Tekla Structural Designer 2020

Tekla Structural Designer

January 2021

Contents

1 Upgrade to this version

When you are ready to upgrade to a new version of Tekla Structural Designer:

1. Read the Tekla Structural Designer 2020 release notes (page 37) to get to know what has changed since the previous version:
2. Check the supported operating system versions and hardware requirements (page 89) for this version.
3. Update the licenses, as explained in Upgrade Tekla Structural Designer to a new version (page 90), or apply a service pack to a previously installed version as explained in Install a Tekla Structural Designer service pack (page 96).

1.1 Tekla Structural Designer 2020 release notes

Welcome to Tekla Structural Designer 2020!

Select your headcode/region from the list below to see the many new features and improvements that apply to you in this version:

US

Issues with Associated Bulletins

- [TSD-6440] - Snow Loading - US Head Code & ASCE 7-2016 - for Roof Projection and Parapet Snow Drift loads to Cl.7.8 of ASCE7-16, the Importance factor value was used directly in the calculation of the drift height h_d rather than its square root per the equation of Figure 7.6-1. This could produce unconservative drift load values for some risk categories. For more information please see: Product Bulletin PBTSD-2003-1.
  - This issue is addressed in this release.

General & Modelling

- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)
- Enhanced Grasshopper Link for Live Parametric Modeling and Analysis (page 42)
- New Beam End Partial Fixity (page 43)
- Improved Selection & Editing and Level Creation at a Point (page 47)

Loading

- Load Combination Generator - enhanced with intelligent generation of default combinations (page 50)
- New Diaphragm Load type - allows application of single lateral and torsion loads to diaphragms (page 51)

Analysis & Results

- Analysis & Design Process - automatic termination when extremely large deflections occur (page 49)
- New Review-View Control of Cracked/Uncracked Setting for Concrete Members (page 53)
- New 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls (page 54)

Design

- Steel Simple Connection Resistance Check - Significantly enhanced with new database of Standard Connections (page 56)
- Concrete Design - Result Line Design saving of reinforcement and manual design forces (page 70)
- Seismic Analysis & Design Enhancements (page 72)

BIM Integration

- New Trimble Connect Integration Link (page 77)
- New Integration with ETABS (page 81)
- New Tekla Open API (page 83)

Other

- Minor Enhancements and Fixes (page 83)

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Eurocode

General & Modelling

- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)

BS

General & Modelling

- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)
- Enhanced Grasshopper Link for Live Parametric Modeling and Analysis (page 42)
- New Beam End Partial Fixity (page 43)
- Improved Selection & Editing and Level Creation at a Point (page 47)

Loading

- New Diaphragm Load type - allows application of single lateral and torsion loads to diaphragms (page 51)

Analysis & Results

- Analysis & Design Process - automatic termination when extremely large deflections occur (page 49)
- New Review-View Control of Cracked/Uncracked Setting for Concrete Members (page 53)
- New 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls (page 54)

Design

- Concrete - Result Line Design saving of reinforcement and manual design forces (page 70)

BIM Integration

- New Trimble Connect Integration Link (page 77)
- New Integration with ETABS (page 81)
- New Tekla Open API (page 83)

Other

- Minor Enhancements and Fixes (page 83)

India

General & Modelling

- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)
- Enhanced Grasshopper Link for Live Parametric Modeling and Analysis (page 42)
- New Beam End Partial Fixity (page 43)
- Improved Selection & Editing and Level Creation at a Point (page 47)

Loading

- Load Combination Generator - enhanced with intelligent generation of default combinations (page 50)
- New Diaphragm Load type - allows application of single lateral and torsion loads to diaphragms (page 51)
- Wind Wizard for Indian Head Code Updated to IS 875 (Part 3) : 2015 (page 51)

Analysis & Results

- Analysis & Design Process - automatic termination when extremely large deflections occur (page 49)
- New Review-View Control of Cracked/Uncracked Setting for Concrete Members (page 53)
- New 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls (page 54)

Design

- Steel Design - New Design of Plated Beam and Columns for the Indian Head Code (page 62)
- Concrete Design - Result Line Design saving of reinforcement and manual design forces (page 70)
- New Design Warning Controls - Indian Head Code (page 76)

BIM Integration

- New Trimble Connect Integration Link (page 77)
- New Integration with ETABS (page 81)
- New Tekla Open API (page 83)

Other

- Minor Enhancements and Fixes (page 83)

Australia

General & Modelling

- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)
- Enhanced Grasshopper Link for Live Parametric Modeling and Analysis (page 42)
- New Beam End Partial Fixity (page 43)
- Improved Selection & Editing and Level Creation at a Point (page 47)

Loading

- New Diaphragm Load type - allows application of single lateral and torsion loads to diaphragms (page 51)

Analysis & Results

- Analysis & Design Process - automatic termination when extremely large deflections occur (page 49)
- New Review-View Control of Cracked/Uncracked Setting for Concrete Members (page 53)
- New 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls (page 54)

Design

- Concrete Design - Result Line Design saving of reinforcement and manual design forces (page 70)
- Foundation Design - New Pile Cap Design for Australian Head Code to AS 3600 : 2009 (page 75)

BIM Integration

- New Trimble Connect Integration Link (page 77)
- New Integration with ETABS (page 81)
- New Tekla Open API (page 83)

Other

- Minor Enhancements and Fixes (page 83)

Compatibility

We suggest that you complete any unfinished models using your current version of Tekla Structural Designer.

This version is not backwards compatible. When you create or save a model in Tekla Structural Designer 2020, you cannot open it in older versions due to database differences.

We suggest that Tekla Structural Designer 2020 is only installed on systems meeting these hardware recommendations (page 89).

If you are upgrading from a version earlier than the latest release 2019i SP3 (version 19.1.3.17 released Jan 2020) you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

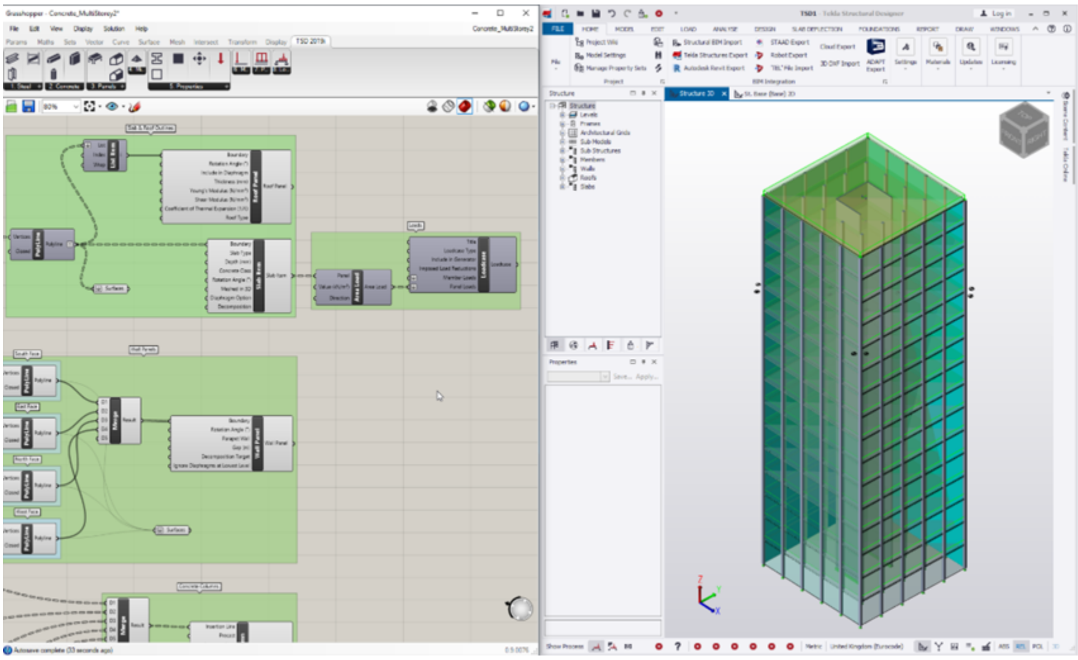
- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Enhanced Grasshopper Link for Live Parametric Modeling and Analysis

The Grasshopper-Tekla Structural Designer (GH-TSD) live link, first released in the 2019i version, enables parametric modelling for Tekla Structural Designer using Rhino (64-bit)/Grasshopper. The link is comprised of a set of Grasshopper components that can create and update objects live in Tekla Structural Designer. For full information on how to get started with the link see the TUA article Grasshopper - Tekla Structural Designer Live Link.

In this release the link is further enhanced with new controls for loading and analysis...

- Note that the Grasshopper Link currently functions for the US Head Code and Eurocode (all Countries/ NA's) Head Code.
- Also see this video demonstrating the link in action.



Related video

Data driven design with Grasshopper

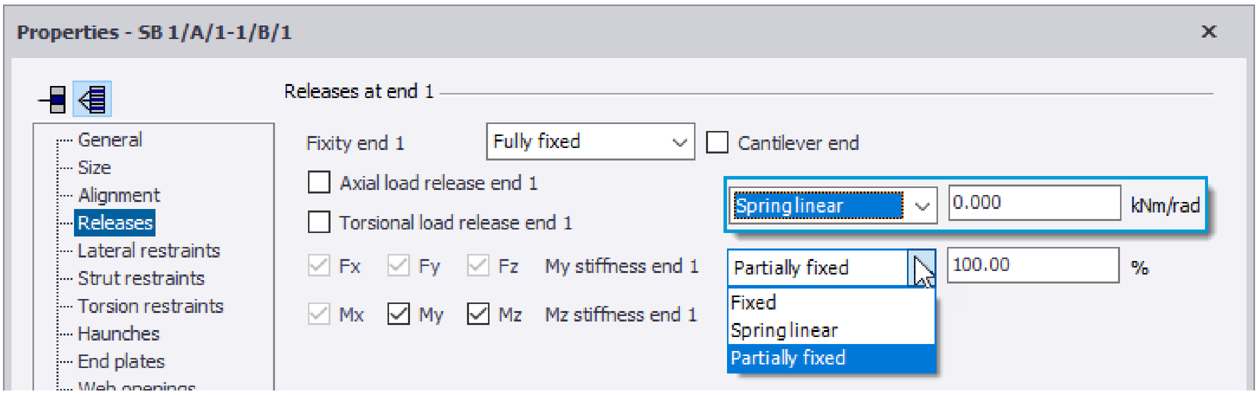
New Beam End Partial Fixity

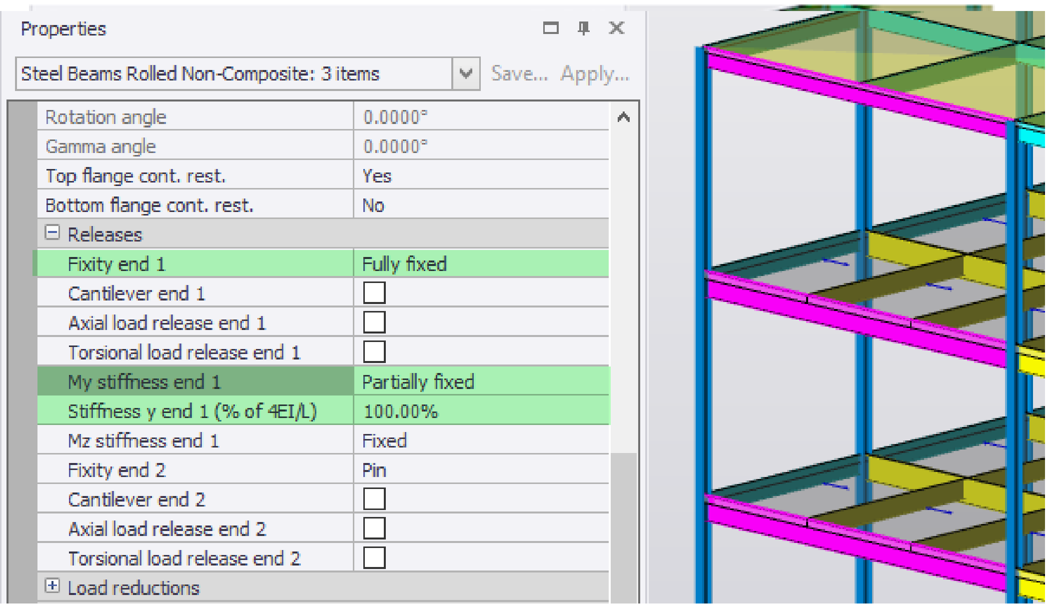
A new partial fixity option is added for beam end fixity. This will be especially useful for example when modelling precast concrete structures in which it is common to to consider some amount of end fixity, but not full end fixity (which is commonly assumed for in-situ concrete buildings).

See this video demonstrating partial fixity in action.

The new end fixity settings are available for review and edit both in the individual member dialog and the Properties Window for selected beam(s) as shown in the picture below.

- To apply the new end fixity setting, first set the end Fixity to “Fully fixed” to enable the new fixity options of “Spring linear” and “Partially fixed”.
    - For “Spring linear” a value of stiffness in moment/rad is directly entered.
    - For “Partially fixed” a % value is entered. Note that this is the % of the member flexural stiffness of 4EI/L (not of full fixity), the value of which is automatically calculated and applied in the analysis model.

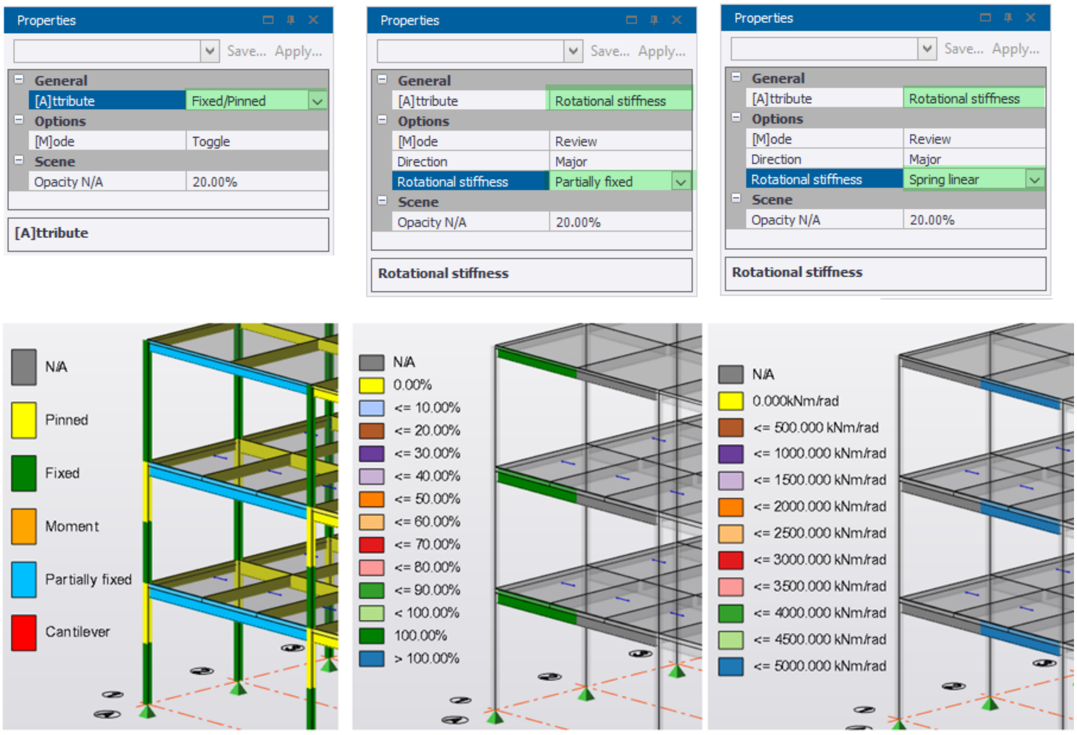




End fixity settings can also be applied and verified efficiently via Review View > Show/Alter State, using the new Rotational Stiffness (page 890) Attribute.

- Use Set mode to rapidly apply or edit the % Partial fixity and Rotational stiffness values.

- Use Review mode, to display the values in an automatic color-coded range as shown below.

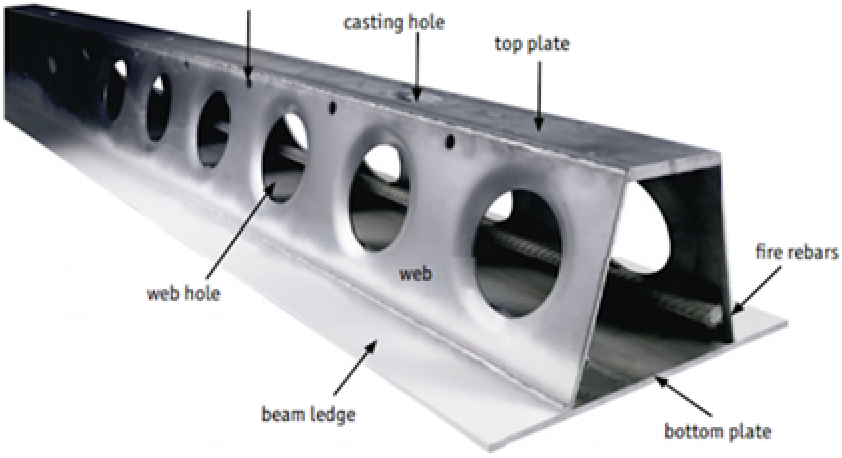


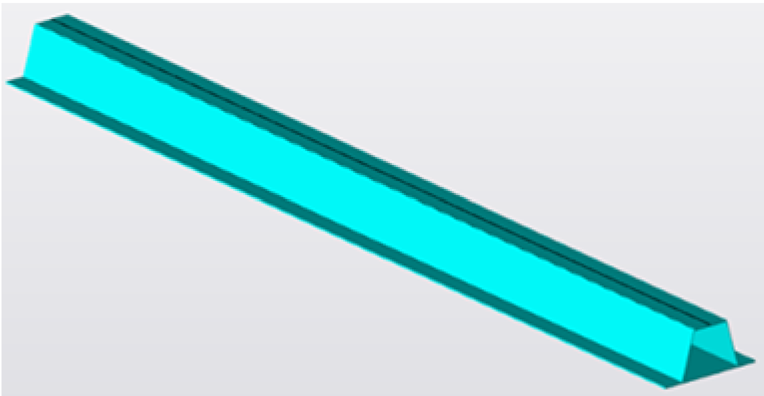
Related video

Automatic modelling of partial fixity

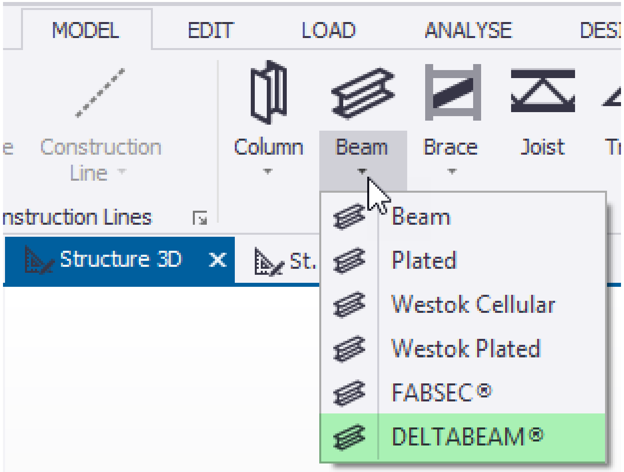
Steel Beams - New Section Database of Peikko DELTABEAM beams

DELTABEAM (page 411)® beams manufactured by Peikko in Finland are now added to the Steel section databases for the Nordic countries of Finland, Sweden and Norway and the general Europe database. This proprietary beam type is typically used in long span. slim floor systems in which they are cast into and act compositely with the floor slab. With this addition, floors of this type can now be modelled and analysed in Tekla Structural Designer.

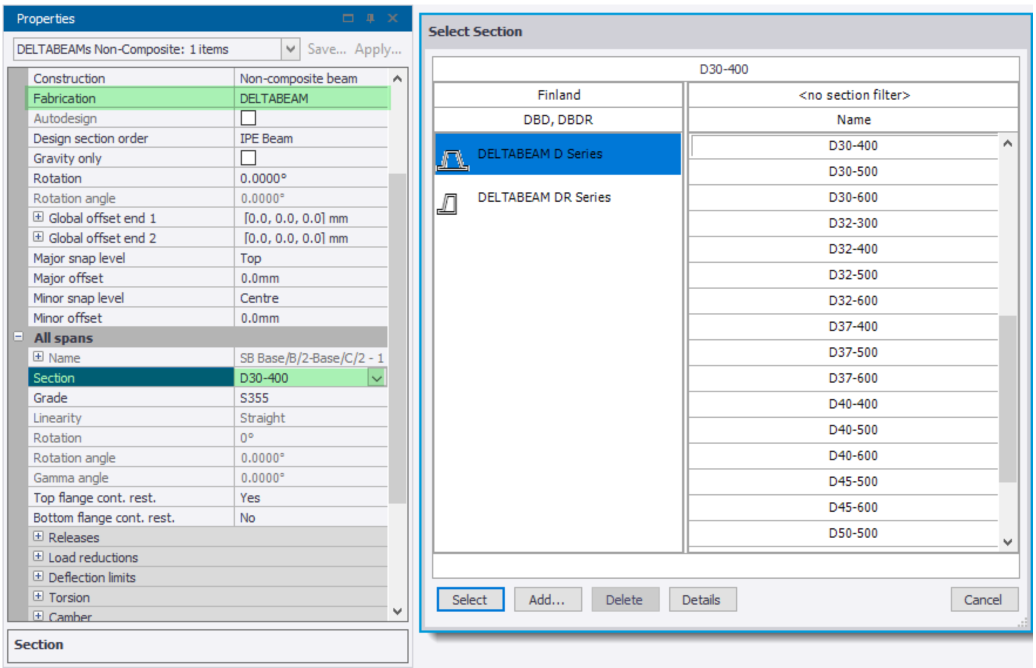




- A new dedicated DELTABEAM (page 411)® option is added to the droplist of Steel Beam types on the Model Ribbon. When working from the Properties Window, set the Fabrication to DELTABEAM® to apply the new sections as shown below.



- Two series of DELTABEAM® sections are then available for selection; *D-type* internal and *DR-type* edge. Currently only the section properties required for analysis are defined in the database hence there is no design for them within Tekla Structural Designer. A report of design forces can be produced for external design.



Related video

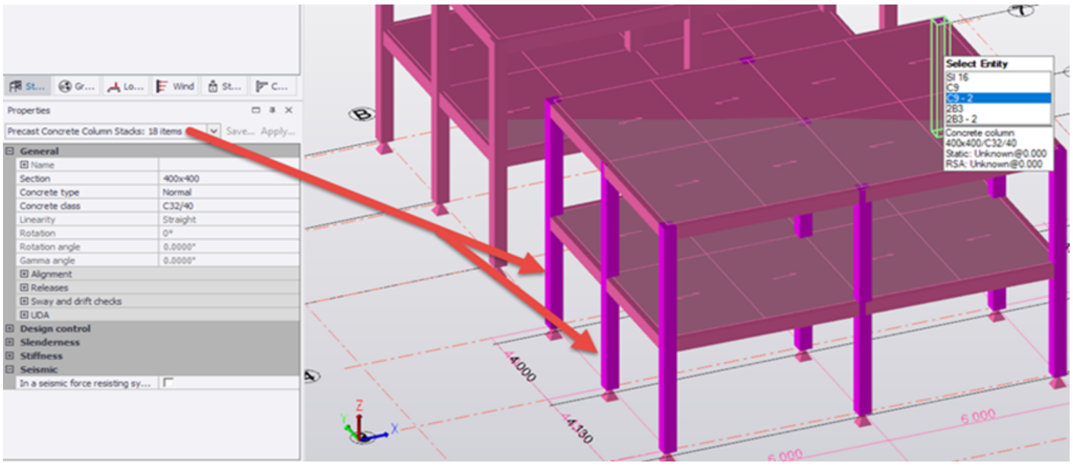
Easy modeling of Peikko Deltabeams

Improved Selection & Editing and Level Creation at a Point

These ease of use enhancements are made in Tekla Structural Designer 2020 to improve the effectiveness and efficiency of modeling.

Selection and Editing

- Selected objects are now highlighted in two colours identifying which are Active (type selected in property grid) and Remaining. This helps clarify the case where a single span/stack of a member is the active selection.
- Additionally, the Tooltip and member selection defaults to objects of the Active type first, making it much easier to focus work on a given object type.

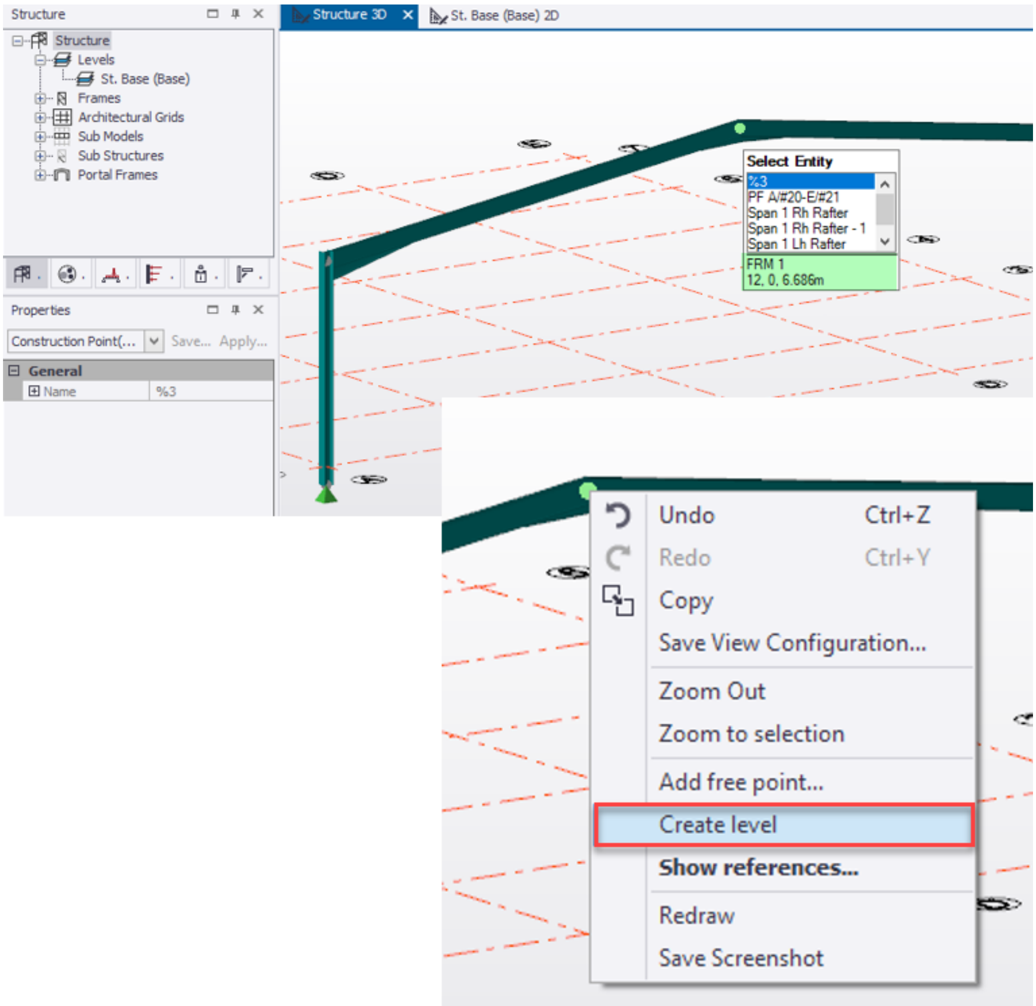


Level Creation at a point

A construction point can now be selected and a level created at its elevation. This is particularly useful for example for creating a level at the apex of a Portal Frame or at a point along an inclined plane. This enhancement adds the following information and capabilities:

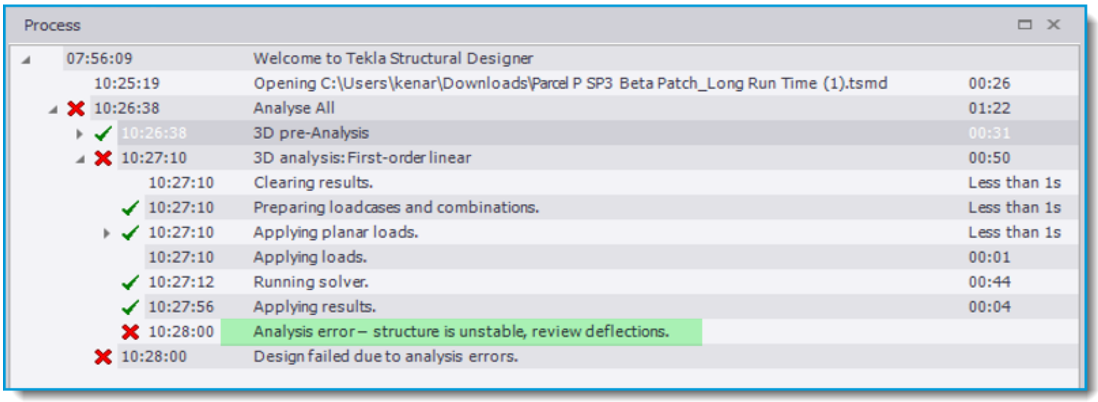
- Construction Points can now be selected, including Free points (note that "points" must be active in scene content).
- The following information is displayed in the cursor Tooltip for the selected point:
    - The associated plane.
    - The Global coordinates of the point (to the precision set in Home > Model Settings > Units).

- When you right click on a point the context menu now includes the option to “Create Level” for the point elevation (if no level exists).



Analysis & Design Process - automatic termination when extremely large deflections occur

[TSD-3399] The Analysis & Design Process will now terminate automatically when extremely large deflections* result from the First Order Linear 3D Building Analysis (which is always at the beginning of any Analysis & Design process). This will prevent the process continuing to higher order forms of analysis - such as non-linear - and the design phase, which may expend considerable processing resources and time running potentially into several hours before the process ultimately fails. This will prove to be a waste of time if the model contains errors which need to be addressed before running design. A typical example of the new termination and the message issued in this case is shown in the picture below.



*Such extremely large deflections are usually the result of *mechanisms* in the model - from for example incorrect connectivity or too many releases etc. We recommend these are **always** investigated and resolved before attempting Design. See the following TUA articles for guidance on what to do in this case; Guide to resolving Mechanisms and (N-1) rule for the number of pins to apply at a node where N members meet.

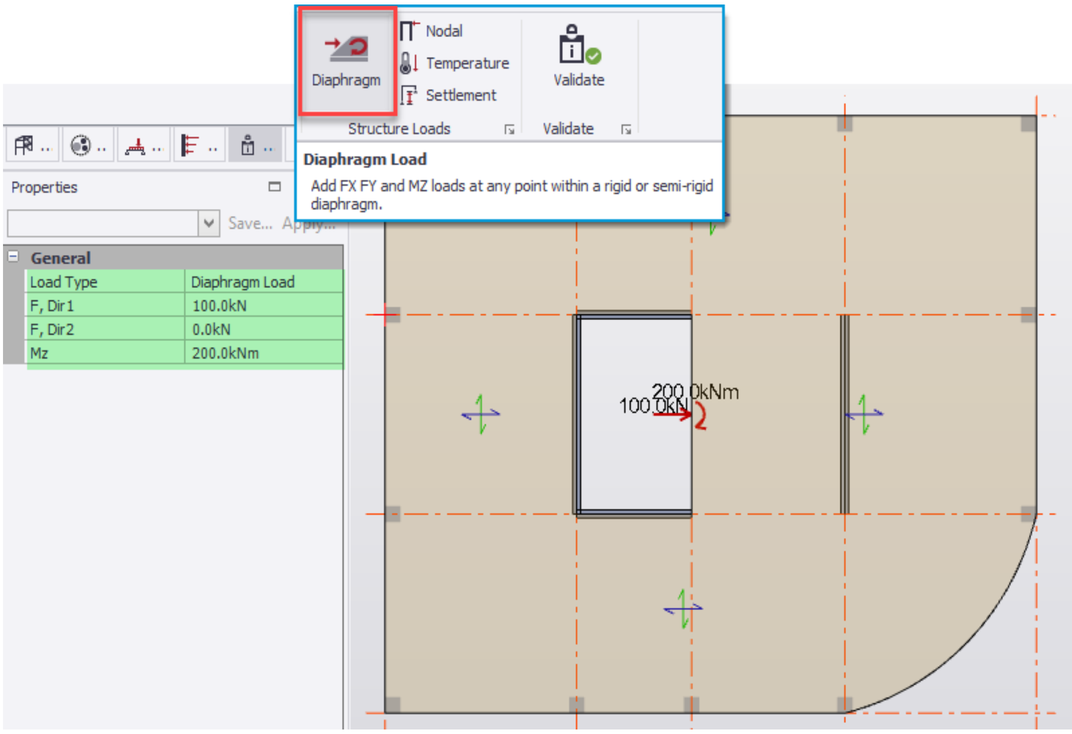
Load Combination Generator - enhanced with intelligent generation of default combinations

[TSD-3624] From investigation and customer feedback on the use of previous releases, it was found that many engineers were simply accepting the combination generator defaults which could produce many unnecessary combinations. Additional intelligence is now built into the Load Combination Generator to produce fewer combinations by default, appropriate to the content and design settings of the model. Since the Analysis and Design process's speed and duration are directly related to the number of combinations (potentially geometrically), this could significantly improve performance.

- For example for the US Head Code, ASD combinations* are not required in general where the LRFD code has been selected for steel design. Not generating the ASD combinations can halve the total number. Such superfluous combinations will no longer be generated by default in this circumstance potentially significantly improving performance.
  - *Note that ASD combinations are NOT required for foundation design for US Head Code models.

New Diaphragm Load type - allows application of single lateral and torsion loads to diaphragms

To assist with applying loads of a single value of lateral and torsion load per floor in a building, a new Diaphragm load (page 547) tool is added. Such loads may be needed for example for tall/ atypical structures requiring wind tunnel testing for wind loading which is then provided as total lateral and torsion level loads.

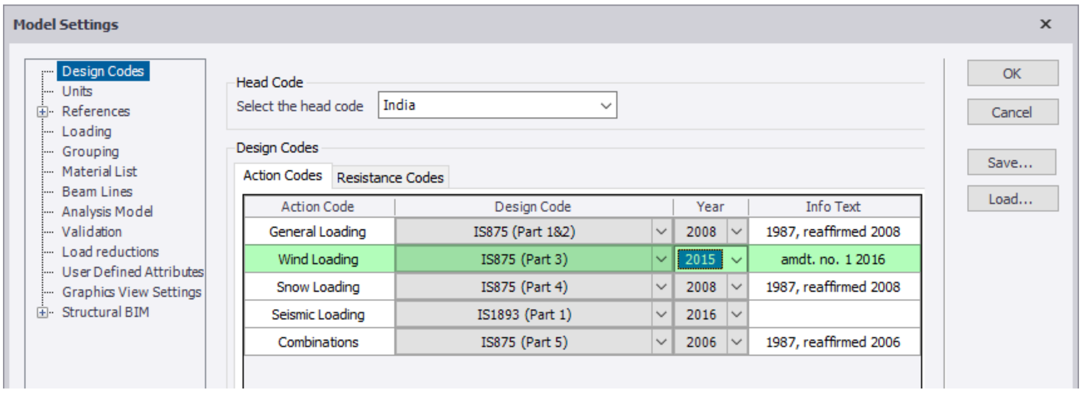


Related video

Diaphragm loading

Wind Wizard for Indian Head Code Updated to IS 875 (Part 3) : 2015

Following popular requests, the Wind Wizard for the Indian Head Code has been fully revised and updated to the new IS 875 (Part 3) : 2015 (incorporating amdt. no.1 April, 2016).

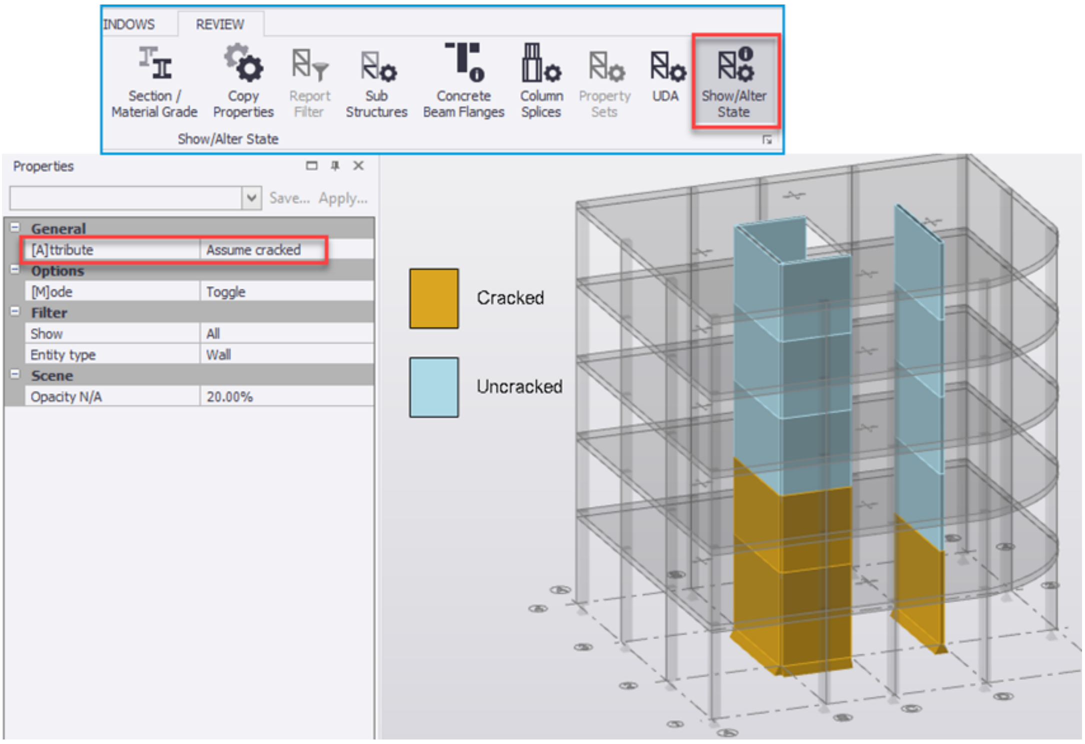


- The new 2015 code year option is selected in the Design Codes > Action Codes settings as shown in the picture above.
- With this selected the Wind Wizard determined pressures are calculated considering all the relevant and newly introduced design factors of the 2015 code version.
- The update allows cyclonic design effects to be considered for structures situated on the Indian east and west coasts (Gujarat coastal region).
- Key aspects of the Wind code update are:
  - Classification of building class (A, B and C) based on structure's greatest horizontal or vertical dimension (exposed to wind) as per Cl.5.3.2.2 (of IS 875 (Part 3) - 1987 (reaff.2003) has been omitted in the new code.
  - A new importance factor for cyclonic region k4 (Clause 6.3.4) is introduced for the wind pressure determination.
  - For the design wind pressure determination the following new factors are introduced:
    - Kd - wind directionality factor (Clause 7.2.1)
    - Ka - area averaging factor (Clause 7.2.2)
    - Kc - combination factor (Clause 7.3.3.13)
  - Design wind speed is now (Vz) = Vb x k1 x k2 x k3 x k4; wind pressure pz = 0.6 x Vz2 and design wind pressure (Pd) = Kd x Ka x Kc x pz
  - Certain values in the table for pressure coefficient, figures for factors associated with cliff and escarpment and ridge and hill topography have been updated.

New Review-View Control of Cracked/Uncracked Setting for Concrete Members

NOTE This feature goes together with the New 2D In-Plane Stress Contouring (page 54) to facilitate more rapid assessment and control of the appropriate analysis properties to use for concrete buildings.

You can now graphically modify the assume cracked property (page 875) for all concrete elements via Show\Alter State.



NOTE The assumed cracked property controls the modification factors used for the concrete members in analysis - for more on this topic see: Where do the default values for cracked and uncracked properties of concrete come from?

- This should prove a significant time saver for those spending time making / applying decisions on what is cracked.
    - The Mode options allows switching between Toggle and Set Cracked or Uncracked - objects can then be clicked or windowed to edit the setting.

- Additionally the Entity type can be set to restrict editing to only certain object types; Members, Walls, Beams or Columns.

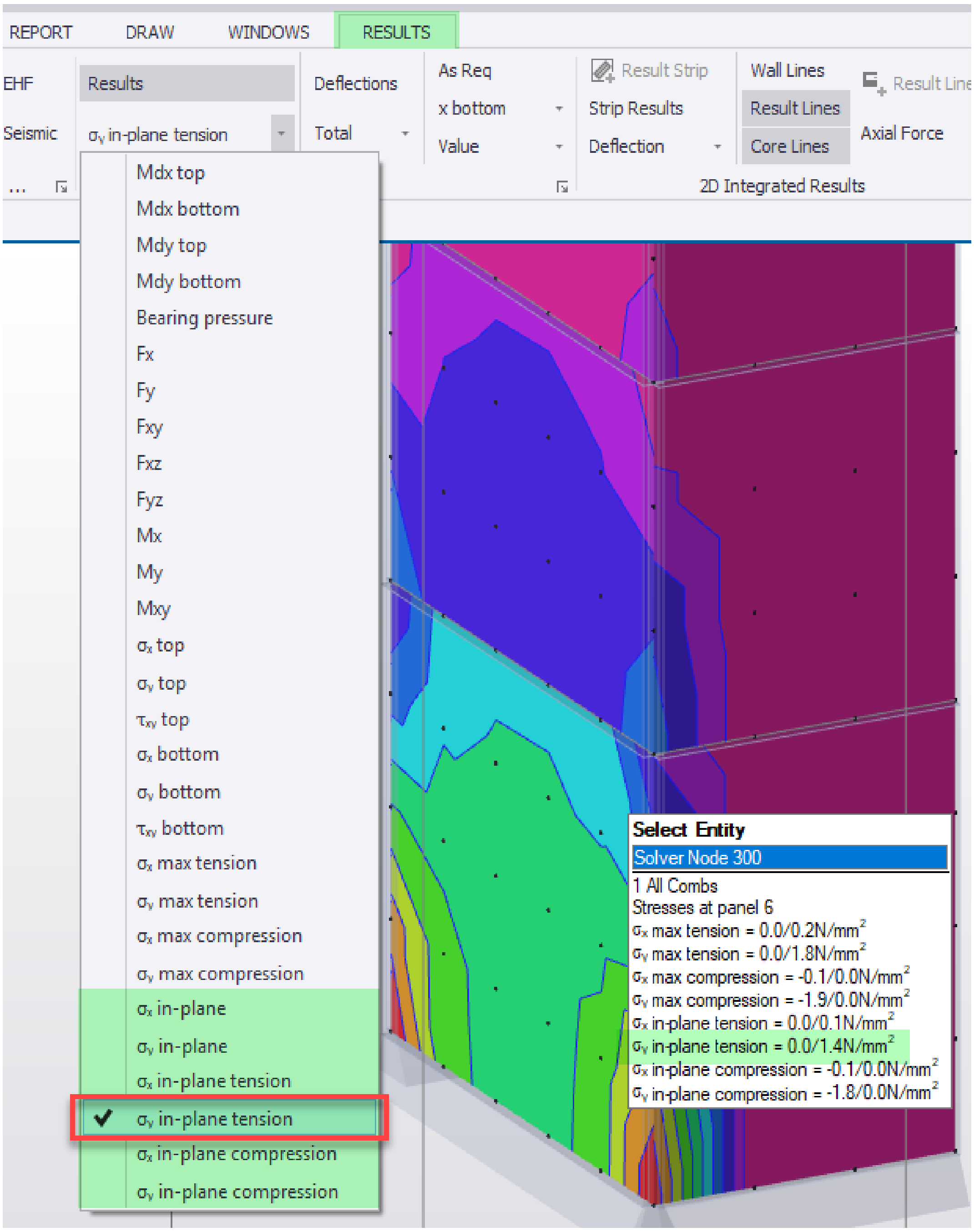
See also

Concrete member cracked or uncracked status (page 1286)

New 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls

New 2D stress contours are added to the existing options giving the in-plane stresses in 2D element meshes.

- The in-plane stress is in effect the average stress through the thickness of the element, which we think will be of greater interest to more engineers especially those wanting to assess the extent of cracking in concrete walls. For this purpose the (vertical) “σ_y in-plane tension” result will be of most use.
- This will be the case especially when the engineer creates envelopes, which will make it very easy to see panels with zero tension (which could then be set to uncracked).
- This feature goes together with the New Review-View Control of Cracked/ Uncracked Setting for Concrete Members (page 53) to facilitate more rapid assessment and control of the appropriate analysis properties to use for concrete buildings. The user can have two side-by-side views open to simultaneously review the stress contour results via one and review/edit the Assume cracked setting via the other.



See also

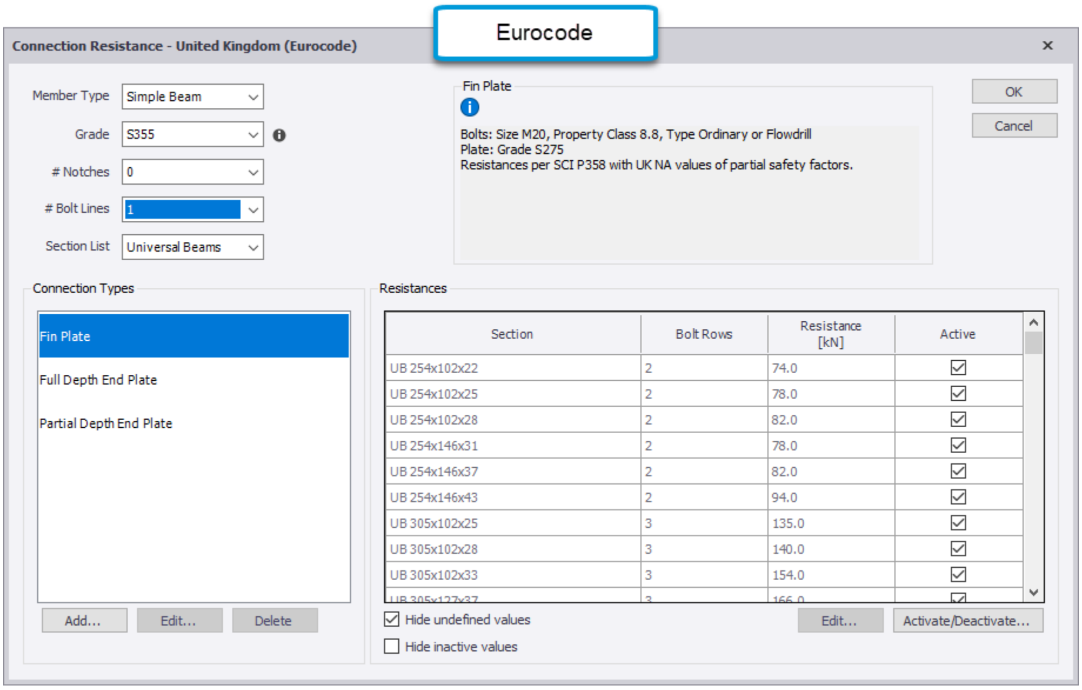
Concrete member cracked or uncracked status (page 1286)

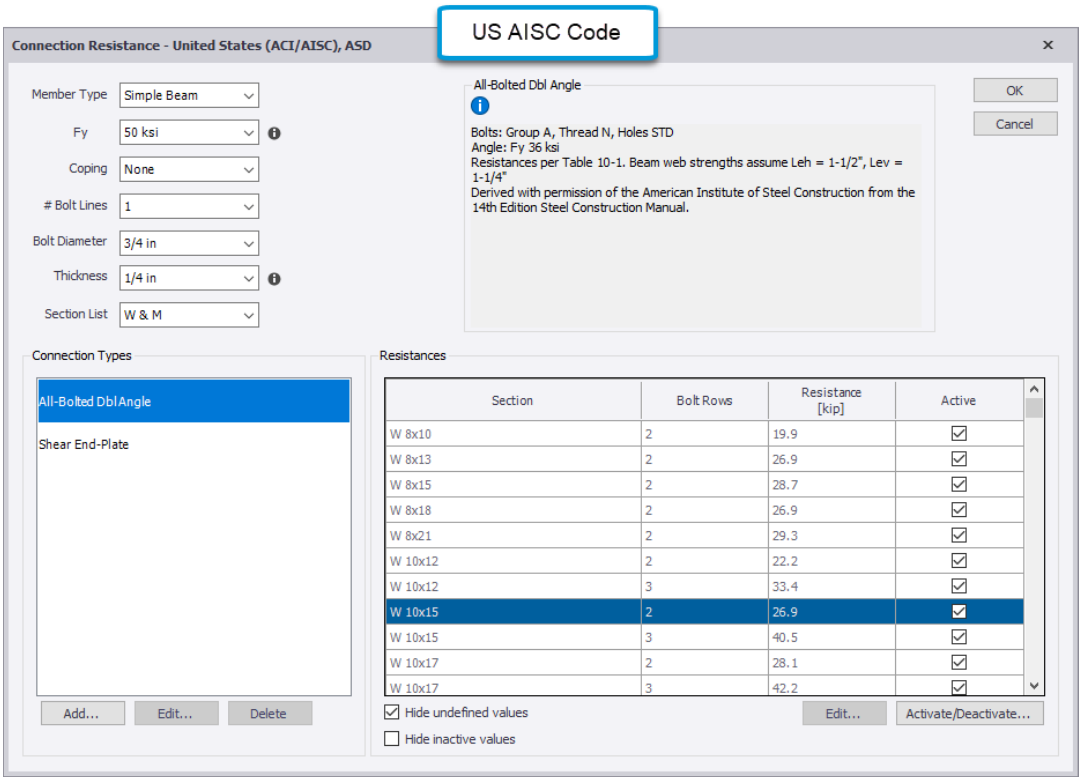
Connection Resistance Check - Significantly Enhanced with new database of Standard Simple Connections

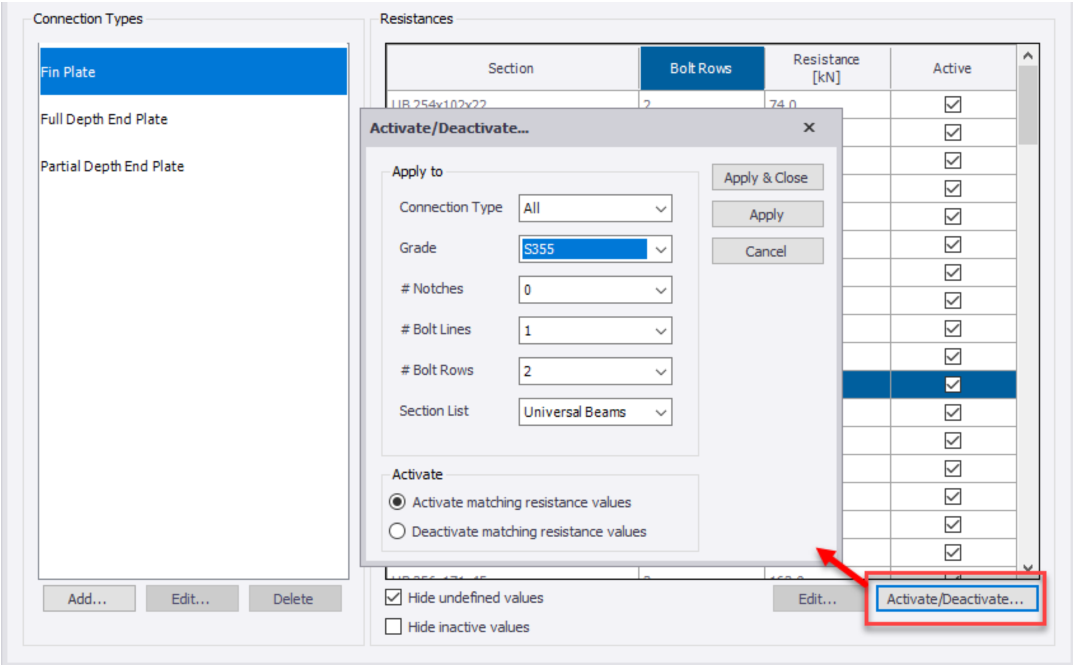
The Connection Resistance check is a quick and simple way of checking the capacity of Simple Connections in beams (major axis shear) and braces (axial compression or tension). It is a long standing feature which was enhanced in release 2018i to allow definition of any number of connection types and resistances. For this release it has been completely overhauled and significantly enhanced, improving the ability for engineers to identify, early in the design process, potential issues with forming standard simple connections.

Details of the enhancements are:

- **Pre-defined Resistances database (page 1009)** - the existing functionality is retained, but now includes a large in-built database of pre-defined resistances for the Eurocode and US Head Codes in new viewer format which includes significantly more detail about the connection configuration.
- **New Connection Resistance dialog (page 2399)** - as shown in the pictures below, the new viewer now separates Beams and Braces and has multiple new controls to define the configuration of the connection in terms of; Steel Grade, Number of Notches, Bolt Lines and Rows and, for the US Code, plate thickness and bolt sizes.
    - The engineer can now also control the connection types and individual resistances to be considered by the check using the 'Active' checkbox in the Resistances Table. Multiple rows can be selected and Inactivated/ Activated simultaneously in the usual manner.
        - Additionally a number of criteria can be set to automatically filter the active list using the **Activate/Deactivate...** dialog (see picture below)
    - New Connection Types and Resistances can also be added.
    - Existing resistances defined in previous releases will be imported. However further action is required for these - please see the Install the upgrade (page 91) section for what to do.


Eurocode
Connection Resistance - United Kingdom (Eurocode)
Member Type Simple Beam
Grade S355
Notches 0
Bolt Lines 1
Section List Universal Beams
Fin Plate
Bolts: Size M20, Property Class 8.8, Type Ordinary or Flowdrill
Plate: Grade S275
Resistances per SCI P358 with UK NA values of partial safety factors.
OK
Cancel
Connection Types
Fin Plate
Full Depth End Plate
Partial Depth End Plate
Add... Edit... Delete
Resistances
Section Bolt Rows Resistance [kN] Active
UB 254x102x22 2 74.0
UB 254x102x25 2 78.0
UB 254x102x28 2 82.0
UB 254x146x31 2 78.0
UB 254x146x37 2 82.0
UB 254x146x43 2 94.0
UB 305x102x25 3 135.0
UB 305x102x28 3 140.0
UB 305x102x33 3 154.0
Hide undefined values
Hide inactive values
Edit... Activate/Deactivate...


US AISC Code
Connection Resistance - United States (ACI/AISC), ASD
Member Type Simple Beam
Fy 50 ksi
Coping None
Bolt Lines 1
Bolt Diameter 3/4 in
Thickness 1/4 in
Section List W & M
All-Bolted Dbl Angle
Bolts: Group A, Thread N, Holes STD
Angle: Fy 36 ksi
Resistances per Table 10-1. Beam web strengths assume Leh = 1-1/2", Lev = 1-1/4"
Derived with permission of the American Institute of Steel Construction from the 14th Edition Steel Construction Manual.
OK
Cancel
Connection Types
All-Bolted Dbl Angle
Shear End-Plate
Add... Edit... Delete
Resistances
Section Bolt Rows Resistance [kip] Active
W 8x10 2 19.9
W 8x13 2 26.9
W 8x15 2 28.7
W 8x18 2 26.9
W 8x21 2 29.3
W 10x12 2 22.2
W 10x12 3 33.4
W 10x15 2 26.9
W 10x15 3 40.5
W 10x17 2 28.1
W 10x17 3 42.2
Hide undefined values
Hide inactive values
Edit... Activate/Deactivate...



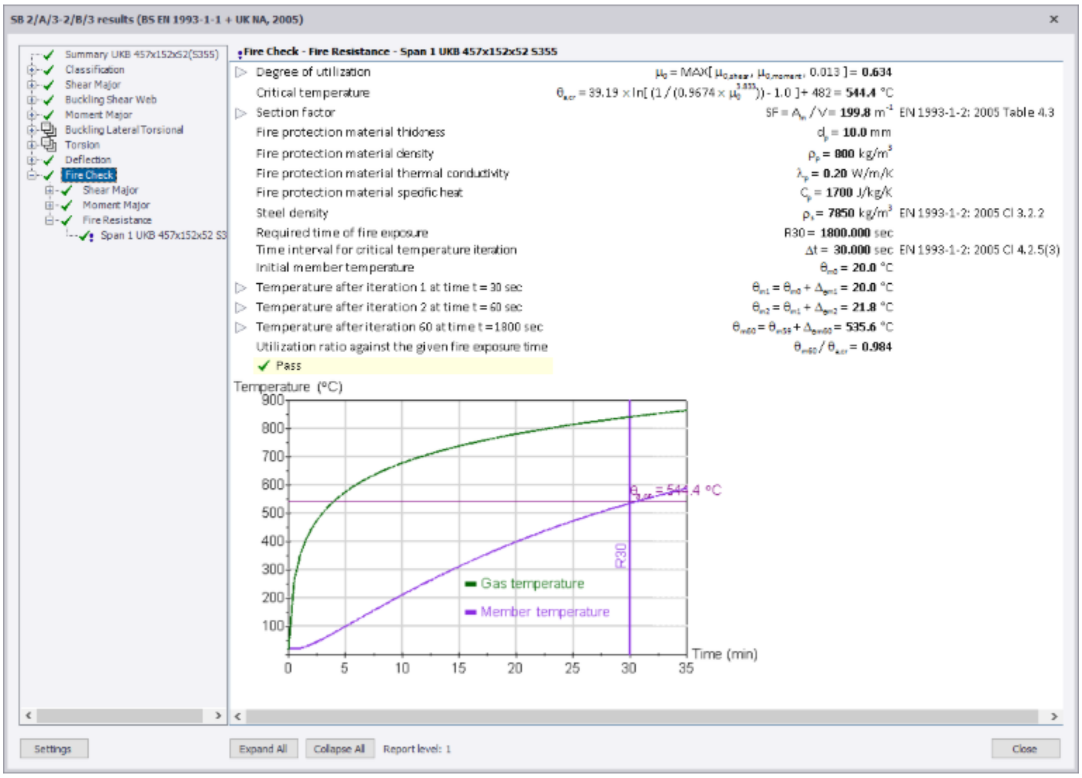
- **Enhanced Check (page 809)** - as well as the new database of pre-defined resistances and additional connection details, the check itself has also been enhanced and refined:
  - Now both (pin) ends of the beam are checked, with the gravity shear now reported as a positive value for the Rh end of beam
  - Optimisation - the check now loops through active Bolt Lines & Rows and reports first passing and last failing resistance (applies to both pre and user-defined resistances)
  - A warning is now issued where uplift occurs (since the connection resistance is not checked for this condition)
- **Report** - a new “Connection Resistance” report is added to the list of available default reports which contains all the available Connection Resistance report items (Beams and Braces for both Steel and Cold Formed).

Related video

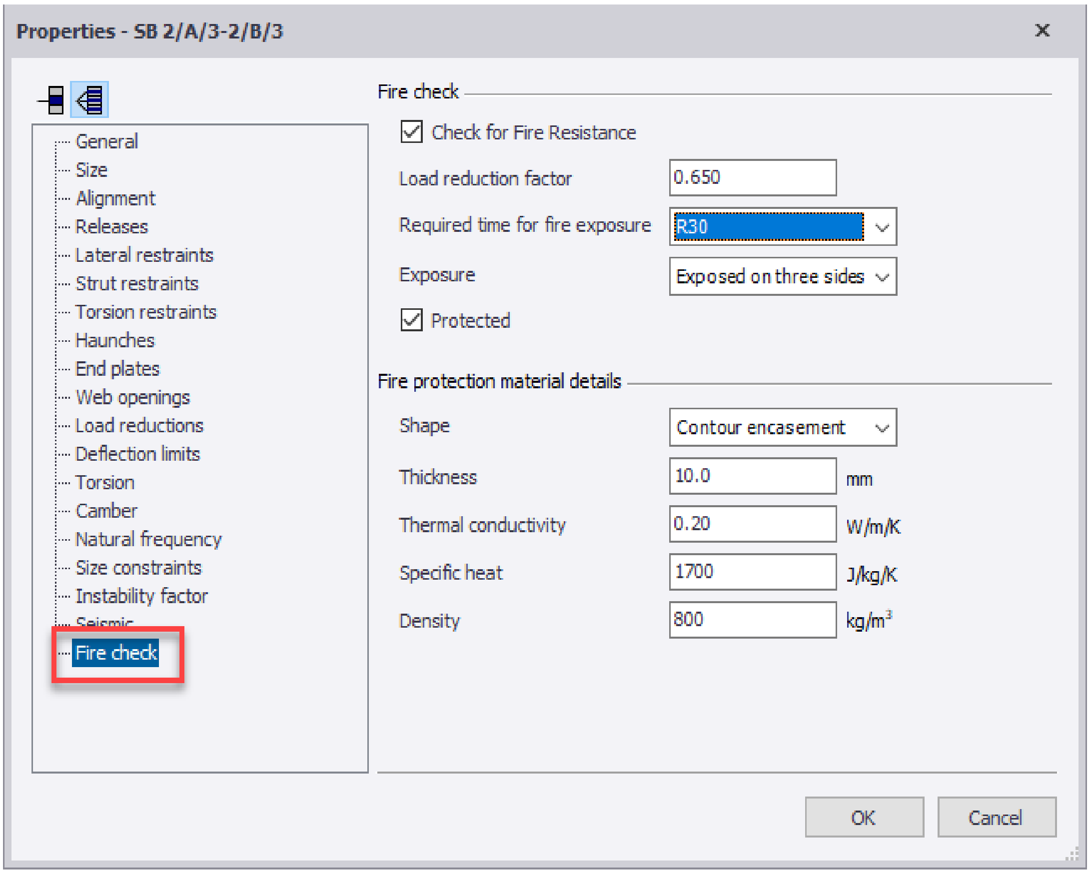
Predefined connection resistance database for Eurocode and AISC

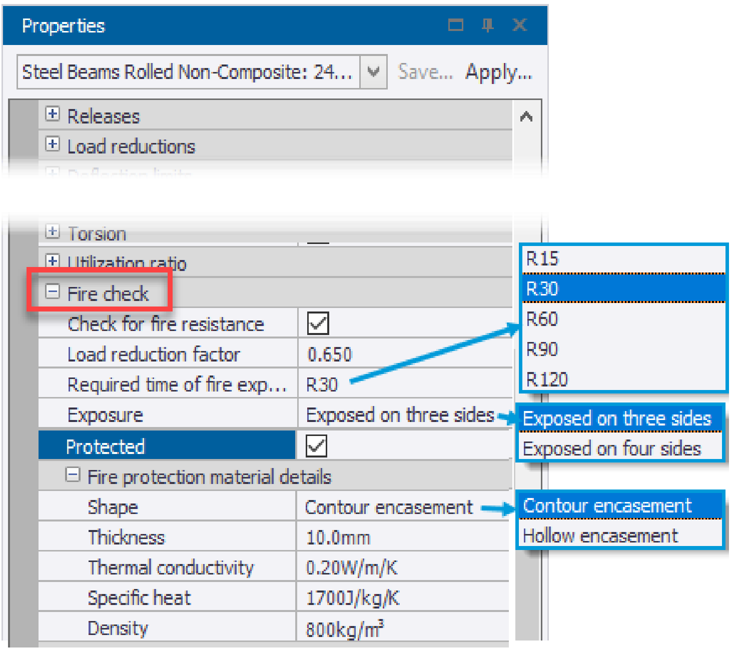
Steel Design - New Fire Resistance Check to Eurocode EN 1993-1-2, Cl 4.2.4 - All NA’s

A new simple to use Steel Fire Resistance design check is added in this release in accordance with EN 1993 & the National Annex for; the UK, Ireland, Singapore, Malaysia, Sweden, Norway, Finland or the recommended Eurocode values. The check is carried out in the temperature domain by using the critical temperature method per EN 1993-1-2, Cl 4.2.4. Detailed, transparent and comprehensive design details are reported, including the Temperature-Time relationship graph per EN 1991-1-2 Cl.3.1 (10) as shown below.



- The design calculation checks the mechanical resistance of a single steel member in case of fire for a required time of exposure which is set by the engineer in the check properties.
- The Fire check parameters can be set in the member dialog for a single member and are also listed in the Properties Windows, as shown in the picture below.
  - There is an additional setting in Design > Settings > Fire check for the time interval for critical temperature iteration.
- The steel member may be unprotected (bare steel) or protected and may be exposed on 3 (under a slab) or 4 sides.
  - Specified protection can be Contour or Hollow encasement for which the engineer then specifies the; thickness, thermal conductivity, specific heat and density.
- The design scope is as follows:
  - Beam characteristic (Non-composite) with fabrication type of Rolled
  - Single span, pin-ended and restrained beam members subjected to Major axis bending only.
  - The check is applied only to gravity combinations and is a check only - i.e. is not considered as an Auto Design criteria.

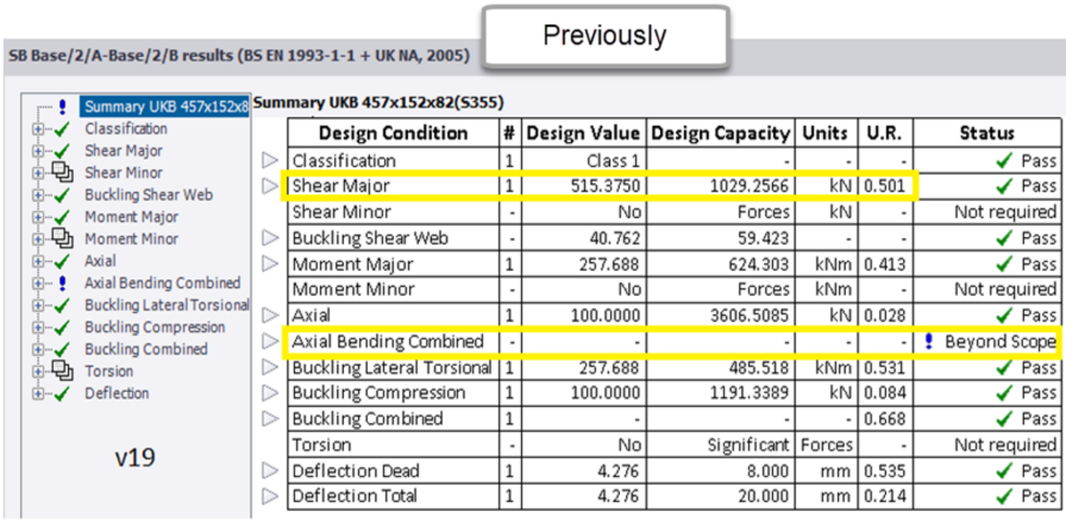

Properties - SB 2/A/3-2/B/3
General
Size
Alignment
Releases
Lateral restraints
Strut restraints
Torsion restraints
Haunches
End plates
Web openings
Load reductions
Deflection limits
Torsion
Camber
Natural frequency
Size constraints
Instability factor
Seismic
Fire check
Fire check
Check for Fire Resistance
Load reduction factor
0.650
Required time for fire exposure
R30
Exposure
Exposed on three sides
Protected
Fire protection material details
Shape
Contour encasement
Thickness
10.0
mm
Thermal conductivity
0.20
W/m/K
Specific heat
1700
J/kg/K
Density
800
kg/m³
OK
Cancel


Properties
Steel Beams Rolled Non-Composite: 24...
Save...
Apply...
Releases
Load reductions
Torsion
Utilization ratio
Fire check
Check for fire resistance
Load reduction factor
0.650
Required time of fire exp...
R30
Exposure
Exposed on three sides
Protected
Fire protection material details
Shape
Contour encasement
Thickness
10.0mm
Thermal conductivity
0.20W/m/K
Specific heat
1700J/kg/K
Density
800kg/m³
R15
R30
R60
R90
R120
Exposed on three sides
Exposed on four sides
Contour encasement
Hollow encasement

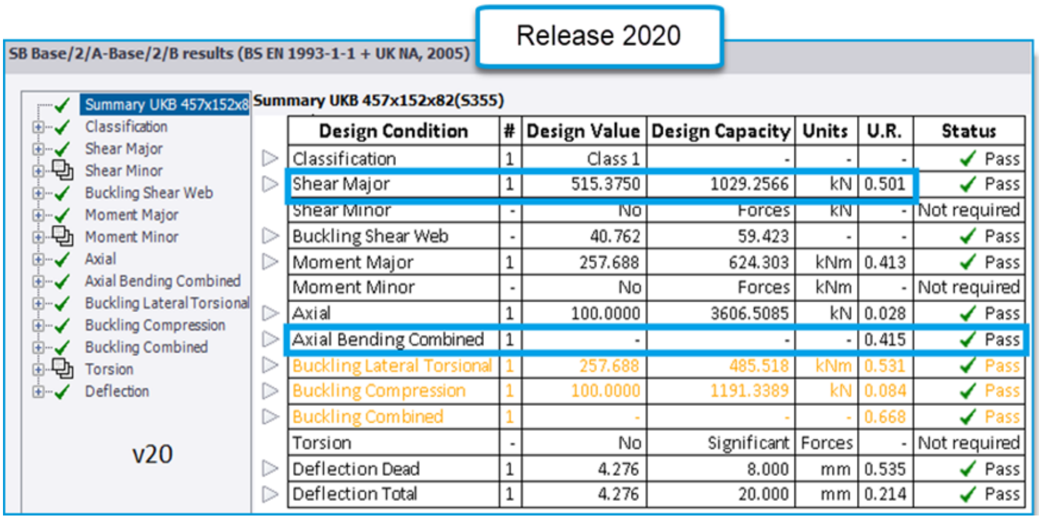
Steel Design - New Design for High Shear in Combined Bending & Axial Check - Eurocode EC3 All NA's

Previously beyond scope in steel beam design, design for high shear in the combined bending and axial check is now checked. The new calculations have been based on Eurocode EC3 part 1 and also with reference to the research paper '*Resistance of steel cross-sections subjected to bending, shear and axial forces*' by Jerzy Goczek and Lukasz Supel, Engineering Structures 70 (2014) 271-277, Elsevier.

- No additional input is required for the new check - valid beam members which satisfy the load condition requirements will automatically be designed for high shear where this exists. The new check is considered in the autodesign routine thus sections failing the check will not be selected and further section sizes will be attempted.
- The scope of the new check is as follows:
    - Members: doubly symmetric rolled or plated I section beams
    - Loading: uniaxial major bending and axial force with major axis high shear is the only condition for which high shear will be designed
        - Note the presence of torsion force (above the ignore torsion force level) will not be a valid load condition for the high shear check, regardless of whether 'check for torsion' is ticked on or off, and will result in Beyond Scope status
        - Note that where the shear web buckling exceeds 50% of capacity, this will not be a valid condition for the high shear check, hence will result in Beyond Scope status.
- A new warning will be displayed in each of the three buckling checks when high shear is present (including minor axis high shear and for non-valid members and loading conditions). This states that buckling design is assumed to be unaffected by the presence of high shear (which is not explicit in EC3 but mirrors the statement in BS5950 clause 4.8.1).



Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
Classification	1	Class 1	-	-	-	Pass
Shear Major	1	515.3750	1029.2566	kN	0.501	Pass
Shear Minor	-	No	Forces	kN	-	Not required
Buckling Shear Web	-	40.762	59.423	-	-	Pass
Moment Major	1	257.688	624.303	kNm	0.413	Pass
Moment Minor	-	No	Forces	kNm	-	Not required
Axial	1	100.0000	3606.5085	kN	0.028	Pass
Axial Bending Combined	-	-	-	-	-	Beyond Scope
Buckling Lateral Torsional	1	257.688	485.518	kNm	0.531	Pass
Buckling Compression	1	100.0000	1191.3389	kN	0.084	Pass
Buckling Combined	1		-	-	0.668	Pass
Torsion	-	No	Significant	Forces	-	Not required
Deflection Dead	1	4.276	8.000	mm	0.535	Pass
Deflection Total	1	4.276	20.000	mm	0.214	Pass

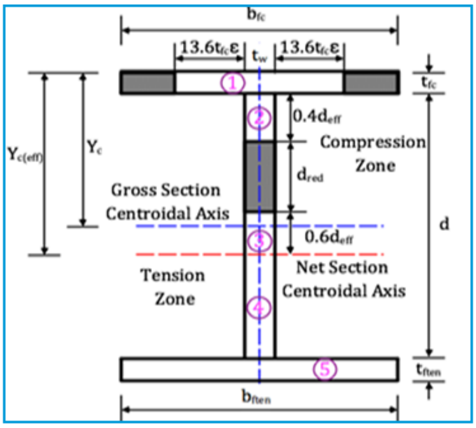


Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
Classification	1	Class 1	-	-	-	Pass
Shear Major	1	515.3750	1029.2566	kN	0.501	Pass
Shear Minor	-	No	Forces	kN	-	Not required
Buckling Shear Web	-	40.762	59.423	-	-	Pass
Moment Major	1	257.688	624.303	kNm	0.413	Pass
Moment Minor	-	No	Forces	kNm	-	Not required
Axial	1	100.0000	3606.5085	kN	0.028	Pass
Axial Bending Combined	1	-	-	-	0.415	Pass
Buckling Lateral Torsional	1	257.688	485.518	kNm	0.531	Pass
Buckling Compression	1	100.0000	1191.3389	kN	0.084	Pass
Buckling Combined	1	-	-	-	0.668	Pass
Torsion	-	No	Significant	Forces	-	Not required
Deflection Dead	1	4.276	8.000	mm	0.535	Pass
Deflection Total	1	4.276	20.000	mm	0.214	Pass

Steel Design - New Design of Plated Beam and Columns for the Indian Head Code

Expanding the design capabilities for India, steel plated beams and columns can now be designed to IS 800 : 2007.

The new design capability caters for both checking and autodesign.



Moment Major - 1 LS$_1$-1.5D+1.5L+1.5Lr - Stack 1 (4.000) PC 750x300/15x112.3 Fe 410 WA - Position 4.000

Major Axis		
Factored applied moment, M_z	= **-5.58** kNm	
Factored applied axial force, P	= **176.28** kN	
Shift in centroid, $e_{N(z)}$	= **57** mm	
Additional design moment for slender section, $M_{z,add}$	$M_{z,add} = P \times e_{N(z)}$ = **10.08** kNm	
△ Total design moment for slender section design, $M_{z,total}$	$M_{z,total} = M_z + M_{z,add}$ = **15.66** kNm	
Partial safety factor, γ_{m0}	= **1.100**	IS 800 : 2007 Clause 5.4.1 Table 5
Major axis effective elastic section modulus, $Z_{eff,z}$	= **2891.56** cm^3	
Yield strength, f_y	= **250.00** N/mm^2	
Classification	Class 4	
Design bending strength, M_{dz}	$M_{dz} = Z_{eff,z} \times f_y / \gamma_{m0}$ = **657.17** kNm	IS 800 : 2007 Clause 8.2.1.2
Ratio	$M_{z,total} / M_{dz}$ = **0.024**	
✓ Pass		

Key aspects of the new design are:

- In developing the design process, reference has been made to BS 5950-1: 2000 and EN 1993-1-1 : 2005 for some aspects of design not covered in IS 800 : 2007.
- The scope of the new check is as follows;
    - Beam characteristic (Non-composite) and Column characteristic with fabrication type of Plated and Plated I sections only.
    - Design of Class 4 slender sections is supported for axial compression and for Flexure.
    - Moment Minor + High shear is beyond scope as is Slender section + High shear.
    - Connection details between the flange plate and web plate are not considered.

New Precast Concrete Member modeling and design via Tekla Tedds 2020 Integration

In an exciting development, integration with Tekla Tedds has been thoroughly enhanced to allow the efficient design of precast concrete beams (page 1540) and columns (page 1554) using the new in Tedds 2020 precast concrete member design calculations.

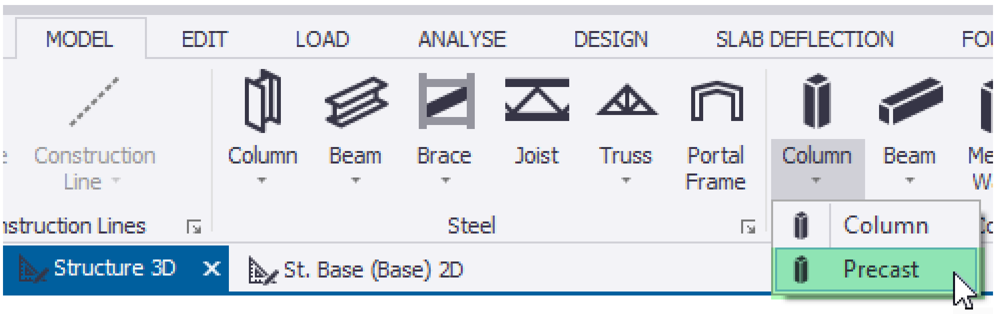
See the feature in action in this video.

NOTE Use of this feature requires an installation of and license for Tekla Tedds 2020.

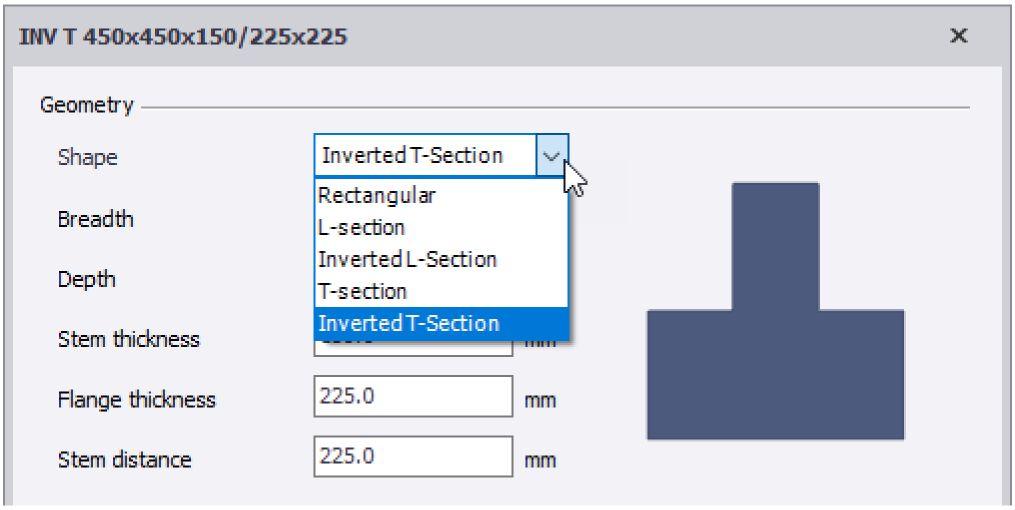
The workflow (page 1528) fully supports multiple analysis and design loops and even Design Groups (when these are enabled, which is the default), just as for in-situ concrete design within Tekla Structural Designer.

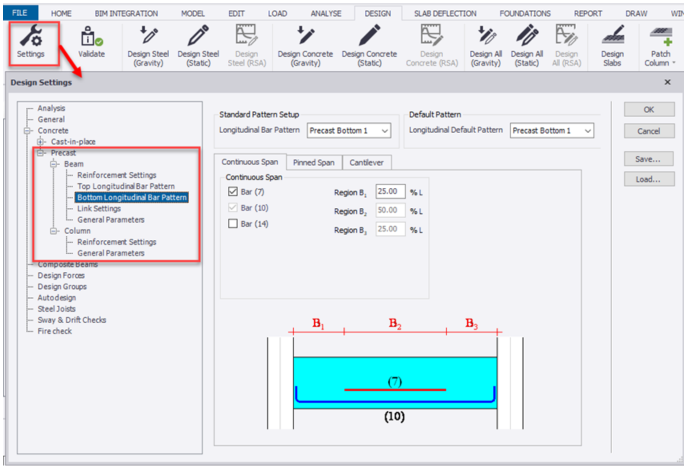
Key aspects of the new integrated design process and workflow are:

- The “Fabrication” property of precast beams and columns is first set to “Precast” to enable the new dedicated precast settings and integrated Tedds design for the members. A dedicated ‘Precast’ option is also added to the drop list options for the Concrete Column, Beam and Wall buttons of the Model Ribbon as shown below.



- The engineer should also decide at an early stage if they wish to use **Design Groups** and enable/ disable these accordingly via Design > Settings > Design Groups for Concrete Beams and/ or Columns.
- New beam section shapes are added in Tekla Structural Designer for precast concrete beams to match the section options in the Tedds calculation; L-section, T-section and inverted options of both these.
  - For precast concrete beams, multiple design sections are populated in the Tedds calculation as required. These are determined by the Design settings for Precast beams which specify the bar patterns and regions as shown below.
  - There are also reinforcement and general settings for precast concrete columns which control initial values set in the Tedds calculation.

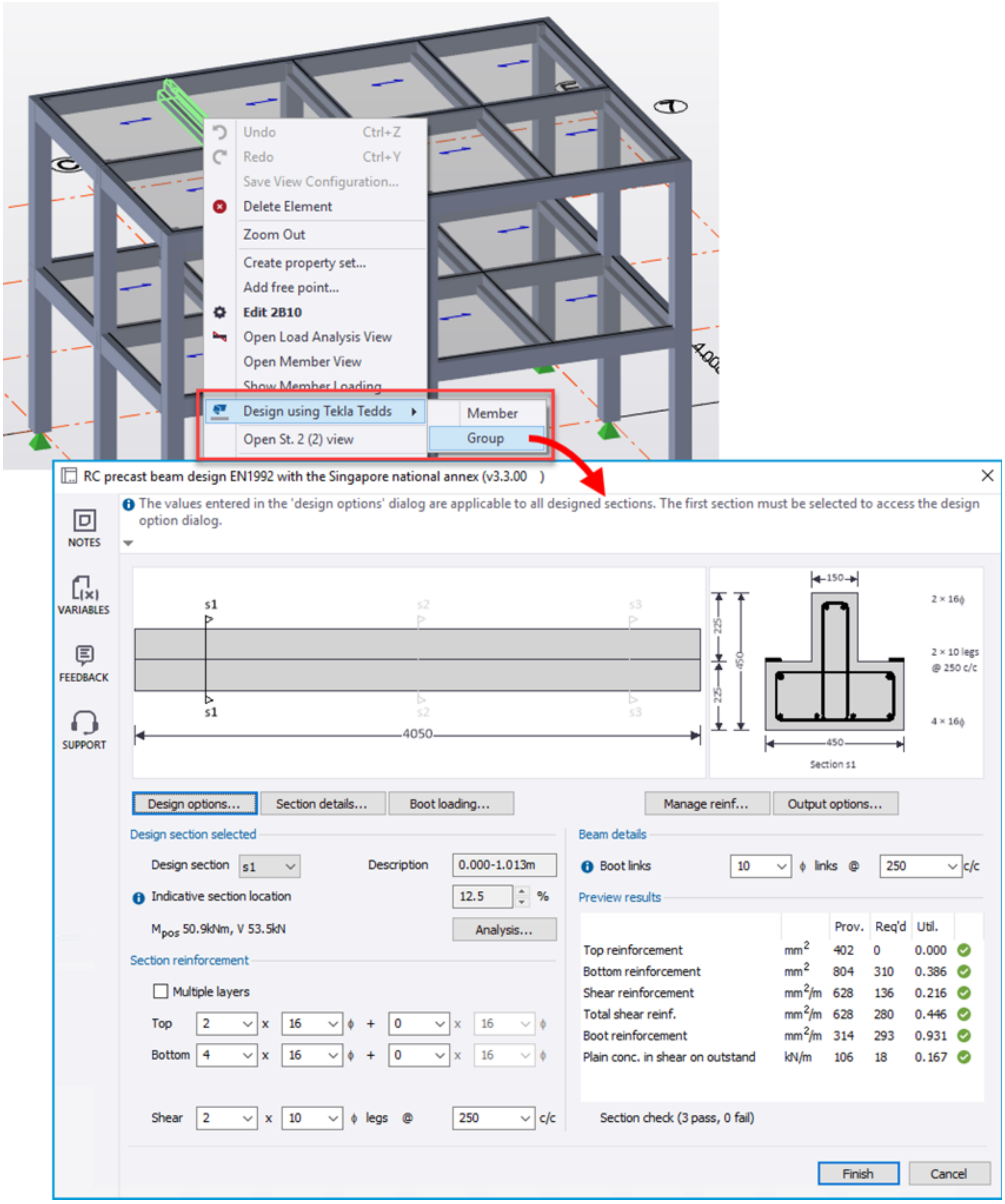




- The design link can be run for individual members in the model via the right-click member context menu > “Design using Tekla Tedds” with options to perform the design for the selected Member or its Design Group (when Design Groups for concrete members are enabled). Design parameters such as concrete grade, section size and design forces are automatically populated from the model to the Tedds calculation. Additional parameters not defined in the model can be fully reviewed and set as required in the Tedds calculation interface. The member can then be fully and interactively designed in the Tedds calculation
    - With Design Groups disabled, Design > Member designs only the individual selected member for its specific design forces.
        - For continuous beams and columns composed of multiple spans/ stacks, the view will automatically zoom to and highlight the span/ stack currently being considered for design.
    - With Design Groups enabled, the process is as follows for each option:
        - Design > Member; the Tedds calculation is populated with the design forces of the selected member. When the calculation dialog is finished, all the design settings made - such as number and size of

reinforcement bars and links etc - are copied to all members of the group which are then checked for their individual design forces.

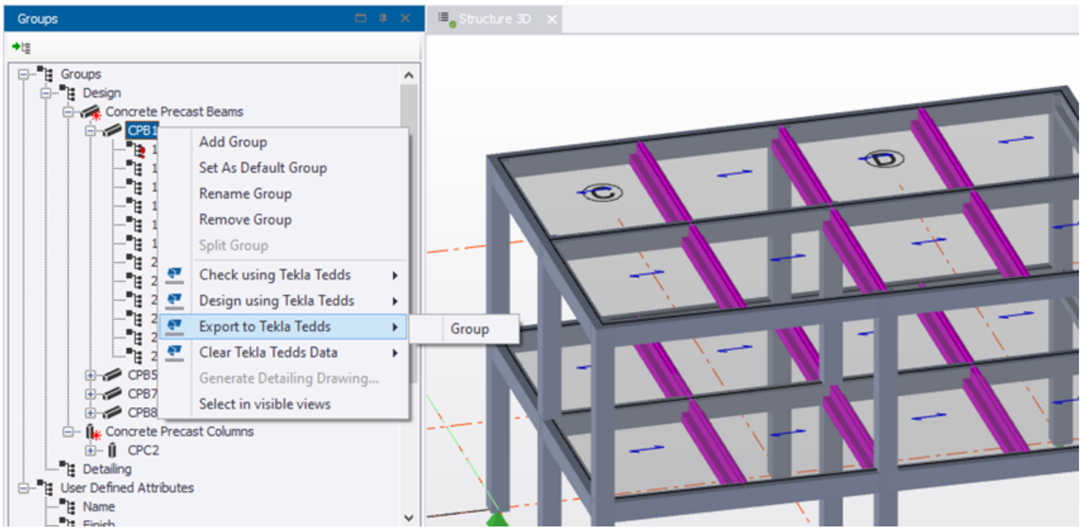
- Design > Group; the Tedds calculation is populated with the worst-case design forces of the group as a whole. When the calculation dialog is finished, the further process is the same as for Design > Member.



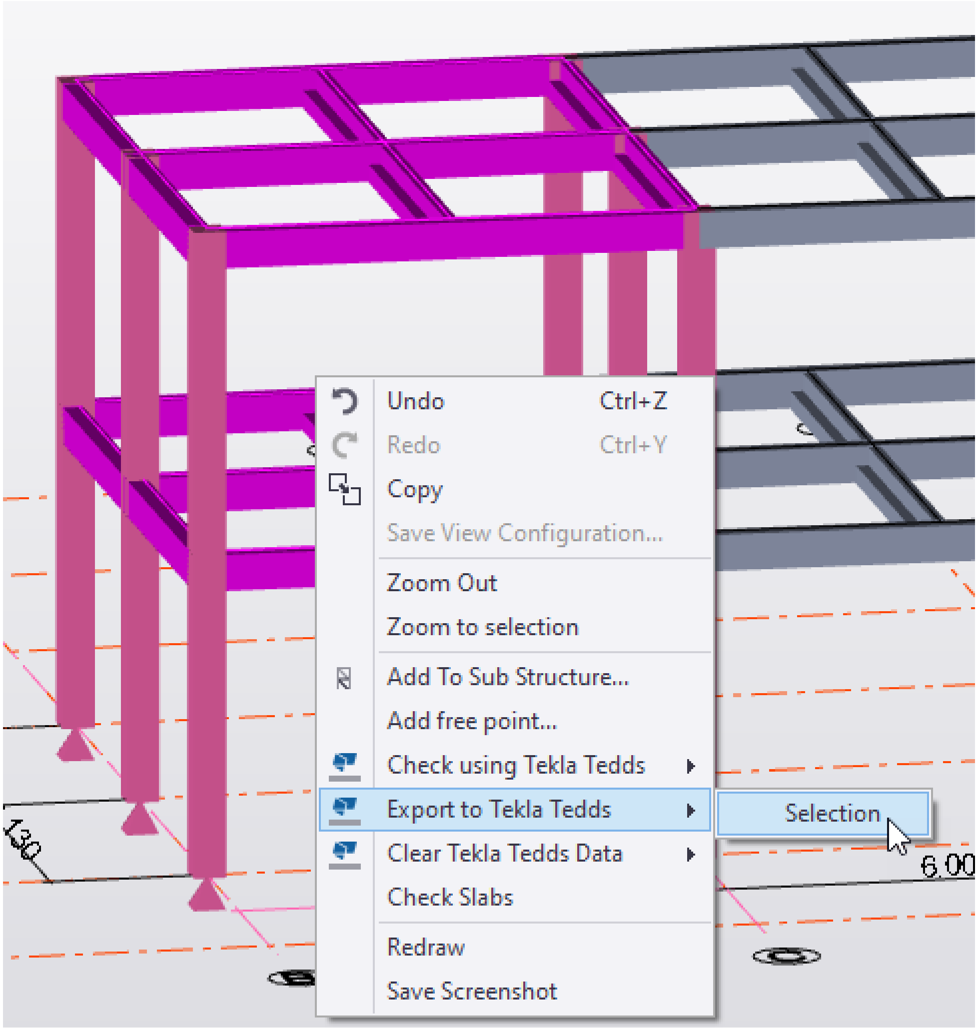
- Once the design is completed and the Tedds calculation dialog is finished, **ALL** of the Tedds design data as well as the results are embedded in the Tekla Structural Designer model for each designed member. This enables looping of analysis and checking/ design to accommodate model changes, such as to loading.

  - The resulting design Pass/Fail status and utilization ratios (UR's) from Tedds can be checked in the Review View and via the member Tooltip in the usual manner just as for design of in-situ concrete members within Tekla Structural Designer. When using Design Groups, the engineer can review the UR's of members and their automatic grouping and make changes to this as required.
  - After the design is complete and the Tedds data has been written to the model, the following additional options will be listed in the member context menu Tedds options; "Check using Tekla Tedds", "Clear Tekla Tedds Data", and "Export to Tekla Tedds" (see below for more on these options).
  - If required, the section size can also be changed in the Tedds calculation and the model will automatically be updated with this change (for all members in the Design Group where these are enabled). In this case, the analysis can then be re-run and all members of the group checked using Tekla Tedds to update the design fully.

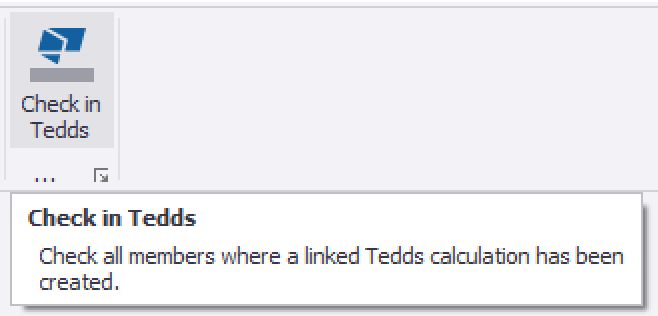
- When Design Groups for concrete members are enabled, Tedds design for Design Groups of precast beams and/or columns is also available from the Project Workspace > Groups Window. Design groups for precast concrete beams and columns are automatically created and populated just as for in-situ concrete members. Group checking/design is run by moving the cursor over a group to select it, then right-clicking and selecting the desired option from the context menu. As for design of individual members from the model, where the group has been previously designed and embedded Tedds Data exists, a "Check using Tekla Tedds" option will be listed as well as "Design using Tekla Tedds".



- The “Check…” option will run the Tedds Calculation ‘silently’ for all members and update the results to the model which can then be reviewed as described above.
- The “Design…” options (either for Group or Member) automatically open the calculation interface as described above, allowing the engineer to review and change the design data if required. On finish, all members in the group are then checked for the revised input without further display of the calculation interface.
- Where the group has been previously designed, a “Clear Tekla Tedds Data” option is also listed allowing the engineer to begin the design from scratch if they wish.
- For Reporting of the design, the engineer selects the “Export to Tekla Tedds” option which will open the Tedds application and create a Tedds Project containing calculation documents for the designed precast member.
  - This option is available for;
    - An individual Member in the model (Design Groups disabled).
    - A single Group (via right-click over a Group in the Groups Window or a member of the model).
    - All Beam or Column Groups together (via right-click over the Concrete Precast Beam/ Column Parent Group).
    - The entire model via right-click over the “Design” Parent Group for Precast Beams/Columns in the Groups Window.
    - A graphical Selection of part of or the entire model.
  - With Design Groups enabled, this will create a Tedds Project containing one calculation document* (.ted file) for each group included in the selection.
  - When Design Groups are disabled, the model member context menu export option will change to “Member” and will create a Project and calculation document for the selected member*.
    - *In the case of continuous multi-stack/span columns and beams there will be a calculation document for each stack/span of the member.



- For the scenario of re-checking multiple existing Tedds designs after changes to the model - made either via the Tedds calculation dialog e.g. for change of section size, or directly in the model e.g. for changes to loading - the analysis results can be updated via Analyze All (Static), then all the designed precast members in the model checked in a single operation using the Design Ribbon's new "Check in Tedds" command.



Related video

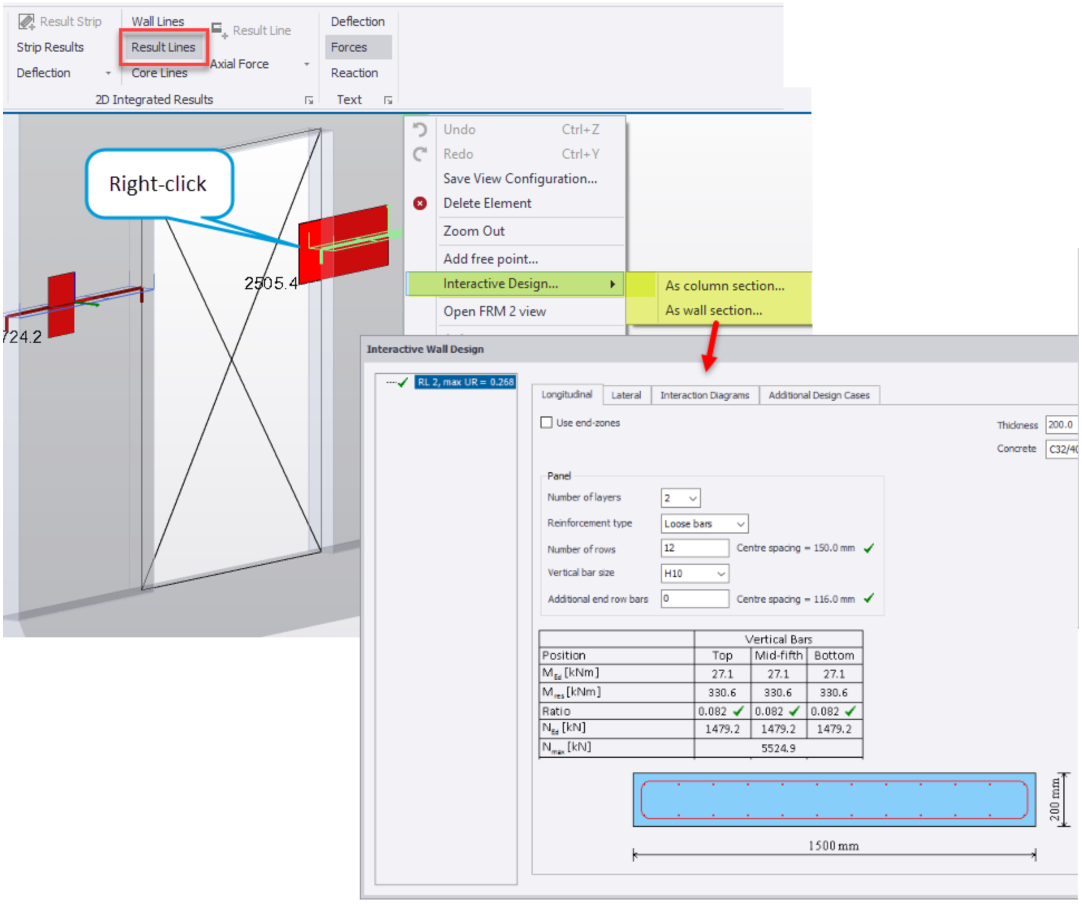
Precast modeling, analysis and design using Tekla Tedds

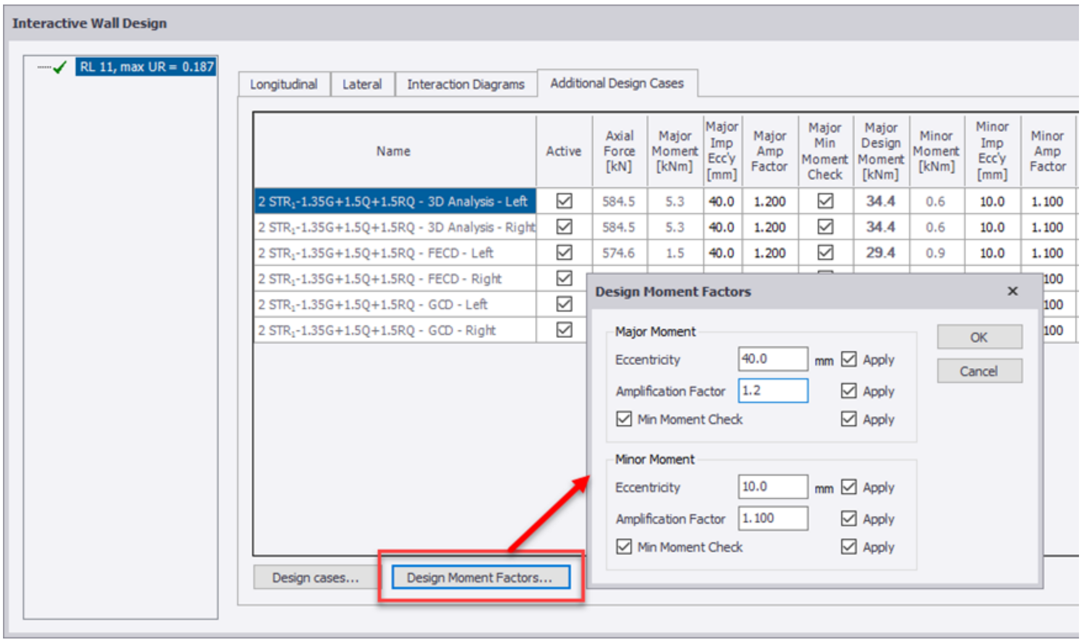
Concrete Design - Result Line Design saving of reinforcement and manual design forces

2D Result Lines were first implemented in Structural Designer release 2019 (released Mar 2019) allowing the engineer to investigate sectional forces around openings in concrete walls for example. Subsequent releases have continually enhanced this feature, adding interactive concrete section design in release 2019 SP1 and enhanced design forces in release 2019 SP2.

See this video demonstrating result lines in action.

This release further enhances the Interactive design using Result lines feature to add the saving of both the reinforcement and manually added/adjusted design forces defined in the interactive design dialog.





- Key aspects of Interactive design using Result Lines are:
  - **The Interactive Design** option is accessed from the right-click context menu for Result Lines as shown above. The engineer then chooses from wall or column section design options and the appropriate Interactive section design dialog is displayed.
    - The wall length/ column depth considered is the Result Line length. The design dialog features full interactive manual selection of both lateral and vertical reinforcement, Interaction Diagrams and auto-design and check options, just as for a regular wall/ column section. All edits to the reinforcement settings are now saved when the Interactive design dialog OK button is clicked (please note that changes to concrete column/wall size is not saved).
    - Result Lines are not constrained to cross sections in the horizontal plane - vertical sections can be used above/between openings to investigate forces and reinforcement requirements in “coupling beams”.
    - The section design always considers the main bars (running perpendicular to the cross section) as being on the inner layer, from a design perspective this will tend to be conservative but the engineer should give this some consideration when working with non-horizontal sections.
  - **Additional Design Cases and Adjustments** - the Result Line forces are listed in the “Additional Design Cases” page. These forces can be

manually added to and adjusted. All additions/ adjustments are now saved when the Interactive design dialog OK button is clicked.

- There are three potential “Design Moment” adjustments for each direction:
  - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
  - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety)
  - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension “h” in the direction considered).
- When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.
- The adjustment values and options can be applied manually to individual Cases and also quickly in a single operation to all Active cases (those with “Active” option checked on) via the [Design Moment Factors...] button as shown above.

- Note that the design does not consider the reinforcement specified in wall properties - only that which is defined in the Interactive design dialog.
- Currently the feature is not linked with Reports and so it is envisaged output will be via screenshots of the interactive design and check results dialogs.

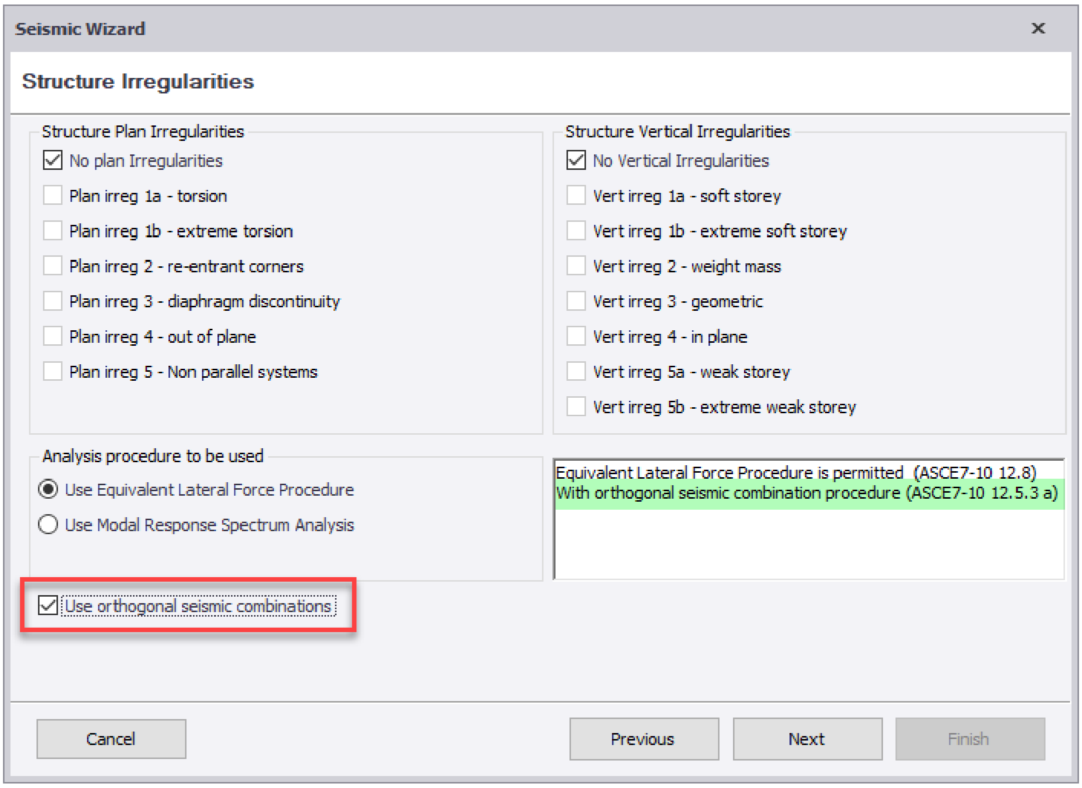
Related video

Interactive design using Result lines

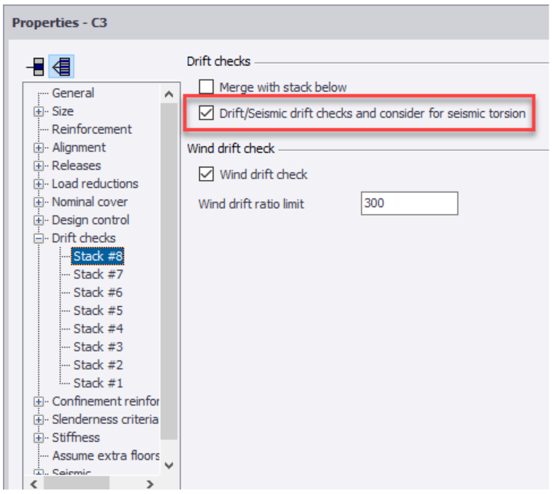
Seismic Analysis & Design Enhancements

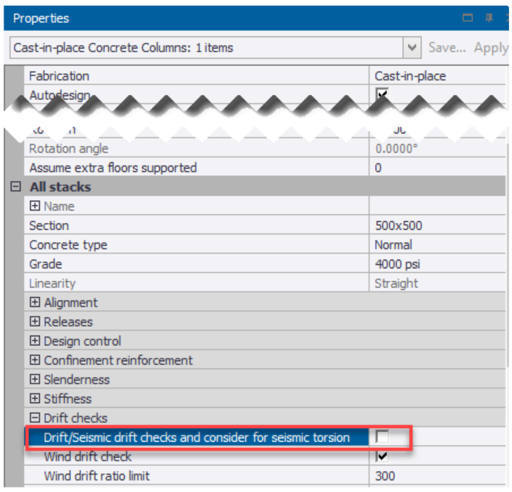
The following enhancements are made in this area following customer feedback.

- **Seismic Wizard and Seismic Combinations** - the engineer can now directly specify the use of orthogonal seismic combinations, in which the seismic load cases for both directions are included. Formerly the creation of these was automated only for certain specific types of Structural Irregularity. The new setting to directly activate these combinations is on the Structural Irregularities page of the Seismic Wizard as shown in the picture below.



- **Seismic Torsion Assessment of Columns & Walls** - in previous releases, every column/wall in the model was considered when calculating the effects of Seismic Amplified Accidental Torsion. However, there may be situations in structures where columns/ walls are remote from the core structure and would be ignored for seismic torsion. The inclusions of columns/ walls in this calculation can now be controlled via the checkbox "Drift/Seismic drift checks and consider for seismic torsion". This can be set both in the Properties Window for all and individual stacks/ panels in the Properties Window and the member edit dialog. When unchecked, the member is not considered for; the assessment of Accidental Torsion; the assessment of Amplified Accidental Torsion.

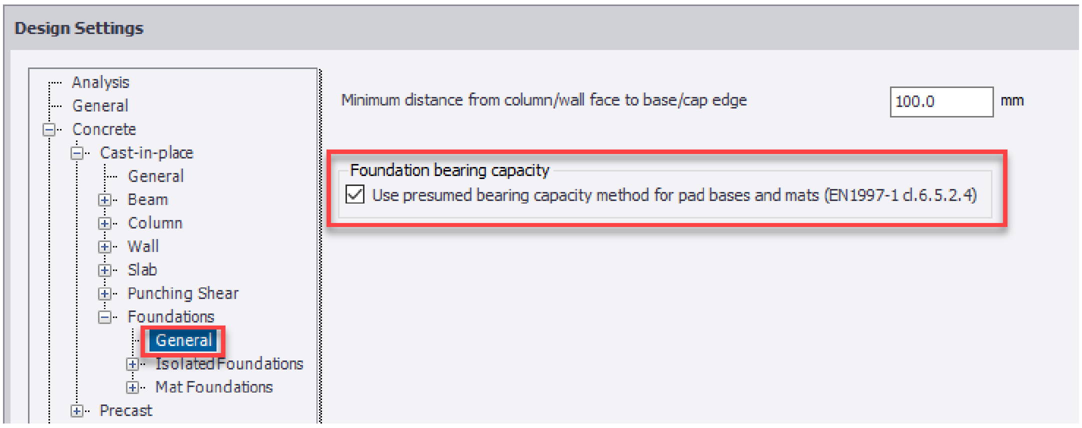


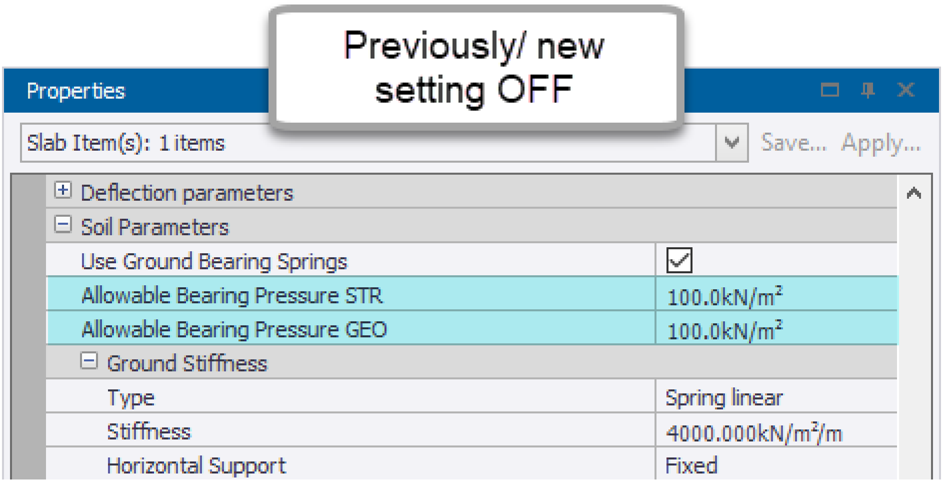


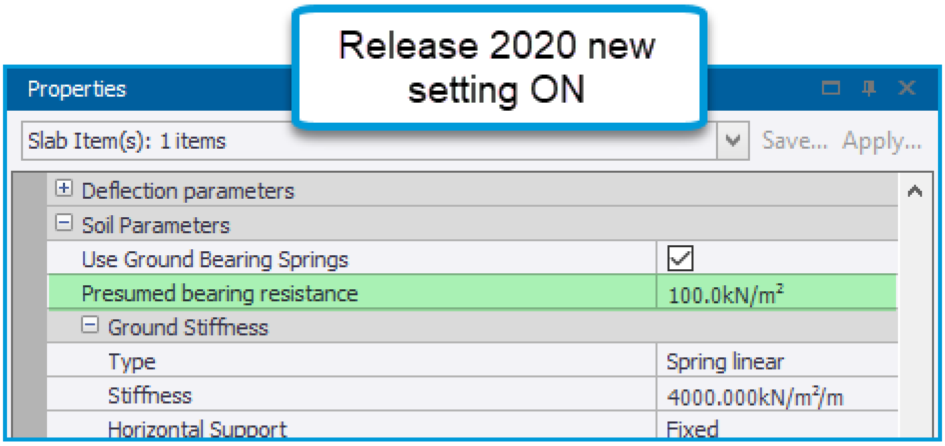
Foundation Design - New option for presumed bearing capacity method for Mat Foundation - Eurocode

The option to use a presumed bearing capacity method (instead of the Ultimate bearing pressure capacity) for per EN-1997-1 cl. 6.5.2.4 was added in an earlier release for isolated foundations. The presumed bearing capacity method uses SLS pad pressure results checked against a presumed bearing resistance entered by the engineer. Following customer requests this method option is now extended to include Mat Foundations.

- The Design setting to control the application of this method is moved from the **Isolated Foundations > General Parameters** group to the **Foundations > General** group of the Foundations Design Settings, as shown below, and now applies to both isolated and mat foundations together.
    - The new combined setting will be off by default for new models. However, for existing models in which it was activated for Isolated Foundations, it will be checked on automatically when they are opened in this release.
- When activated, the mat foundation Soil Parameter properties change to allow input of the Presumed bearing resistance of the check as shown below.



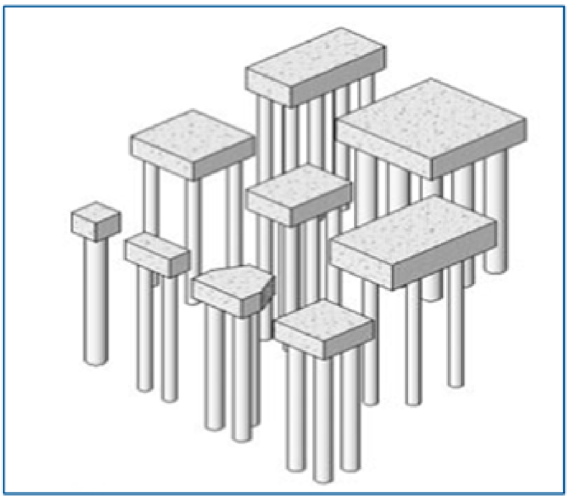


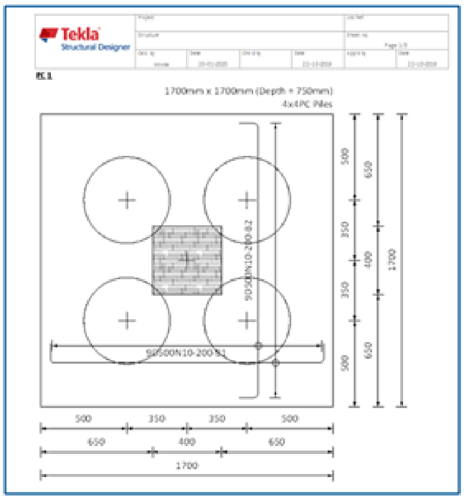


Foundation Design - New Pile Cap Design for Australian Head Code to AS 3600 : 2009

Bringing foundation design to the Australian Head Code in line with that in other Head codes, this release adds the design of Pile Caps to AS 3600 : 2009. The functionality matches that of the existing design for the Eurocode and US design codes and supports auto-design of both the pile cap and pile selection.

Comprehensive detail drawings of pile caps and piles can also automatically be produced.





- Key aspects of the design are:
  - Pile caps can be modelled with 1 to 9 pile configurations.
  - There are options to auto-design; cap depth, number of piles and reinforcement.
  - The following checks are performed with extensive design details reported for each:
    - Pile capacity check
    - Pile cap bending check
    - Pile Cap shear capacity check
    - Column and pile punching shear check
    - Pile pair punching check
    - Pile one-way shear check.
    - Check for limiting parameters (spacings, edge distance, thickness etc.)
  - The engineer can choose the pile capacity to be considered for pile punching and shear checks.

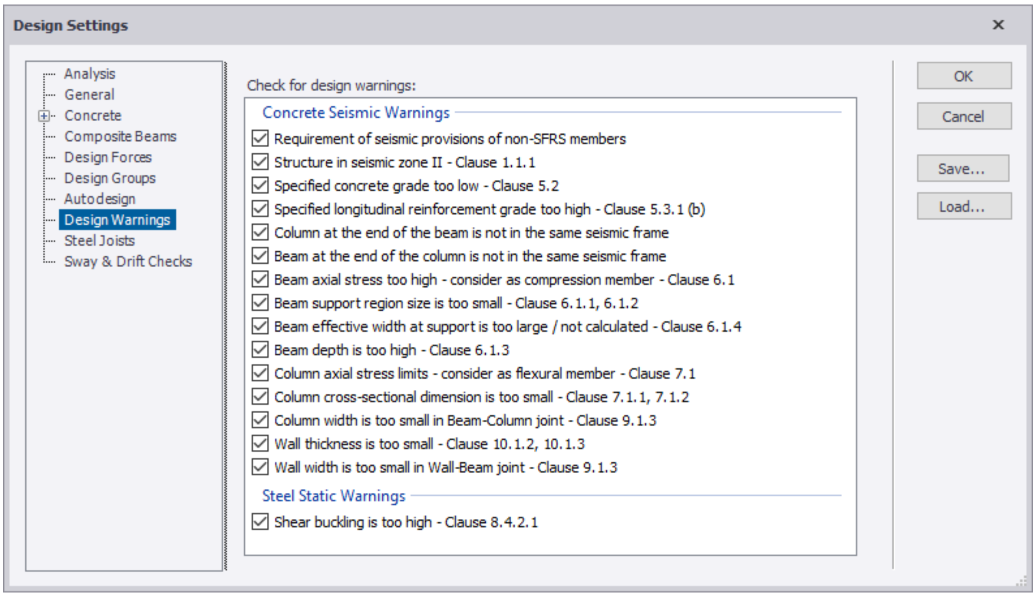
New Design Warning Controls - Indian Head Code

Following customer requests, controls are now added for design warnings for Concrete Seismic Design, and Steel static design, for the Indian Resistance

Codes of Seismic Design and Detailing to IS 13920: 2016, and Steel Design to IS 800: 2007.

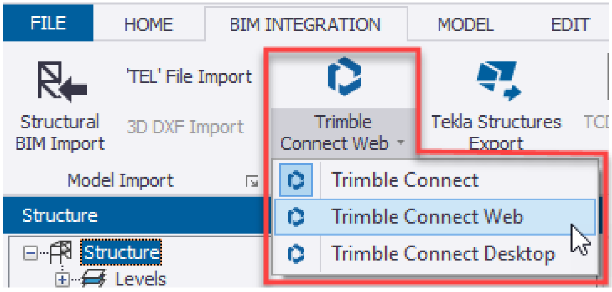
This feature allows the warning design status and messages for selected checks/ considerations to be removed at the engineers discretion.

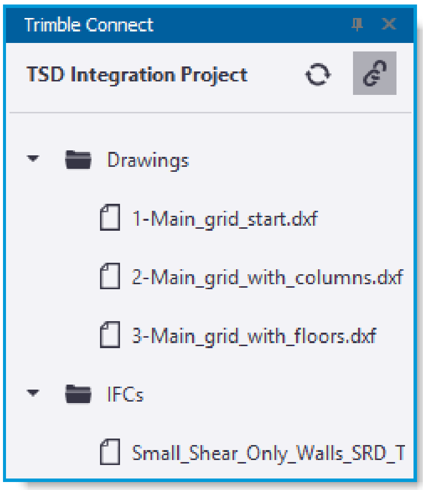
- The new controls are accessed via Design Settings > Design Warnings as shown in the picture below.
- Warning controls are listed for the following areas and include Clause references to guide the engineer; general seismic topographical and SFRS type considerations; Material properties; Geometric properties; Beams, Columns, Walls and Joints.



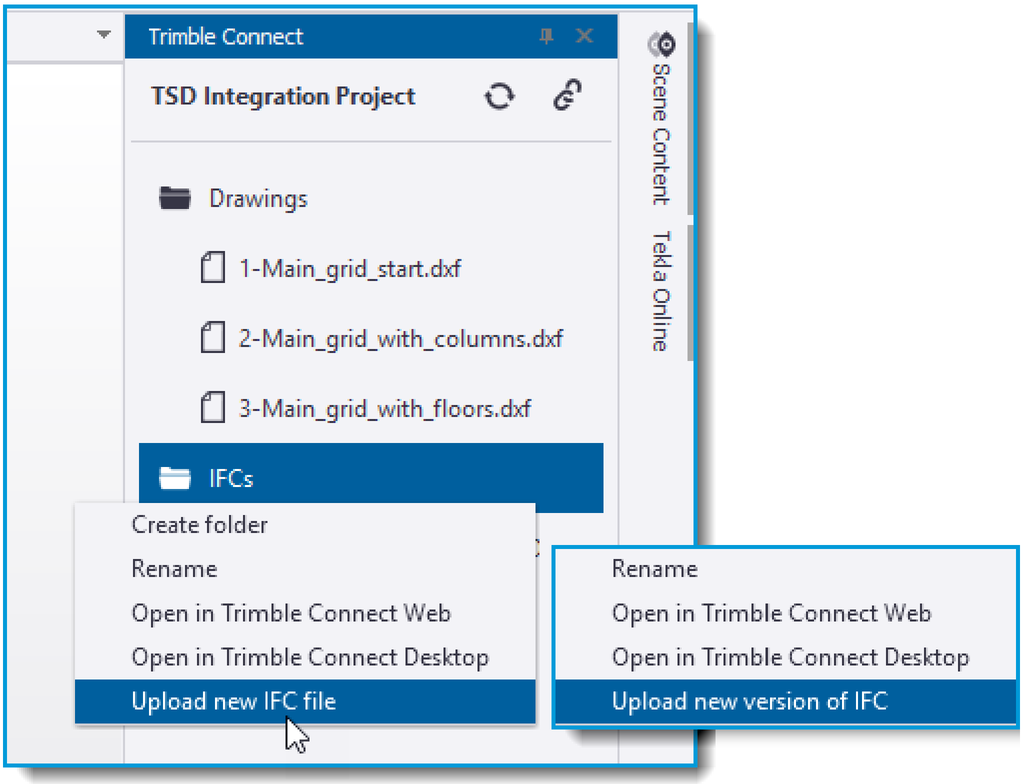
New Trimble Connect Integration Link

Furthering our drive towards a Connected World, the new Trimble Connect Link in this release allows easy collaboration between project stakeholders, making sharing of Tekla Structural Designer model information as easy as possible. The link is accessed via Trimble Connect command on the new dedicated BIM Integration Ribbon, as shown in the picture below. You can then browse your Trimble Connect Projects, create new folders in them and send Tekla Structural Designer model data to them.



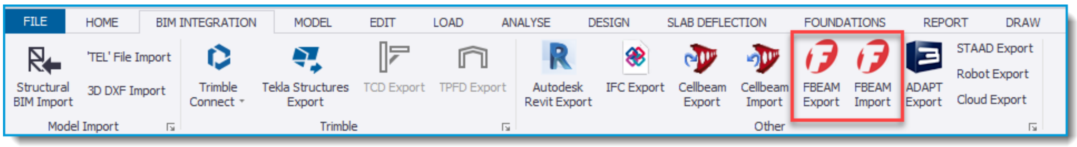


- Note that in order to use the link you must have a Trimble Connect license and be signed in to your Tekla Online Account (using your Trimble Identity credentials).
- Selecting the main “Trimble Connect” option opens the new dedicated Trimble Connect Window, which is docked to the right of the main window by default. Your Trimble Connect projects will then be listed in the Window.
  - The Trimble Connect button has additional options to launch either “Trimble Connect Web” (in your default browser) or “Trimble Connect Desktop” if you have this installed.
- Once your Trimble Connect projects are listed you can then click the link icon  adjacent to a project to open and browse it. Options to create and rename folders within the project are then available from the right-click context menu for the Window.
- To add the open Tekla Structural Designer model to Trimble Connect, select the context menu “Upload new IFC file” command which launches the IFC Export dialog. When this is completed the IFC file will automatically be added to the project.
  - Once added, the IFC file can then be opened in Trimble Connect (both Web and Desktop) from the context menu options. To accommodate model changes, the IFC can also be updated using the context menu option “Upload new version of IFC” as shown in the picture below,

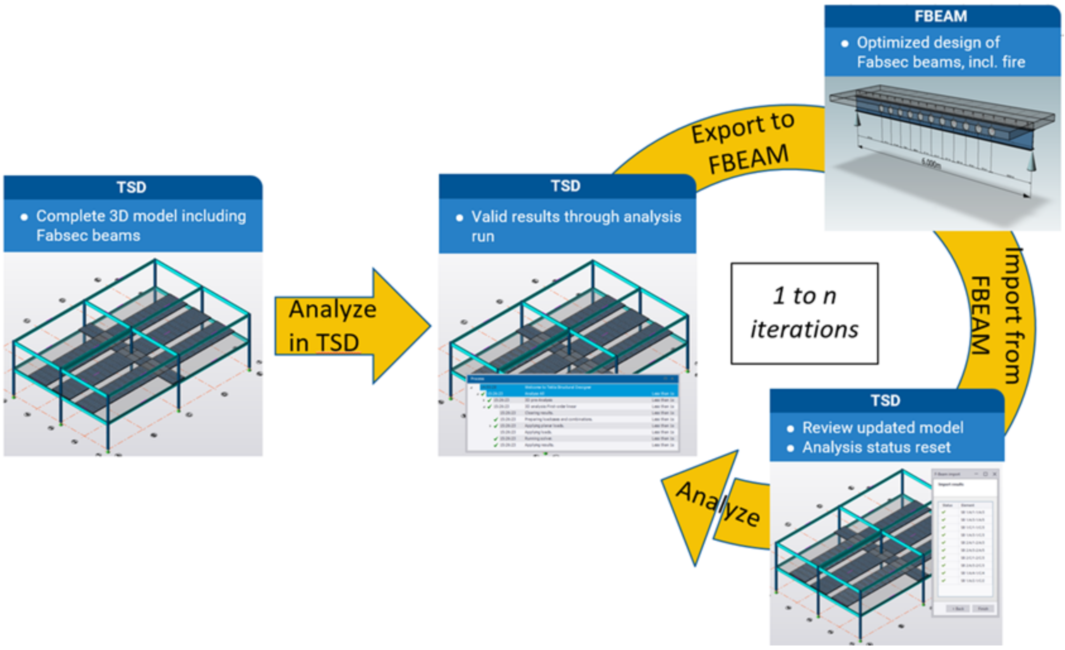


New Bi-directional FBEAM® Link for design of FABSEC® Beams

Listed on the new dedicated BIM Integration Ribbon, as shown below, is a new link with the FABSEC® FBEAM® software for the design of beams with their Fabrication set to FABSEC in Tekla Structural Designer models.



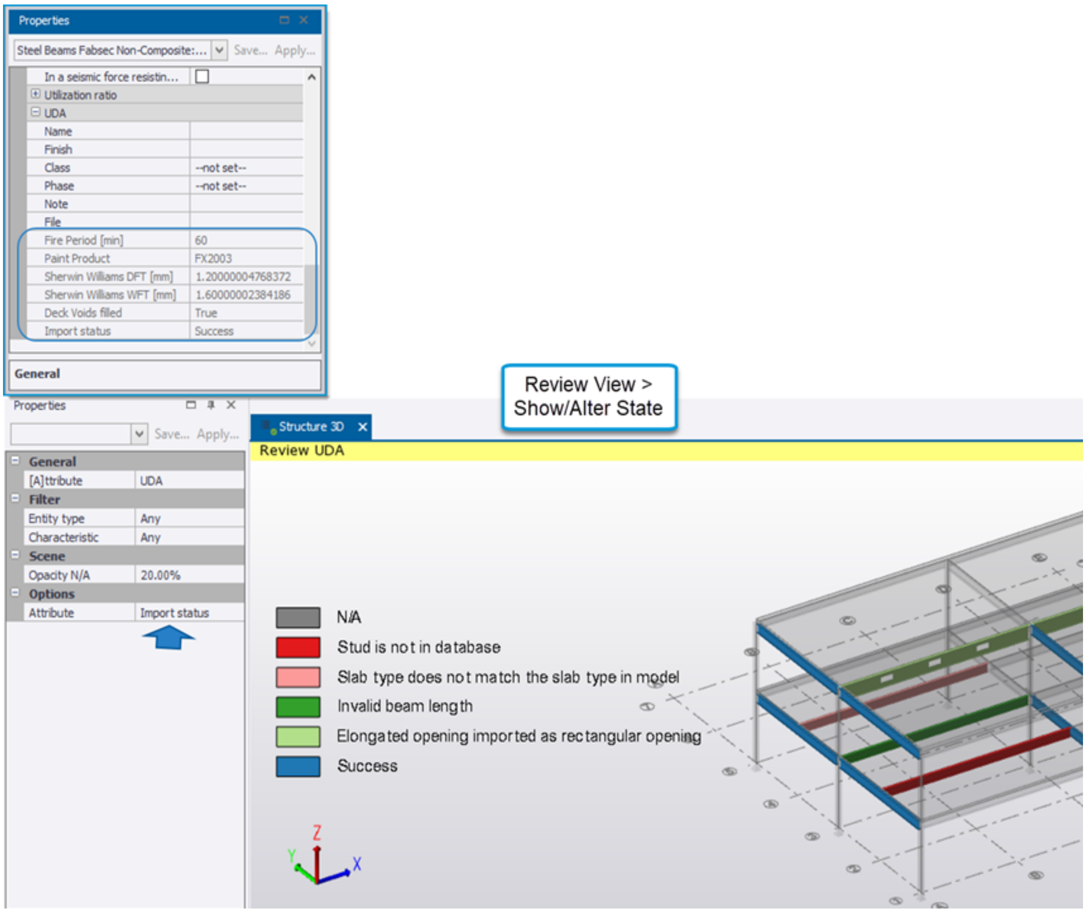
- FABSEC sections are a form of proprietary plated beams used in the UK construction market. There is no native design for such sections within Tekla Structural Designer.
- The new link now allows beams specified as FABSEC in the Tekla Structural Designer model to be sent to FBEAM via Export (page 333) for detailed design and optimization. Finalized design data can then be passed back to the Tekla Structural Designer model from FBEAM via the Import (page 334) function.



- The table below shows the data transferred between the programs for each stage:

	Exported to FBEAM	Imported to TSD
Beam section properties	✓	✓
Beam name, span properties	✓	-
Beam layout, secondary beams and restraints	✓	-
Loading and load combinations	✓	-
Floor (slab) properties	✓	-
Web openings and stiffeners (rectangular and circular only)	✓	✓
Shear connector properties and layout	✓	✓
Transverse reinforcement and slab reinforcement	✓	✓
Fire design details	-	✓

- Import status and data imported from FBEAM populates the following new UDAs in the Tekla Structural Designer model which can then be reviewed in Review View > Show/Alter State/UDA as shown below:
  - Import status,
  - Position of elongated openings
  - Fire design details



New Integration with ETABS: ETABS-Tekla Structural Designer Link Plugin

By popular request in a number of regions, this release adds a link to export an ETABS model to Tekla Structural Designer. This caters for the scenario where for example the engineer has an ETABS model and wishes to investigate design options using Tekla Structural Designer.

For full details on using the link and the data imported see the new Help Topic on this feature.

Key details of the link are:

- The link is via a plugin to ETABS which can be downloaded from the Tekla Warehouse.
- The ETABS plugin export creates a CXL file which is the same file format used for integration with other applications such as Tekla Structures and Revit. The CXL file is then imported into Tekla Structural Designer in the same manner as for other CXL files using the BIM Integration > Structural BIM Import command.
- The following data is transferred by the link in this first release:
  - Grid lines, Levels and Materials
  - Beams and Columns:
    - Concrete - alignment, orientation, end releases, material type and grade and section size:
      - *Note that inclined concrete members are imported as beams, so section sizes not permitted for beams will be converted to rectangular and curved beams are imported as straight.
    - Steel beams, columns, and braces are imported if their section names can be mapped to an equivalent section in Tekla Structural Designer sections database - alignment, orientation, end releases, material type and grade are maintained.
    - Other material beams, columns and braces (and steel sections that can’t be mapped) - are imported as analysis elements with their alignment, orientation and end releases maintained. Section sizes can then be easily applied as required in Tekla Structural Designer.
  - Concrete Walls - imported as meshed concrete walls with geometry, thickness and alignment maintained (sloping walls are excluded).
  - Slabs - imported as two-way spanning slabs on beams
    - Openings - rectangular & circular holes in slabs are imported as slab openings (holes of other shapes imported as rectangular)
    - Curved slab edges are imported as straight edges and sloping slabs are excluded.
    - Thickened slab areas - such as column drops - which are created in ETABS by overlaying slab panels are imported as two panels. They will then be flagged by a validation error and can be easily remodelled in Tekla Structural Designer using the dedicated column drop tool or the slab thickness override option.

- Diaphragms - ETABS diaphragm objects are not transferred. However all imported slabs have their diaphragm option set to rigid. This can then be reviewed/ edited by the engineer as required.
- Supports - ETABS support objects are not directly transferred so supports are created as follows:
    - Pinned supports (Fx, Fy, Fz and Mz fixed) are created under every column (unless a supporting member such as a transfer beam is detected)
    - Supports pinned about the minor axis created under every wall (unless a supporting member is detected)
    - All other ETABS joint restraints are excluded, including Springs.
- Trusses - imported as analysis elements
- Loads - are excluded.

New Tekla Open API

An Open API is now available allowing the integration of Tekla Structural Designer with other applications...This will shortly be available in the Tekla Developer Center so check back here regularly for updates.

Related video

Tekla Structural Designer API

Minor Enhancements and Fixes

General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability.
- Steel Section Database - Nordic Countries - some missing sections are added for both steel and cold formed materials to the section databases of Norway, Finland and Sweden. Additionally we have:
    - [TSD-5898-9] - some missing sections are added for both steel and cold formed materials to the section databases of Norway, Finland and Sweden.
    - [TSD-5906] - the naming of plated sections is changed to use the * symbol instead of lower-case x.
    - [TSD-4501] - new sections are added to existing order files.

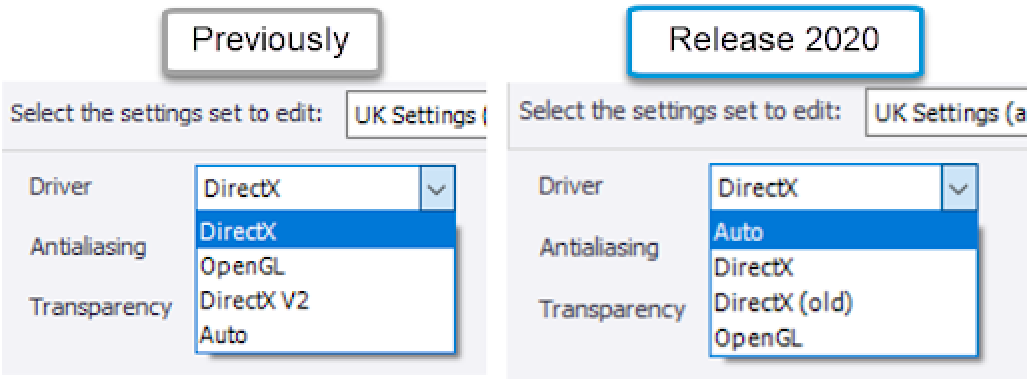
    - [TSD-5551] - Section and order file defaults are updated to use only commonly available ones.
- [TSD-5224] - Properties Window - the note window at the bottom of the Properties window will now auto size to display the full text of the property where formerly this could be truncated.

BIM Integration

- Structural BIM Import
    - [TSD-4014] - for imported concrete models, the Autodesign member property for concrete 1D elements is now considered as follows:
        - When importing a CXL file into a new model file, the Autodesign member property is checked ON (previously it was OFF).
        - If importing a CXL file into an existing model, the current member autodesign property is retained irrespective of the value in the CXL file.
    - [TSD-4337, 4338] - a new UDA entitled Pour phases has been added to allow pour phase value to be passed between Tekla Structures and Tekla Structural Designer for concrete elements. The UDA is created on import where Pour Phase attribute exists in the CXL file.
        - Existing Pour phase UDA's are checked on import and the BIM Status tree will list messages for any updated Pour phase UDA's.
- [TSD-2965] - Tekla Structures/ Revit Export - for transfer columns (e.g. a column supported on a beam) the misleading warning for analysis forces at bottom of column not including additional moment - issued in the BIM > Structural BIM Export status tree - is removed (as it applies only for column splice loads).
- [TSD-2756] - Structural BIM Integration (CXL) File - the CXL file data structure is enhanced to include member fabrication, slab deck types and wall types:
    - On export from TSD, concrete members will be assigned a fabrication type.
    - On import to TSD if Other Data is ON in the import wizard, the fabrication/deck type will be updated as appropriate. Associated messages are shown in the BIM status tree.

Performance

- [TSD-6075] - Graphics - the graphics driver settings in Home > Settings > Scene > Graphics are updated and adjusted.

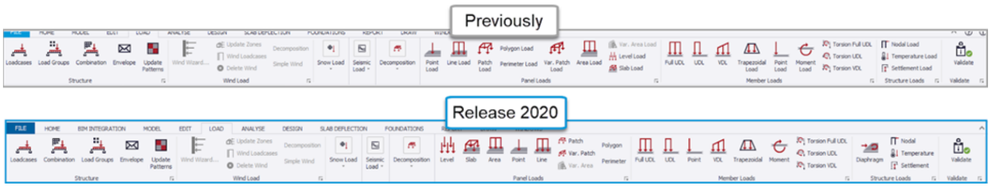


- *DirectX* is now called *DirectX (old)*
- *DirectX V2* is now called *DirectX*
- DirectX (old) is based on DirectX 9 and is becoming obsolete while the new DirectX is based in DirectX 11 and is currently the primary HOOPS driver for 3D acceleration.
  - For existing installations (which will carry forward old settings) we recommend that the Driver setting is checked and set to the Auto option if necessary, as continuing with Direct X (old) could impair performance and stability.
  - To check your adapter's compatibility visit the HOOPS developer website for HOOPS 20.x, DirectX and OpenGL requirements.
- The "Auto" setting will let HOOPS pick the best driver (and is now the default)
- [TSD-5870] - Design Process- the Process window now reports the progress % of the Presizing stage of autodesign (for steel members). The Cancel button can also now be selected during this process to terminate design.

Loading

- [TSD-6200] - Loading Ribbon - buttons on the loading ribbon have been reorganized to keep the most commonly used items at large size as the application window size or screen resolution is adjusted.
  - "Load Groups" is moved after Combination.
  - "Panel Loads" re-ordered to promote the use of "Level" load when appropriate.
  - "Point" load moved to the left.

- The redundant word “Load” is removed from each button.



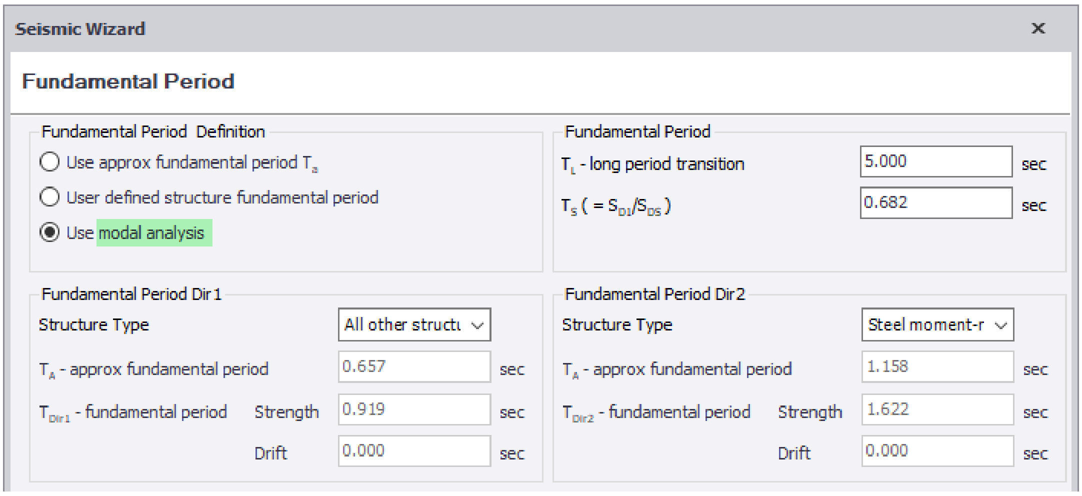
- [TSD-4017] - Load Combination Generator - US Head Code - the default service factors for dead loads in the combinations ASD-1.0D and LRFD-1.4D have been changed from 0 to 1.0.
    - The previous default value of zero, if not corrected, would produce incorrect beyond scope status for isolated foundation design where these combinations were generated and active. This issue should no longer occur.
- [TSD-5827] - Perimeter Load - previously the Perimeter Load command could only be used provided that the 'create as line loads' option was checked on in the load properties. This has now been corrected.
- [TSD-6004] - Temperature Loads - Slabs - temperature loading applied to slabs set as semi-rigid diaphragms was not applied if the diaphragm mesh type was also changed to be triangular (not the default). Temperature loads are now applied to semi-rigid diaphragms for any mesh type selection.
- [TSD-6154] - Seismic Loading - US Head Code - for Seismic Loading to ASCE7-16, the revised capping provision for short period regular structures per Section 12.8.1.3 is now applied for building meeting all requirements of the clause with $S_{DS} < 1.0$ using the actual S_{DS} value. Previously in this circumstance $S_{DS} = 1.0$ was used, resulting in over-conservative seismic loads.

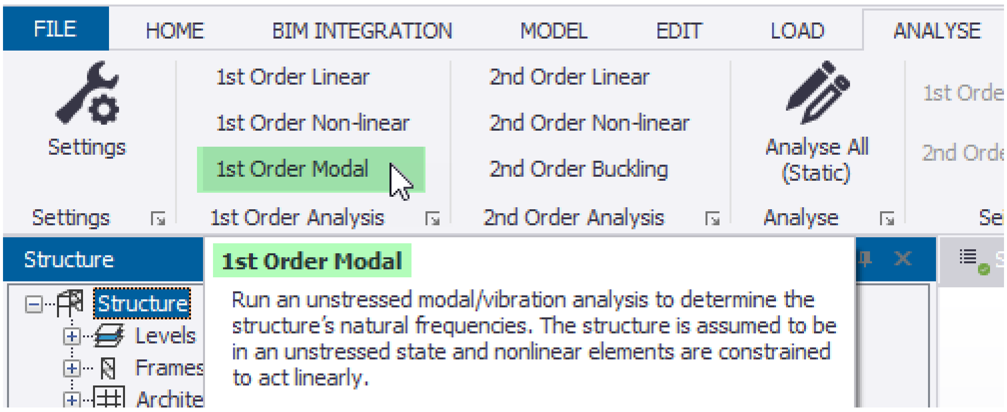
Validation

- [TSD-5795] - Composite beams - the validation time for models containing composite beams (of their effective widths) is considerably reduced.
- [TSD-5459] - Wind Wall Panels - Decompose to Diaphragms - an additional model validation check is now performed to check the validity of area loads applied to wind panels that have been set to decompose to rigid diaphragms. For such panels the load can only be decomposed provided that it has been applied perpendicular to the plane of the panel. If the load is acting in any other direction a validation warning "Wind wall load beyond scope" is now issued.

Analysis & Results

- Vibration Analysis renamed to “Modal” Analysis - in many regions, the term “vibration analysis” is more commonly associated with the perceptible vertical movement of floors (such as is checked by the DG11 Floor Vibration Check for example) or lateral sway of tall building subjected to service wind load, and the term "modal analysis' is used to refer to an analysis which determines vibration modes and their properties (such as period etc). Hence the term "vibration analysis" is replaced by "modal analysis" in all instances within the program, such as in the Seismic Wizard and on the Analyze Ribbon as shown below.





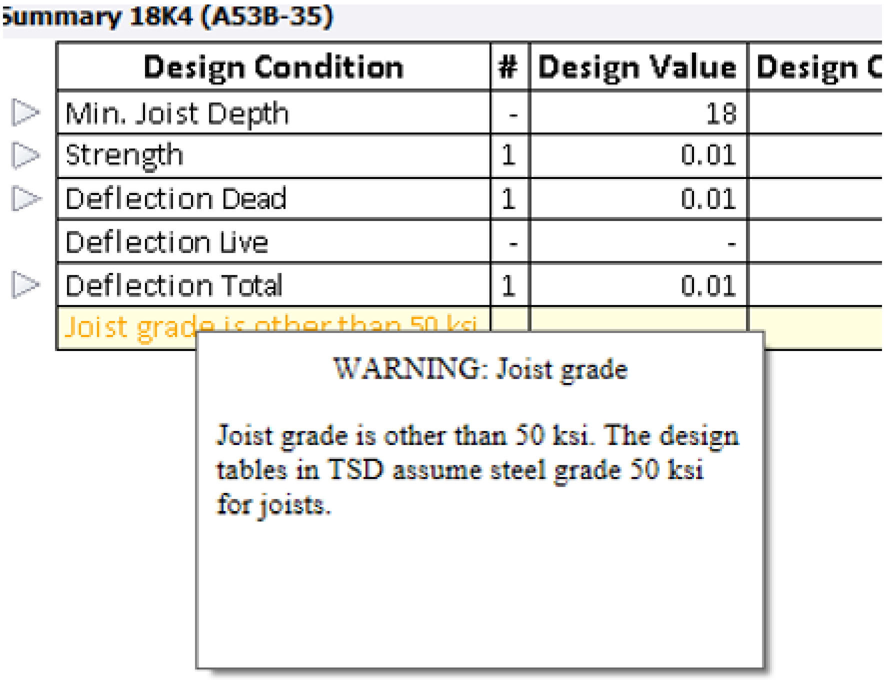
- [TSD-6382] - 2D Results - imposed load reductions are now implemented for meshed concrete wall F_y (vertical force) 2D contour values and the associated vertical stresses σ_y etc. Click the "Reduce Axial Force" toggle button in the Results View to see the results with imposed load reductions applied.

Design - General

- [TSD-6248] - Timber Design using Tedds Link - design forces sent to the Tedds Timber design calculation now include the maximum forces of only active load combinations. Previously, maximum values considering load combinations, load cases and envelopes were used causing conservative design results in some scenarios.

Design - Head Code US

- [TSD-5445] - Steel Joist Design - when a steel grade other than 50 ksi is applied to steel joists, the design will now details now issue a warning stating that the design assumes steel grade 50 ksi as shown below.



Summary 18K4 (A53B-35)

Design Condition	#	Design Value	Design C
Min. Joist Depth	-	18	
Strength	1	0.01	
Deflection Dead	1	0.01	
Deflection Live	-	-	
Deflection Total	1	0.01	
Joist grade is other than 50 ksi			

Reports

- [TSD-5219] - Punching Checks - the “Report for Member” option is added to the right-click context menu for a selected punching check, allowing the engineer to produce an individual report for a punching check object.
- [TSD-5224] - Material Listing - the Cold formed table now includes a column for the total number of cold formed members for each grade and the total.
- [TSD-6052] - Portal Frames - Wind Drift Table - in the Wind Drift table of Review View > Tabular Data and the associated Report item, Portal frame columns are now given a unique reference which includes the Portal Frame reference for example “PF F/1-F/3 Span 1 Rh Column” rather than just “Span 1 Rh Column”.
- [TSD-5797] - Report List - the Report Styles in new and existing models are now listed in alphabetical order. This also applies to the Member Type list in the Member Reports dialog. With this new feature, user-defined reports can be listed in a grouped order by careful prefixing.
- [TSD-4991, 6010] - 2D Result Diagrams & Views - in some circumstances, reports including analysis diagrams of 2D contour results would not display the contours correctly. This issue has now been corrected.

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.2 Tekla Structural Designer 2020 hardware recommendations

System requirements for effective operation

CPU: Multi core Intel i5 Series or above, Xeon or AMD equivalent

- Highest affordable performance recommended.

Memory: 16GB (32GB or more recommended)

- Memory requirements are highly dependent on model content.

OS: 64-bit Microsoft Windows 8.1 / 10

- Operating systems must be running the latest service packs / updates.
- **Graphics**: 1600 x 900 resolution (1920 x 1080 or higher recommended)
  - 1GB or higher of dedicated RAM.
  - Utilizes HOOPS Visualize, a third party graphics engine available from Tech Soft 3D. To check your adapter's compatibility visit the HOOPS developer website for HOOPS 20.x, DirectX and OpenGL requirements.
- **Disk space**: 1GB or more of free space for installation
  - Operational disk space requirements are highly dependent on model content.

Internet connection: Required for access to Online Services and some documentation.

License Service:

- Tekla Structural License Service 3.00 including Sentinel RMS 9.5

License Server: The latest version of the Tekla Structural Licence Service, at time of release, is shipped and installed with the software. If you have chosen to have a separate licence server, it is always our recommendation that you also run the latest version of the Tekla Structural License Service on it to ensure compatibility. Please see System Requirements for specific version details.

Test environments

The application is tested and supported on the following business versions of Microsoft Windows with the latest updates applied:

- Windows 10 64-bit
- Windows 8.1 64-bit

1.3 Upgrade Tekla Structural Designer to a new version

NOTE The following instructions are specifically for users who already have a earlier version of Tekla Structural Designer installed.

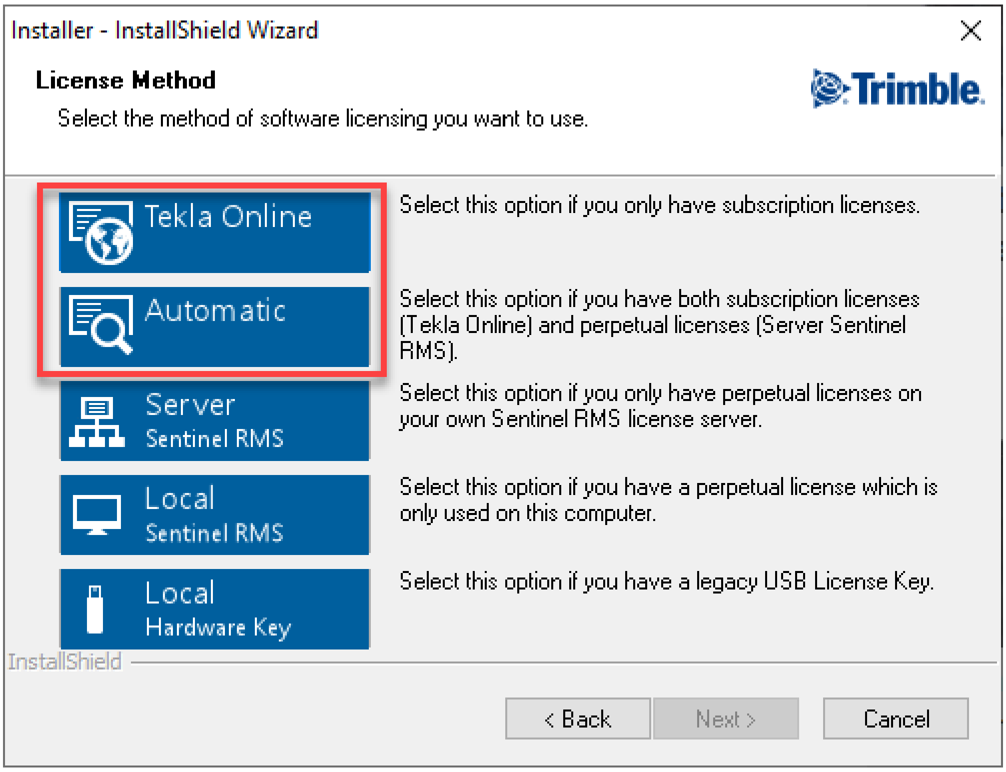
You can have multiple Tekla Structural Designer versions on your computer. When you install a new version you are not required to immediately uninstall the previous version.

Service packs are cumulative updates, which replace the previous service pack installation for the same Tekla Structural Designer version. So you should only have the latest service pack installed per version.

NOTE We recommend that you complete any models you are already working on using your current version of Tekla Structural Designer. Once you save a model in the new version, you cannot open it in a previous version.

Licensing information specific to Tekla Structural Designer 2020

- **Upgrade Licenses** - Tekla Structural Designer 2020 will require the activation of a new license. You should already be in possession of your Product Activation Key (PAK) as these are usually distributed prior to the software release. Please contact your local Service Department now if you do not have your PAK. To minimise any down time we advise that your PAK is activated BEFORE installing Tekla Structural Designer 2020.
- **License Server Version** - the Tekla Structural License Service **must** be updated to the new version 3.00.0001 (incorporating Sentinel RMS 9.5) to be compatible with this release. Licensing will not function correctly if this update is not performed! Please see this article for more information about this, and see System Requirements for specific version details.
- **Subscription Licenses & Tekla Online Licensing** - if you have switched to subscription licensing select the new **Tekla Online** License Method when installing the program as shown in the picture below. Please note the following:



You can download the installation package for Tekla Structural Designer 2020 from the Tekla Downloads service.

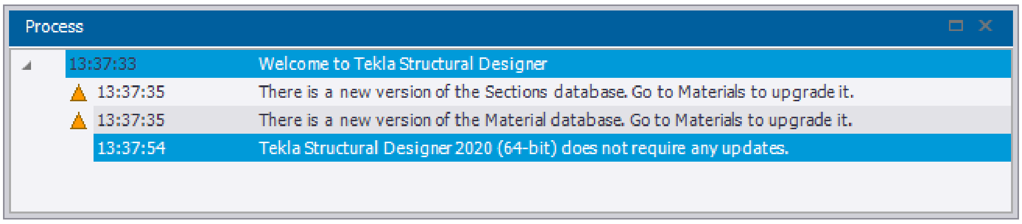
Install the upgrade

Complete the upgrade in this general order (follow the links in the text for detailed instructions):

1. Install and license Tekla Structural Designer (page 238) as if you were a new user.
2. Upgrade your local databases

NOTE In the Tekla Structural Designer 2020 release and for future releases, the local database is now version-specific. A version directory (called “20.0” for this release) is added to the local database directory which is located in C:\Users\username \AppData\Local\Tekla\Tekla Structural Designer\Database. A copy of the current local Database from your previous version will be made in the version-specific directory.

When you first run Tekla Structural Designer 2020 a message informing you that new databases are available will be displayed in the Process Window as shown below.



These should be upgraded as detailed below.

a. Open “Materials” from the Home Ribbon

b. Select the “Sections” page in the list of options

c. Click the “Upgrade” button, then click this button again in the subsequent “Upgrade Database” dialog

d. Repeat this process for all the other databases (Material, Reinforcement...etc) to ensure all your databases are up to date.

3. Transfer Connection Resistances

In Tekla Structural Designer 2020 the Connection Resistance check and associated data has been considerably expanded to include such information as steel grade (of supported beam) number of bolt rows & lines etc. If you have added your own Connection Resistance values in a previous release, the following action is required:

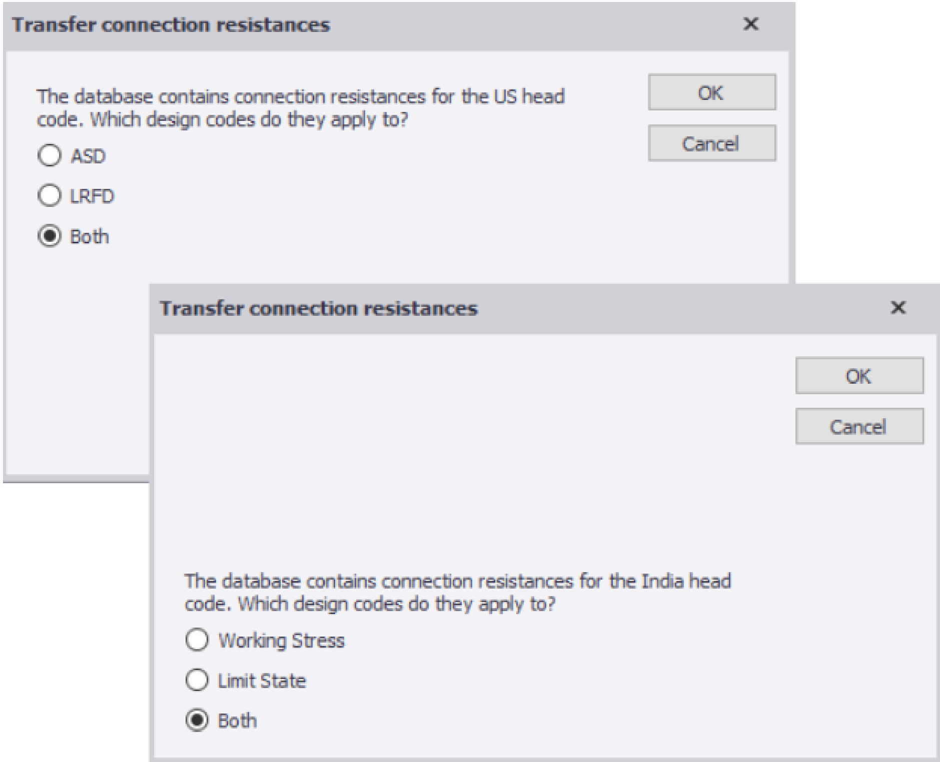
a. When Tekla Structural Designer 2020 is run for the first time, existing Connection Resistance data will be imported and listed in the new Connection Resistance dialog and flagged with a warning status as requiring review/ updating (see pictures below).

b. As shown below, use the Edit buttons to review/ set the Connection Resistance data as follows:

- Select each imported Connection Type then click the Edit... button below this section of the dialog to review then set/ confirm the required data; Steel Grade and number of notches and bolt lines.
- Then for each row of the Resistances table for each type, click the Edit... button below this table to review/ set the number of Bolt Rows

c. Engineers using India or US head codes with connection resistance data from a previous version will see a query at the point of

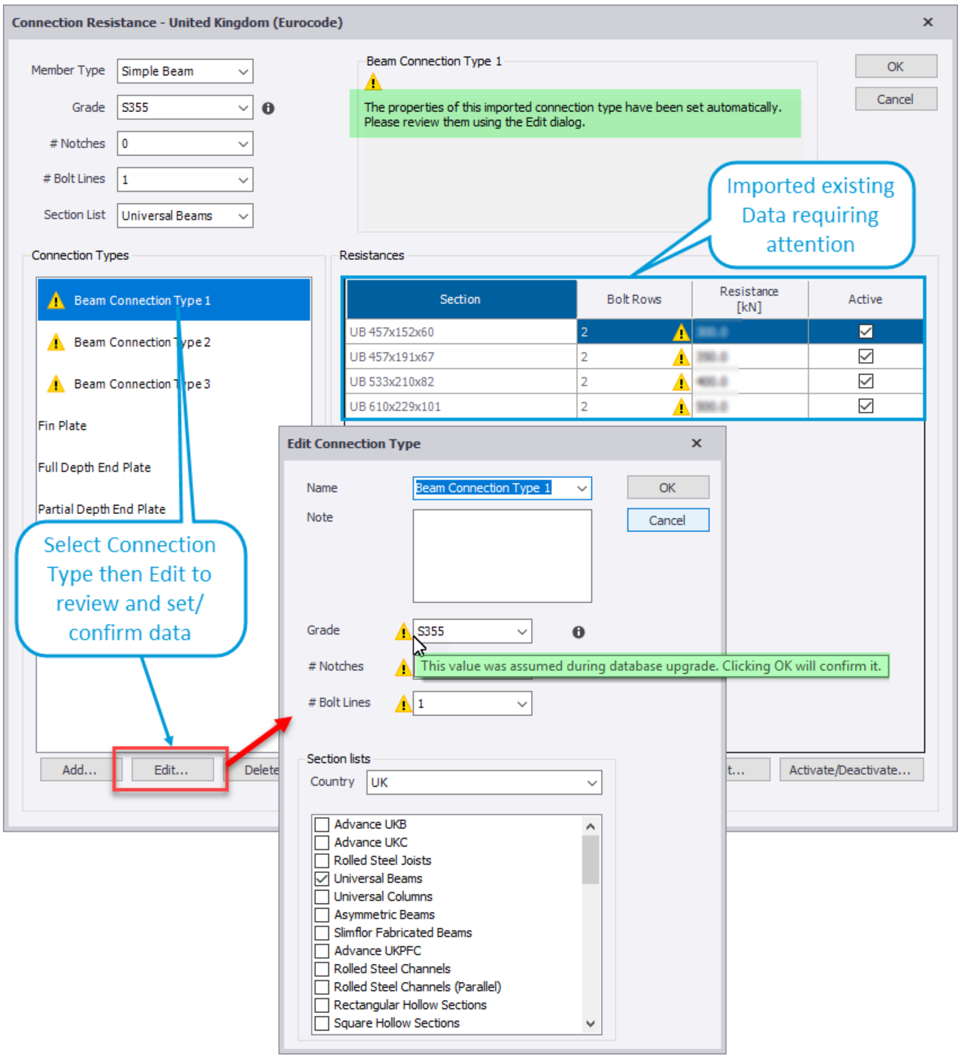
Upgrading the database asking if they want to import the data for ASD/Working Stress, or LRFD/Limit State, or both as shown below.

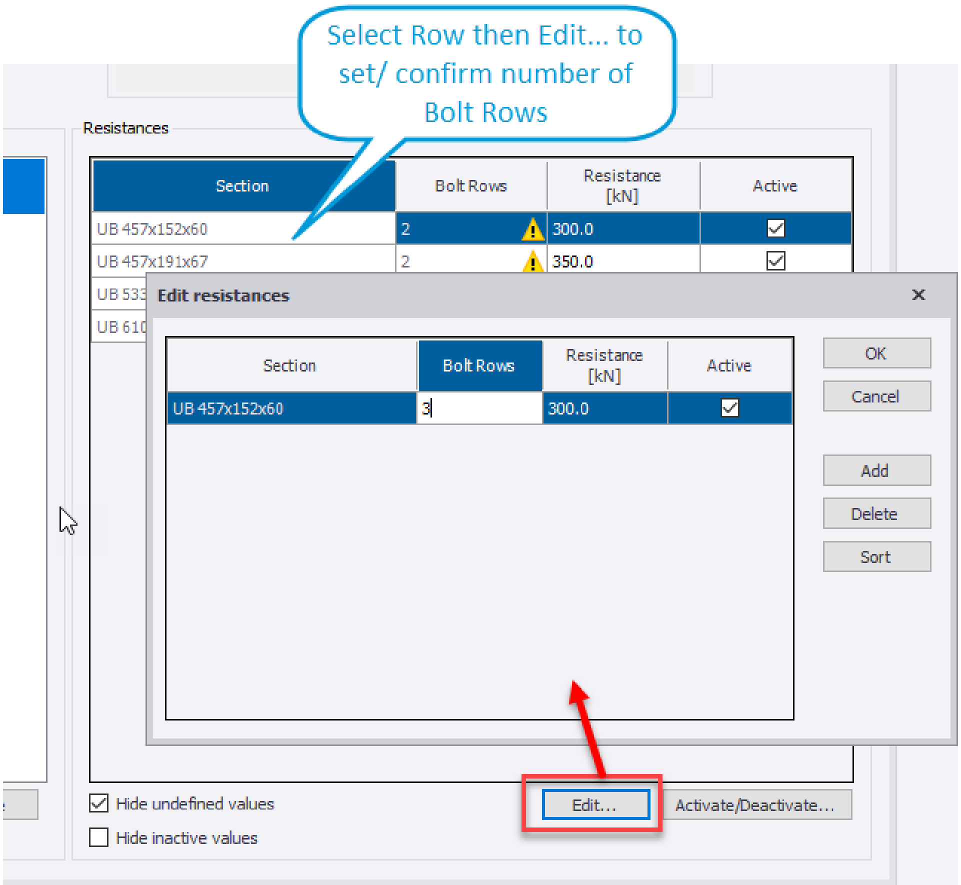


d. During the data transfer and Upgrade the value of parameters for 'old' beam data are assigned based on defaults as follows:

- Eurocodes: Grade S355, # Notches 0, # Bolt Lines 1.
- US codes: Fy Any, Coping None, # Bolt Lines 1, Bolt Diameter 3/4 in (20 mm), Thickness 1/4 in (6.4 mm).
- IS codes: Grade Fe 410, # Notches 0, # Bolt Lines 1.
- AS codes: Grade 300 (AS/NZS 3679.1), # Notches 0, # Bolt Lines 1.

- BS codes these: Grade S355, # Notches 0, # Bolt Lines 1.





1.4 Tekla Structural Designer service packs

Tekla Structural Designer service packs are Tekla Structural Designer version updates.

Tekla Structural Designer service packs update the existing installed Tekla Structural Designer version software. You do not need to separately install previous service packs in order to install the most current service pack. For example, you can install service pack 2 without installing service pack 1.

- Service packs can include new features, and improvements and fixes to existing features. We recommend that all users install the latest service pack.

You can find the service packs in Tekla Downloads.

See also:

Install a Tekla Structural Designer service pack (page 96)

Latest Releases:

Install a Tekla Structural Designer service pack

You can install a service pack to update a Tekla Structural Designer version or a previous service pack. Service packs can contain new features, and improvements and fixes to existing features

If you have the related Tekla Structural Designer version or a previous service pack installed on your computer, you do not need to remove it before installing a new service pack.

To install the service pack

1. Download the service pack software installation file from Tekla Downloads to your computer.
2. Double-click the installation file to run the installation.
3. Follow the steps in the installation wizard to complete the installation.

Release notes: Tekla Structural Designer 2020 SP6

This release will update your Tekla Structural Designer installation to version number 20.0.6.30 and should be installed to ensure optimum function of the program. It includes a number of new features, enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than the immediately prior release 2020 SP5 (version 20.0.5.56 released Nov 2020), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing

- No new license is required for this version.
- **Tekla Structural License Service** - the latest Service Pack for this was released in July 2020. This is available in Tekla Downloads and should be installed on all clients for optimum functionality.
  - For more information see the Release Notes page for the Tekla License Service update July 2020 (v3.1.3.4).
- **License Server Version** - for Server licensing, the latest version of the Tekla Structural License Service 3.1.2 (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) or later to be installed and will replace your current version.
  - Please note that because the Trimble digital certificate used to digitally sign the original installation (Trimble Solutions Oy) expired in December 2020, users will require Administrator privileges to install this latest update which they would not have required for previous updates. This update is signed with the Trimble Solutions Corporation digital certificate which is valid until April 2021
- **Integration**
  - **Tekla Tedds** - to use the new Timber Design using Tekla Tedds feature (page 174), you must install the Tekla Tedds Engineering Library update (September 2020) or later. All Tedds updates can be obtained from Tekla Downloads or via the Update Service.
  - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate this release with Tekla Portal Frame Designer/ Tekla Connection Designer, you should install Tekla Portal Frame Designer/ Tekla Connection Designer 20 Service Pack 2. This was released in Sept 2020 and can be obtained from the Tekla Download Service.
  - **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.0) was released on 1st July 2020 and updated to version 7.01 in September 2020. The installation and update are available in Tekla Downloads. If you are using Autodesk Revit© 2021, you can install this to perform BIM integration with Tekla Structural Designer.
    - All fixes and enhancements included in the 2021 release and update were also included in updates for the Integrators specific to the

other currently supported Revit© versions (2018, 2019, 2020). For more information see the Tekla Structural Designer Integrator September 2020 updates. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - Depending which version you are upgrading from, some Databases may be updated by this release. When this release is first run, your Databases will now automatically be updated and it is no longer necessary to do this manually via Home > Materials. See the Release notes: Tekla Structural Designer 2020 SP4 for more about this.

Highlights

General & Modeling

- Review View - Design Status Enhancements (page 102)
- Material List Enhancements (page 106)
- Review View - Show/Alter State Reinforcement command extended to Isolated Foundations (page 112)

Issues with Associated Bulletins

- [TSD-8551] - Seismic Design - Steel Columns - USA Head Code - this issue relates to the the design of steel columns assigned to Seismic Force Resisting Systems (SFRS) to the seismic provisions of the 2016 version of AISC 341 (years 2005 and 2010 were not affected). In some circumstances the expected seismic axial force amplification for the axial strength check of section D1.4a was not applied which could result in an unconservative design. For more information please see Product Bulletin PBTSD-2101-1.
  - This issue is fixed in this release.
- [TSD-8637] - Concrete Beam Design - British Standards (BS) Head Code (all countries) - the overall deflection check design status was always that of the last load combination instead of the most critical from all combinations. Therefore, in the most extreme case, this check could fail for one or more load combinations but the beam would still be marked as passing if the last combination had a passing status. For more information please see Product Bulletin PBTSD-2101-02.
  - This issue is fixed in this release.

General & Modeling

Highlights

- Review View - Design Status Enhancements (page 102)
- Material List Enhancements (page 106)
- Review View - Show/Alter State Reinforcement command extended to Isolated Foundations (page 112)
- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of this is given directly below:
  - [TSD-8382] - Analysis - A memory corruption issue has been fixed that could cause the analysis process to abort and/or the program to force close unexpectedly for some models that included short beam spans with partially fixed ends.
- [TSD-7317] - Keyboard Functions - new keyboard shortcuts have been added **Z1**, **Z2** and **Z3** to zoom to a 1-3m area around the current cursor location. These new shortcuts compliment the existing **ZA** and **ZS** shortcuts - for more on these and others see the Help topic Keyboard shortcuts (page 291).
  - Use the shortcuts as follows; first locate the cursor where you want to zoom, then press the “Z” keyboard key, followed by the “1”,”2” or “3” keyboard number key.
- [TSD-8043] - Connections - when a model has been populated with connections (see the Help topic Design connections (page 819) for how to do this), they are now shown graphically in construction plane (Levels, Slopes and Frames) and sub model views.

Interoperability

- [TSD-7851] - Grasshopper - Tekla Structural Designer Live Link - the Slab component in the Grasshopper plugin has been enhanced to allow creation of the previously released General material slabs. For more information about this feature see the article Grasshopper - Tekla Structural Designer Live Link. Please note the following:
  - This enhancement requires the installation of the new Grasshopper plugin specific to this release. This can be obtained from the Tekla Warehouse.
  - In existing scripts, the user can replace the old slab component with the new one to make use of the new parameters.
- [TSD-8557] - Export to One Click LCA - changes have been made to the format of the exported data to improve data mapping in One Click LCA. Also data exported from US Customary unit models is now in Metric units, since One Click LCA does not support the use of US Customary units for the

uploaded data. For more about this feature see the Help Topic Export to One Click LCA (page 341).

- [TSD-8361] - BIM Integration - Revit - Column Splices - improvements are made in this release to the way in which splices added in the BIM application (i.e. in Revit) are handled before updating the Tekla Structural Designer model. Some situations that were causing this process to behave unexpectedly have been identified and the following enhancements made to improve this:
  - If one or more stacks are excluded from the update process (BIM status = Exclude in Tekla Structural Designer model) then we shall exclude the whole column
  - [TSD-8495] - Update messages have been improved to ensure they are associated with an object in the Tekla Structural Designer model.
    - Note that there is one related outstanding issue when round tripping Tekla Structural Designer > Revit > Tekla Structural Designer without making any changes that may cause confusion. This can cause erroneous "Position Update" messages for some of the column stacks in Tekla Structural Designer when splices with offsets exist.

Loading

- [TSD-8428] - Load Decomposition - an issue is fixed that could cause some slab loads to not be decomposed correctly in the rare circumstance of models with only one decomposition plane (i.e. no other planes at all with loaded slab/roof panels) which included 1-way spanning slabs supported by 2-way spanning slabs. This issue relates only to release 2020 SP4 (v 20.0.4.55 released Sept 2020) - in which it was introduced - and later.
  - This issue has only been encountered internally and we expect it is highly unlikely customers will have, given its unusual conditions. We would note also that it would be very clear if the issue occurred and resulted in any significant missing load, as it would be flagged by all of; a decomposition warning “Some loads have not been applied > Load LL xx lies outside of panels”); a warning in the Process Window "Total reactions vs Applied loading do not balance" and errors for the affected load case(s) in the Project Workspace > Loading Window.

Design

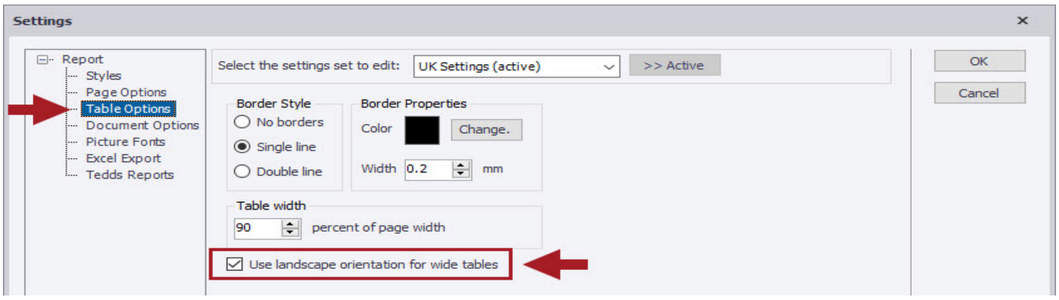
Head Code US

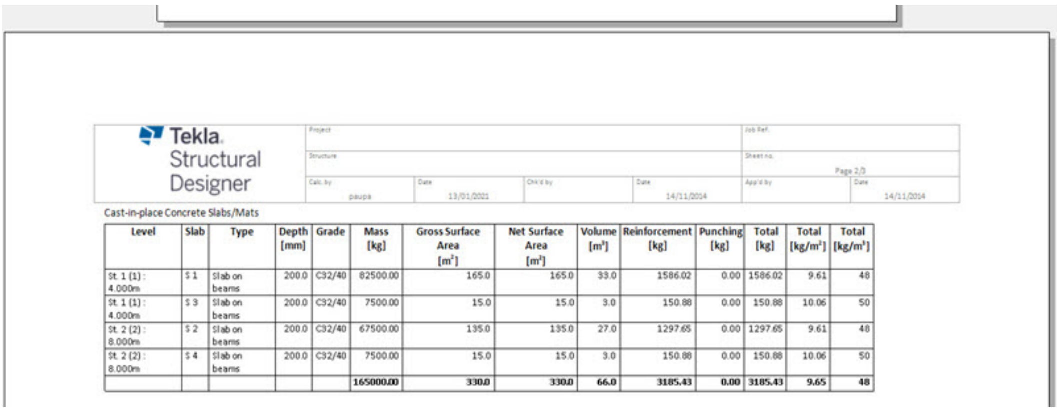
- [TSD-8415] - Concrete Design - Beams - the deflection check has been corrected so that it now compares the position of the neutral axis to the depth to the compression surface reinforcement, identifying if this reinforcement lies in the compression zone and a decision is then made by the program if its beneficial effect should be considered in the calculation of the long term additional deflection multiplier.

- [TSD-1298] - Seismic Design - Steel Braces - for the 2010 and 2016 years of the AISC 360/341 steel resistance code, tension only braces are not permitted for the Seismic Force Resisting System (SFRS) type Special Concentric Braced Frame (SCBF - as directed by the design code). In this release new validation is added to check for this circumstance and a Validation error “Tension only brace not permitted in SCBFs” is issued when this is the case.
    - Tension only braces are permitted in both OCBFs and SCBFs for the 2005 year of the AISC 360/341 steel resistance code, and OCBFs only for the 2010 and 2016 years of the code.
- [TSD-4146] - Steel Beam Design - Torsion - for the Torsion Combined Forces check, in some situations the wrong flange was considered when determining the laterally unbraced length used for the calculation of the critical stress $F_{cr,bx}$ when governed by LTB. This could produce an incorrect failure status for the check when the laterally unbraced length of the bottom flange was incorrectly used for positive bending moment + torsion in beams with the top flange either fully or partially restrained. This issue is fixed in this release.

Reports and Drawings

- [TSD-8532] - Reports - as detailed above, many Material Listing tables have been enhanced with additional columns, making them wider. To assist with their effective display in reports, new automatic Landscape table orientation is introduced in this release (applies to all report tables). This is controlled by a new setting Report Settings > Table Options > “Use landscape orientation for wide tables” as shown in the picture below. When enabled (default) the report page orientation will automatically change to landscape mode when the table header will not fit in portrait orientation.
    - Note that you can reduce the need for landscape orientation and number of pages by; reducing page margins (default values may be quite wide); increasing the allowable table width from 90 to 95% of the available width





Cast-in-place Concrete Slabs/Mats

Level	Slab	Type	Depth [mm]	Grade	Mass [kg]	Gross Surface Area [m²]	Net Surface Area [m²]	Volume [m³]	Reinforcement [kg]	Punching [kg]	Total [kg]	Total [kg/m²]	Total [kg/m³]
St. 1 (1) : 4.000m	S 1	Slab on beams	200.0	C32/40	82500.00	165.0	165.0	33.0	1586.02	0.00	1586.02	9.61	48
St. 1 (1) : 4.000m	S 3	Slab on beams	200.0	C32/40	7500.00	15.0	15.0	3.0	150.88	0.00	150.88	10.06	50
St. 2 (2) : 8.000m	S 2	Slab on beams	200.0	C32/40	67500.00	135.0	135.0	27.0	1297.65	0.00	1297.65	9.61	48
St. 2 (2) : 8.000m	S 4	Slab on beams	200.0	C32/40	7500.00	15.0	15.0	3.0	150.88	0.00	150.88	10.06	50
					165000.00	**330.0**	**330.0**	**66.0**	**3185.43**	**0.00**	**3185.43**	**9.65**	**48**

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Review View - Design Status Enhancements

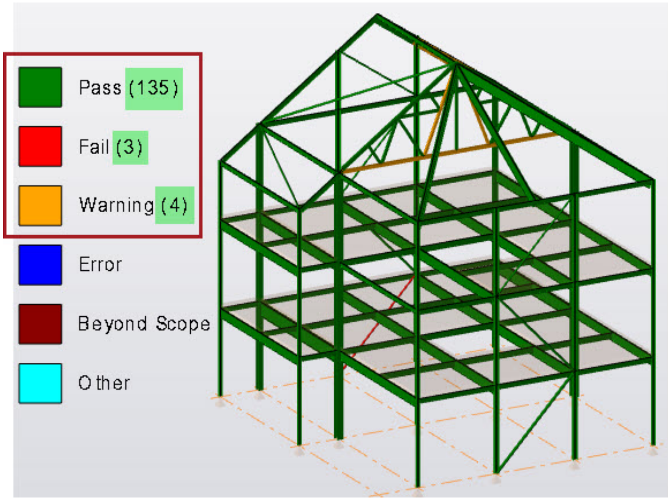
There is a growing emphasis on mitigating the Carbon Impact of design and construction, so we have been focusing on new and improved features to assist engineers with this. For more about this and existing features related to it, see the Help Topic Measuring the carbon impact of a structure (page 1643). This emphasis has led to a renewed focus on efficiency of design in terms of minimizing *material usage* - and thus maximizing utilization - rather than on reduction of direct construction costs through rationalization of sections/ details, potentially at the expense of some increased material and reduced utilization efficiency.

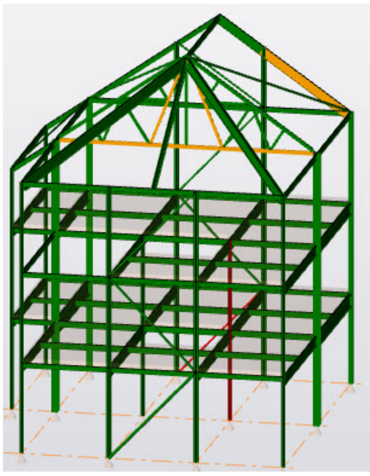
For a detailed discussion of this topic, please see for example this paper from the UK Institute of Structural Engineering: Rationalisation versus optimisation – getting the balance right in changing times, by Ian Poole.

Enhancements are made in the Tekla Structural Designer 2020 SP6 release to assist the engineer in assessing the efficiency of their design in these terms. These go in hand with the enhancements to Material listing (page 106) information also made in this release. Improvements are made to the Review View > Design Status functionality, with the Legends being made more informative and the addition of new viewing filters. Together we believe these enhancements will make the Design Status view even more effective, allowing the engineer to remove 'clutter' and, for example, to focus on objects with low utilization so they can improve the efficiency of their design. For more information see the new Help Topic Design review filters (page 853).

- These enhancements apply principally to the Status and Ratio views for: Member Design, Foundations, Piles, Slab/Mat Design, Connections.
    - Note however that the filters can also be used in the Review View with no Ribbon buttons selected.

- **Legend** - when the Status view is active, the total number of entities with each design status is now given adjacent to each color group in the Legend as shown in the picture below. This highlights that Fails/ Warnings exist, even when the failing entities may be obscured in the current view - the engineer can then adjust the view to find where they occur. This is also the case for the (Utilization) Ratio view - the total number of entities in each ratio band is shown in brackets adjacent to the band legend color.



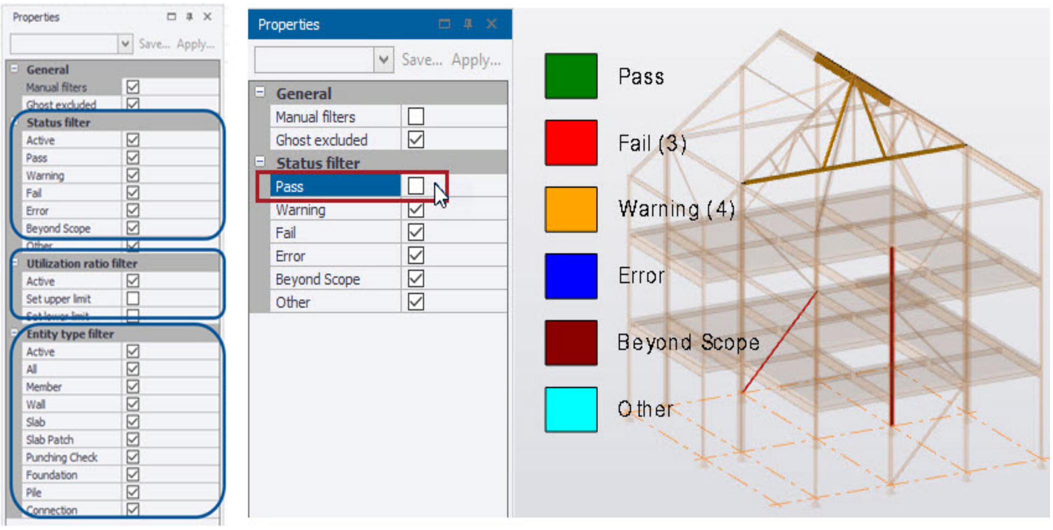


NOTE For the count of entities with each status, each span/stack/panel of continuous beams/columns/walls respectively counts as one entity. So, for example, if all stacks of a four-stack continuous column are failing, then this will contribute 4 to the count of failing entities.

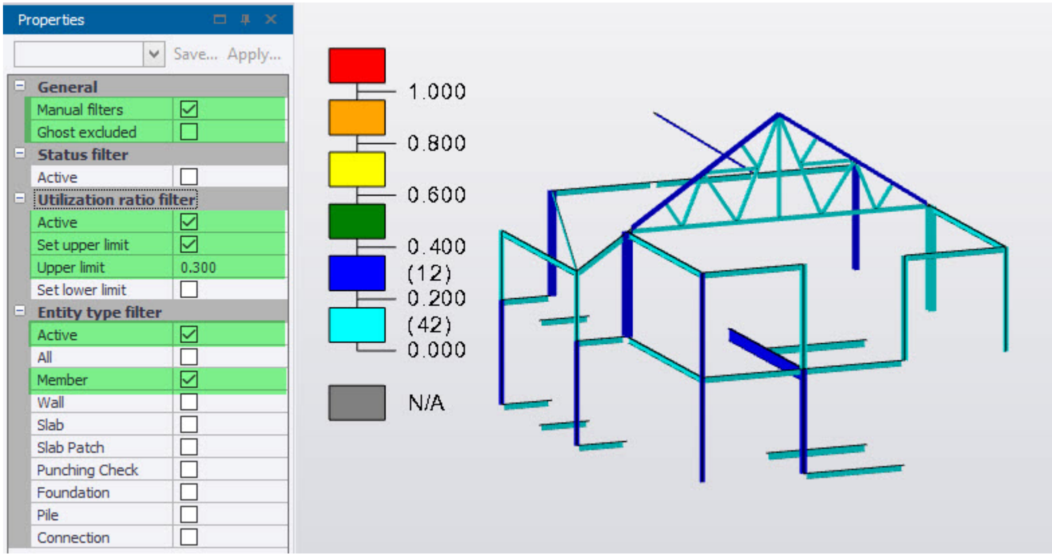
- **New Filters** - the following new Review View filters are now available and are accessed via the Properties window: Status, Utilization ratio, Entity type.
  - These allow the engineer to show/ emphasize in the active view only entities that comply with the filter settings.
    - By default, with the “Ghost excluded” general setting enabled, other entities are ghosted, as shown in the picture below right, making the location of filtered entities in the model clear. With “Ghost excluded” disabled, excluded entities are removed entirely from the view.
  - By default, with the “Manual filters” general setting disabled, when the Status or Utilization ratio Ribbon buttons are selected, only the

associated Status/ Utilization ratio filters are automatically listed in the Properties window.

- You can see all the filters simultaneously by enabling the “Manual filters” setting - you then check the “Active” box to enable each set of filters you require as shown below left.



- The filters can be used individually or in combination and allow the engineer to focus on specific results, typically for status/utilization displays - for example the picture below shows simultaneous use of the Utilization ratio (UR) and Entity type filters, with “Ghost excluded” off, to see only members with a low UR (below 0.3).
  - Note also the number of entities listed for each band in brackets adjacent to the Legend band color as mentioned above - so in this example we know there are 12 entities with UR between 0.2 and 0.3, and 42 between 0 and 0.2



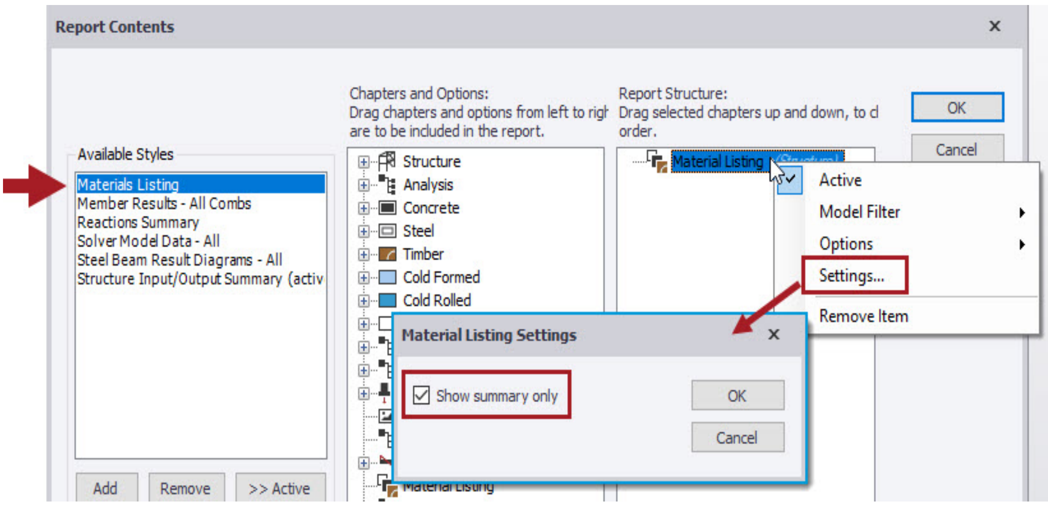
Material List Enhancements

In Tekla Structural Designer 2020 SP6, as a key part of the development work for the recent and ongoing *Carbon Impact* features (for more information on this see the Help Topic Measuring the carbon impact of a structure (page 1643)), the Material List data tables are significantly enhanced, both for the Review View > Tabular Data and the associated Report item. Having the material content of the model in more detail allows the engineer to more readily assess many aspects of design, including efficient material usage and thus Carbon Impact. For more information on the enhancements, see the new Help Topic Review material list tabular results (page 902) and its subtopics.

- **Reports Levels** - A key enhancement is that there are now two levels of Material List tables - *Summary Only* and *Detailed* - where previously there was just one level. See the pictures below showing the difference this makes for concrete beams for example. The new available levels are:
    - **Summary Only** - this is the default level and is a new very condensed report, giving only the total quantities for each section size for beams/ columns for example.

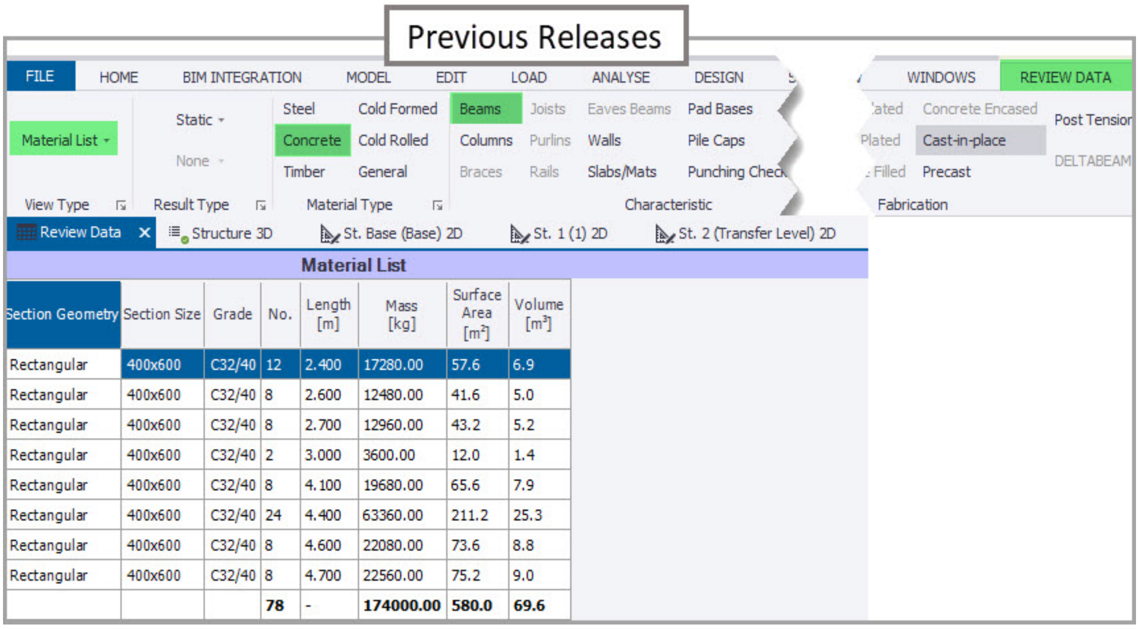
- **Detailed** (“Summary Only” Ribbon button disabled) - the tables for concrete beams / columns / walls now give quantities for each span/ stack/ panel, making them much more detailed than previously. Additionally, the tables for beams and columns now include the detailing group reference.

- **Export to Excel** - note that all the new/ enhanced Material List tables can be exported to an Excel file directly from Review View > Tabular Data for further sharing/ further processing, just as for all other tables.
- **Reports** - the enhancements detailed here also apply to the associated Material List Report item. By default the Materials Listing report will also give the Summary level. A new Setting “Show summary only” is added to control the Report level as shown in the picture below - set this Off to produce a Detailed level report.

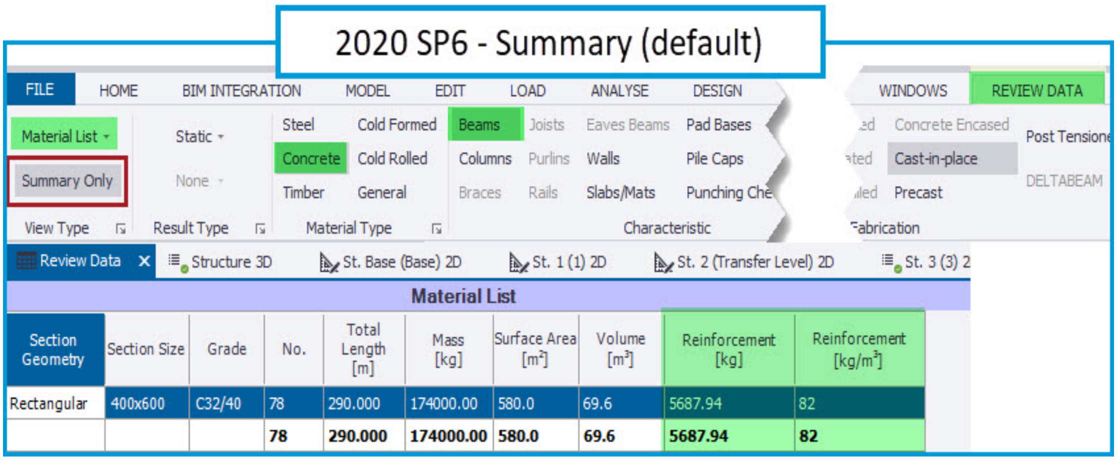


- **Reinforced concrete members** - new Reinforcement quantities - the tables for both levels also now include new columns for reinforcement quantities* giving the total weight (e.g. kg) and weight/ unit volume (e.g. kg/ m3). This applies to all of the following concrete elements; Beams, Columns, Wall, Slabs, Pad/Strip bases and Pile Caps. Note the following:
  - *In general reinforcement quantities are given only for cast-in-place (CIP) fabrication. An exception to this is slabs, for which reinforcement can be specified for example for precast concrete planks (with topping) - such reinforcement will also be included in the material list table for these.

- For concrete beams and columns, at the Detailed level every span/stack is listed separately (or when group design is active this reduces to every group span/stack)
  - For beams, reinforcement running between spans is shared.
  - There is no additional detailing weight allowance (as previously)
- Concrete Slabs - the Detailed table for concrete slabs also includes additional columns giving quantity values for; slab reinforcement, punching shear reinforcement, total reinforcement (slab + punching) as well as the total reinforcement weight and weight/ unit volume. Note the following:
  - The calculation of overall reinforcement quantities is as previously - a user defined additional detailing allowance is still added (specified in Design Settings > Concrete > CIP > Slab > General Parameters).
  - Punching shear reinforcement - this gives an approx weight for punching shear rails (based on ACI guidance)
  - Some approximation also results from slab patch and punching reinforcement - this is shared equally between slab items patches/ punching checks touch, not apportioned by covered areas.



Section Geometry	Section Size	Grade	No.	Length [m]	Mass [kg]	Surface Area [m²]	Volume [m³]
Rectangular	400x600	C32/40	12	2.400	17280.00	57.6	6.9
Rectangular	400x600	C32/40	8	2.600	12480.00	41.6	5.0
Rectangular	400x600	C32/40	8	2.700	12960.00	43.2	5.2
Rectangular	400x600	C32/40	2	3.000	3600.00	12.0	1.4
Rectangular	400x600	C32/40	8	4.100	19680.00	65.6	7.9
Rectangular	400x600	C32/40	24	4.400	63360.00	211.2	25.3
Rectangular	400x600	C32/40	8	4.600	22080.00	73.6	8.8
Rectangular	400x600	C32/40	8	4.700	22560.00	75.2	9.0
			78	-	**174000.00**	**580.0**	**69.6**



Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m²]	Volume [m³]	Reinforcement [kg]	Reinforcement [kg/m³]
Rectangular	400x600	C32/40	78	290.000	174000.00	580.0	69.6	5687.94	82
			78	**290.000**	**174000.00**	**580.0**	**69.6**	**5687.94**	**82**

2020 SP6 - Detailed

Review Data | Structure 3D | St. Base (Base) 2D | St. 1 (1) 2D | St. 2 (Transfer Level) 2D | St. 3 (3) 2D

Material List

Detailing Group	Section Geometry	Section Size	Grade	No.	Length [m]	Mass [kg]	Surface Area [m²]	Volume [m³]	Reinforcement [kg]	Reinforcement [kg/m³]
CRB2-D6 - 3	Rectangular	400x600	C32/40	1	4.400	2640.00	8.8	1.1	66.23	63
CRB3-D2 - 1	Rectangular	400x600	C32/40	4	4.700	11280.00	37.6	4.5	253.62	56
CRB3-D2 - 2	Rectangular	400x600	C32/40	4	4.100	9840.00	32.8	3.9	237.33	60
CRB6-D1 - 1	Rectangular	400x600	C32/40	1	2.400	1440.00	4.8	0.6	102.80	178
CRB6-D1 - 2	Rectangular	400x600	C32/40	1	2.700	1620.00	5.4	0.6	97.90	151
CRB6-D1 - 3	Rectangular	400x600	C32/40	1	4.400	2640.00	8.8	1.1	82.93	79
CRB6-D1 - 4	Rectangular	400x600	C32/40	1	[illegible]	[illegible]	4.8	0.6	43.6[illegible]	76
CRB7-D1 - 2	Rectangular	400x600	C32/40	2	2.700	3240.00	10.8	1.3	223.25	172
CRB7-D1 - 3	Rectangular	400x600	C32/40	2	4.700	5640.00	18.8	2.3	176.83	78
CRB7-D2 - 1	Rectangular	400x600	C32/40	2	2.400	2880.00	9.6	1.2	289.95	252
CRB7-D2 - 2	Rectangular	400x600	C32/40	2	2.700	3240.00	10.8	1.3	223.81	173
CRB7-D2 - 3	Rectangular	400x600	C32/40	2	4.700	5640.00	18.8	2.3	177.27	79
				78	-	**174000.00**	**580.0**	**69.6**	**5687.94**	**82**

- Other notable aspects of the enhancements are:
- The table order can now be sorted by any column by clicking the column header. Additionally, a new arrow symbol adjacent to the header text indicates the sort order; ▲indicating ascending and ▼ descending order.
- In a similar manner to the Review View > Design Summary table, you can now double click on a row in the active table to locate the entity - or entities - it references in the 3D View of the model. Together with the sort facility discussed above, this makes it very easy for example to locate elements with a high reinforcement weight/ unit volume, that the engineer may wish to focus on.
- **Composite beams** - as shown in the picture below, the table now gives Transverse Shear Reinforcement quantities both in terms of size/spacing and weight (calculation of the weight value follows guidance taken from SCI publication AD 437).

Previously

Review Data

Material List

Section Size	Grade	No.	No. Connectors	Length [m]	Mass [kg]	Surface Area [m^2]
UKB 203x133x25	S355	9	122	6.000	1355.13	49.4
UKB 356x127x33	S355	4	80	6.000	793.75	28.1
		13	**202**	-	**2148.88**	**77.5**

2020 SP6

Review Data

Material List

Section Size	Grade	No.	No. Connectors	Transverse Reinforcement	Length [m]	Mass [kg]	Surface Area [m^2]	Reinforcement [kg]
UKB 203x133x25	S355	7	98	H8 @ 225.0	6.000	1053.99	38.4	125.39
UKB 203x133x25	S355	1	10	H8 @ 300.0	6.000	150.57	5.5	7.90
UKB 203x133x25	S355	1	14	H8 @ 225.0	6.000	150.57	5.5	16.87
UKB 356x127x33	S355	4	80	H8 @ 200.0	6.000	793.75	28.1	80.58
		13	**202**		-	**2148.88**	**77.5**	**230.74**

- **Slabs**:
  - Composite - the table now includes decking and reinforcement details. The reported mass value is also corrected to account for the profiled steel decking section where previously it was for a solid rectangular section and so was an overestimate.
  - Precast (concrete planks) - the following enhancements are made:
    - Manufacturer and reference (e.g. Bison Hollow Core) and reinforcement information is added to the table.
    - Reinforcement data is calculated on the same basis as in concrete slabs (which allows for laps at slab item boundary and user defined detailing allowance). To see more reinforcement detail, you can set the Ribbon Content button = “loose bars” or “mesh”.

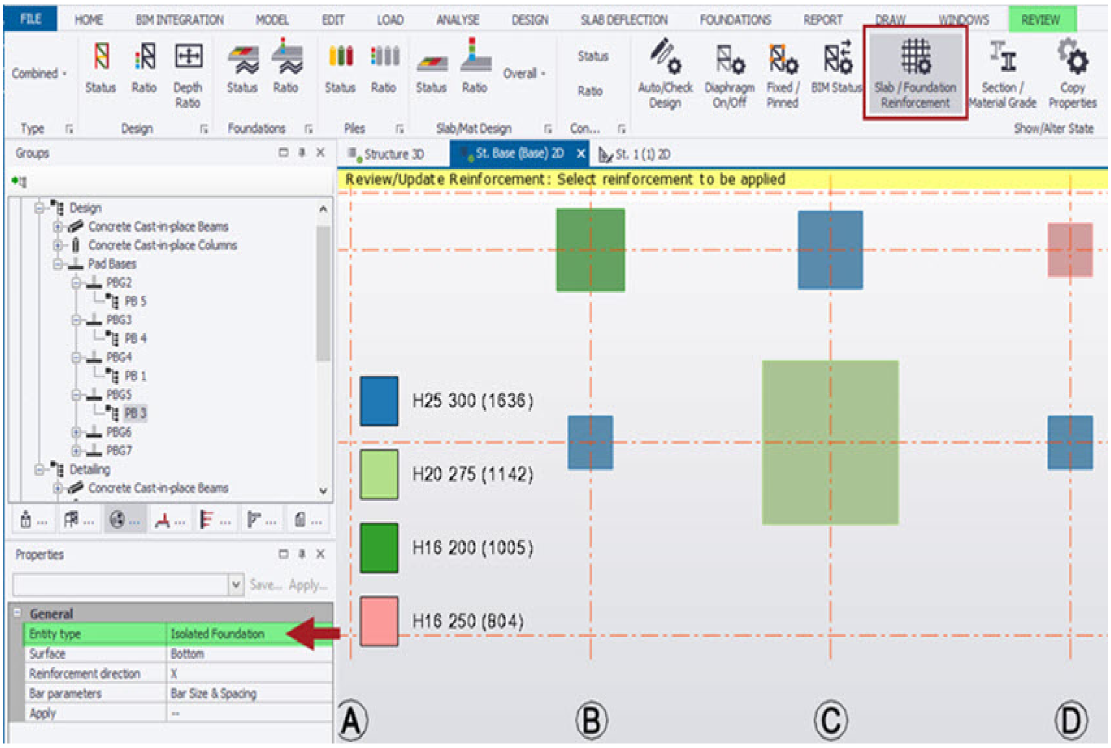
NOTE The Detailing allowance specified in design settings for CIP slabs applies to all rebar defined in any slab type.

    - The reported concrete volume is corrected so it relates only to the topping (if it is specified at all).
- Timber/ General Deck - there is now much more detailed reporting than previously giving; Level, thickness, material grade, and in the Detailed table each slab item reference.

Review View - Show/Alter State Reinforcement command extended to Isolated Foundations

In Tekla Structural Designer 2020 SP6 the Review View > Show/Alter state Reinforcement command is now extended to include Isolated Foundations, enabling rapid graphical review and editing of their reinforcement. This applies to: Column pad bases (spread footings), Wall strip footings and Pile caps.

- To use this new feature, select Entity type = Isolated Foundations in the Properties window, as shown in the picture below. Just as for slabs, you can then select the Surface and or Reinforcement direction for review/ editing.



Release notes: Tekla Structural Designer 2020 SP5

This release will upgrade your Tekla Structural Designer installation to version number 20.0.5.56 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than the immediately prior release 2020 SP4 (version 20.0.4.55 released Sept 2020), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing

- No new license is required for this version.
- **Tekla Structural License Service** - the latest Service Pack for this was released in July 2020. This is available in Tekla Downloads and should be installed on all clients for optimum functionality.
    - For more information see the Release Notes page for the Tekla License Service update July 2020 (v3.1.3.4).
- **License Server Version** - for Server licensing, the latest version of the Tekla Structural License Service 3.1.2 (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) or later to be installed and will replace your current version.
- **Integration**
    - **Tekla Tedds** - to use the new Timber Design using Tekla Tedds feature (page 174), you must install the Tekla Tedds Engineering Library update (September 2020) or later. All Tedds updates can be obtained from Tekla Downloads or via the Update Service.
    - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate this release with Tekla Portal Frame Designer/ Tekla

Connection Designer, you should install Tekla Portal Frame Designer/ Tekla Connection Designer 20 Service Pack 2. This was released in Sept 2020 and can be obtained from the Tekla Download Service.

- **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.01) was released in September 2020 and is available in Tekla Downloads. If you are now using Autodesk Revit© 2021, you can install this to perform BIM integration with Tekla Structural Designer.
  - All fixes and enhancements included in this release are also included in updates for the Integrators specific to the following currently supported Revit© versions; 2020 (Integrator version 6.02); 2019 (Integrator version 5.03); 2018 (Integrator version 4.04). For more information see the Tekla Structural Designer Integrator September 2020 updates. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - Depending which version you are upgrading from, some Databases may be updated by this release. When this release is first run, your Databases will now automatically be updated and it is no longer necessary to do this manually via Home > Materials. See the Release notes: Tekla Structural Designer 2020 SP4 for more about this.

Issues with Associated Bulletins

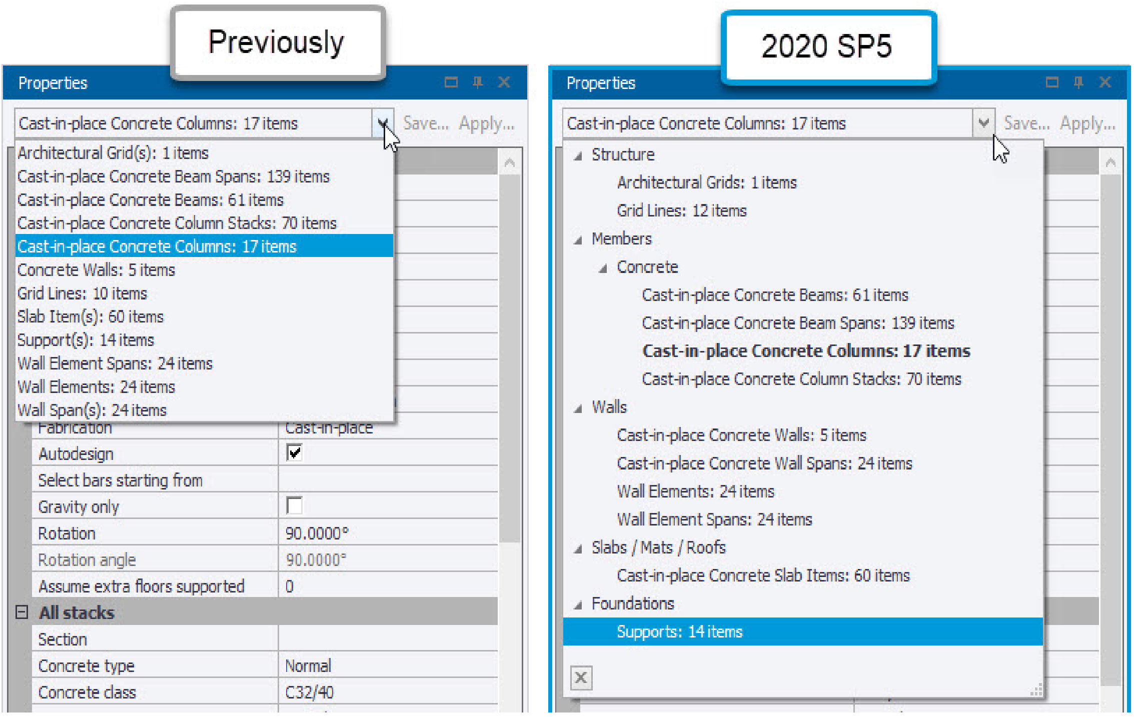
- [TSD-8103] - Piled Mat Foundation Slabs - RSA Seismic Design - For the seismic combinations Static+RSA & Static-RSA and where tension occurred in these, piles were not checked for tension capacity in some circumstances. This could produce an unconservative result where an unchecked tension capacity would govern the pile design. For more information please see Product Bulletin PBTSD-2011-1.
  - This issue is fixed in this release. In addition the pile design report now includes the pile capacity check for all selected RSA combinations

General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of this is given directly below:
  - [TSD-7940] - Report Generation - Analysis Elements with no material (e.g. Linear and non-linear axial/torsional springs) no longer prevent the

generation of reports that contain the following report chapters: Material Listing, Analysis>Elements, Analysis>Element-Member, or Analysis>Element Properties.

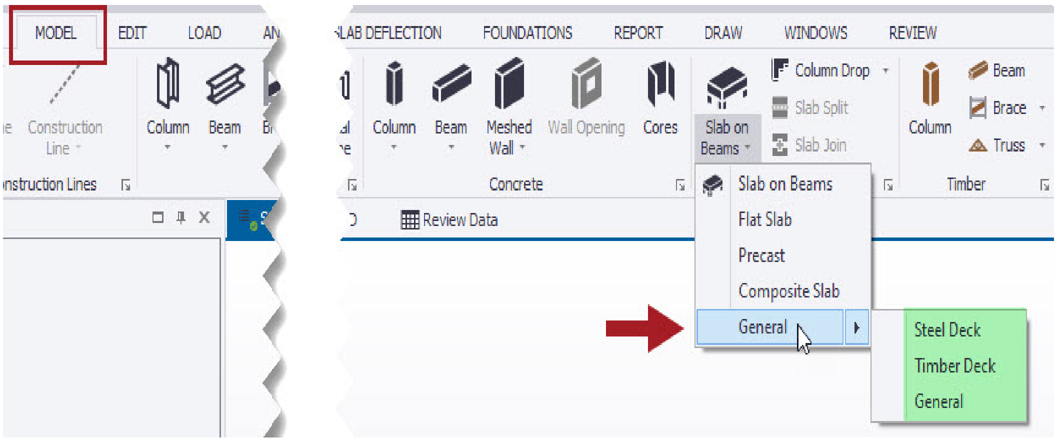
- [TSD-7773] - Selection Properties Grid - the organisation of the property grid dropdown list of currently selected entities has been changed as shown in the picture below:
    - As the range of objects available in the program has increased, a simple list organised alphabetically has become unwieldy in some circumstances, so it is now organised by object, material and fabrication types (in a similar manner to the Entities to be deleted filter dialog).
    - We believe that all users will appreciate this more clear and logical organisation, however we do acknowledge there may be an initial sense of unfamiliarity for longer-term users.



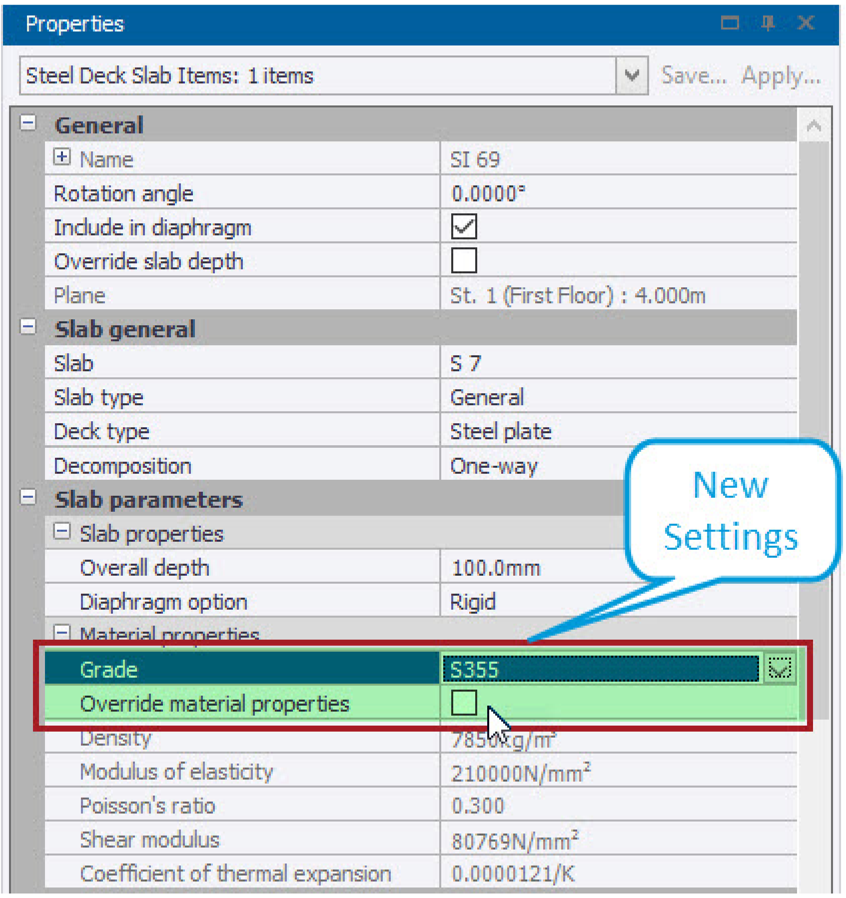
- [TSD-6135] - Export to Excel - the send to Excel export option (for Model and Results and Design tabular data, Wind Tunnel Data and Reports) no longer relies on Microsoft Excel being installed on your computer; if it is not installed an Excel (*.xlsx) file is still created. In addition, the time taken to export large files has been significantly reduced.

- Note that a further difference to previous releases behaviour is that an Excel file (*.xlsx file) is now created in your Windows user Temp directory (C:\Users***user_name***\AppData\Local\Temp where "***user_name***" is your Windows user name). You can then elect to save the file to another name/ location as you require.

- [TSD-7691] - Slabs - Steel & Timber Deck & General - slabs of any General material (e.g Aluminium or any user-defined material) can now be created. In addition, on the Model ribbon the Steel and Timber Deck types are now combined under one common tab 'General' with the inclusion of the new 'General' deck option, as shown in the picture below. General material slabs are created in the same manner as other slabs (slab on beams) and have the same one and two-way decomposition, properties and loading options as the existing deck types.



- Other aspects of this enhancement are:
  - Timber decks can now be set to two-way decomposition.
  - The Imposed Load Reduction % option is now available for all these General slab types.
  - The material of Steel and Timber (and the new general type) decks can now be selected from the material database where previously it was only user-defined. As shown in the picture below, there is a new property for selection of the steel/timber/general material grade. There is also a new "Override material properties" checkbox to allow user-defined properties (when enabled the grade selection option is removed). This option is off by default for new slabs.



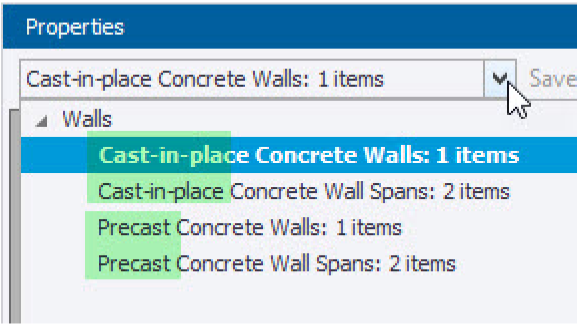
Note the following:

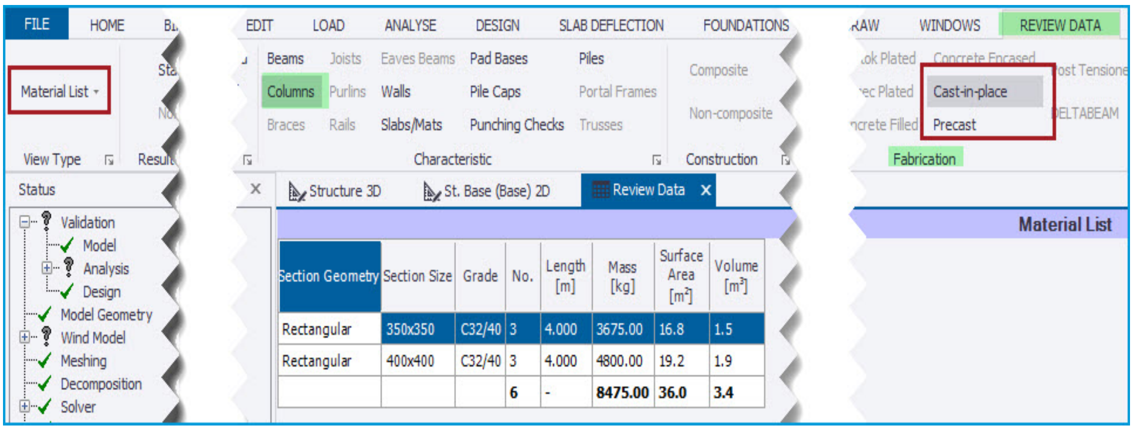
- Existing models - in those with Steel or Timber deck slabs, the "Override material properties" checkbox for these will be enabled and thus the existing properties retained. The user can then decide if they wish to edit this and instead apply a material grade.
- Timber grades cannot currently be directly set for Timber decks because the Poisson's ratio is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. Hence, if you disable the override and select a timber grade, you must then enable it again to enter a valid poisson's ratio (other grade properties such as density and modulus of elasticity will be retained).

- Two-way General material slabs can be meshed in 3D analysis and graphical analysis 2D results obtained just as for two-way concrete slabs (including result lines and strips), however there is no design for them. New Analysis modification factors are also added for them (default value 1.0).

- BIM Integration - General slabs can be imported/exported as per the current functionality of Structural BIM Import/Export. Note that for optimum integration (in relation to material mapping) it is recommended to use database materials for General slabs rather than applying 'override' properties.
- General slabs are also added to the Scene Content settings, and the Review View > Tabular data for material listing (and the associated report item settings) is now separated and enhanced for general material slabs.

- [TSD-7590] - Structural Walls - different types of wall fabrication - e.g. Cast-in-place and Precast - are now separated in the property grid as shown below to allow easier editing of similar wall types in complex models. These different wall types are also now separated when reviewing Material list data, both in the tabular data view - via new Fabrication filter buttons as shown in the picture below - and in the settings of the associated Reports item.

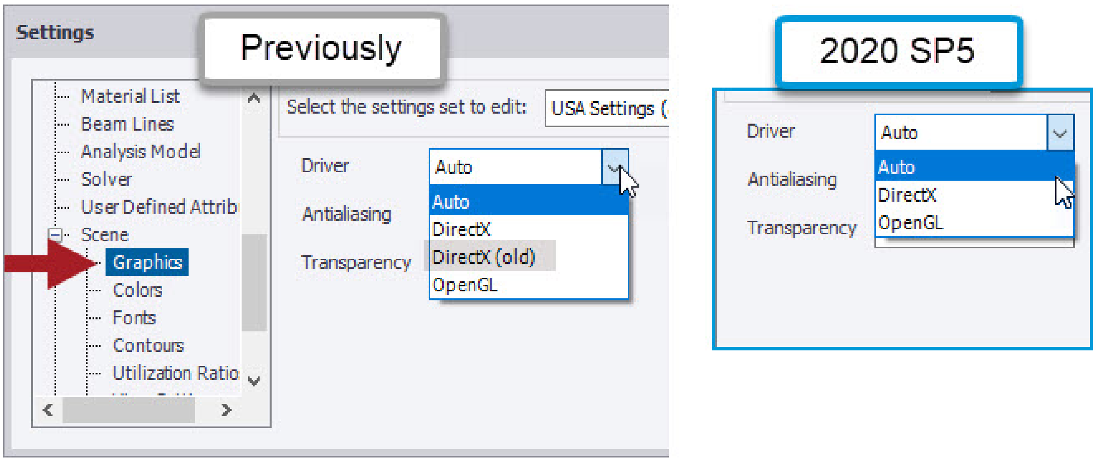




- [TSD-3038] - Copy/ Move Beams and Construction Levels - copying or moving a beam that is created at an existing construction level to a location which is NOT at a level - e.g. between floors - no longer creates a new construction level at this location height. This is important since it has implications for the creation of column stacks and hence design. Note the following:
    - It is not necessary to have a construction level in order to place beams (or cladding rails etc) at a level - e.g. to span between columns between floors. Beams can be connected to columns at any point within their length.
    - Where a beam connects to a column at a construction level, the column will be divided into stacks above/below this level. This has implications for design and is an engineering decision since the stack lengths are used to determine effective lengths and in sway/drift checks.

Performance

- [TSD-7221] - Graphics - the graphics driver settings in Home > Settings > Scene > Graphics are updated and adjusted as shown in the picture below.



    - Note the following regarding this change:
        - The “DirectX (old)” option was based on DirectX 9. This is no longer supported by the latest version of the graphics engine and has been removed from the list.
        - Existing installations set to “DirectX (old)” by default will see this setting automatically updated to the “Auto” option. This setting allows HOOPS (the 3rd party graphics engine used by Structural Designer) to pick the optimum driver and is now the default for new installations. We do recommend checking your current Graphics Driver setting and setting it to the Auto option if necessary.

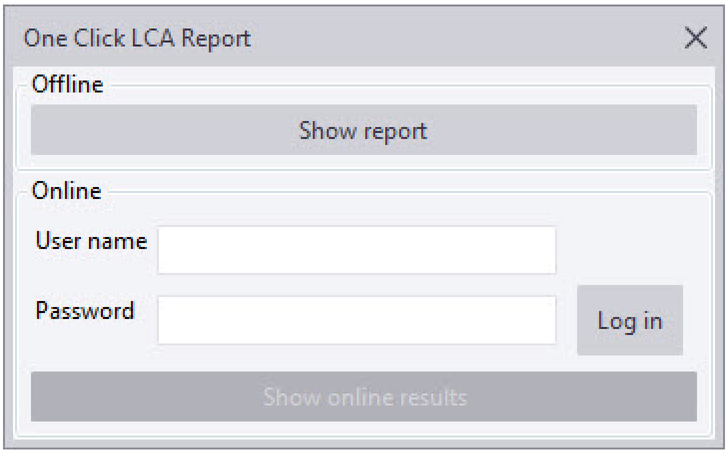
- To check your graphics adapter's compatibility visit the HOOPS developer website for HOOPS 25.xx, DirectX and OpenGL requirements.
- [TSD-7523 & 7545] - Graphical Issues - it is expected that this update will render the program graphics more reliable in general and potentially resolve some more rare graphics issues such as gridlines becoming intermittently invisible and associated crashes.

- [TSD-4843] - Analysis Process - the handling of analysis warnings/errors during analysis is improved to limit the number of these stored and listed with results. Previously, for larger models containing analysis issues (such as instabilities due to mechanisms), this process could occupy significant analysis time/ resources and prevent saving after analysis. The improvement resolves the latter issue and can reduce analysis times for such models from several minutes to a matter of seconds, enabling much more rapid troubleshooting and resolution of analysis problems.

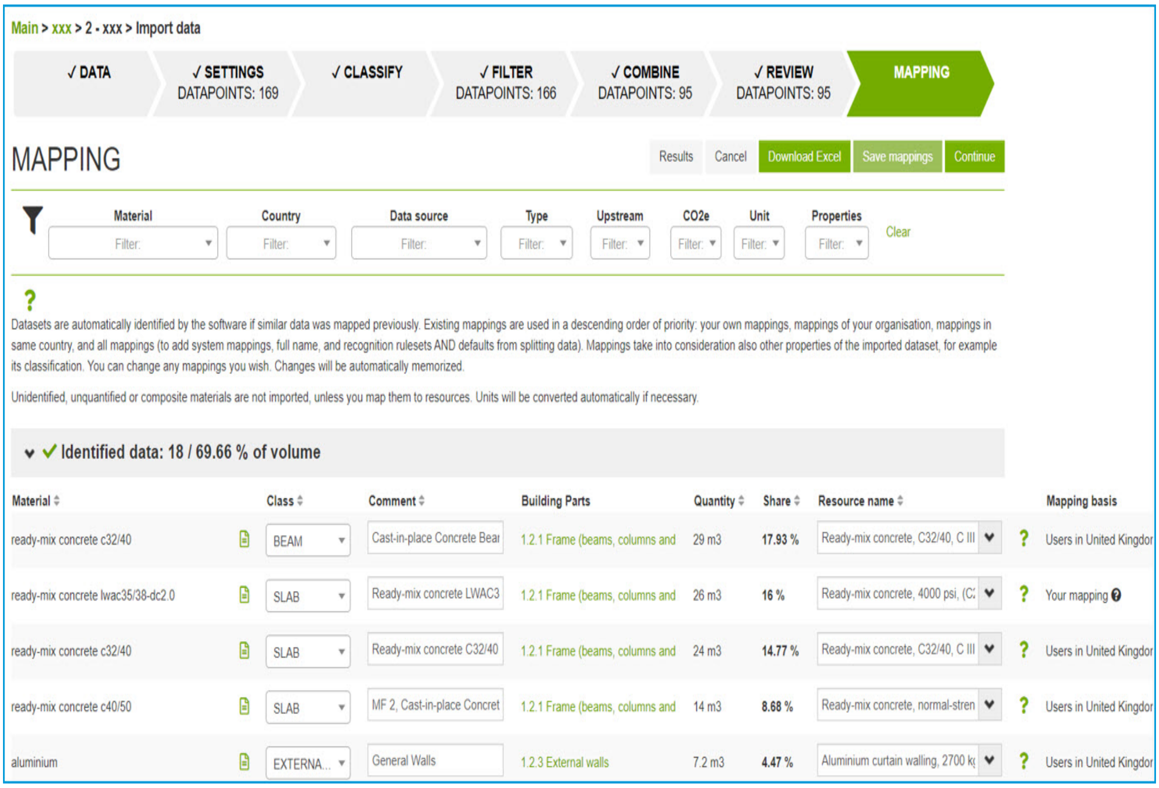
Interoperability

- [TSD-8064] - Carbon Calculation - Export data to "One Click LCA" - the carbon impact of construction is under ever greater scrutiny and engineers are often now required to assess the Building Carbon Footprint of their projects. An exciting and powerful new feature is added in this release to assist with this. The engineer can now easily create a report of the project materials data to quickly determine its embodied carbon using One Click LCA. Note that a subscription to the One Click LCA App is required in order to use this (see the website One Click LCA for more details). As shown in the picture below, a new button for this is added to the BIM Integration ribbon. This automatically compiles a very compact summary of materials in the project which can then either be uploaded to One Click LCA or viewed directly. For more information please see the Help Topic Export to One Click LCA (page 341).

- When the One Click LCA export button is selected, the Report options dialog is displayed as shown below. To view the report directly, click the “Show report” option - the report is written to an Excel (*.xlsx) file, which will open automatically provided you have an installed application associated with this file type. This data file can then be shared with any other parties. Alternatively you can send the data directly to One Click LCA cloud by entering your sign-in credentials (your **One Click LCA credentials** not those of your Trimble Identity) and selecting Log in.

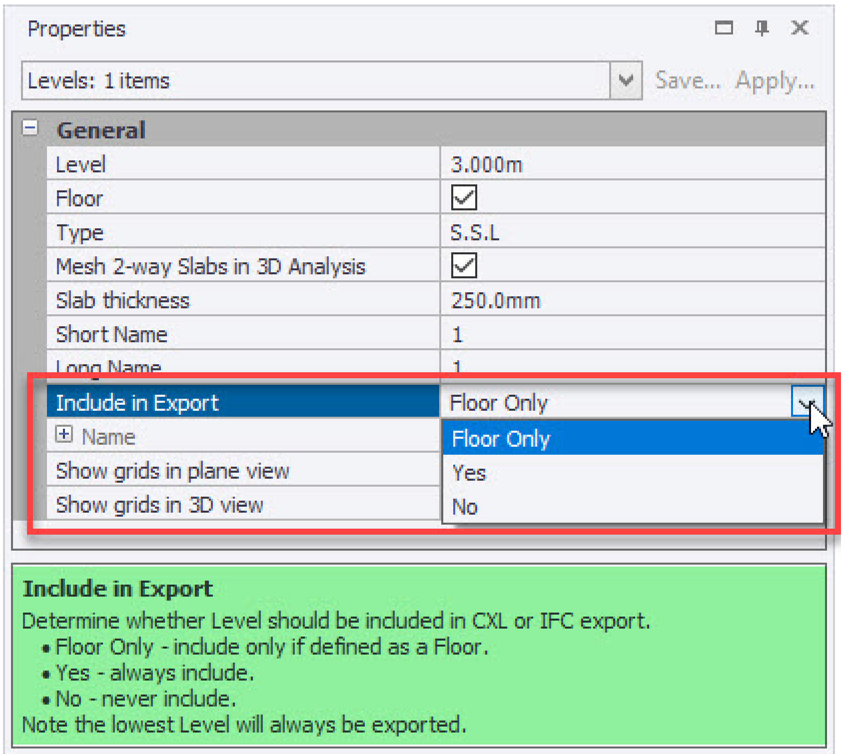


- In One Click LCA you can then create or choose a project for your data, consider groupings and review/ apply mapping. An example of how this looks is shown below. For further information on using One Click LCA see the One Click LCA website.



- [TSD-6321] - Export to CXL (Structural BIM) and IFC - Levels - to give greater control over the export of levels to BIM Applications (both Tekla Structures and Revit via CXL file) and IFC, a new setting “Include in Export” is added to Level properties as shown in the picture below. The setting option determines whether the level is exported as follows:
    - Floor Only - included in export only if defined as a floor (the “Floor” option is enabled in Level Properties/ Construction Levels)

- Yes - always included in export
- No - never included in export.



- [TSD-6970] - Structural BIM Import - Foundations - Wall Strip Bases (Footings) - the import of valid strip footings via the Structural BIM import process is improved to cater for cases where the footing does not extend to one or both ends of the wall. Previously, such footings would be imported automatically as foundation mats. New tolerance has been added such that this will only occur when the overlap between the wall and the footing is smaller than 20% of the wall length on either side, effectively allowing for strip footings up to 40% shorter than the wall length.
- [TSD-6615] - BIM Integration - Column Splices - an issue is fixed which caused an error when importing a CXL file to update an existing Structural Designer model when column splice locations were altered in the BIM application (e.g. Revit).

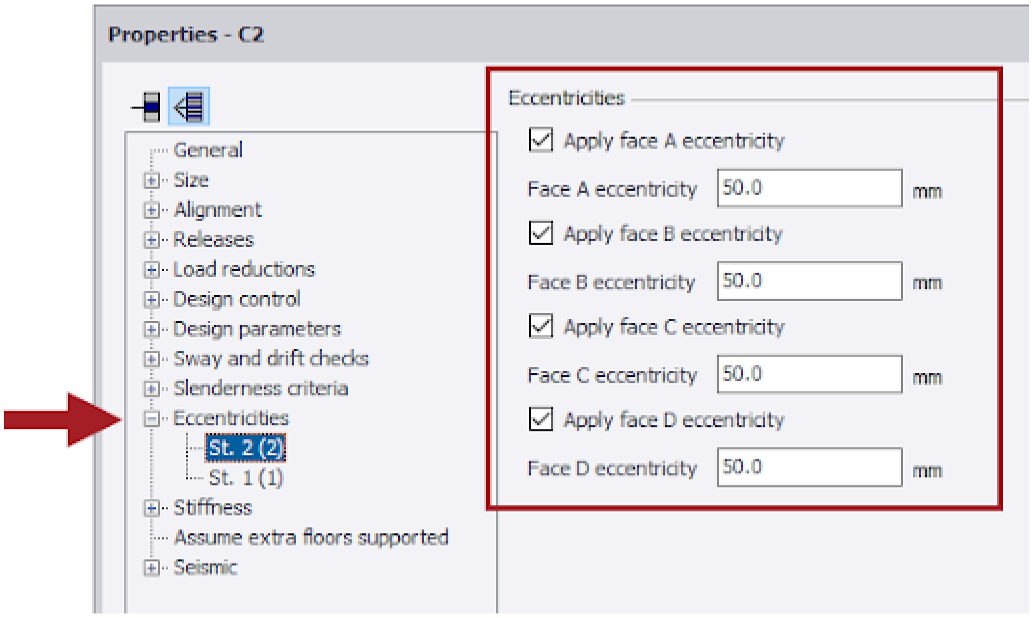
Analysis & Results

- [TSD-7634] - Trusses - Releases - for any truss bottom chord, the axial load release option can now be enabled at both ends simultaneously, where previously only one end was allowed. This allows the easy modeling of “slotted” type connections - which allow some amount of horizontal movement and hence do not transfer axial load - which are commonly used where parallel chord truss bottom chords connect to a column.

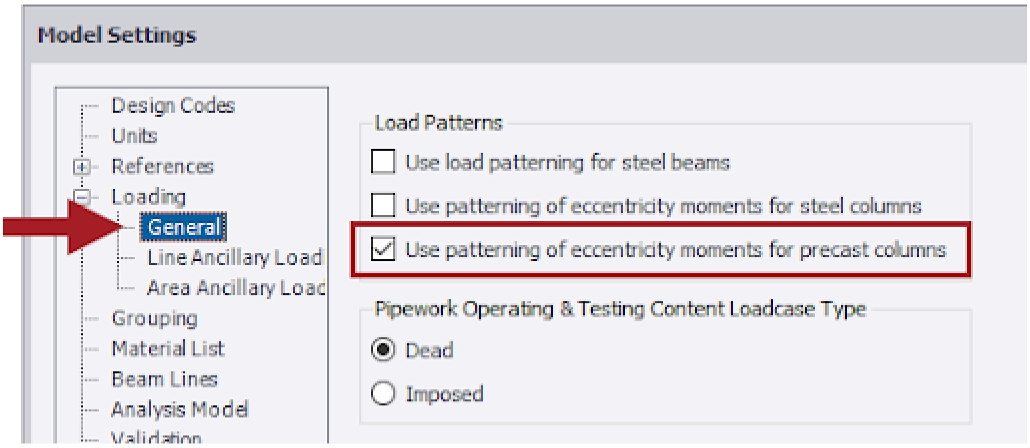
- [TSD-6597] - Solver Model - Rigid Offsets - a more rare issue causing rigid offsets not to be updated correctly in the solver model following some model edits is fixed in this release. For existing models with this issue the solver model will be rebuilt and any existing analysis results cleared when opened in this release - results can then be re-established by running analysis or design.

Design

- [TSD-7930] - Design Groups - Columns - In previous releases, the grouping criteria for steel, timber and cold formed columns were not as stringent as those for concrete columns in that there was no requirement for all group members to share the same section size. This significantly hindered the workflow when attempting to design using groups. This is addressed In this release by the introduction of a new grouping criterion for timber and steel columns requiring section sizes to be consistent within each group. Note the following:
    - For timber design using Tekla Tedds, this removes the previous restriction that made it the users responsibility to ensure all members of column groups were the same size.
    - When a model from a previous version is opened in 2020 SP5, the new grouping criterion is automatically applied. Any existing timber or steel column groups containing mixed section sizes have the inconsistent members removed, (either to another suitable existing group, or if none exists a new group is automatically created).
- [TSD-7606 & 7644] - Precast Concrete Columns - Eccentricity Moments and Load Patterning - eccentricity moments are now automatically calculated and considered in the determination of the moments for the design of precast columns using Tekla Tedds. Additionally, eccentricity patterning of live loads can also now be considered for determination of both axial forces and design moments for precast columns in the same manner as that introduced for Steel columns in the 2020 SP4 release.
    - The following new settings are added for this feature:
        - Column properties - new settings for Eccentricities for each face and level at which beams connect to the column.



- Global and Model setting - new setting under Loading > General to enable patterning “Use patterning of eccentricity moments for precast columns” (on by default for new and existing models).



- Note the following:
  - Analysis of eccentricity moments applies to all Head Codes, however design using Tekla Tedds is only available currently for the Eurocode.
  - Eccentricity moments are calculated only for connected beams that are pinned. They can be viewed in the Load Analysis view in the same manner as for steel columns.
  - The minimum axial force considered by design using Tekla Tedds is capped by the Axial force ignorable limit if it is less than this (default 0.5 kN).

    - A warning for tension is issued if the axial force in a lower stack is below 0.0.
    - For more details on this feature, see Precast column connection eccentricity moments (page 1558).

- [TSD-3801] - Isolated Foundations - Pad/Strip Footings - All Head Codes - the auto-design process for Pad/Strip Footings is improved to increase the depth to attempt to pass the uplift condition where previously only the footing size (plan dimensions) was increased. Where uplift governs this should produce a passing solution in more cases, or a more logical and useful failure where the design size constraints (Design > Settings > Concrete > Foundations > Isolated Foundations > Foundation size) are too restrictive for a passing solution to be found.
  - Note that the user can reduce the maximum side length/ strip width constraint to promote solutions of smaller plan dimension/ deeper footings.
- [TSD-7606] - Slab Design - Column Drops - the design of slab drops and punching shear checks draw some of their design data from the parent slab they are assigned to. Although these entities can only be assigned to one parent slab, in previous releases they would be allowed to overlay different parent slabs and this could sometimes create issues such as incorrect geometry, unexpected design data or invalid objects. This is addressed in this release as follows:
  - Both slab drops and punching shear checks are restricted to the parent slab they are assigned to, on the assumption that connected slab items of identical type are expected to have the same slab parent.
  - For existing models with more unusual situations this could result in changes in slab drop and punching shear check geometry or these objects becoming invalid. Hence for existing models opened in this release, you are advised to review these for any such occurrences. Where they do occur you can consider alterations to the slab items or parent slabs to ensure drops/ punching checks overlay them as expected.
    - Note that for invalid slab drops, their properties can be exposed from Model Geometry > Slab Items in the Status tree in order to change their parent slab.
- [TSD-8109] - Concrete Beam Design - Links (Stirrups) - improvements are made to the creation of top closures for open links. Only one top closer is now created at each position. When there are multiple links at the position, if there is no "outer link" then the top closer will still link the outer legs. Note that this change could result in a (very small) reduction in quantity of reinforcement.
- [TSD-8049] - Review View - Sway/ Drift tabular data - a problem introduced in the previous 2020 SP4 release preventing the sorting of the Sway/ Drift

tables via the column headers (e.g. by lambda/alpha/ drift ratio value) is fixed in this release.

Reports and Drawings

- Reports:
  - [TSD-7728] - Report Creation and Send to Excel - when a report is generated, or report data is sent to Excel, the status of existing analysis and design data is checked and the following warning issued if this is out of date - "Some results are out of date. Do you want to continue?". The warning could often be issued erroneously when it did not relate to the data being reported/ exported. Improvements are made in this release to reduce the occurrence of such unnecessary warnings and the warning message is enhanced to state which results (of Analysis, Concrete Design or Steel Design) are out of date.
- Drawings:
  - [TSD-7601] - General Arrangement - Cold Formed Trusses - new DXF layers for cold formed trusses have been added so that they are now displayed in General Arrangement Drawings where previously they were omitted. The default drawing settings have been updated with these in the settings sets. Note that these can be loaded into existing models via Draw > Settings > Load.

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Release notes: Tekla Structural Designer 2020 SP4

This release will upgrade your Tekla Structural Designer installation to version number 20.0.4.55 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than release 2020 SP3 (version 20.0.3.28 released July 2020), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing & Installation

Licensing

- No new license is required for this version.
- **Tekla Structural License Service** - the latest Service Pack for this was released in July 2020. This is available in Tekla Downloads and should be installed on all clients for optimum functionality.
  - For more information see the Release Notes page for the Tekla License Service update July 2020 (v3.1.3.4).
- **License Server Version** - for Server licensing, the latest version of the Tekla Structural License Service 3.1.2 (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) or later to be installed and will replace your current version.
- **Integration**
  - **Tekla Tedds** - to use the new Timber Design using Tekla Tedds feature (page 174), you must install the Tekla Tedds Engineering Library update (September 2020) which is released at the same time as this Service Pack and can be obtained from Tekla Downloads or via the Update Service.
  - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate this release with Tekla Portal Frame Designer/ Tekla Connection Designer, you should install Tekla Portal Frame Designer/ Tekla Connection Designer 20 Service Pack 2. This is released at the same time as this Service Pack and can be obtained from the Tekla Download Service.
  - **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.0) was released on 1st July 2020 and is available in Tekla Downloads. If you are now using Autodesk Revit© 2021, you can install this to perform BIM integration with Tekla Structural Designer.
    - All fixes and enhancements included in this release are also included in updates for the Integrators specific to the following currently supported Revit© versions; 2020 (Integrator version 6.01); 2019 (Integrator version 5.02); 2018 (Integrator version 4.03). For more information see the Tekla Structural Designer Integrator July 2020

updates Release Notes. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - Some Databases are updated in this release. When this release is first run, your Databases will now automatically be updated and it is no longer necessary to do this manually via Home > Materials. See **Other Enhancements & Fixes**, in the **General & Modeling** section below for more on this.

Highlights

General & Modeling

- New Line and Area Ancillaries - application of loading from ladders, stairs, pipework etc (page 148)
- New Inactive Members (page 153)
- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)

Interoperability

- Enhanced Grasshopper Live Link - Design Results and Reporting (page 162)

Analysis & Results

- Dynamic Analysis - Wind Tunnel Data Report Generation (page 165)
- Walls - Distributed Wall Reactions (page 168)

Design

- Timber Design - Comprehensively Enhanced Timber/ Wood design using Tekla Tedds 2020 - US & Eurocode (page 174)
- Steel Design - New Patterning of Eccentricity Moments (page 188)
- Minimum Design Forces & Rounding Increment (page 192)
- Steel Design - Compound Section Design - Indian Head Code (page 195)
- Concrete Design - Pile Punching Checks on Mat Foundations - Eurocode and US Head Codes (page 197)

Issues with Associated Bulletins

- [TSD-7035] - Shear only walls - these were introduced in the 2020 first release (version 20.0.0.129 March 2020). However, the initial modeling formulation for these had the limitation of failing to report the vertical ‘push-pull’ effects at the ends of the panel that would be expected to be present to resist overturning. Due to the potential for mis-use and mis-understanding this caused, the option to use shear only walls was accordingly removed in the 2020 SP2 release (version 20.0.2.33 May 2020). For more information please see Product Bulletin PBTSD-2005-2.

  - This issue has been resolved in this release and use of shear only walls reinstated. See New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158) for more information.

- [TSD-2529] - Wind Wizard - Finland (Eurocode) Head Code - for Terrain category 0 (Sea), the calculated peak velocity pressure q_p was incorrect and too low, thus producing wind loads which were not conservative. Loads for other Terrain categories were not affected. For more information please see Product Bulletin PBTSD-2007-1.

  - This issue is fixed in this release. Note that for existing models with this circumstance, the Wind Wizard must be run again to correct the peak velocity pressure q_p value.

General & Modeling

Highlights

- New Line and Area Ancillaries - application of loading from ladders, stairs, pipework etc (page 148)
- New Inactive Members (page 153)
- New Lateral Force Resisting System Wall Type - Shear Only Walls (page 158)

Enhancements and Fixes

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of this is given directly below:

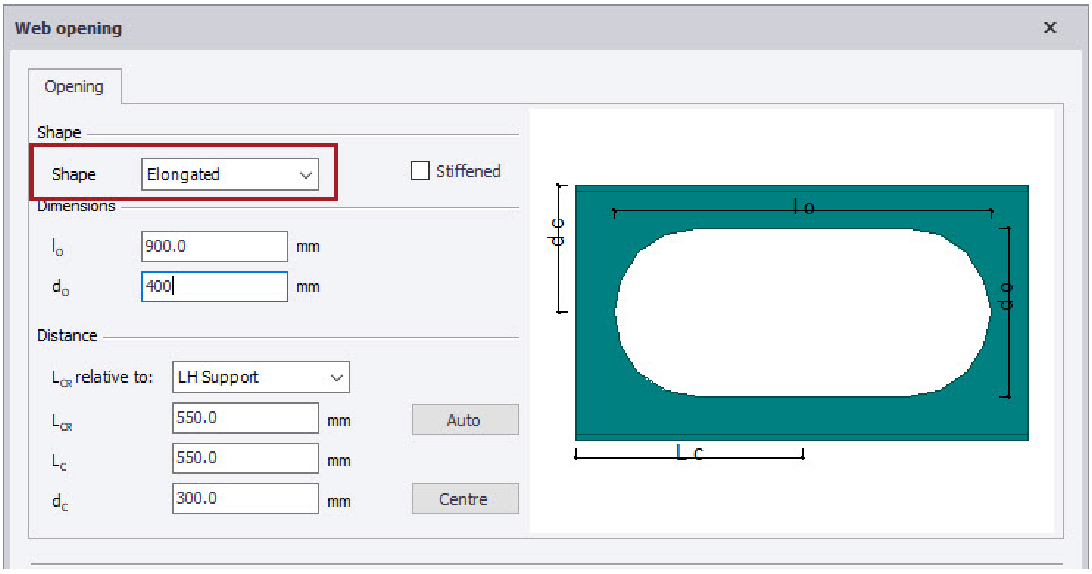
  - [TSD-5746] - Composite Beams and Concrete Beams - significant improvements are made to the calculation process for both the effective width of composite beams (both floor and natural frequency) and flange widths of concrete beams. This improves the performance particularly for very large models with 1000's of beam spans:

    - Composite beams - this primarily impacts the time taken to populate the Properties Window for a selection of beams - a selection of many beams in larger models which previously could occupy a few minutes now takes a matter of seconds.

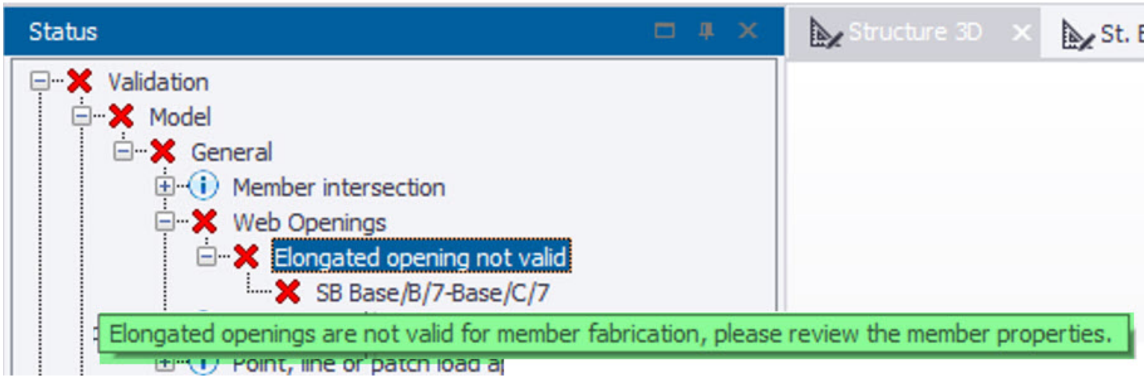
    - Flanged concrete beams - this impacts the time taken for analysis and design and can result in an improvement in performance (in terms of reduced overall analysis and design time) of 2-3 times or more.

- [TSD-4943] - Sub Structures - further to recent enhancements to the creation and management of Sub Structures, additional enhancements have been made in this release so that members created/edited/copied/moved while working in a Sub Structure view are retained in the Sub Structure.

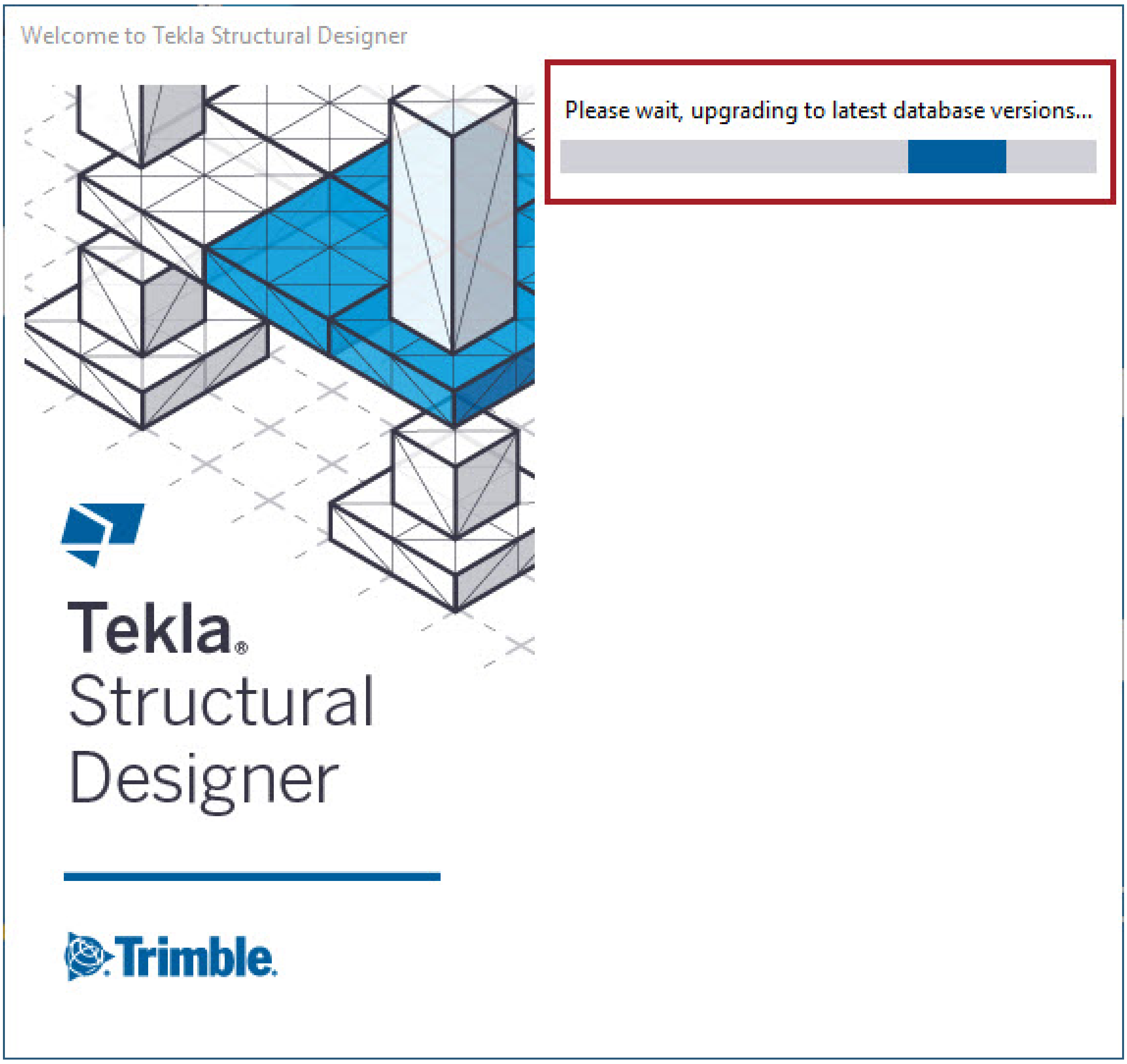
- [TSD-4890] - Isolated Foundations - when the base level of an isolated foundation is moved, the current foundation along with support moves to the new level where in previous releases it would be deleted. This applies to foundations of; columns, walls and portal frame columns.
- [TSD-3672, 7377] - Steel Beams - Plated Fabrication - different types of plated beams can be modeled, including those used in proprietary systems (such as Westok and Fabsec]. Previously, whenever there was a change in plated beam fabrication, the default section size would be applied. This behavior is improved in this release to retain the dimensions of the beam section in this circumstance thus enabling a more convenient workflow.
  - If the corresponding section does not exist in the Steel section database, it is created on the fly (in the model) and can be added to the local database at the engineer’s discretion by reviewing the new sections in the model (via Home > Materials > Model).
  - Note that the same section size - namely the one from the first span - will be assigned to all spans of multi-span plated beams when changing fabrication.
- [TSD-6831, 6834] - FABSEC Fabrication - Openings - elongated openings can now be defined in beams set to the FABSEC fabrication option, as illustrated below. Elongated openings are also imported/ exported correctly when conducting integration with the FBEAM program (for more on this see Export to and import from FBEAM (page 330)).
  - The Web opening dialog features automated validation checks that the elongated opening satisfies recommended dimension limits, as for existing opening shapes. As part of this enhancement, for all openings a warning triangle icon is now displayed rather than an error icon for dimensions that exceed recommended limits but can still be OK’d (at the engineer's discretion).



- Note that, prior to this release, such openings were handled using a UDA - this method is now removed and the UDA no longer used. Any existing models containing equivalent elongated openings can be updated by re-importing the appropriate FBeam .xml file.
- Elongated openings are not valid for other fabrication types, however they will be retained if this is changed from FABSEC. Hence new validation is added to check for this circumstance and a validation error will be reported for it as shown below.



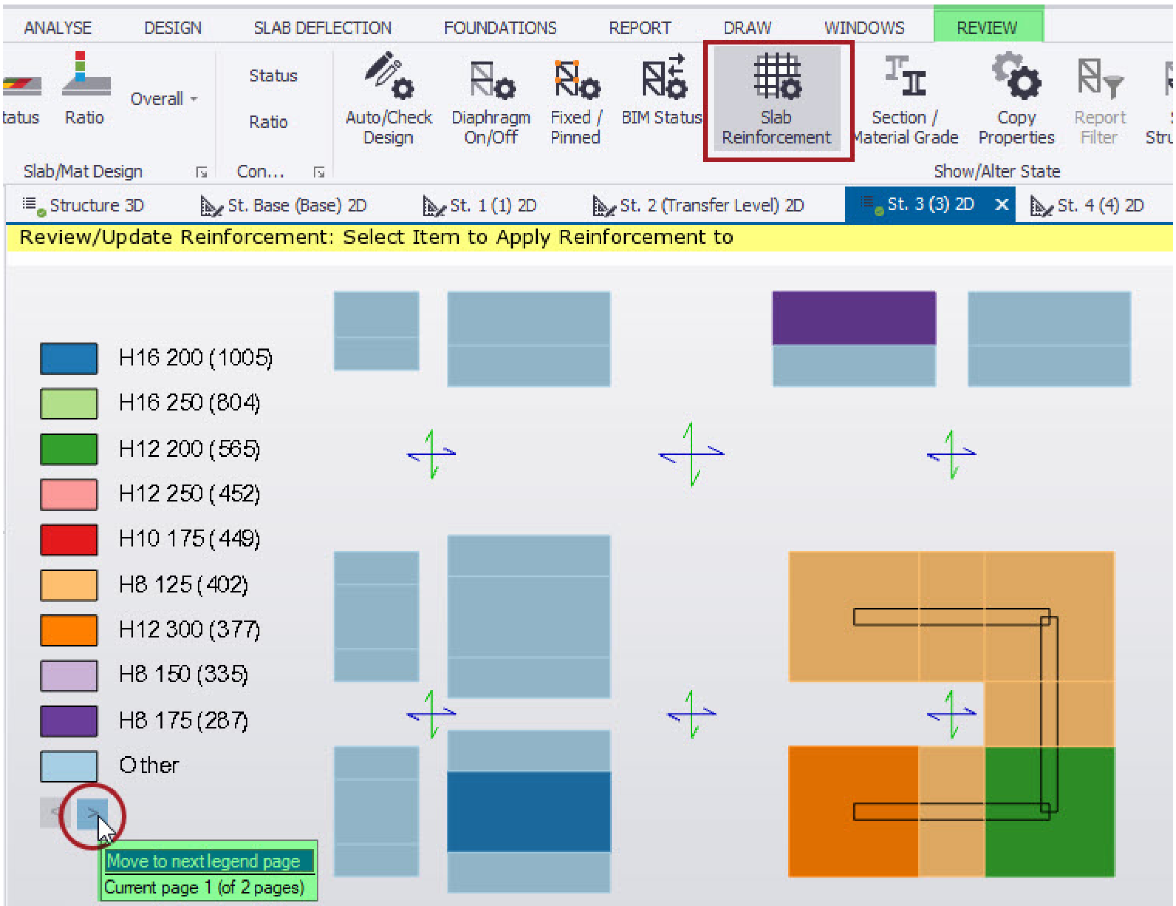
- [TSD-5578] - Databases - as noted in the **Installation** section above, this release introduces automatic updating of your local Databases, so it is no longer necessary to do this manually. This process works as follows:
  - Providing all databases can be updated automatically without user intervention, when this release is first run then the updates are applied and a message is displayed stating "updating to latest database versions" as shown below. On successful completion this closes and the program will open.



- [TSD-7194] - Timber Columns - Splices can now be introduced into timber column stacks in the same manner as for steel columns (via the stack properties) to enable having different section sizes above/ below the splice.
- Section Properties:
    - [TSD-5618] - Timber - the major and minor Shear Area values for rectangular Timber sections are revised to (5/6)*A_g from the previous 0.5xA_g (where A_g = Gross section area). This change aligns the shear area value with that used by Tedds. Note the following:
        - The shear area affects only the component of deflection from analysis due to shear deformation and is not used elsewhere by the program. Moreover the proportion of the gross area used to calculate the shear area - commonly termed the "form factor" - is essentially an estimate and different references on elastic theory

may differ on the value for various section shapes. The factor of 5/6 is commonly accepted for rectangular sections and elastic behavior.

        - Since shear deflections are inversely proportional to the shear area and - for normally proportioned beams - usually account for a smaller proportion of overall displacement, the previous smaller $0.5xA_g$ value would have produced a generally minor overestimate of the shear component of deflections only, hence this update will produce a reduction in deflections of a similar order.
        - To update the shear area values for existing models, go to Home > Materials > Model > Sections then select the timber section(s) listed and select “Update from Database”.
    - [TSD-5427] - Steel - the names of Continental UB “350x175” sections (Nominal depth D = 350 mm and nominal breadth B = 175), have been corrected from the previous “300x175” name according to the manufacturer's catalog. This change is relevant for the regions of Singapore and Malaysia. Note that the issue relates to the name only - the section values were correct per the manufacturer's catalog for “350x175” sections.
- [TSD-3992] - Review View - Slab Reinforcement - the legend for Show/Alter State > Slab Reinforcement is now expanded with multiple page options in the same manner as other options - such as Section/ Material Grade - have been in previous releases. This removes the previous limit of 9 reinforcement variations and enables the graphical review and editing of any number of these. When the number of variations exceeds 9, the next legend page right arrow **>** is enabled as shown in the picture below - you can then move forward/ backward between legend pages to review all the reinforcement variations.



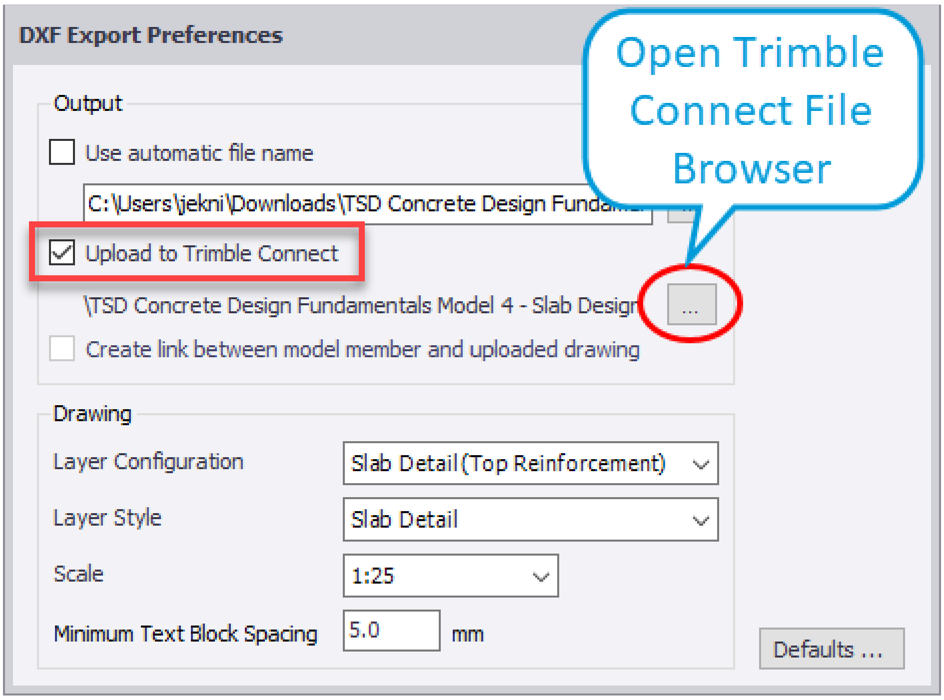
Interoperability

Highlights

- Enhanced Grasshopper Live Link - Design Results and Reporting (page 162)

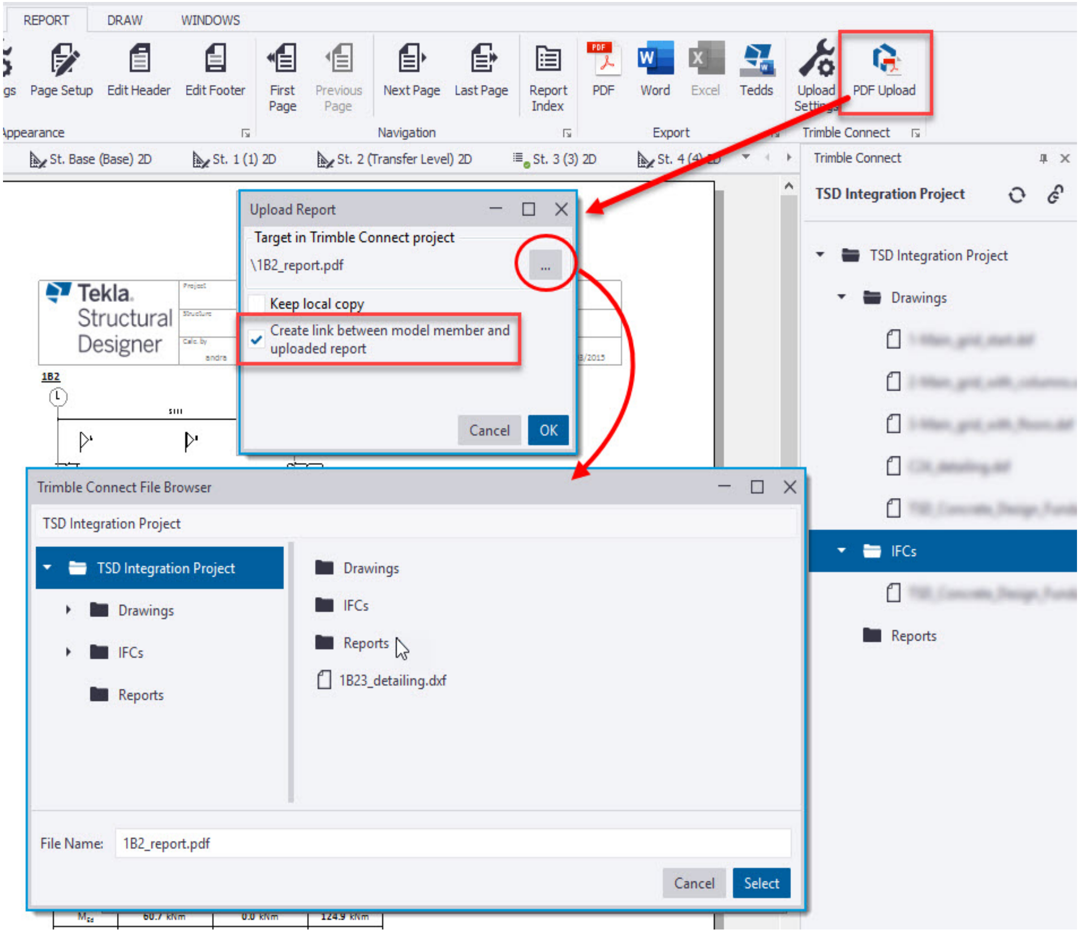
Enhancements and Fixes

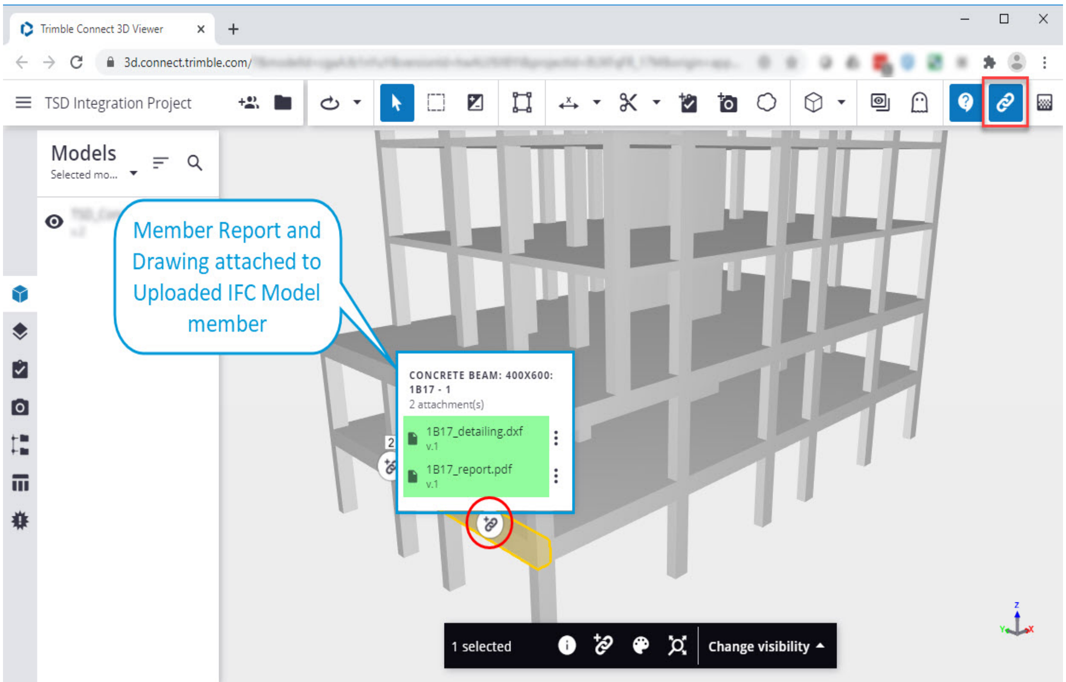
- **Trimble Connect** - Integration with Trimble Connect is enhanced with the following new options:
  - **Drawings** - Upload of any DXF drawing to a linked Trimble Connect Project. This option is listed in the DXF Export Preferences dialog as shown below.



  - **Reports** - Upload a PDF of any Report to a linked Trimble Connect Project via the new “PDF Upload” button in the Trimble Connect group of the Report Ribbon.
    - Both this and the DXF Export Preferences dialog include an option to open the new Trimble Connect File browser to allow the selection of the target project folder for created Drawing/ Report (see picture below).
    - Defaults for the default Target folder in the Trimble Connect project, and whether to keep a local copy of the report, can also be set via the “Upload Settings’ button.
  - **Model Links** - Uploaded Member Drawings/ Reports can also optionally be linked to their specific member in the uploaded IFC model, as shown

in the pictures below. They can then be conveniently accessed directly from the IFC model attachments in Trimble Connect.





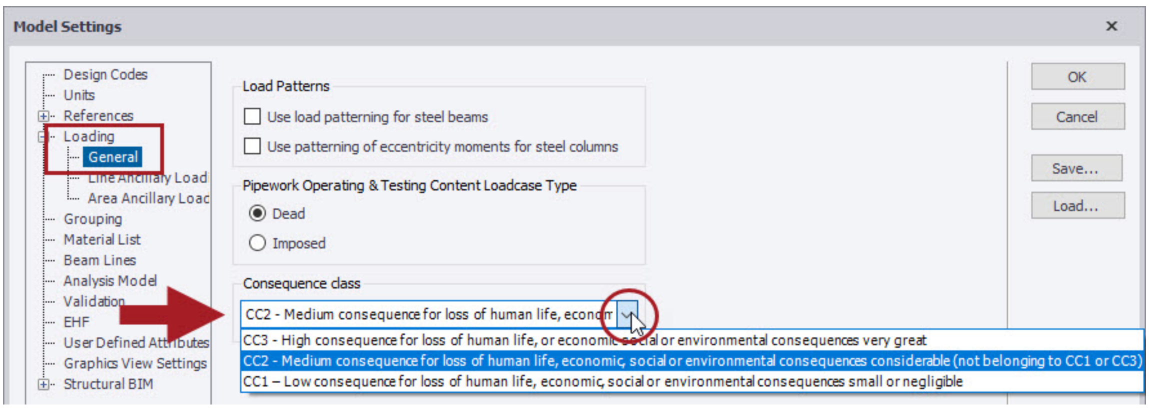
Open API

This release includes further enhancements to the Tekla Structural Designer Open API. For how to get started with the API, see the Tekla Developer Center. New additions to the API enable interrogation by an external application / process of the following objects and items:

- Construction points & point groups
- Planes (Levels / Frames / Slopes)
- Slab items / slabs / slab openings / decks / slab patches
  - For slabs: Reinforcement, decomposition types, diaphragm options, mesh types
- Roofs
- Structural walls
- Wind walls
- Piles

Loading

- Load Combinations - US Head Code:
  - [TSD-3294] - Seismic Combinations - In a similar manner to Gravity and other Lateral combinations, Notional Loads (NL) can now be included in seismic combinations. Addition of NL's to seismic combinations is a manual process at the engineer's discretion via the combinations dialogue - there is no automatic generation option in the Combination Generator.
    - The notional load value is calculated in all cases from the combination's constituent "gravity" load cases - i.e., Self weight, Slab Dry, Dead, Live, Live (Other), Roof Live, Snow.
  - [TSD-4018] - Combinations Generator - the LRFD 1.4D combination is now enabled by default in the Load Combination Generator.
- [TSD-7668] - Wind Loading - US Head Code - the Wind Loadcases dialog "Auto" command now correctly enables the "Roof Loads" option for the "±X'±Y'" cases when the Roof C_p checkbox is selected in load case generator, so all Wind loadcases now have Roof Loads enabled by default.
- [TSD-4678] - Consequence/ Reliability Class - the setting for "Consequence Class" for Finland (Eurocode NA) and "Reliability Class" for Sweden (Eurocode NA) has been moved from the Structure Tree Properties to Settings (Global and Model) > Loading > General as shown below.
  - Note that if the class is changed, existing combinations are deleted and must be re-created as some dead load combination factors are dependent upon the class.



- [TSD-7574] - Notional Loads, Equivalent Horizontal Forces (NL, NHL, EHF) - NL's for previously analysed loadcases from which all loading had subsequently been removed were not updated to reflect this, leading to higher than required NL's being considered. We expect this would be an extremely rare occurrence, however it has been encountered and is addressed in this release. The issue no longer occurs for new models. For existing models which you suspect may have this issue you should:
    - Review the applied notional forces and check they appear reasonable. If required the stored results can be cleared and hence NL's updated by doing one of the following:
        - Use the Save model only command - will remove all stored analysis results and update the NL's when you next analyze/ design.
        - Edit objects in the model such as the geometry of a member or it's supporting conditions - will set results to out of date so NL's are updated when you next analyze/ design.
        - Use the Ctrl+right-click context menu from the Structure tree heading and select the "Clear solver results" option.

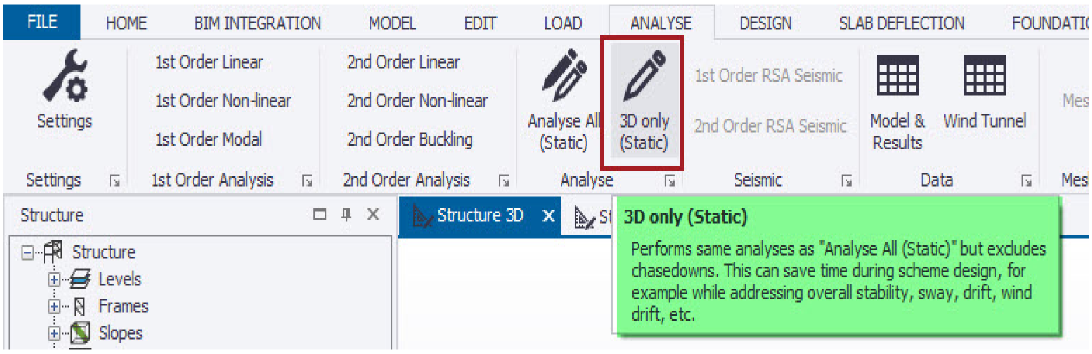
Analysis & Results

Highlights

- Dynamic Analysis - Wind Tunnel Data Report Generation (page 165)
- Walls - Distributed Wall Reactions (page 168)

Enhancements and Fixes

- [TSD-6770] - 3D only (Static) - a new analysis option "3D only (Static)" is added to the Analyze Ribbon as shown below. As the tooltip for this details - this performs the same analyses as "Analyse All (Static)" but excluding the chase-down analyses. It is thus a more rapid form of overall analysis which can be used to obtain/ update the results for all processes that use the 3D Building Analysis results, such as sway/ drift* checks and steel design.
    - *Seismic drift check results are not updated by this analysis when the RSA method is selected for Seismic - as in previous releases, RSA Design must be run to update these.



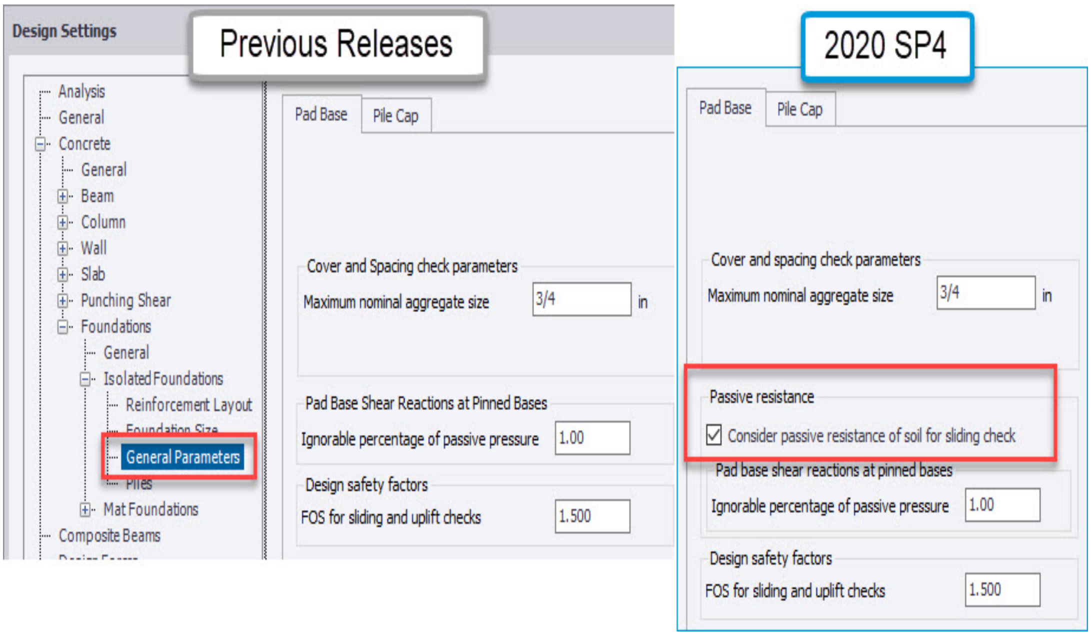
Design

Highlights

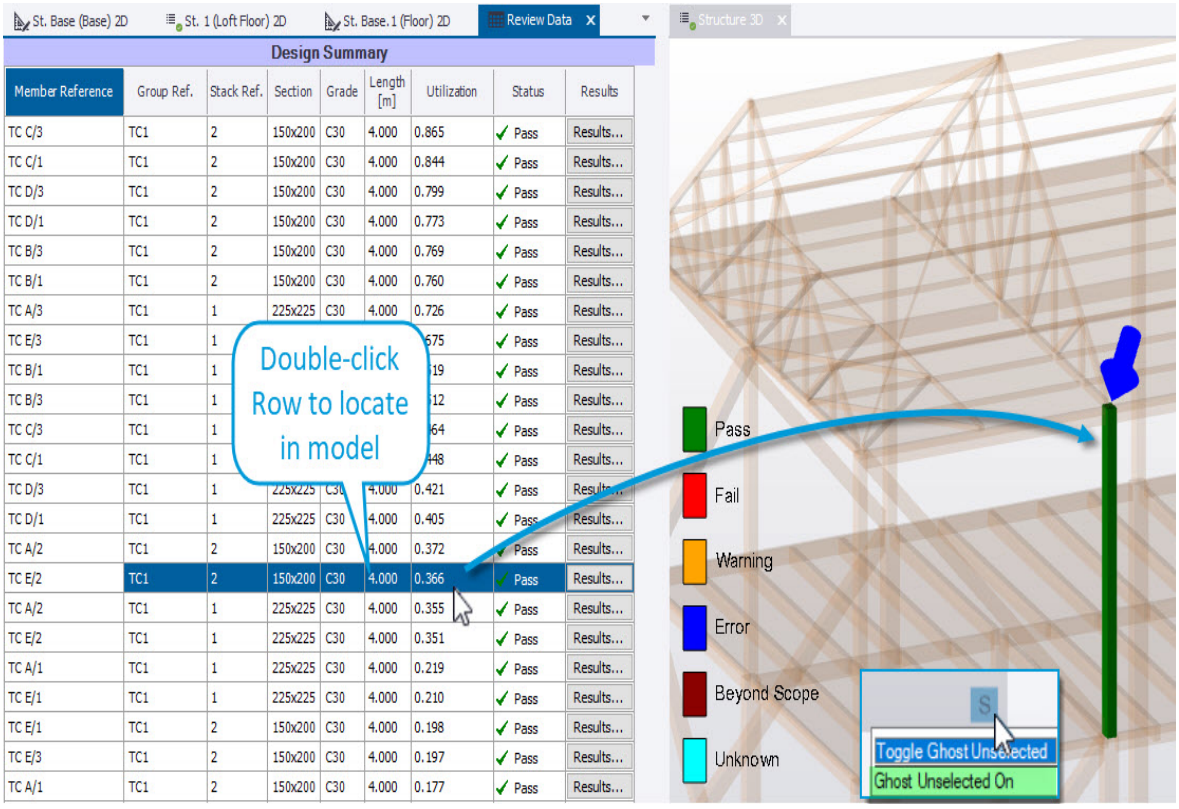
- Comprehensively Enhanced Timber/ Wood design using Tekla Tedds (page 174)
- Minimum Design Forces & Rounding Increment (page 192)
- Steel Design - New Patterning of Eccentricity Moments (page 188)
- Steel Design - Compound Section Design - Indian Head Code (page 195)
- Concrete Design - Pile Punching Checks on Mat Foundations - Eurocode and US Head Codes (page 197)

General Design Enhancements and Fixes

- [TSD-4551] - Foundation Design - Isolated Pad Bases - Sliding Check- a new option is added allowing the engineer to control whether the passive resistance of the soil is included in the check of Sliding Resistance. In previous releases, the soil passive resistance was always included for all Head Codes except the Eurocode (all country NA's) which always omitted it. As shown in the picture below, the new control is a check box in Design Settings > Concrete > Isolated Foundations > General Parameters > "Consider passive resistance of soil..."
  - To match previous behaviour, the setting defaults to on for all Head Codes except the Eurocode (all country NA's).
  - In addition to the new control for all Head Codes, the soil passive resistance is also included in the sliding checks for the Eurocode Head Code (all country NA's) when the new setting is enabled (default off).



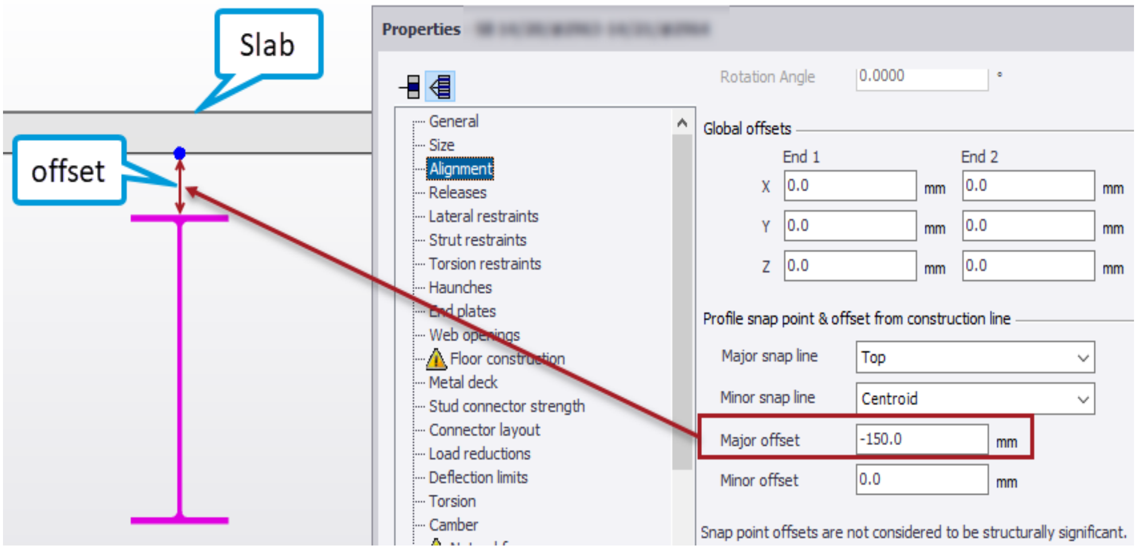
- [TSD-7527] - Design Summary Table - of particular use during the Design Review stage, you can now double-click a row of the Review Data > Design Summary table to select the associated entity in the Structure 3D view. This functions for all entities - beams, columns, walls, slabs etc - and materials which have a Design Summary table, including those for Timber Members introduced as part of the Timber design using Tekla Tedds feature detailed above and illustrated below.
  - As shown, you can arrange the table and Structure 3D view windows so they are both visible to assist with this process (the active view will automatically switch to the Structure 3D view after double-clicking the table row if this is not the case).
  - On double-clicking, the view automatically zooms/ pans to the location of the associated entity which is both selected and indicated with an arrow. Simultaneously the Properties Window is also populated with those of the entity.
  - This feature works particularly well in conjunction with Ghost Unselected On, making the selected entity exceptionally clear as illustrated.



- [TSD-6674] - Concrete Design - Beams - All Head Codes - the design shear identification process has been enhanced such that the design shear force can originate from anywhere within the respective design region. In effect this increases the scope of the design to ensure that the maximum shear force in the region is considered for beams under special and less usual loading conditions such as forces applied within a distance ‘d’ of the support. Also included are minor adjustments and clarifications to the reported check results to make the origin of the design shear more transparent.
- [TSD-7607] - Member Design - this issue relates only to release 2020 SP3 in which it was introduced and design/ check of individual members using the right-click context menu. In the 2020 SP3 release, a change was introduced

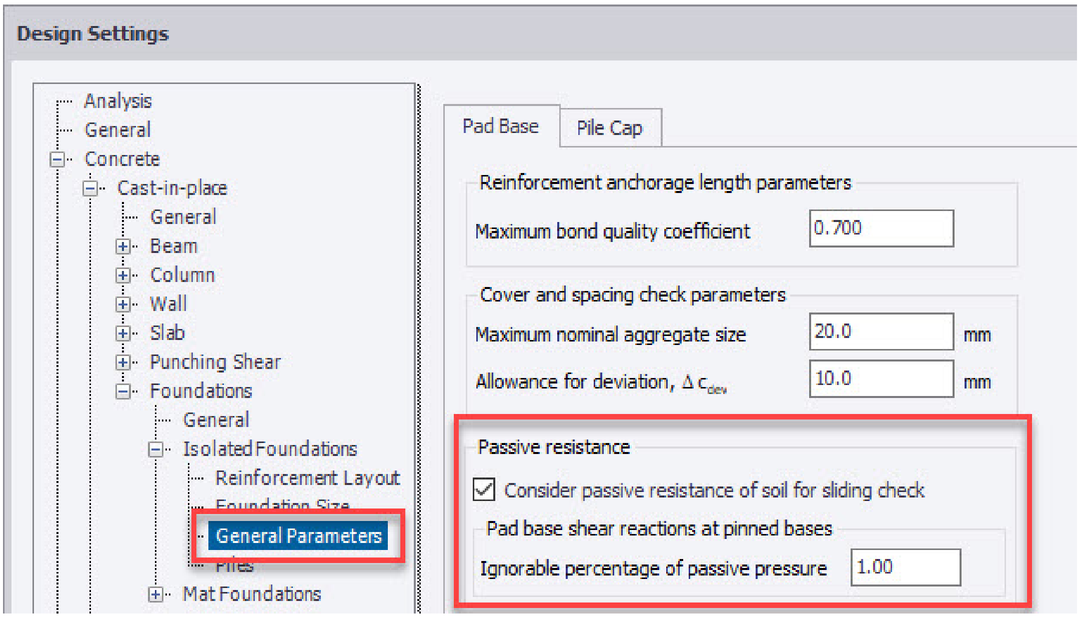
which cleared analysis results on members which were changed by a direct edit - e.g. of their section size. This removed the workflow available in previous versions of investigating alternative member sizes using analysis results which could be out of date, but still considered useful. The change has been reversed in this release enabling this workflow once more.

- [TSD-7223] - Composite Beams - Effective Width - in the unusual circumstances of an alignment offset being applied to composite beams such that the top of the beam was dropped below the bottom of the slab creating a gap - as illustrated in the picture below - the detection of adjacent beams in the automatic effective width calculation was impacted. This could result in the automatically calculated effective width (which can be reviewed/ edited by the engineer at their discretion) not being correctly limited by the beam spacing. Although we consider this to be an unlikely circumstance, it has been encountered and is addressed in this release. Such offsets (which are not structurally significant as noted in the properties dialog) are now correctly ignored by adjacent beam detection and the effective width limited by the beam spacing, where this governs.
    - Note that existing models with this circumstance will be flagged/ corrected by validation in this release depending on the setting Design > Settings > Composite Beams > “Update effective width prior to design check”:
        - When Off - affected beams will be flagged with a validation warning.
        - When On - the effective width of affected beams will automatically be corrected by validation and check/design.



Eurocode Design Enhancements and Fixes

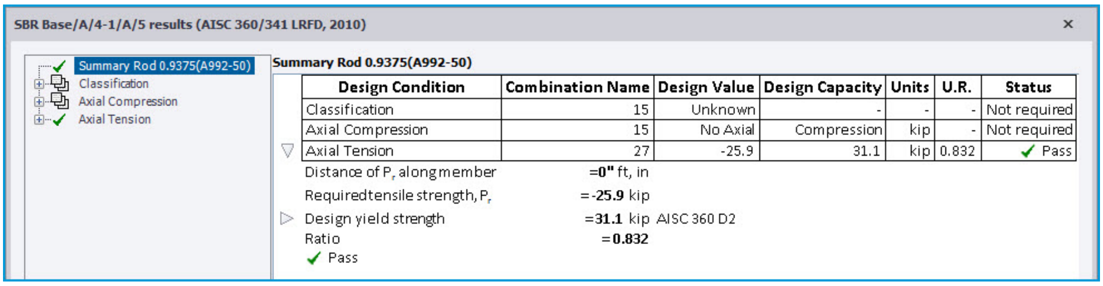
- [TSD-4551] - Foundation Design - Isolated Pad Bases - Passive Resistance - as noted above, the soil passive resistance can now be included in the sliding checks for the Eurocode Head Code (all country NA's) by enabling the new setting for this; Design Settings > Concrete > Isolated Foundations > General Parameters > "Consider passive resistance of soil...".
  - This new option also enables the "Ignorable percentage of passive pressure" value for pinned bases, which simplifies the design for small values of shear values which would commonly be ignored. Previously, this was only available for Head Codes which considered the soil passive pressure (always).
    - The % value default is 1.00 - set the value to zero to consider all forces no matter how small.



- [TSD-6738] - Precast Design (using Tekla Tedds) - Columns - the Tedds Precast column design (EN1992) calculation was recently enhanced (in the Tedds Engineering April 2020 Update) with a basic shear check. The link with Tekla Tedds for precast concrete design is accordingly enhanced in this release to populate the calculation with shear force values from the Structural Designer model.

US Design Enhancements and Fixes

- [TSD-5994, 7579] - Steel Design - Braces - Design is now carried out for steel braces with Rod section type for all years of AISC 360 (ASD and LRFD). The design is only carried out for axial tension, since rod sections are commonly only used for tension-only bracing and ties. Compression is beyond scope - a design details note is issued stating this and that the section is assumed slender and additional hand calculations are required.
    - A new Order list for Rod sections is also added in this release.



Reports & Drawings

Reports

- [TSD-5518] - on selecting Show Report, the report is now opened with the continuous display mode active (so the report can easily be scrolled through with the mouse wheel) and the Project Workspace Report Index is also automatically activated.

Drawings

- [TSD-7000] - Gable Posts - General Arrangement (GA) Drawing - Gable Posts (Steel, Timber & Cold Formed) are now included in GA drawings and in Scene Content > Plan View.

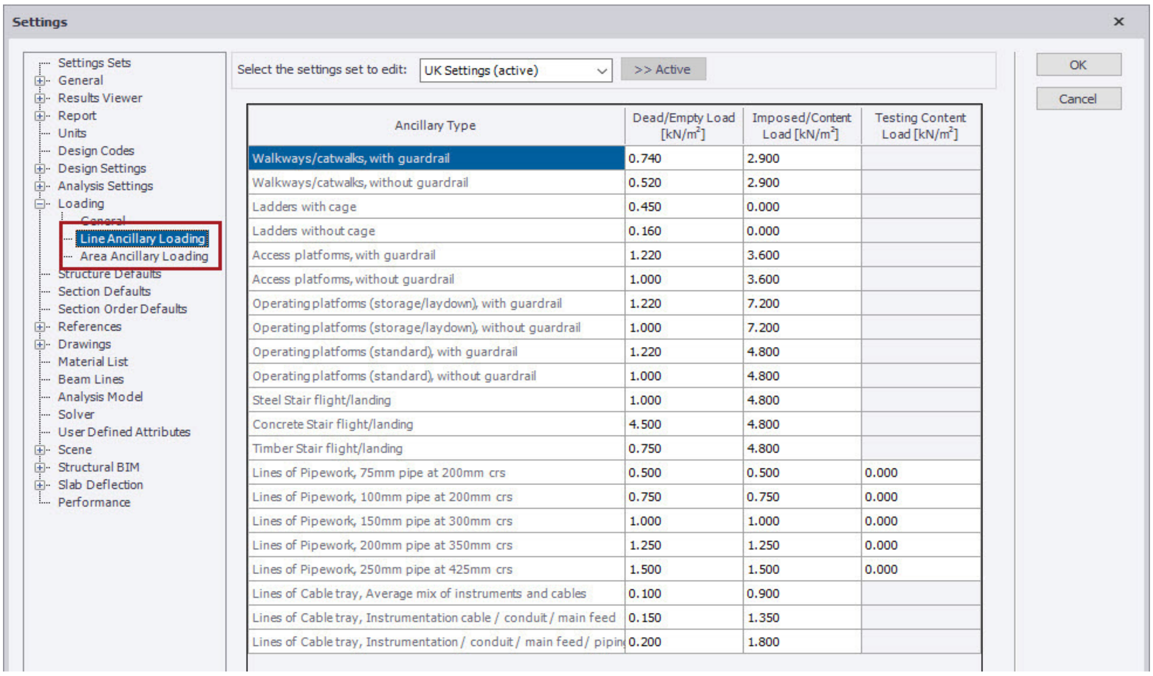
Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

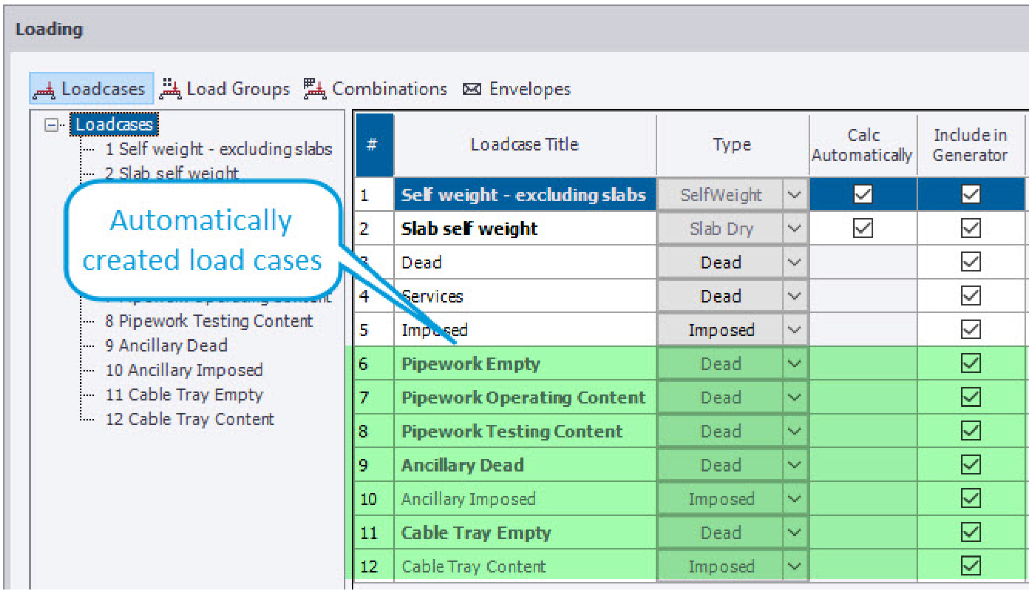
New Line and Area Ancillaries Loading

This powerful new feature enables the quick and efficient application of loading from ancillary items - such as ladders, stairs and pipework etc - that are not part of the main structural frame. These are termed “Ancillary Loading” and can be applied as either line or area objects. While the feature has been developed especially with Industrial Structures in mind, it has a wider potential applicability to many kinds of structures. For more information see the new Help Topic Ancillaries

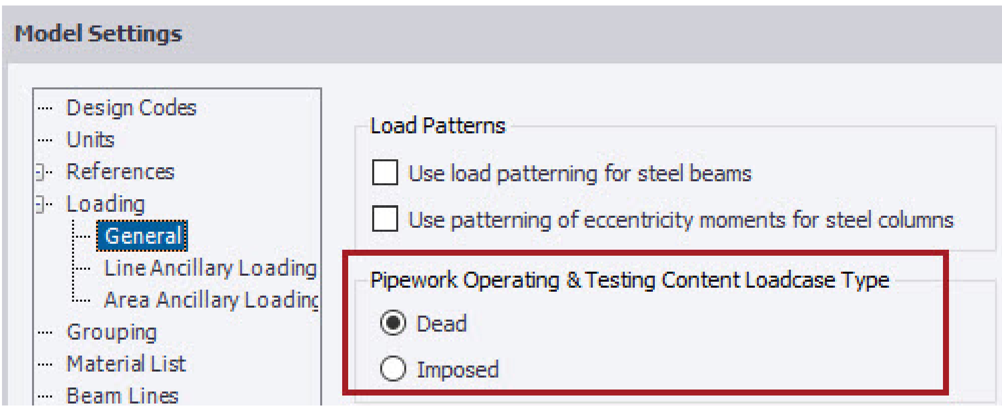
- A comprehensive list of Ancillary types is pre-defined with default loading for both lines and areas. The Global settings for these can be reviewed and their load values edited in the Home > Settings dialog as shown in the picture below. The same settings are also replicated in Home > Model Settings for an individual model.



- Key aspects of Line and Area Ancillaries are:
  - Dedicated Dead and Imposed (Live) load cases for the ancillary loads are automatically created and can be included in combinations produced by the Combination Generator (note that if working to Eurocodes, Imposed case Ψ and ϕ factors default to zero and must be manually defined).

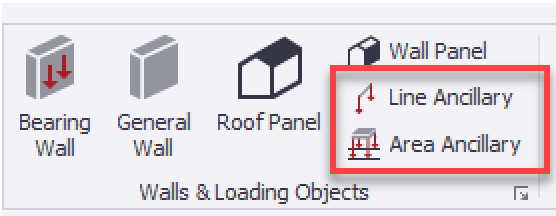


- Default Dead Loads and Imposed/Live Loads for each type are set and can be overridden by the User. These can also be pre-set to be project specific via Home > Model Settings > Ancillary Loading.
  - Note that there is an additional setting for Pipework Operating & Testing Content Loadcase Types, to define whether these are considered as Dead or Imposed (Live) loads. This is found in both Home > Settings and Home > Model Settings on the Loading > General page as shown below.

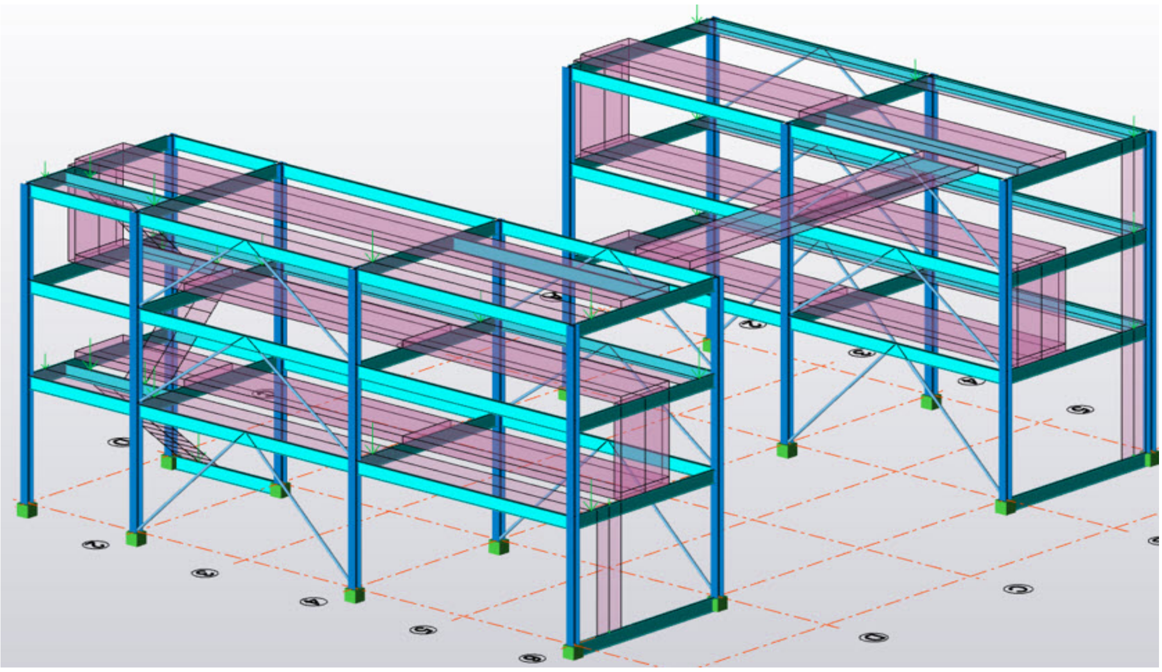


- Imposed stair loading can be projected on the horizontal or along the member.

    - All decomposed loads from Line and Area Ancillaries are present in the Analysis and Design. Once loads are decomposed from Line and Area Ancillaries, they play no further part in Analysis and Design.
- Since they represent physical objects, Ancillaries are applied via dedicated new tools added to the “Walls & Loading Objects” group of the **Model** tab of the Ribbon.

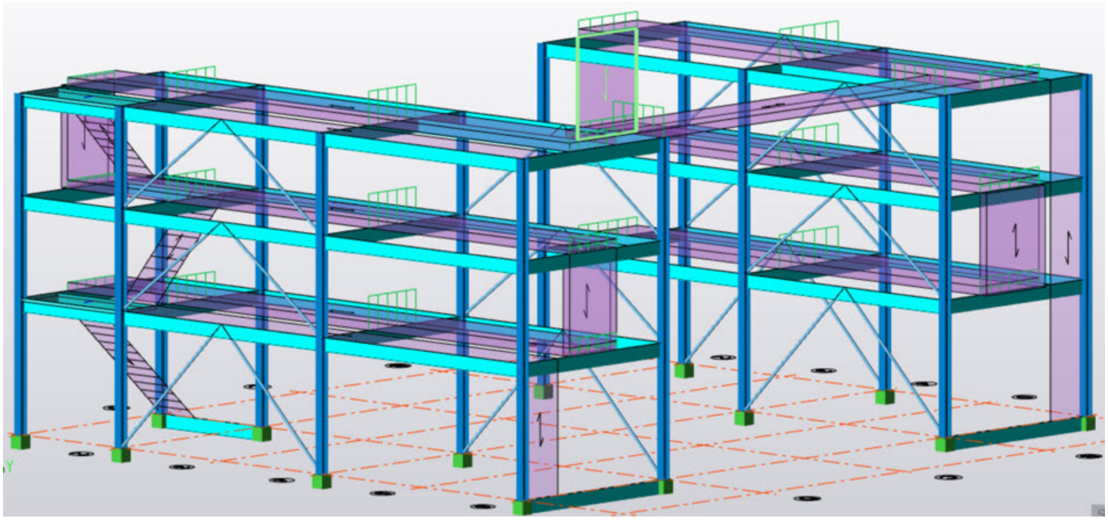


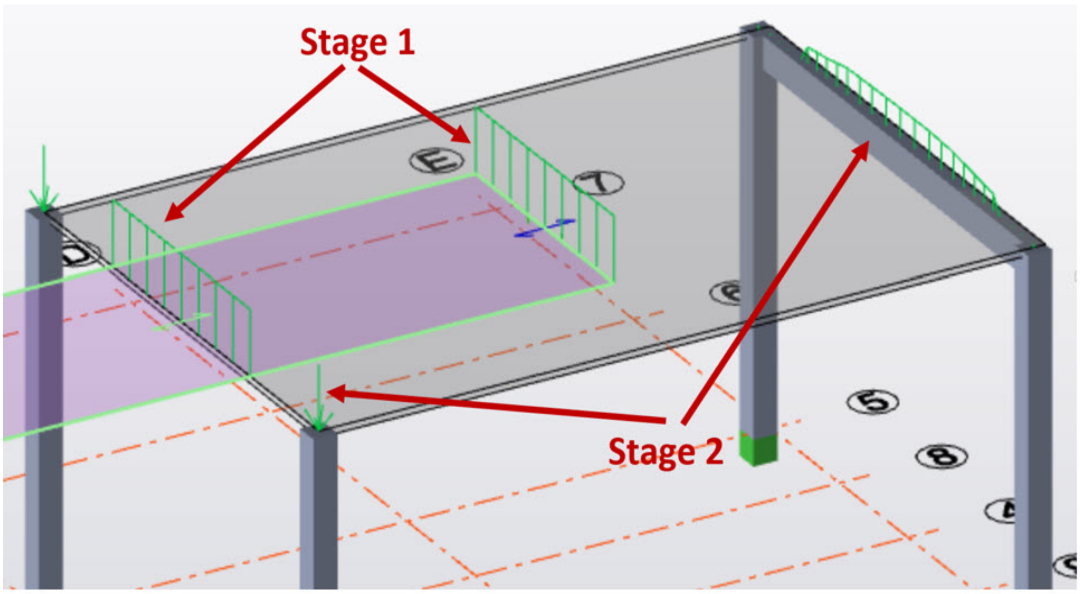
- **Line Ancillary** - creates a rectangular panel of a selected type and defined width which are set in the Properties window during creation (and can subsequently be edited). The *applied* loading is thus line loads determined from the ancillary type area load(s) (as shown above) multiplied by the line width.
    - The panel can be defined singly or in a continuous run of a number of lengths, which can be at different angles and in different planes. They are created just like beams; by single mouse clicks to define the start/end points of each length. Similarly to creating continuous beams, after clicking the final point of the last length you escape to end the operation.
    - The plane of a panel is automatically determined when a length is drawn in an existing plane such as a level or frame. When this is not the case, an additional click after that of the end point is prompted for to define the plane, with a dynamic graphic of the panel displayed as the cursor is moved to aid the user.
    - Once created, each length of a panel is a discrete object which can be selected and edited individually. Its geometry can be edited graphically just like a beam; by selecting and moving the start/ end nodes to different points. The properties of selected individual or multiple panels can be reviewed and edited via the properties window.
    - The Line Ancillary default load values (shown above) are displayed in the properties window during creation. These can be overridden by checking off the “Default Load Values” properties setting, either during creation or subsequently for one or more selected ancillaries.
    - Line Ancillaries are treated as simply supported beams, spanning one-way onto supporting members/slabs. Decomposition is in two stages; from the Line Ancillary to its supports then onwards - the initial load decomposition thus produces point loads at the support points of the line ancillary, which may then be further decomposed - for example in a 2-way manner where support is provided by 2-way slabs.



- **Area Ancillary** - creates a single panel of a selected type of any shape - the *applied* loading is then that of the ancillary type area load(s) (as shown above) over the defined panel area.
  - They are defined in the same manner as existing Panel types by selecting the vertices points and can be horizontal, vertical or sloped. Once created a selected panel's geometry can be graphically adjusted in the same manner as a roof panel.
  - Operation of the panel properties during creation and subsequently is the same as that of Line Ancillary described above.
  - Area Ancillaries have a span direction and load distribution is in this direction onto surrounding supporting members/slabs, just like a Roof Panel. The initial load decomposition thus produces line loads at the end support points of the line ancillary - see "Stage 1" below. These may then be further decomposed by the supporting members - for example where support is provided by 2-way slabs (must be in the same plane

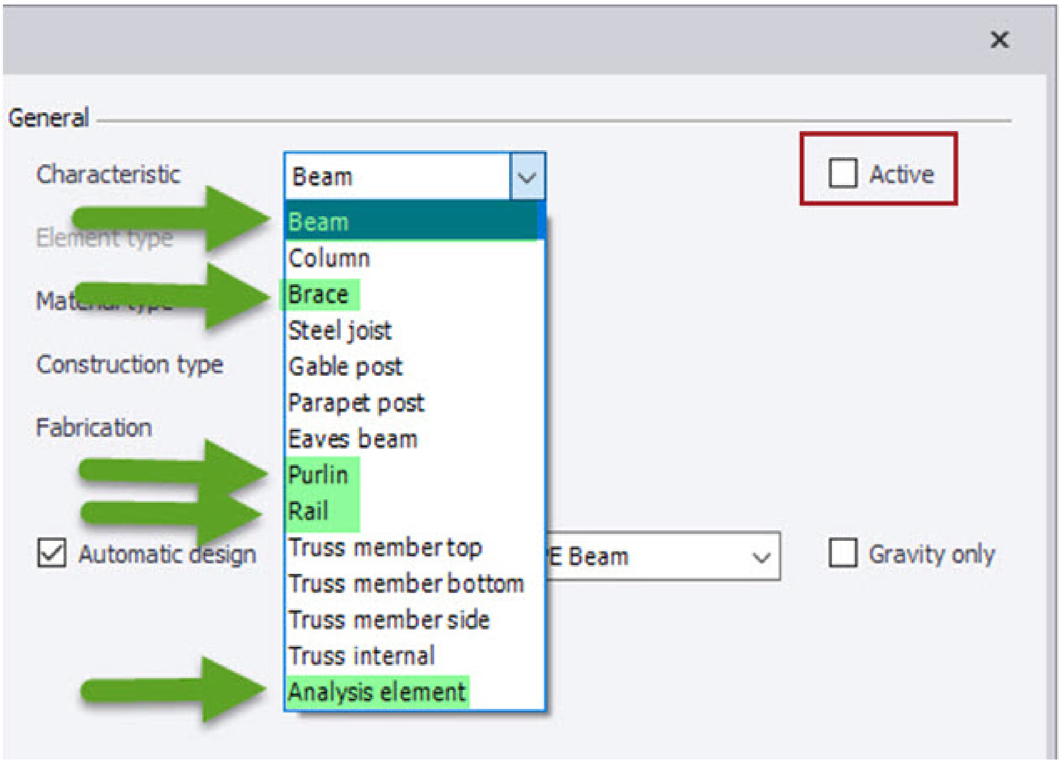
as the area ancillary to provide support) they will be decomposed in a 2-way manner - see “Stage 1” below.

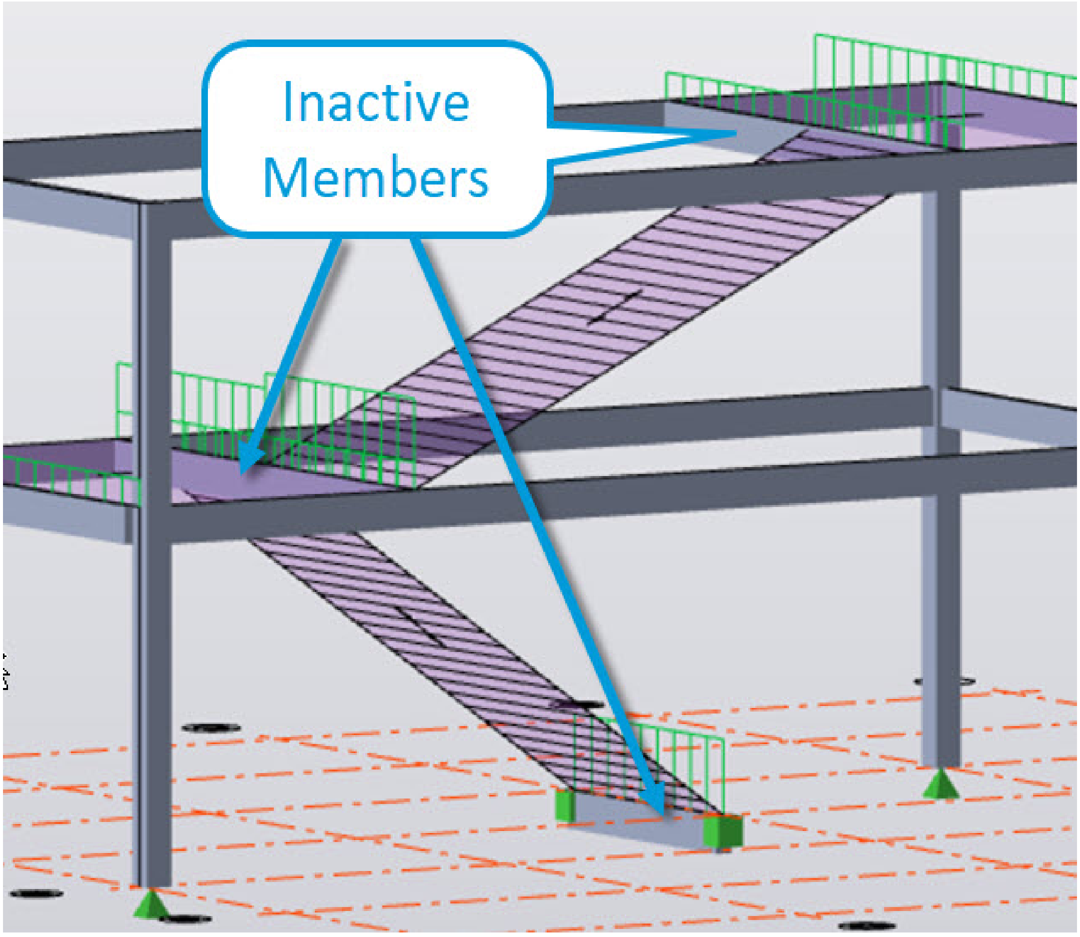




New Inactive Members

This new feature enables the definition of members which are required for load decomposition but that you do not wish to be considered otherwise by the analysis, nor to be designed, and is a companion feature to the new Line and Area Ancillaries feature described above. However it potentially has a considerably wider use and appeal for many types of structure, for example in defining staircases as shown in the second image below. For more information see the new Help topic Inactive members (page 482).

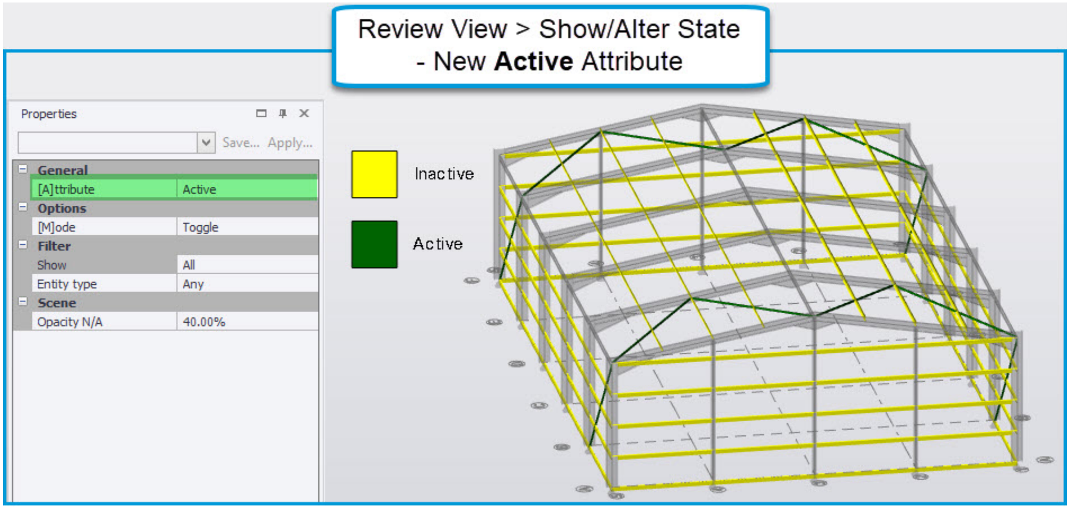


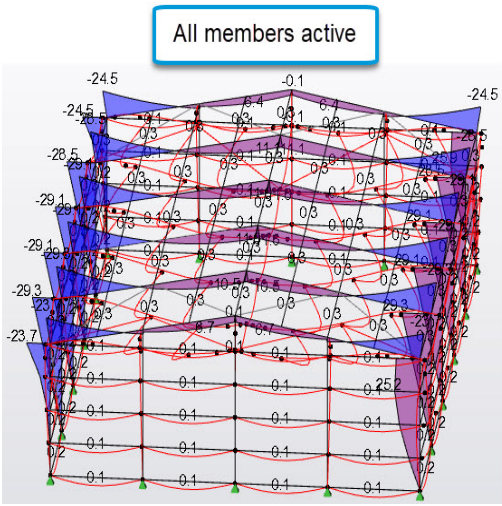


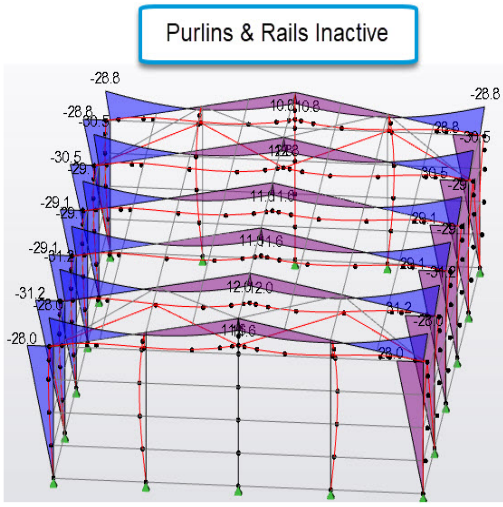
- As shown in the picture above, the option to set members inactive - by unchecking the member properties “Active” checkbox - is now extended to the Beam, Purlin (Track) and Rail (Stud) characteristics (formerly limited to Brace and Analysis elements). Furthermore, all loads applied to inactive members are now decomposed to the rest of the structure.
  - **Load decomposition** - during the pre-analysis load decomposition stage, the inactive members receive decomposed loads as normal (which could be from two-way as well as one-way spanning slabs etc). The resulting end reactions are then calculated and applied during this stage to supporting primary members. Only these resulting loads are then carried forward into the subsequent global analyses.
  - **Review View > Show/Alter State** - a new “Active” Attribute is also added to the Review View > Show/Alter State options, enabling rapid graphical review and editing of the Active status of members (see picture below)

- **Staircases** - A clear general use-case is for staircases and members (that the engineer does not wish to be designed or add additional stiffness to the structure) supporting the landings and top/bottom of staircase Area Ancillaries, as illustrated above right. As shown, the inactive members transfer the area load of the staircase to the supporting primary members/ supports, but will not add any additional stiffness to the analysis model.
    - There can be many other use-case scenarios for industrial structures and line/ area ancillaries e.g. the engineer may be required to allow for equipment and associated access arrangements that will be supplied by others during fit out of a plant.
    - Note another use-case for staircases, could be an area ancillary stair spanning onto a slab edge. This is currently beyond scope, however an inactive member can be added along the slab edge to provide the required load path (requires slab to be meshed in 3D). Note that, while this ensures the load is applied, it will introduce some approximation as the ancillary line load (see “Stage 1” above) becomes two point loads at the ends of the inactive member.
- Other potential use-cases are:
    - **Cladding Members** - secondary/ tertiary members such as Purlins (Studs) and Rails (Joists) supporting cladding are typically modeled primarily to decompose wind wall and roof loads and to provide design restraints to main structural members. They are commonly cold rolled/ formed and so are not designed within the program.
        - However, they may develop internal forces - such as axial loads - and/ or cause additional forces in the main structural members that are neither expected nor desired by the engineer. Furthermore, their results can clutter the Results View Analysis result diagrams, obscuring those of the main structural members in which the engineer is principally interested.
        - Both these problems can now be avoided by setting such members to be inactive, as illustrated below - load applied to the inactive members is applied as required to the main structure (during the pre-analysis load decomposition stage), but a) they are not included in the analysis model and so do not develop any internal forces and

b) hence there are no analysis results for them to clutter the Results View diagrams or output.







- **Grouped Design** - Inactive members could be used to speed up the Design process where group design is enabled, by limiting the number of active members in a group.
    - So for example for regular floor arrangements, where all the beams in the group have very similar forces, only one beam could be set as active and designed rather than every beam in the group (which could run into 10’s or more) thus improving performance. For this scenario note that:

        - Only secondary beams can be inactive - primary beams must be active to provide support to secondary members in the analysis model.
        - The user is responsible for choosing critical beam(s) (an initial design run could establish this then all other beams inactivated for subsequent design runs).
    - **Wind Wall & Roof Panel Supports** - where previously the engineer may have used inactive Analysis elements to assist with the support of Wind Wall and or Roof Panels - for example for overhangs or at the base of vertically spanning walls - inactive beams can be used.
        - All loads decomposed to such members will now be applied to the structure/ supports where previously they would not (errors in the Load Summary check would highlight this issue to the engineer).

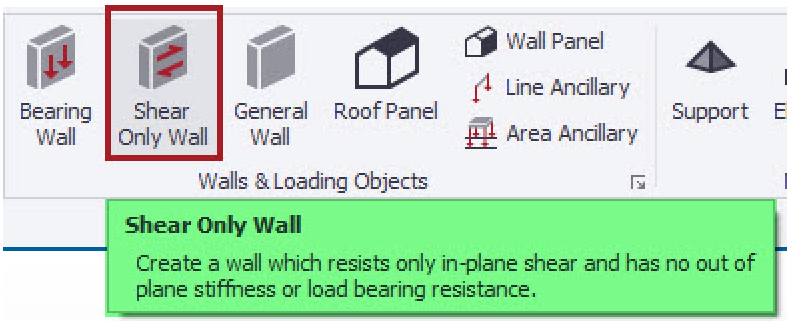
New Lateral Force Resisting System Wall Type - Shear Only Walls

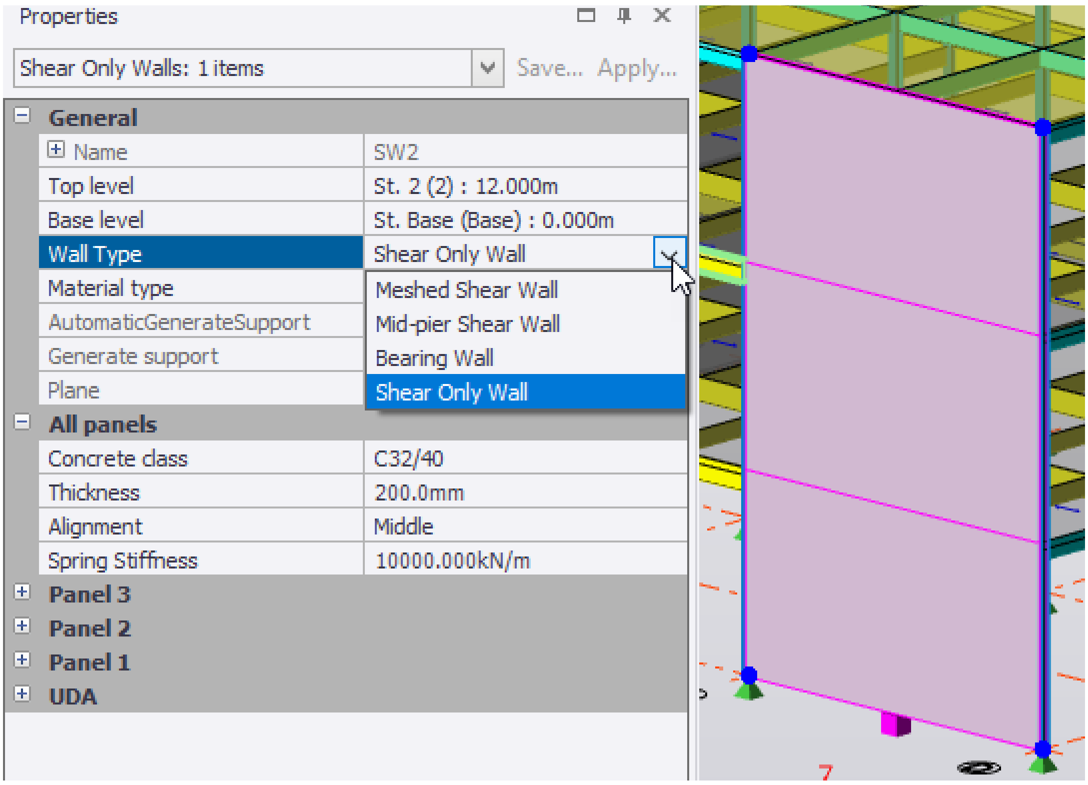
A new type of Wall is available in this release for modeling for example a lateral system of infill masonry panels. Typically engineers wish to model these with only in-plane lateral stiffness of a defined value, and zero vertical and out of plane stiffness. The new wall object automatically models this idealized behavior with a sophisticated underlying analysis model involving axial spring and link elements (new and enhanced in this release). The new wall type can be used in models with both rigid and semi-rigid diaphragms. For more information see the new Help Topic on this feature How shear only walls are represented in solver models (page 757).

NOTE Shear Only walls were originally included in the first 2020 release but due to the issue described in Product Bulletin PBTSD-2005-2 they were removed in the 2020 SP2 release (version 20.0.2.33 May 2020). Full details about them are therefore included in these release notes.

- Shear Only Walls are added via a new button command on the Model Ribbon in the “Walls & Loading Objects” group as shown below. A single wall object of the full building height can be placed within a bay and will automatically be divided into panels for each level.
    - The requirements for Shear Only walls are that they must be; within a single bay (i.e. do not overlap one or more columns); strictly rectangular; vertical and surrounded by columns and beams (other than at the bottom edge).
- For existing walls, the Wall Type can also be reviewed and edited via the Properties Window as shown in the picture below.

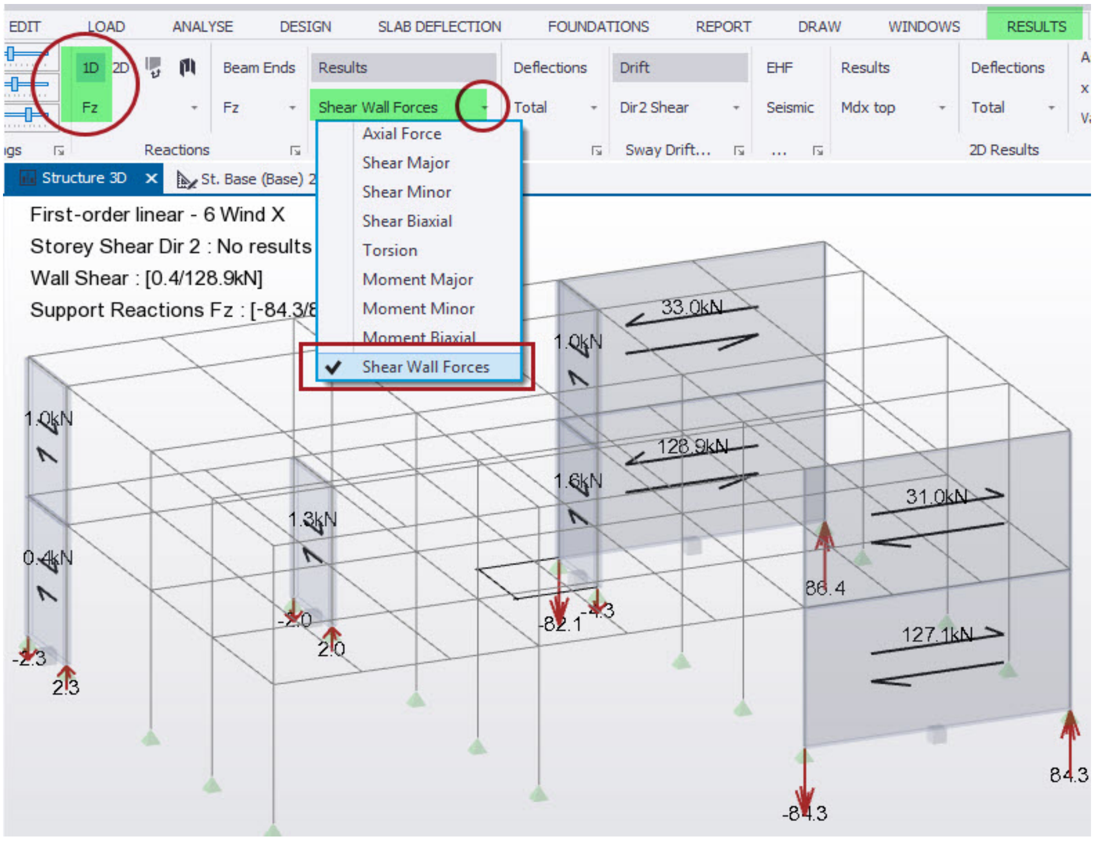
- The material and thickness of the wall can be set, but note that these control only the wall self weight, not its stiffness. The latter is specified directly by the engineer.

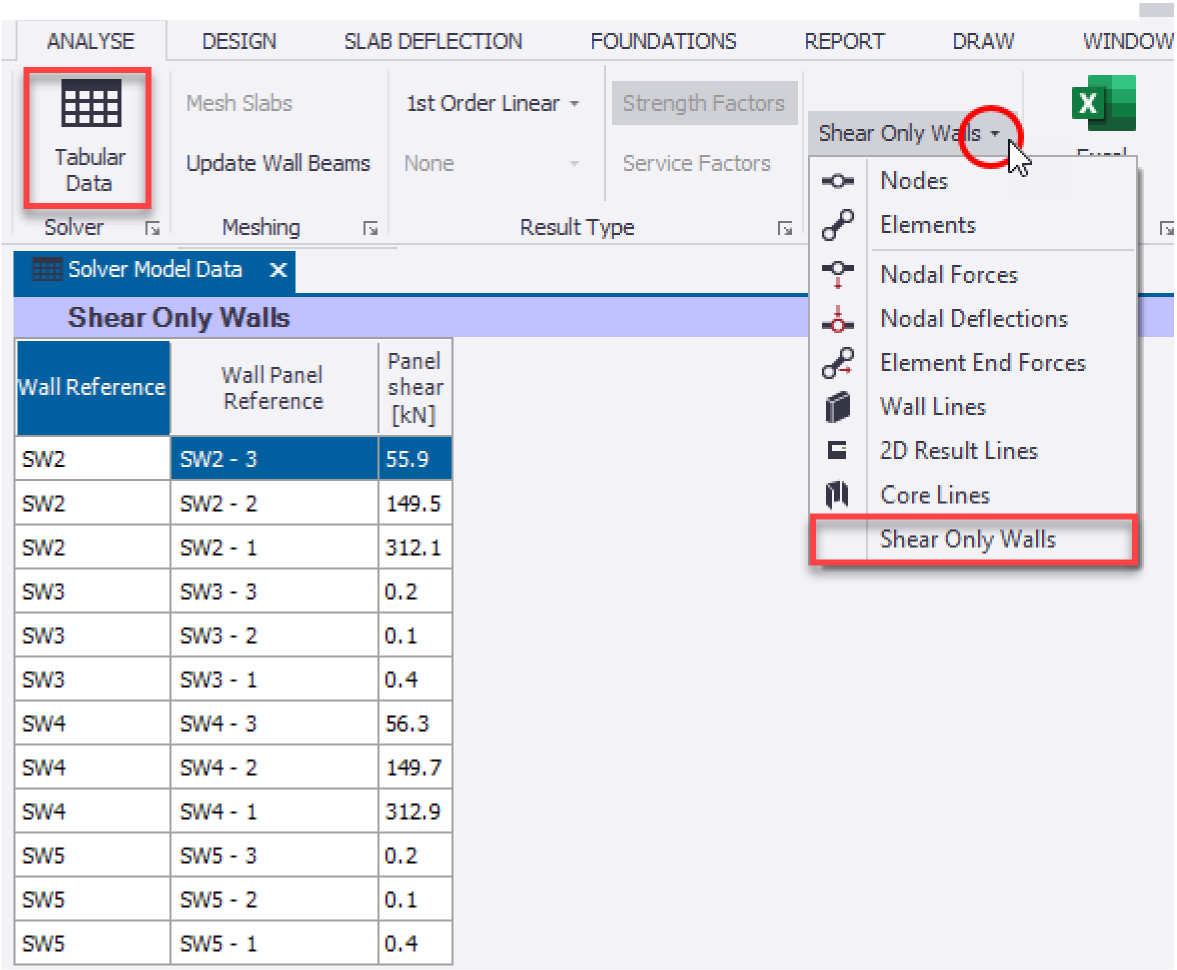




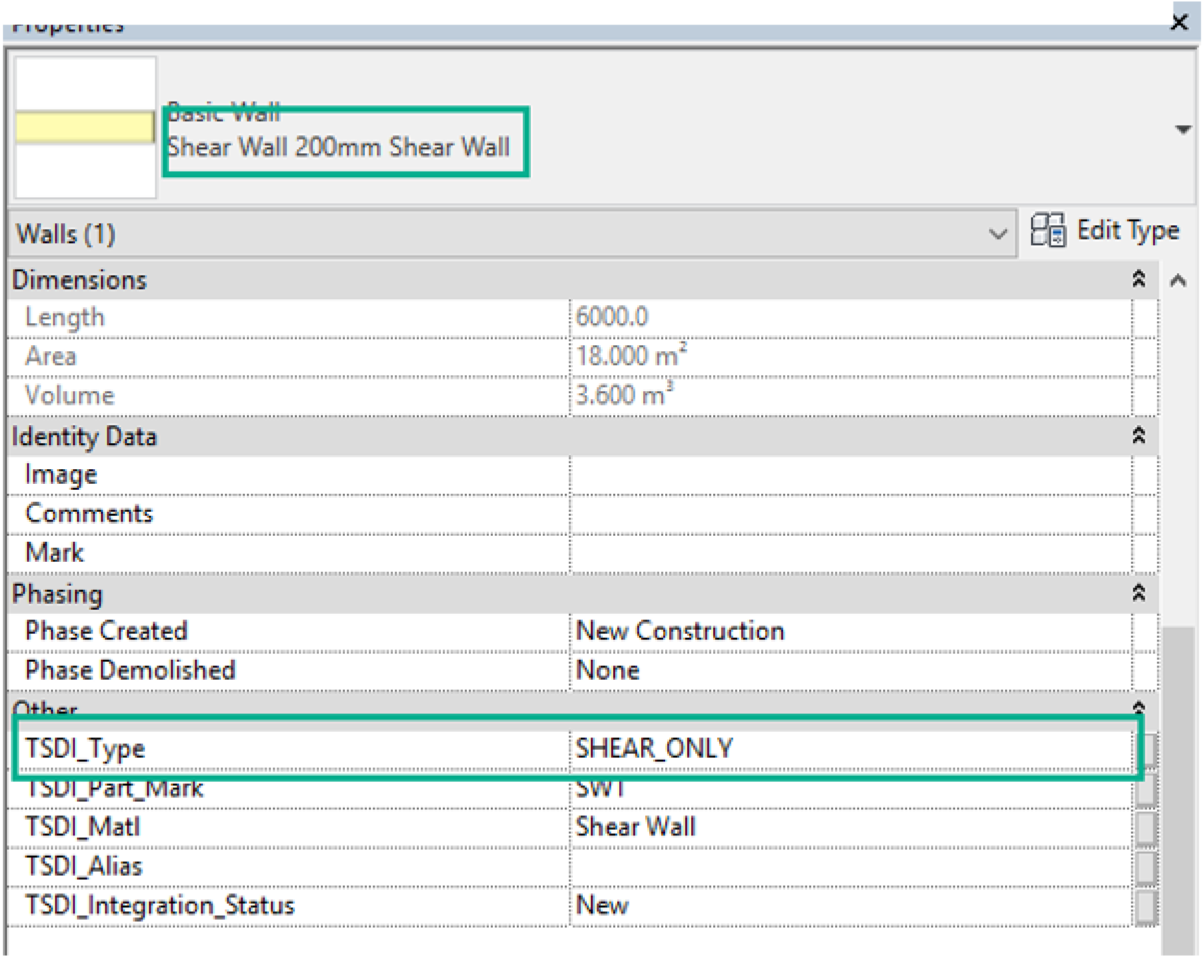
- **Results** - Shear Only wall results are available both graphically and numerically and give the total value of shear in each panel of the wall and the reaction at the wall base. These can be reviewed:
  - In the Results View via 1D Results > ***Shear Wall Forces*** (in 1D Results group) for the panel forces and the base reactions via Reactions (Fy controlled by the [2D] reactions toggle button).
  - in Analyze > Tabular Data > ***Shear Only Walls*** and in Reports via the associated Report item.

- The expected vertical (Fz) push-pull overturning reactions can also be viewed using the 1D reactions option.





- The new shear only wall object is included in the Integrator and CXL file.
- For Tekla Structural Designer → Revit, a new TSDI_Type object tag "SHEAR_ONLY" is included in the CXL file and written to the Revit model for all exported Shear Only wall objects as shown in the picture below.
    - Note that the Revit Name for the wall Material Grade must be manually selected on import into Revit (this selection can then be saved in the Mapping file).

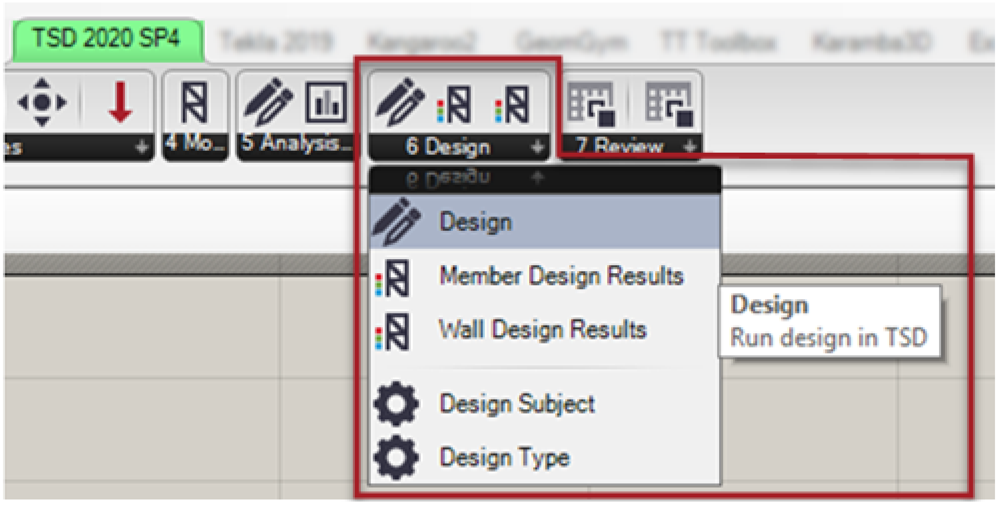


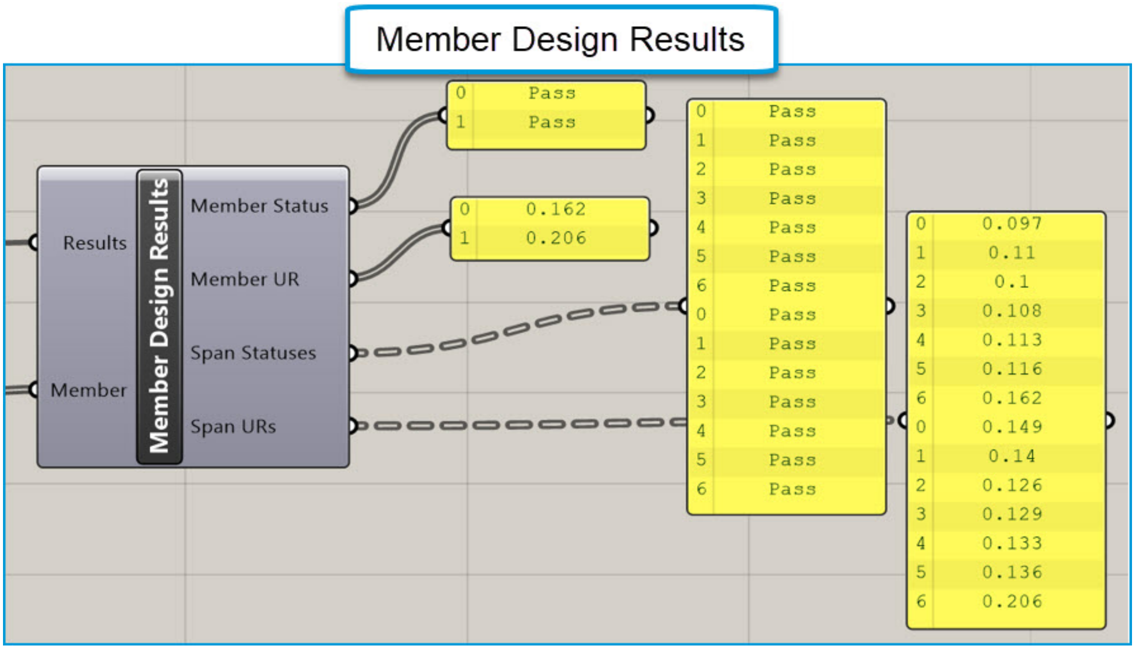
Enhanced Grasshopper Live Link - Design Results and Reporting

The Grasshopper - Tekla Structural Designer Live Link (GH-TSD) is enhanced with new Design and Reporting components to extend the Optioneering & Optimization workflows. For more information about the Grasshopper link see the TUA article Grasshopper -Tekla Structural Designer Live Link (GH-TSD)

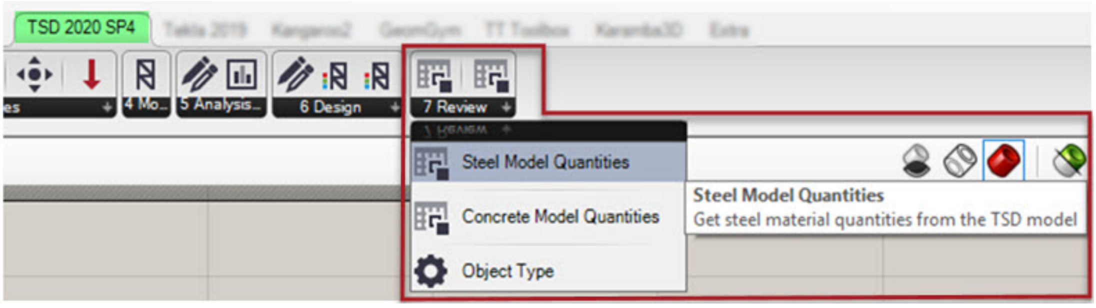
- To investigate different design configurations using the new components you can now:
  - Generate several configurations of a structure, in different materials or with different geometry
  - Run design for each configuration
  - Determine if that configuration is a passing design (and the URs for each beam plus for example average/max/min UR)
  - Measure the 'cost' of that configuration using material quantities
  - Calculate the efficiency of each design configuration

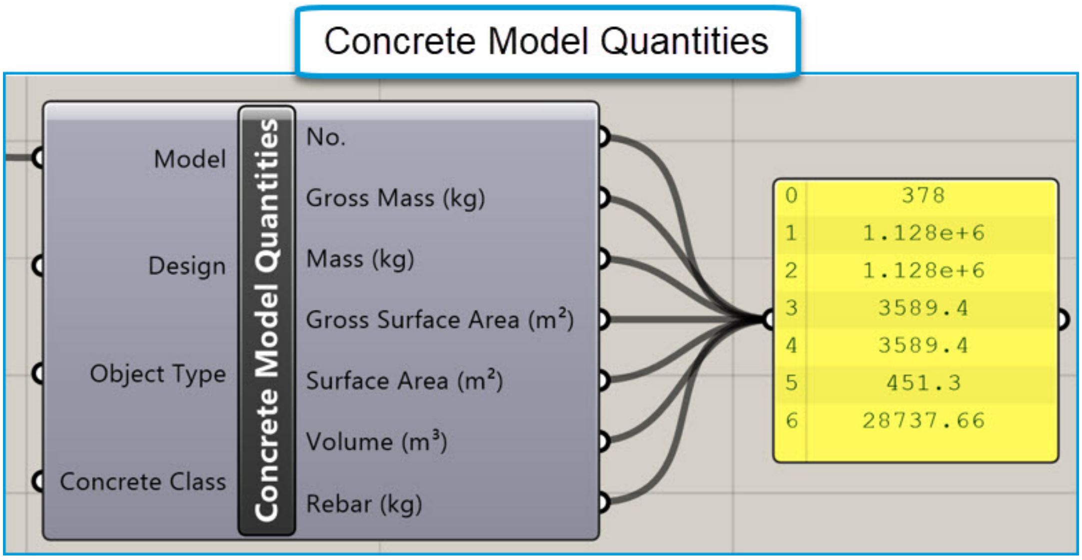
    - Find the best design configuration using automation
- **Design** - the following three new design components are added and are available from the Design group of the ribbon as shown below:
    - Run design - allows the use to run design for the model (provided by the model component) with the following options; Subject - Steel, Concrete or All; Type - Gravity or Static (no RSA)
    - Member & Wall Design Results:
        - Member Design Results - reports Design Status & Utilization Ratio for a member or list of members. Results are available by whole member and for all spans/stacks within that member (see picture below illustrating this).
        - Wall Design Results - reports the same data for Walls - again per whole wall or per wall panel.





- **Reporting** - two new components are added to report steel and concrete material quantities in a new “Review” ribbon Group. These can report in Grasshopper quantities from the Tekla Structural Designer materials list such as; number of objects, mass, surface area, volume and rebar mass (concrete models).



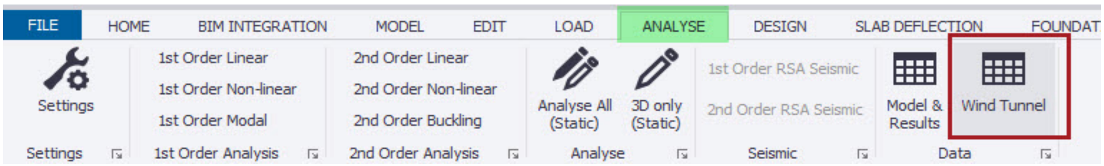


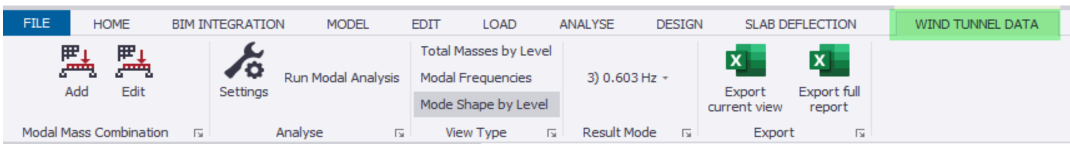
Dynamic Analysis - Wind Tunnel Data Report Generation

Specialist wind tunnel consultants are sometimes employed to determine structure specific wind loading on a project, typically this applies to tall or unusual structures. Following customer requests a new feature set has been added to enable easy and rapid generation of Dynamic Analysis data required by the wind tunnel testing consultant. Note that the Modal analysis capability and output principally required for such data has always existed in Structural Designer - this feature adds a new simple workflow and automated report tables to produce the commonly required data. A new dedicated “Wind Tunnel Data” Ribbon Tab - shown in the picture below - walks the engineer step-by-step through the required input and generated tabular output which can then be conveniently exported straight to Excel for sharing with third parties.

Tall Building Features - note that this feature adds to and will likely be used in conjunction with a number of more recently added features targeted particularly at the design of tall buildings. These are:

- Design for high strength concrete grades (Eurocode) - for more on this see the Tekla Structural Designer 2019i Release Notes
- Stresses in 2D elements (page 679)
- 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls (page 54)
- Concrete member cracked or uncracked status (page 1286)
- Modify assumed cracked settings (page 875)
- Diaphragm loads and diaphragm load tables (page 546)

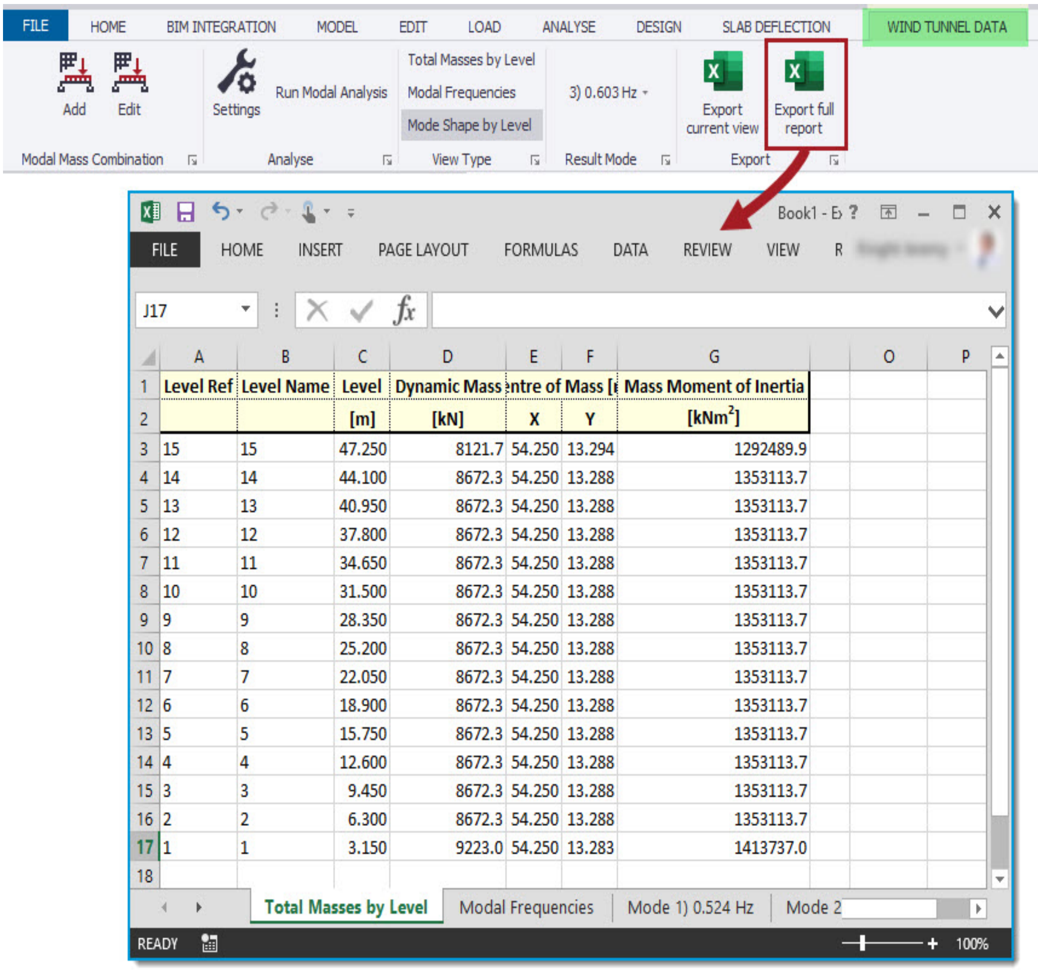




The key aspects of this new feature are:

- The new dedicated ribbon tab is opened by selecting the “Wind Tunnel” button in the Data group of the Analyze Ribbon as shown above. The tab contains all the commands required to define the required Mass combination, configure the analysis settings, run the analysis and review and export the report data.

- The basic process to produce a Report is as follows:
  - Select the “Add” button to create a new Load Combinations of the “Modal Mass” class which is required for Modal Analysis - add the required load cases with their appropriate factors to this.
    - Also review the “Applied mass” tab; while all directions are active by default, you can uncheck the “Dir Z” option so only lateral modes are included.
    - Should it be required, you can the select the “Edit” button to go directly to the Modal Mass combination(s)
  - Select “Settings” to directly access the Analysis settings for 1st Order Modal to review/ edit these.
    - The default settings are likely to be adequate for many circumstances, but the experienced engineer may prefer to enable the Lumped Mass Model and Simple Mass reporting options.
  - Select “Run Modal Analysis” - on completion the automated report tables will be populated and can be reviewed.
  - The report tables are listed in the View type group of the ribbon and are:
    - **Total Masses by Level** - a completely new output table summarizing the dynamic mass at each level*; the Dynamic Mass Total**, the Center of Mass (COM) coordinates and the Mass Moment of inertia (often termed the Rotational Inertia).
      - *Note that only levels with the “Floor” checkbox enabled in Construction Levels are included.
      - **The existing tables giving the mass distribution to all nodes (Total and Active) are also still available.
    - **Modal Frequencies** - the same table of frequencies that has always been available after Modal Analysis
    - **Mode Shape by Level** - a completely new output table giving a single mode shape deflection value at the COM for each floor - select the mode shape to view results for in the drop list in the “Result Mode” ribbon group to the right.
      - Since mode shape deflection values are typically very small* - especially for very large structures - for improved precision the engineer may like to set the Exponential format threshold for values lower than to 1E-3 (Home > Model Settings > Units).
        - *For why this is see the TUA Article What is the magnitude of the deflections from Modal Analysis?
  - Finally, the entire report - or the currently selected table - can be exported directly to Excel via the Export current/ full report buttons in the Export group of the ribbon.

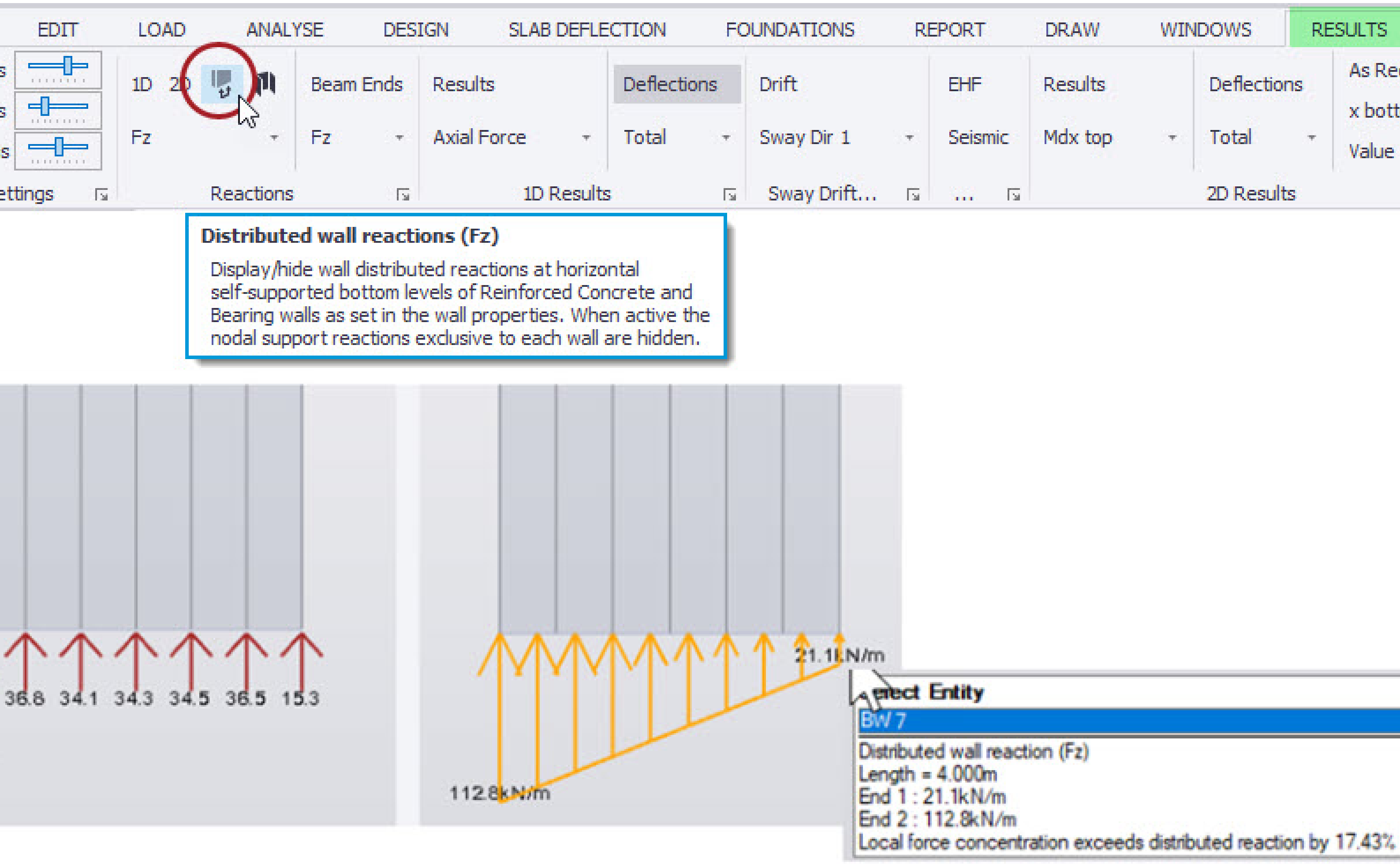


Level Ref	Level Name	Level	Dynamic Mass	Centre of Mass [m] X	Centre of Mass [m] Y	Mass Moment of Inertia
		[m]	[kN]	X	Y	[kNm^2]
15	15	47.250	8121.7	54.250	13.294	1292489.9
14	14	44.100	8672.3	54.250	13.288	1353113.7
13	13	40.950	8672.3	54.250	13.288	1353113.7
12	12	37.800	8672.3	54.250	13.288	1353113.7
11	11	34.650	8672.3	54.250	13.288	1353113.7
10	10	31.500	8672.3	54.250	13.288	1353113.7
9	9	28.350	8672.3	54.250	13.288	1353113.7
8	8	25.200	8672.3	54.250	13.288	1353113.7
7	7	22.050	8672.3	54.250	13.288	1353113.7
6	6	18.900	8672.3	54.250	13.288	1353113.7
5	5	15.750	8672.3	54.250	13.288	1353113.7
4	4	12.600	8672.3	54.250	13.288	1353113.7
3	3	9.450	8672.3	54.250	13.288	1353113.7
2	2	6.300	8672.3	54.250	13.288	1353113.7
1	1	3.150	9223.0	54.250	13.283	1413737.0

- Reminder - when the wind consultants complete their work they will typically pass back wind forces as a series of level loads. The recently added Diaphragm Loads option can be used to apply these to the model. Using this, the level loads can easily be imported as described in Diaphragm loads and diaphragm load tables.

Distributed Wall Reactions

In line with customer requests, distributed line (force/ unit length) reactions are now automatically calculated and reported for all wall types; Meshed, Mid-pier, Bearing and General. A new filter button for these is added to the Reactions group of the Results Ribbon as shown below. The distributed reaction is generally more directly useful in additional calculations and checks for foundation design.

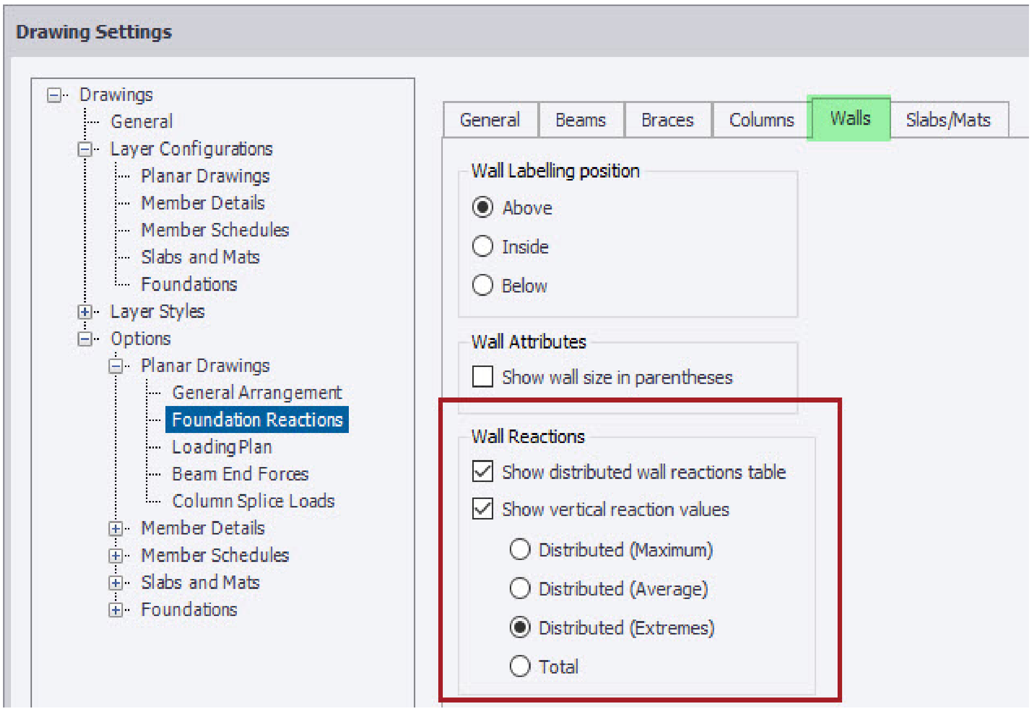


Key points of this feature are:

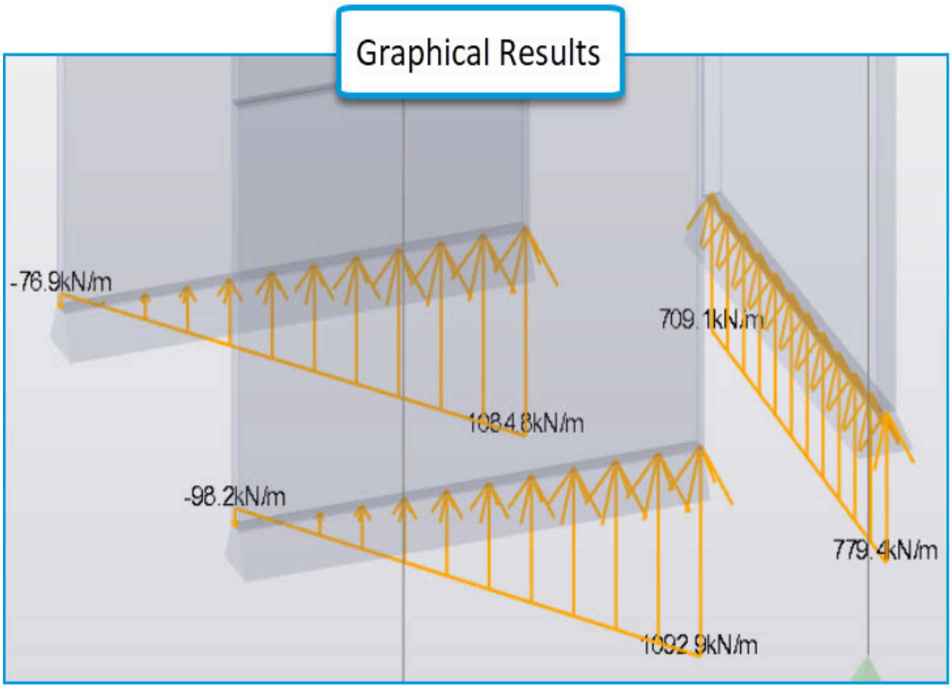
- Distributed wall reactions are:
  - The continuous line reaction along the length of the analytical chord at the wall bottom
  - Available for all structural wall types and all Head Codes
  - Applicable to the vertical (Fz) component of the discrete analytical wall reactions of meshed walls
  - Available for all result types except Envelopes and Seismic RSA

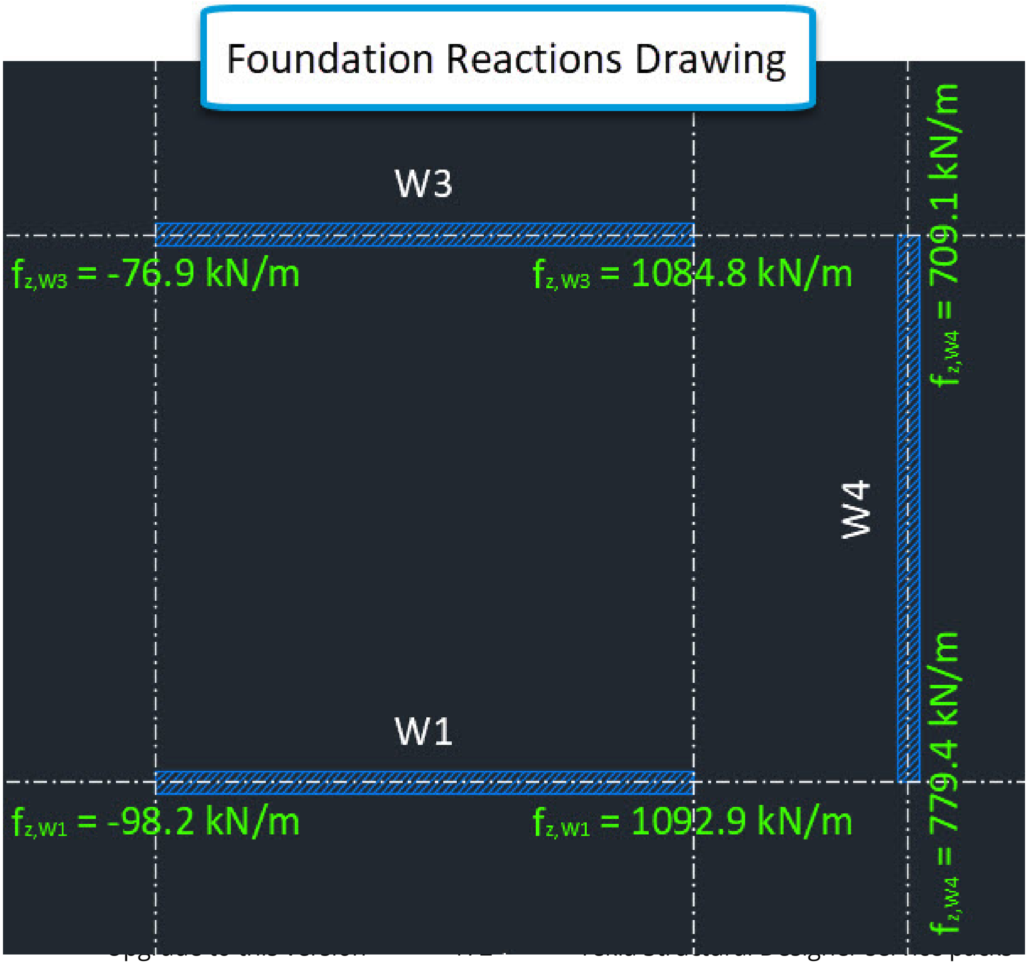
- Reported only for the horizontal bottom levels (analytically) of self-supported walls
- Derived from elastic theory formulae used to calculate stress/unit length based on the total moment and vertical force acting at the centroid of the reaction line.
  - (this assumes that the wall and foundation material are sufficiently homogeneous and isotropic to allow for load distribution according to the Saint-Venant's Principle)

- Graphical Results - Distributed wall reactions can be viewed in the Results View as follows:
  - The new Distributed wall reactions filter button is active only when Fz is selected in the reaction type drop-down list (Fz, Fxyz, Total).
  - When active, the discrete vertical reactions of the wall (1D and 2D) supports are hidden. Note also that the distributed reaction is not displayed for walls in a core when Core reactions are active.
  - The cursor tooltip gives the reaction length and the force/length values at each extreme.
  - The graphical representation of the distributed reaction is typically the same color as other reactions. An orange color indicates a high nodal force concentration - more information on this is then listed in the tooltip.
- Drawings - Distributed wall reactions can be included in the Foundation Reactions drawing as follows:
  - A new Layer Settings option is added in Draw > Settings > Layer Configuration for the Foundation Reactions drawing - “Foundation Reaction values” in the set of controls for Walls (on by default).
  - A new summary table of Distributed wall reactions can also be included, the borders, text and notes for which are controlled by the existing “Notes” and “Tables” Layer Configuration options.
    - Note the text color and size of the reaction values in the drawing is controlled by the “Foundation Reaction Values” Layer Styles item for the Foundation Reactions drawing.
  - New options are also added to Draw > Settings > Options > Planar Drawings > Foundation Reactions > Walls to control their inclusion as shown below.



- A summary table of reaction values can be included (see picture below).
- Vertical reaction values can be shown on each wall, the options for these being:
    - Distributed (maximum): Maximum value UDL,
    - Distributed (Average): Average value UDL,
    - Distributed (Extremes): Calculated linear UDL/VDL extreme values,
    - Total - Integrated total
- Note that the new Layer Configuration options will be active by default in new models - for existing files you can load these default settings by selecting the “Load...” button in the Drawing Settings dialog.


Graphical Results
-76.9kN/m
1084.8kN/m
709.1kN/m
-98.2kN/m
779.4kN/m
1092.9kN/m


Foundation Reactions Drawing
W3
$f_{z,W3}$ = -76.9 kN/m
$f_{z,W3}$ = 1084.8 kN/m
$f_{z,W4}$ = 709.1 kN/m
W4
W1
$f_{z,W1}$ = -98.2 kN/m
$f_{z,W1}$ = 1092.9 kN/m
$f_{z,W4}$ = 779.4 kN/m

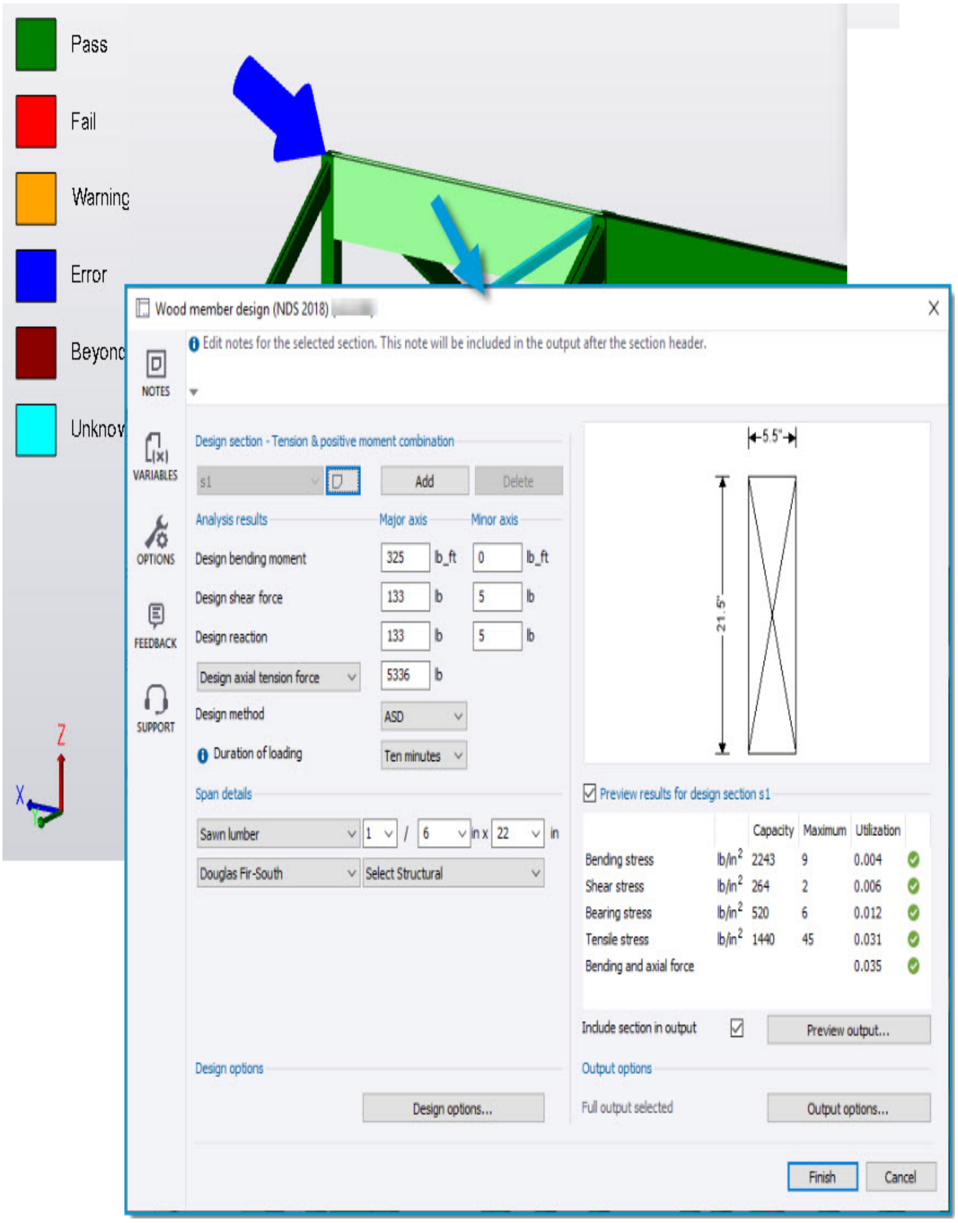
Foundation Reactions Drawing
wall reactions table

Distributed Wall Reactions[1]					
Wall ref.	Length [m]	$F_{z,tot}$ [kN]	Moment [kNm]	f_z [kN/m]	
				End 1	End 2
W1	5.000	2486.8	2481.6	-98.2	1092.9
W3	5.000	2519.8	2420.2	-76.9	1084.8
W4	5.000	3721.3	-146.4	779.4	709.1

[1] Distributed wall reactions where available are given as an alternative to wall nodal reactions. Limitations and assumptions apply.

Comprehensively Enhanced Timber/ Wood design using Tekla Tedds

In an exciting development, integration with Tekla Tedds for Timber member design has been thoroughly enhanced, enabling seamless and efficient Timber design to both the US Head Code and Eurocode. The new workflow fully supports multiple analysis and design loops and Design Groups (enabled by the default). The design process for timber members now works in the same manner and matches the power and capability of that for Precast concrete members using Tekla Tedds introduced in the 2020 first release (for more on this see the Precast member design handbook (page 1527)). Note that use of this feature requires an installation of and license for Tekla Tedds 2020. For more details, see the Timber member design handbook (page 1564).

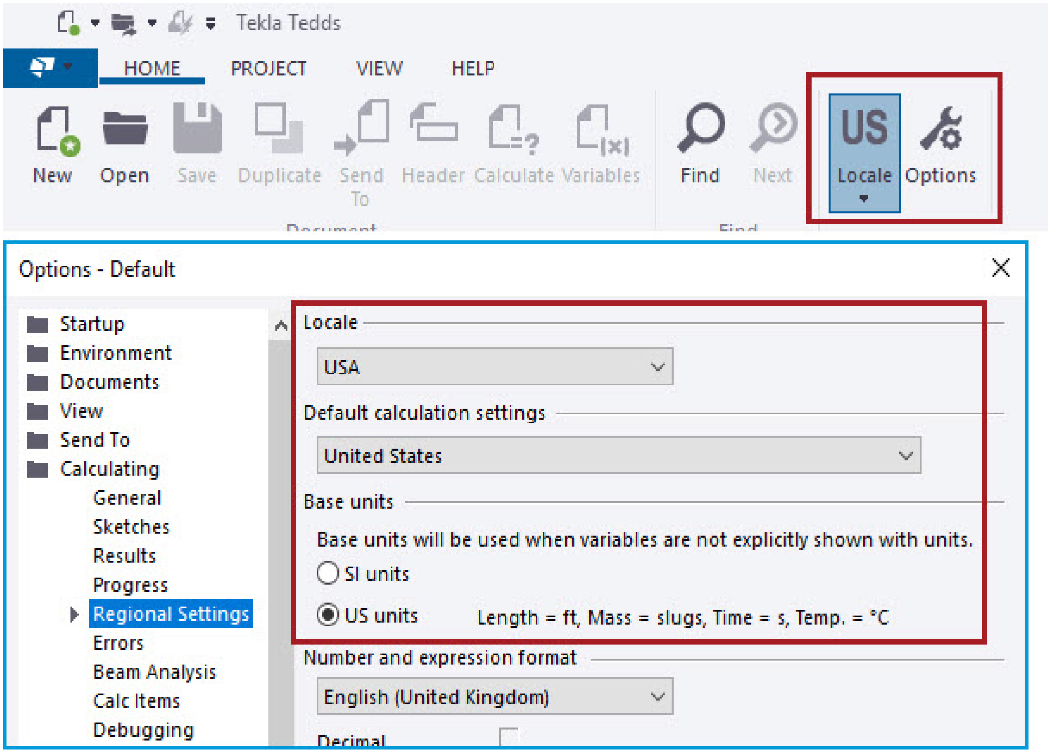

Pass
Fail
Warning
Error
Wood member design (NDS 2018)
Edit notes for the selected section. This note will be included in the output after the section header.
NOTES
VARIABLES
OPTIONS
FEEDBACK
SUPPORT
Design section - Tension & positive moment combination
Add
Delete
Analysis results
Major axis
Minor axis
Design bending moment
Design shear force
Design reaction
Design axial tension force
Design method
Duration of loading
Span details
Sawn lumber
Douglas Fir-South
Select Structural
Preview results for design section s1
Capacity
Maximum
Utilization
Bending stress
Shear stress
Bearing stress
Tensile stress
Bending and axial force
Include section in output
Preview output...
Design options
Design options...
Output options
Full output selected
Output options...
Finish
Cancel

Timber Design Workflow - the essentials of design of Timber members (and Precast concrete) using Tekla Tedds can be thought of as being very similar to Interactive concrete member design (page 1310) within the program:

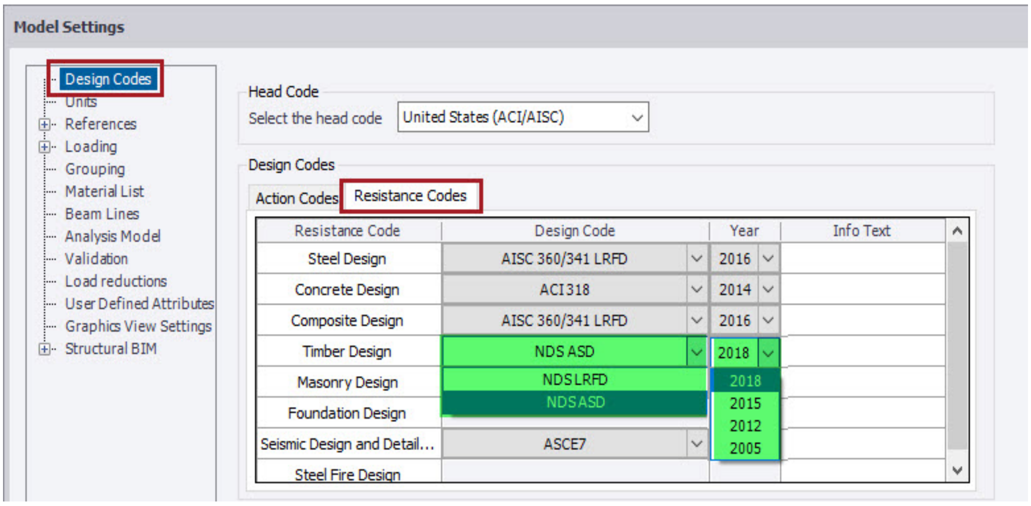
- Just like the concrete beam/ column/ wall interactive design dialog, the Tedds Timber member design calculation interface lists the design forces and settings for a selected member/ group, populated from the model and analysis, the design pass/fail status and individual check results and utilization ratios. You can make changes in the interface - such as to section size and/ or grade - and immediately see the design results for these. All the data you input and changes you make are updated to and stored in the model. You can then re-analyse and run a check on all designed members to ensure the design results are up to date. And you can perform design and analysis loops as necessary so that all members are passing and to cater for any other changes to the model or loading.
- Design Groups work in a similar manner also - these are automatically created for Timber members and listed in the Project Workspace Groups Tree, and can be reviewed and customized as necessary. Group design is enabled by default (in Design Settings > Design Groups > Timber Beams/ Columns/ Braces) and can be run both via the model and from the Groups Tree

Key aspects of the new Tedds integrated design process and workflow are:

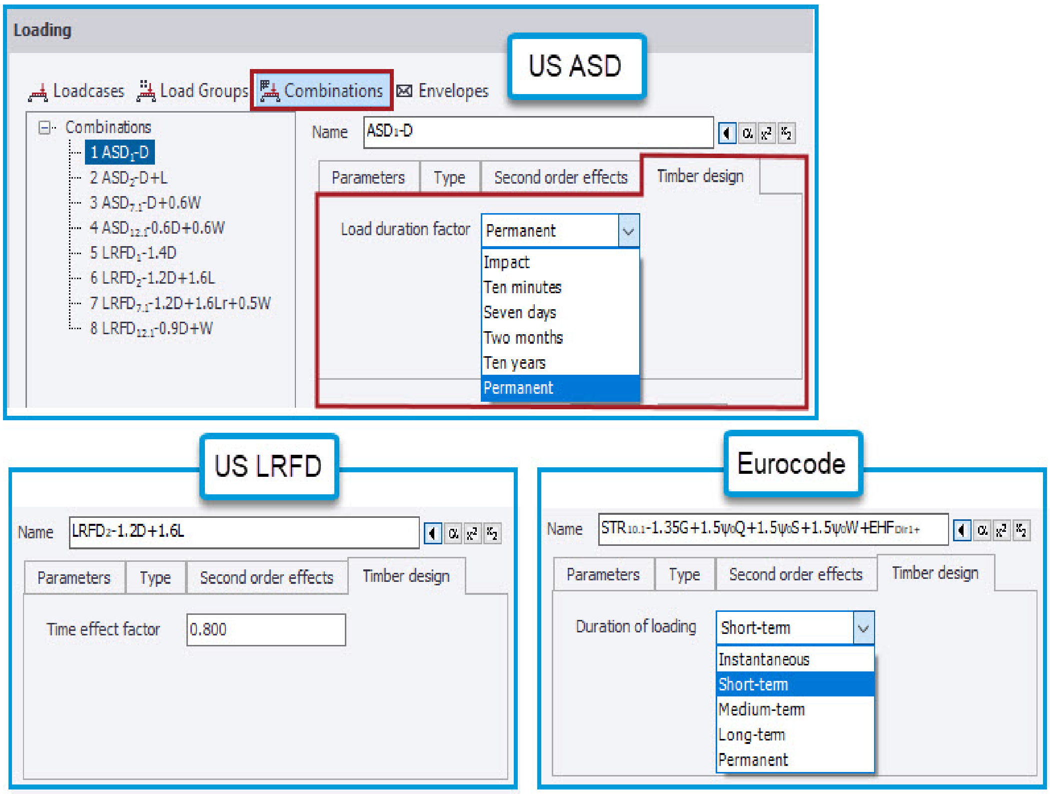
- **Tedds Settings** - First ensure that you set the Tedds **Locale** and **Default calculation settings** appropriate to the Head Code of your Tekla Structural Designer model - e.g. for a US Head Code model using US Customary units you would set the Locale = US and and Tedds Options > Regional Settings as shown below.
    - Design is supported for the US Head Code and Eurocode for the following National Annexes (NA); UK, Ireland, Norway, Finland, Sweden. Note that for the Singapore and Malaysia NA’s, the Tedds calculation adopts the base Eurocode Recommended values.



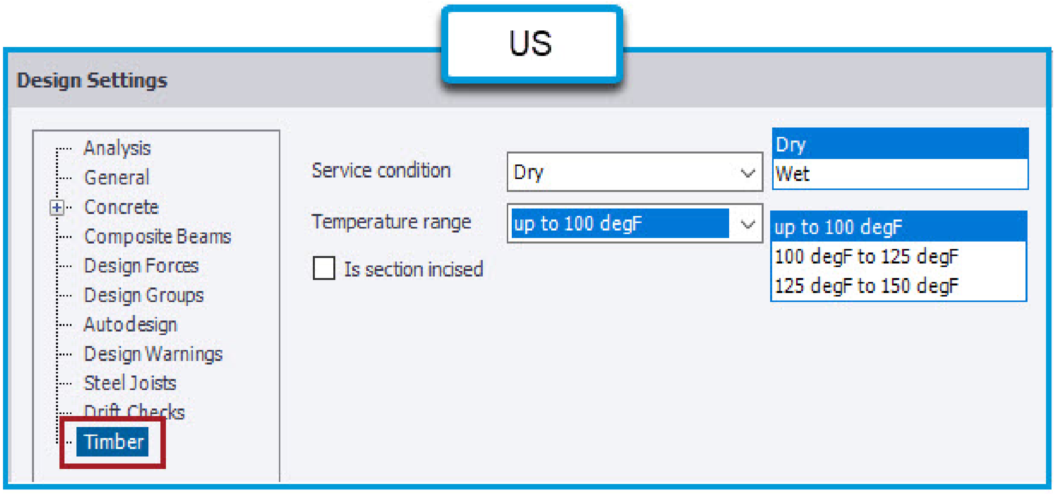
- **New Settings and Controls** - as part of the enhancement, Design Code options have been expanded and new settings added to Load Combinations and Design settings as follows:
  - New Resistance codes for NDS ASD and LRFD for code years 2005, 2012, 2015 and 2018 are added for the US Head Code. These can be set in both Home > Settings > Design Codes for new models and Home > Model Settings > Design Codes for an open model.

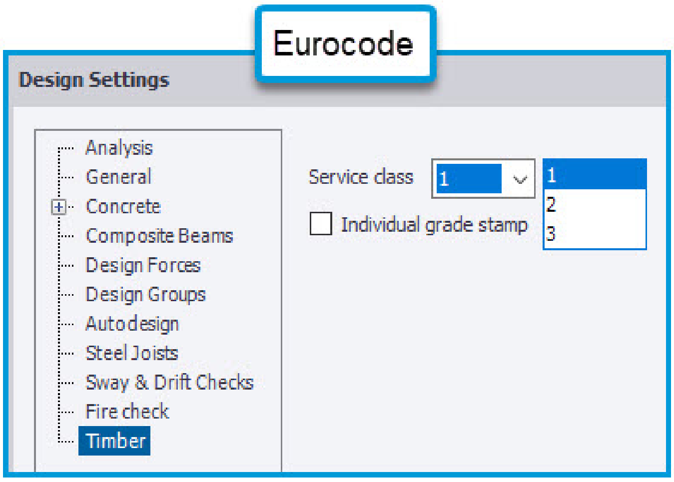


- A new “Timber design” parameter set is added to Load Combinations for the Load duration/ Time effect factor for the US code and Duration of Loading for the Eurocode as shown below. The factor/ duration is automatically assigned to combinations based on their constituent loadcases and can then be reviewed/ set by the engineer as they require.

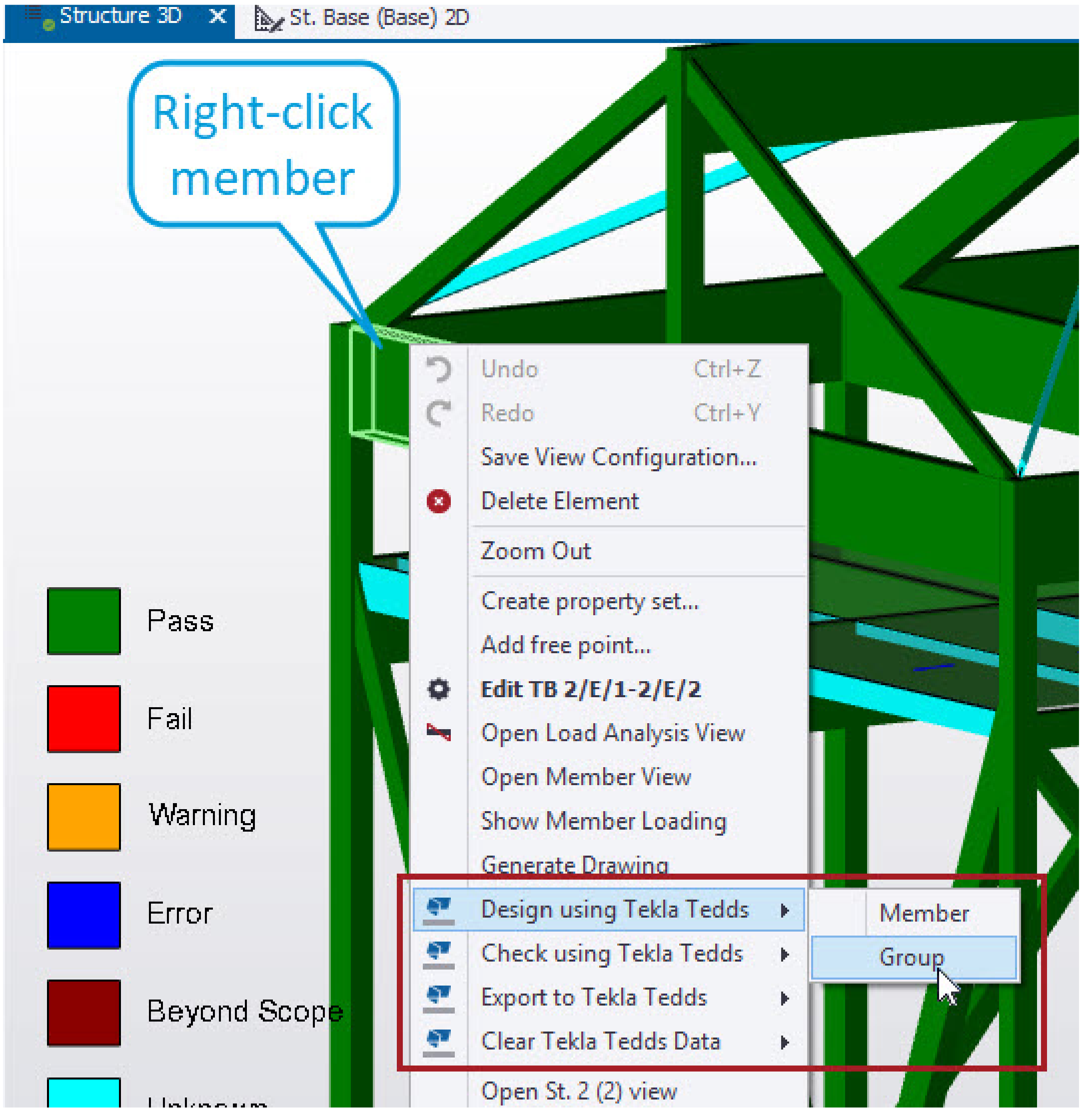


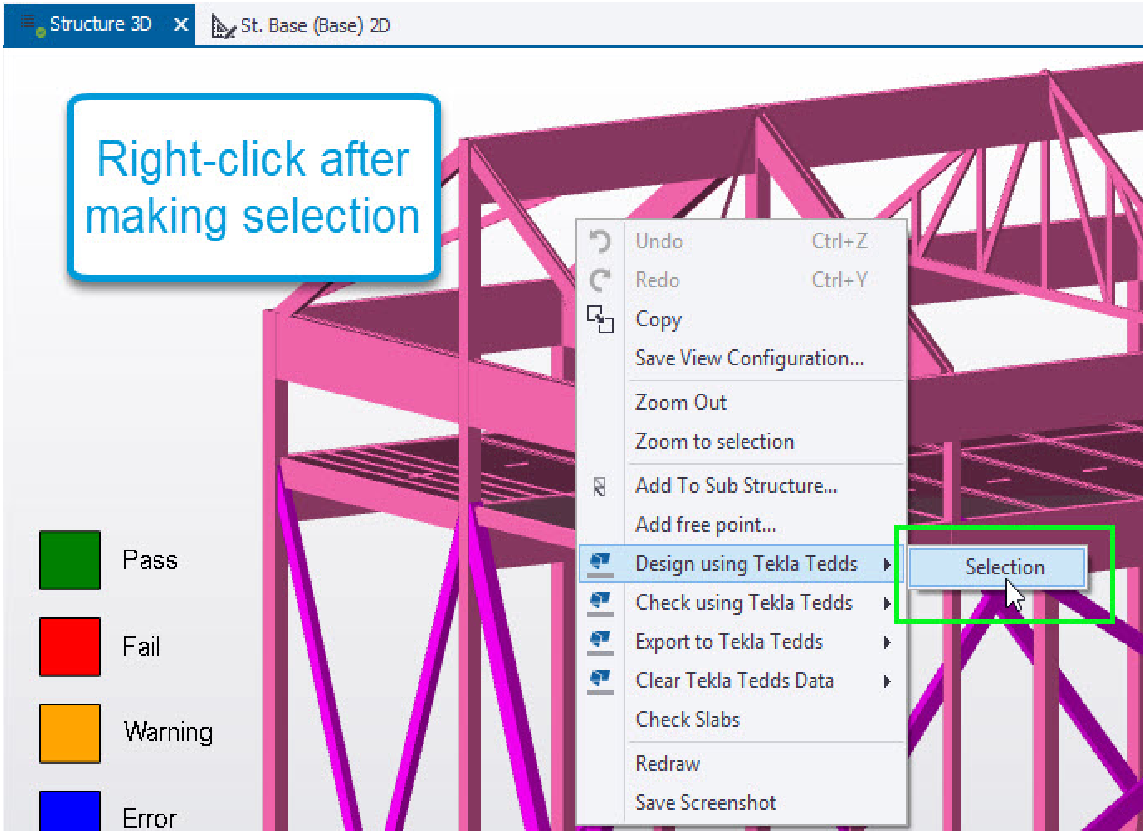
- These new settings enable automatic selection of the appropriate design code and year and setting of the load duration factor in the Tedds calculation - the critical factor/ duration associated with member design loads being populated to Tedds.
- Timber design defaults - a new set of Timber Design settings is added for those that typically apply to an entire structure such as the Service condition/ class and temperature range - these are then populated to the Tedds calculation for all members.



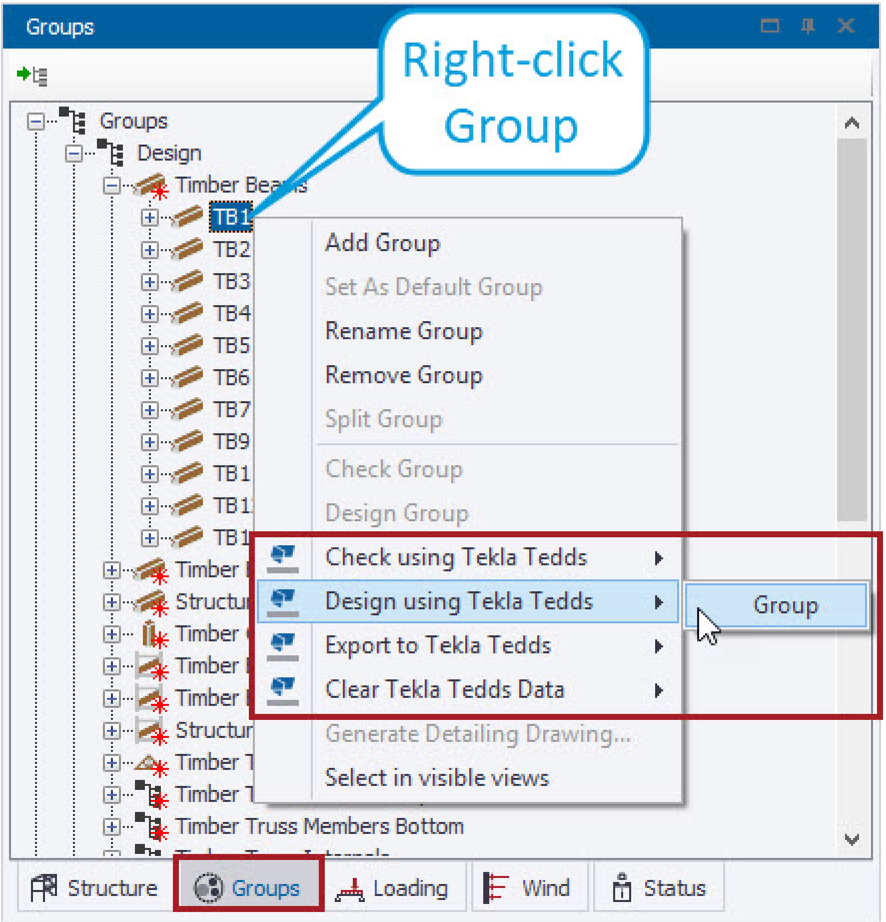


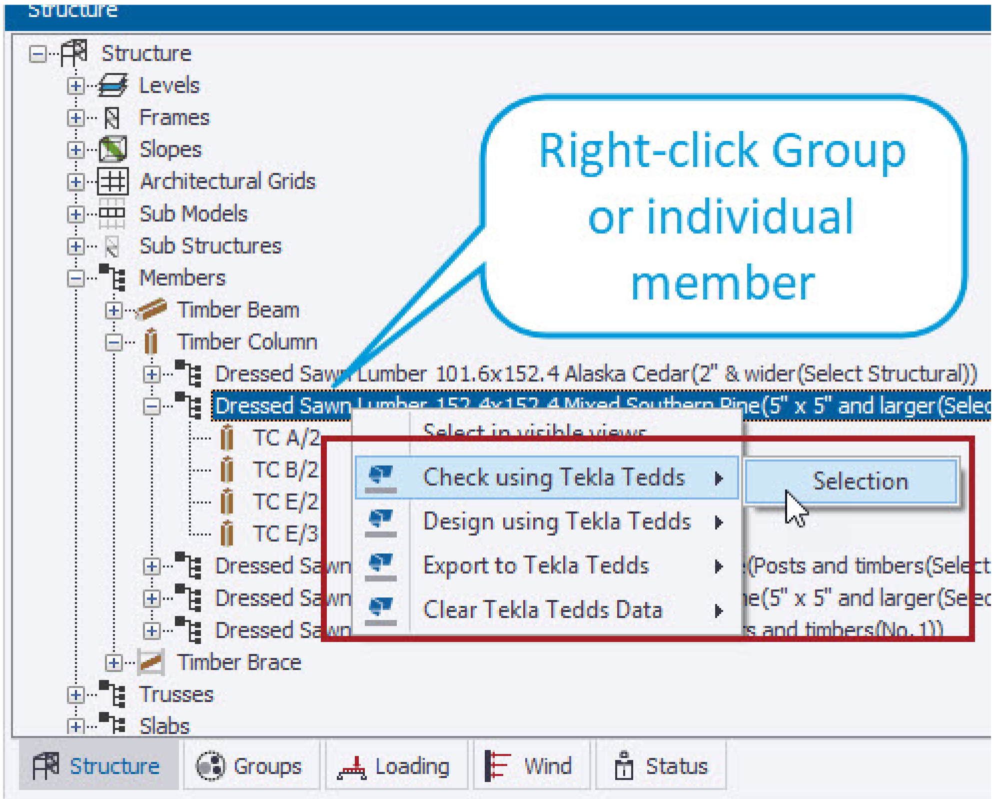
- **Design Process** - after Analysis, Design or Check using Tekla Tedds can be initiated from both the Project Workspace and any graphical view of the model:
    - You can right-click over any individual member in any view, and also make a graphical selection, then right-click to open the context menu listing the Tekla Tedds design options as shown below. You can then select Design or Check of the Member, Group or Selection as appropriate (see below for more on this). Design/ check of a Selection can also be used for example to design all members of a truss - which can be selected with one click - in one operation.


Structure 3D
St. Base (Base) 2D
Right-click member
Pass
Fail
Warning
Error
Beyond Scope
Undo Ctrl+Z
Redo Ctrl+Y
Save View Configuration...
Delete Element
Zoom Out
Create property set...
Add free point...
Edit TB 2/E/1-2/E/2
Open Load Analysis View
Open Member View
Show Member Loading
Generate Drawing
Design using Tekla Tedds
Check using Tekla Tedds
Export to Tekla Tedds
Clear Tekla Tedds Data
Open St. 2 (2) view
Member
Group



- You can also right-click over any individual member or Group listed in the Project Workspace to access the Tekla Tedds design options, in both the **Structure** Tree (including Sub Structures) and **Groups** Tree as shown below:

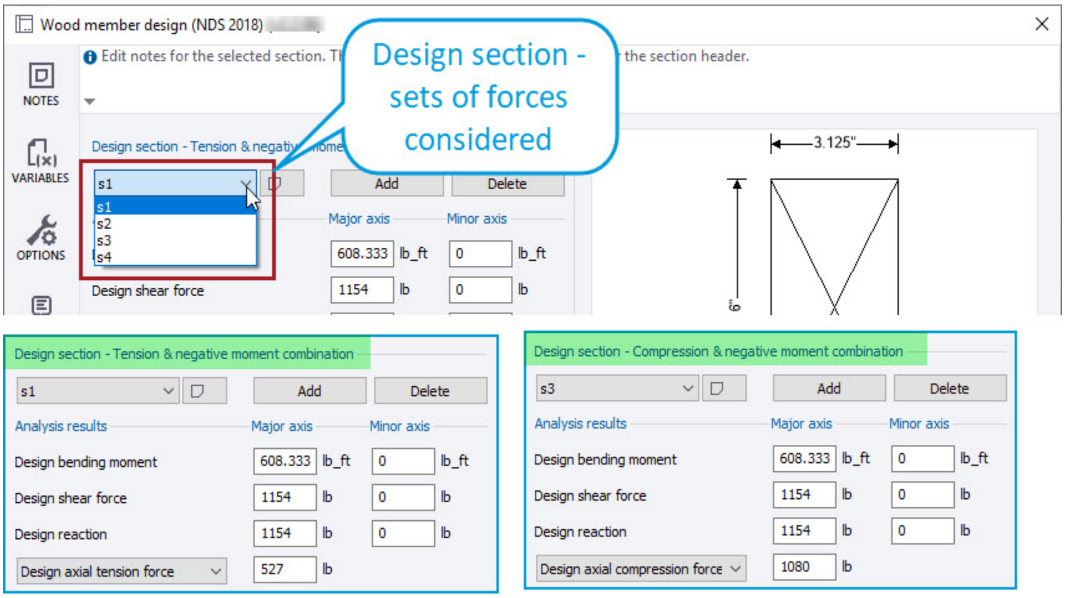

Groups
Right-click Group
Groups
Design
Timber Beams
TB1
TB2
TB3
TB4
TB5
TB6
TB7
TB9
Add Group
Set As Default Group
Rename Group
Remove Group
Split Group
Check Group
Design Group
Check using Tekla Tedds
Design using Tekla Tedds
Export to Tekla Tedds
Clear Tekla Tedds Data
Generate Detailing Drawing...
Select in visible views
Group
Timber Truss Members Bottom
Structure
Groups
Loading
Wind
Status



- The Tekla Tedds options are:
    - **Design using Tekla Tedds** - opens the Tekla Tedds Timber member design calculation interface and populates it with member and analysis data from the model for the selected member/ group.
    - **Check using Tekla Tedds** - this option is available once a Design has been run for the member/ group, and runs a check of the selected member/ group without displaying the Tedds calculation interface and updates the model with the results of this.
    - **Export to Tekla Tedds** - opens Tekla Tedds and creates a new Project containing the design calculation document(s) for the selected member(s).
    - **Clear Tekla Tedds Data** - clears the stored Tedds data if you wish to start again.
- For the **Design** option, the process is as follows:
    - The active view automatically zooms/ pans to the location of the member being currently designed and highlights it with an arrow - or the first span/stack of continuous beams/ columns - so that this is clear, then opens and populates the Tedds calculation interface.
        - For design of a selection and continuous members, after the design of the first member (or span/ stack) is complete, the view and

highlight arrow automatically shift to indicate the next member/ span/ stack considered, and so on, so this is always clear.

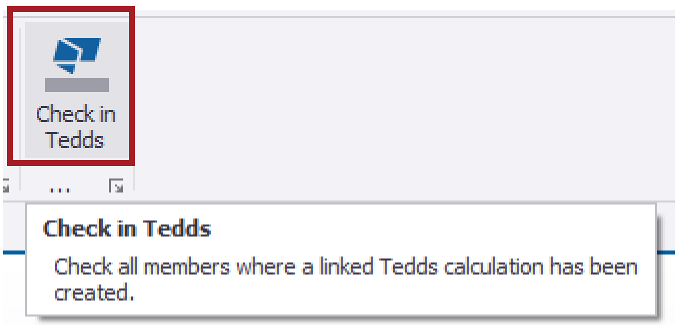
- Design forces - the Tedds calculation is automatically populated with results appropriate to the design method (set in Design Options > Resistance Code) so, for example, for US Head Code models only the results of LRFD combinations are used when LRFD is set as the Resistance Code option.
    - All critical load combinations are considered in one Tedds calculation by populating where necessary separate conditions of tension + bending and compression + bending as separate *Design Sections*. The engineer can then select, review and design for each of these conditions, as shown below.

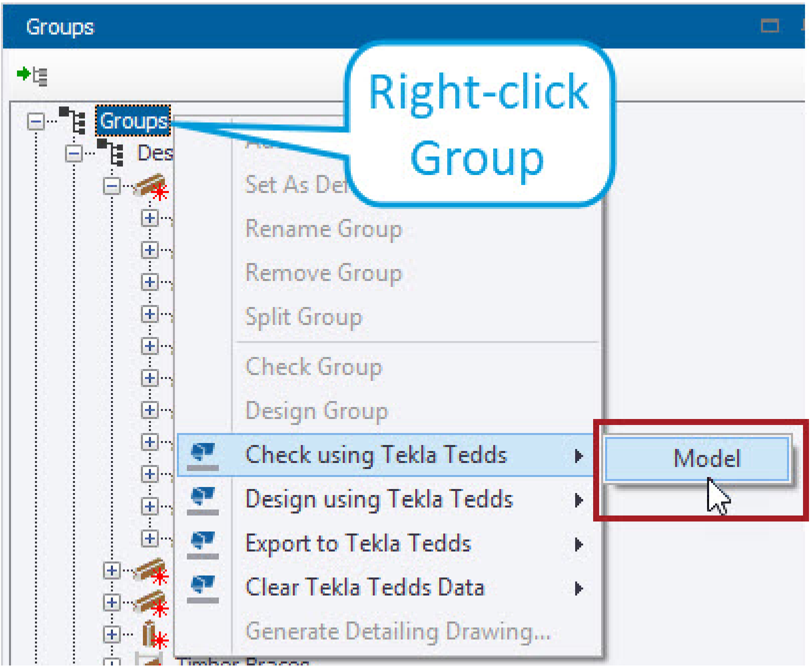


- The Tedds calculation is also populated with the design settings defined in the model and mentioned above, such as Load duration factor and Service condition/ class. Other design settings - such as effective length factors - will be set to default values. All the settings can then be reviewed/ edited as the engineer requires to obtain an adequate passing design.

- Following a completed design, the model is automatically updated with all the input made in the Tedds calculation interface for each member designed, including any changes made to timber sections and/ or grades.

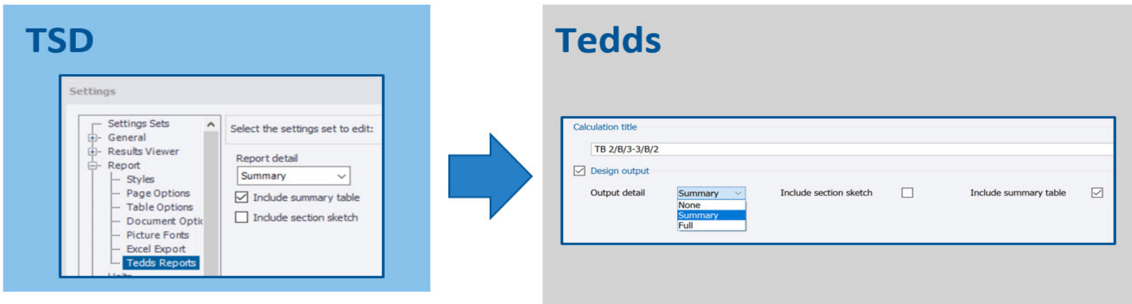
- **Design Status & Review** - the design status and governing utilization ratio (UR) are updated and stored in the model and can be reviewed in exactly the same manner as for steel and concrete design within Tekla Structural Designer; via the Tooltip; in the Review View > Design Status and Tabular Data > Design Summary.
  - The Tedds calculation for a specific member can even be opened directly from the Tabular Data > Design Summary table by clicking on the Results... button at the right hand end of a row.
- **Group Design** - with Design Groups enabled (the default), the process is as follows for each option of Design > Member and Design > Group:
  - Design > Member; the Tedds calculation is populated with the design forces **of the selected member**. When the calculation dialog is finished, all the design settings made - such as section size/ grade - are copied to all members of the group which are then checked for their individual design forces.
    - With Design Groups disabled (Design Settings > Design Groups), Design > Member designs only the individual selected member for its specific design forces.
  - Design > Group; the Tedds calculation is populated with **the worst-case design forces of the group as a whole**. When the calculation dialog is finished, the further process is the same as for Design > Member.
- **Checking & Re-Checking** - following any changes to the model (such as changes of section/ grade during design), all designed members can be easily checked in one operation by re-analysing then running the “Check in Tedds” command on the Design Ribbon. This can also be done by right-clicking at the top of the Groups Tree in the Project Workspace and selecting Check using Tekla Tedds > Model, as shown in the picture below (Design for the entire model is also available here).





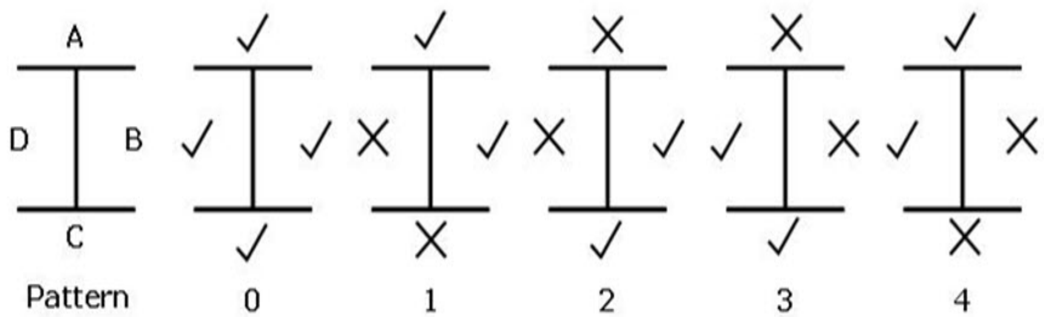
- **Reporting** - For reporting of the design, the engineer selects the "Export to Tekla Tedds" option which will open the Tedds application and create a Tedds Project containing calculation documents for the designed member(s).
  - This option is available for;
    - An individual Member in the model (Design Groups disabled).
    - A single Group (via right-click over a Group in the Groups Window or a member of the model).
    - All Beam or Column Groups together (via right-click over the Timber Beam/ Column/ Brace Parent Group).
    - The entire model via right-click over the "Design" Parent Group for Timber members in the Groups Window.
    - A graphical Selection of part of or the entire model.
  - With Design Groups enabled, the Tedds Project will contain one calculation document* (.ted file) for each group included in the selection.
  - When Design Groups are disabled, the model member context menu export option will change to "Member" and will create a Project and calculation document for the selected member*.
    - *In the case of continuous columns and beams there will be a calculation document for each stack/span of the member.

- Note also that there is a new (Global) setting for the default output level for the Tedds timber calculations in Home > Settings > Report > Tedds Reports.
    - By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds timber calculation as it is run. The setting applies to all Tedds linked calculations (precast concrete and timber).

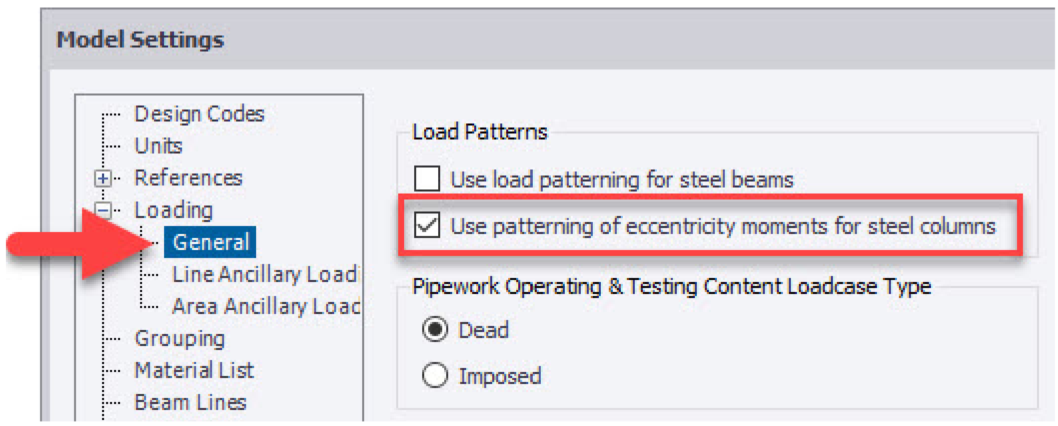


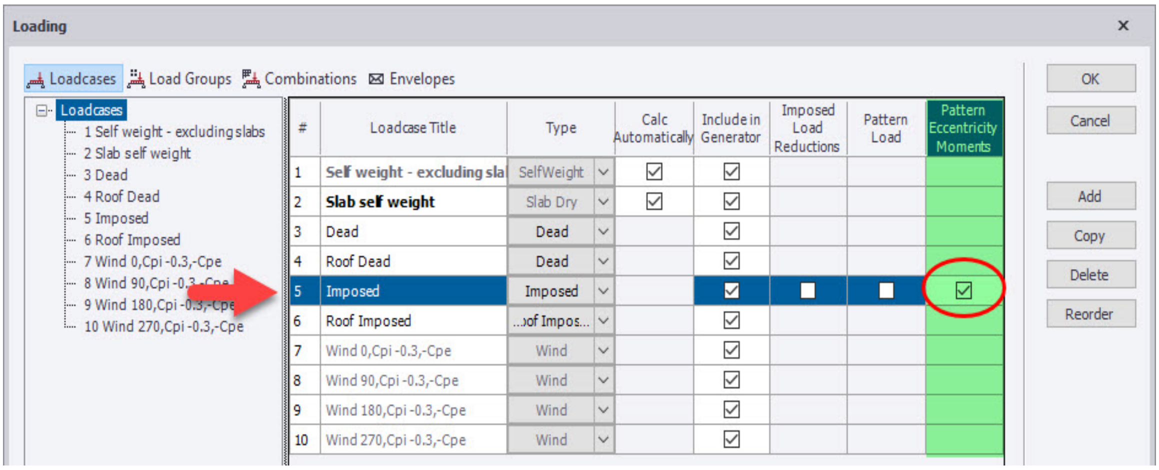
Steel Design - New Patterning of Eccentricity Moments

Since the very first release of Tekla Structural Designer, additional “eccentricity moments” - resulting from the eccentricity of supported beam reactions - have automatically been calculated and considered in steel column design. For more on this feature see the Help topic Steel column connection eccentricity moments (page 1256). In this release the feature is enhanced with the option to consider patterning of eccentricity moments, resulting from different permutations of loading of the supported beams producing unbalanced load and hence potentially increased moments. Five patterns are considered as illustrated in the picture below, in which the ✔ indicates full loading of the beam connected to that face of the column and the ✕ reduced load. The analysis of patterned eccentricity moments applies to all Head Codes, while they are considered by design in the Eurocode and US Head Codes.

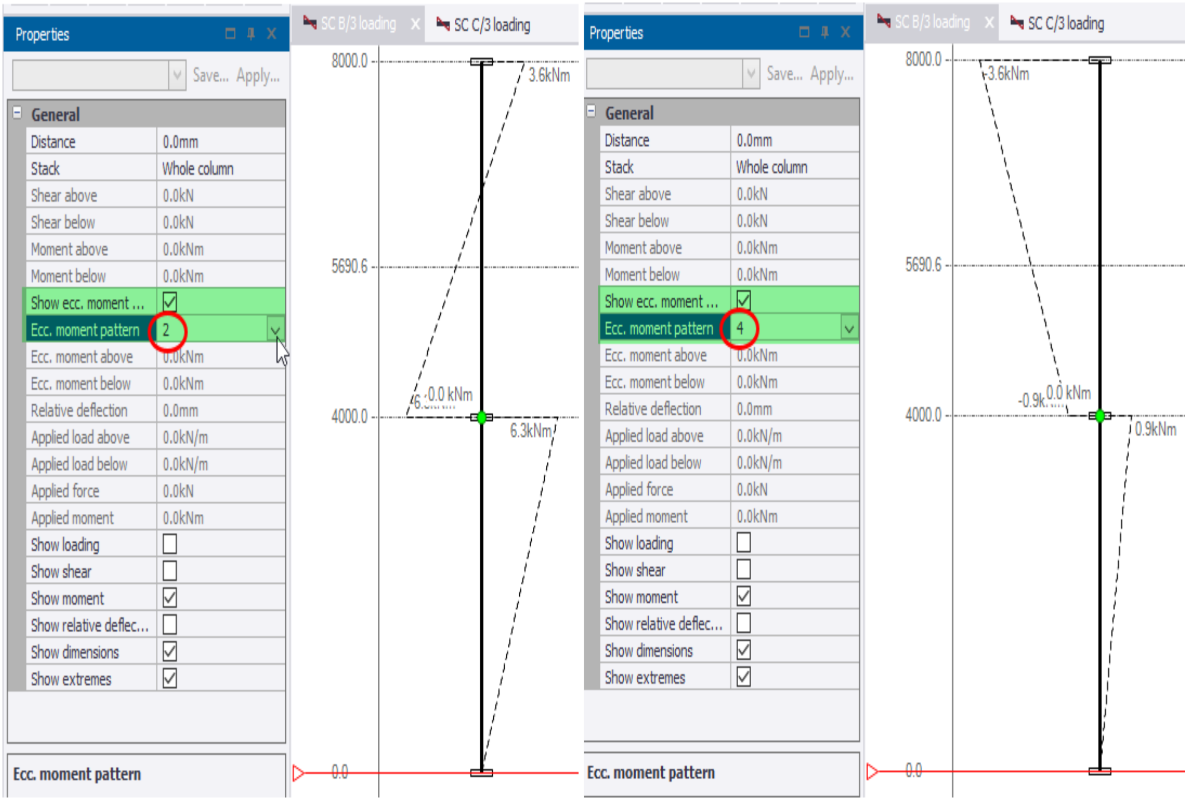


- The feature is fully automatic and requires no additional modeling or loading. To activate it, as shown in the pictures below, simply:
  - Check on the new "Use patterning of eccentricity moments…" option (in Home > Settings > Loading > General for Global Settings and new models, and the same location in Home > Model Settings for an existing open model)
  - Check on the "Pattern Eccentricity Moments" for the Imposed (Live) Load case(s) in the Loading dialog (no other load types can be patterned)
    - (Note that the default for both settings is Off)

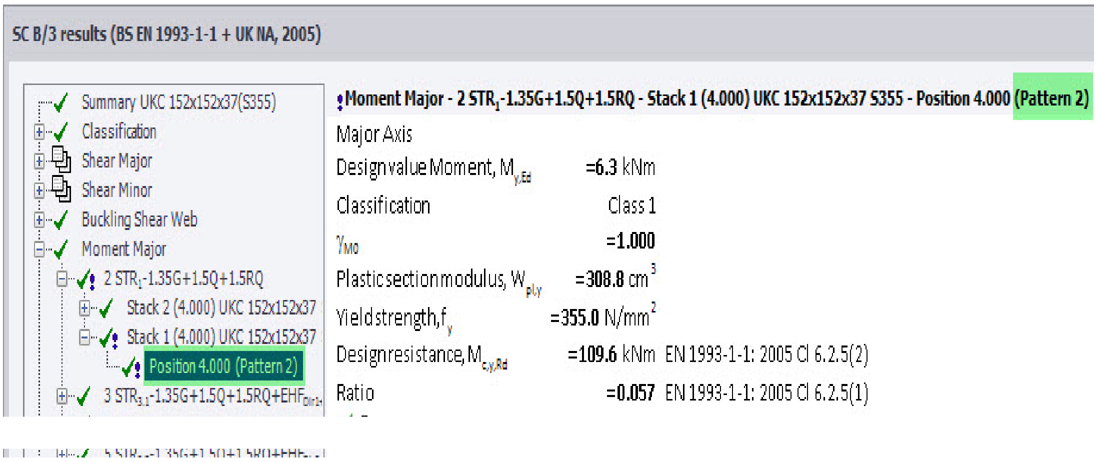




- Key aspects of this new feature are:
  - The inferred ecc. moment profile assumes the same pattern at the top and bottom of a stack e.g. it is not possible to have P1 at the top and P4 at the bottom of a stack.
  - Patterned ecc. moments are combined with ‘real’ moments in the same way as in previous releases (which essentially considered only P0).
  - Axial force; this is reduced (by patterns P1-4) at the current level while that from floors above is from P0 (which is a conservative assumption)
  - All design checks apart from ‘Shear’ can be affected by patterning
    - Note that, in this and all previous releases, there is no rigorous implementation of ‘Simple Construction’ in which load patterning is explicitly ignored (per BS 5950 Clause 4.7.7)
  - Eccentricity moments are determined by a post-analysis process which applies to all Head Codes.
    - The Load Analysis View is also enhanced to enable the viewing of results for patterns to see how they affect the major and minor moments, and axial force. The picture below shows the effect on the major axis moment. A pattern number drop-down list is added allowing the results of a specific pattern to be selected (the eccentricity moments are denoted by a dashed line).

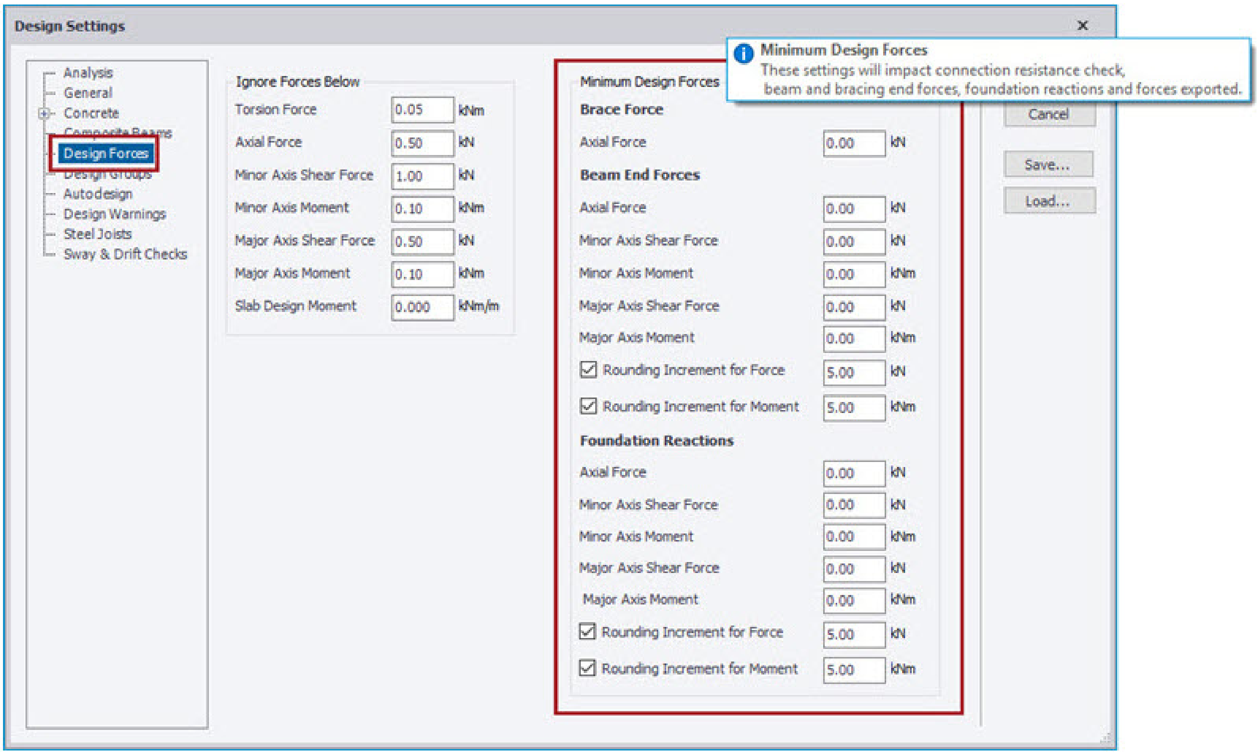


- The eccentricity moments are considered by the steel column design process for the Eurocode (all country NA's) and US Head Codes.
    - To keep the design details to a manageable level, results for every pattern are not listed; the pattern which produces the governing design forces is listed in the check combination and location tree and details heading, as shown in the picture below.



Minimum Design Forces & Rounding Increment

To aid efficiency and rationalization of design, comprehensive new settings are added giving the engineer control and flexibility of Design forces; both those considered by checks within the program and reported and exported. As shown in the picture below, the new settings are added to the “Design Forces” page of Design Settings. Using these the engineer can now control minimum values considered and set optional rounding increments for both Brace and Beam End Forces and Foundation reactions.



- This feature applies to all Head Codes and note that it has no impact on forces considered in member design.
- These settings impact the values considered/ reported in the following checks and areas:
  - Brace & Beam End Forces:
    - Connection Resistance Checks for both Beams and Braces.
    - Brace/ Member End Forces Report Items and Beam End Forces Drawing.
    - Forces populated to Connections (Beam to Beam, Beam to Column and General) - both those designed by/ exported to Tekla Connection Design (TCD) and exported to IDEA Statica.
  - Foundation Reactions:
    - Foundation Reaction Report Items and Drawing.

- Forces populated to Connections (Base Plate and General) - both those designed by/ exported to Tekla Connection Design (TCD) and exported to IDEA Statica.

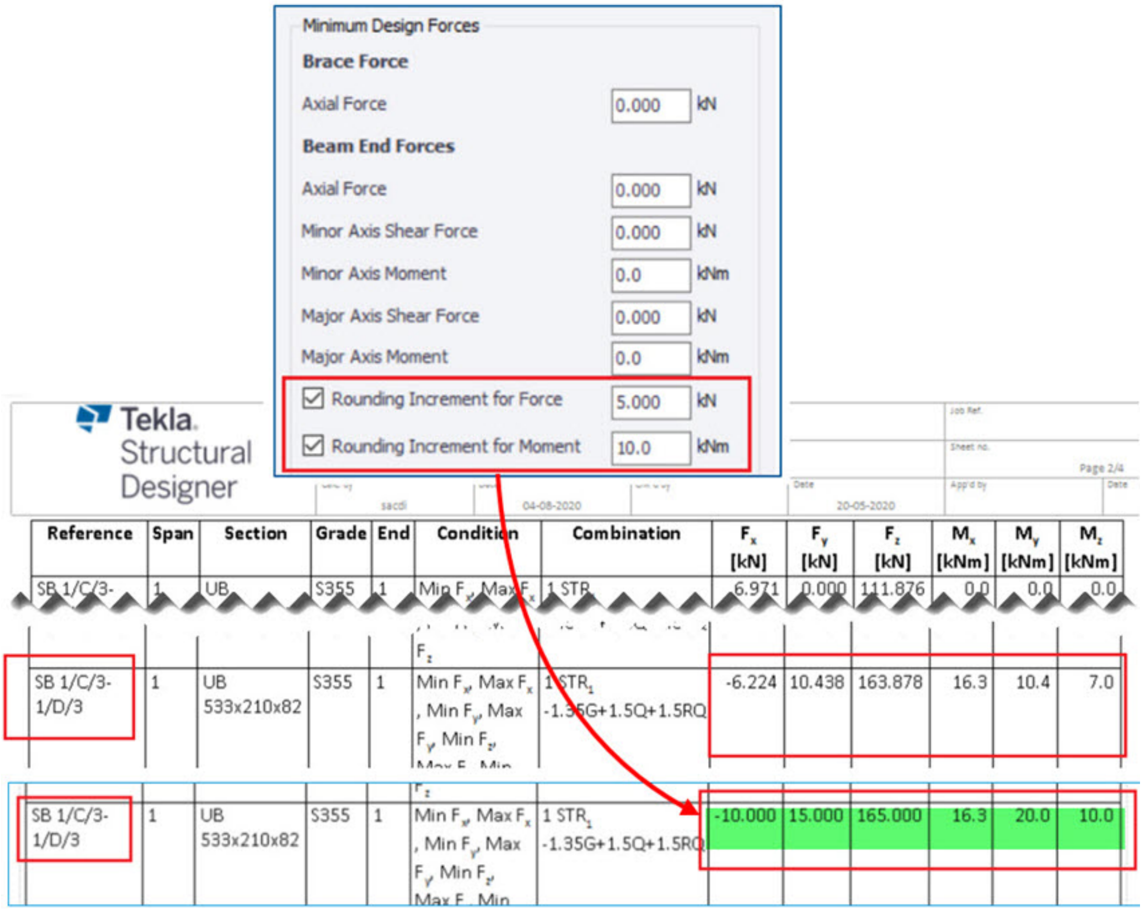
- Minimum Design Force - this replaces the force from analysis where it is the greater - e.g. if for the beam connection resistance checks to consider a minimum shear value of 75 kN, enter this value in the Beam End Forces > Major Axis Shear Force box, and so on.
  - When the Minimum value is used, a note to this effect is displayed in the row for that check in the Tabulated Data Connection Resistance check table and associated report item.

Connection Resistance

Reference	Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status	Note
SB 1/A/1-1/A/2	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	0	2	2	92.000	0.815	Pass	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	1	2	2	92.000	0.815	Pass	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	2	2	2	67.000	1.119	Fail	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Full Depth End Plate	0	1	2	301.000	0.249	Pass	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	0	1	2	158.000	0.475	Pass	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	1	1	2	158.000	0.475	Pass	i
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	2	1	2	99.000	0.758	Pass	i

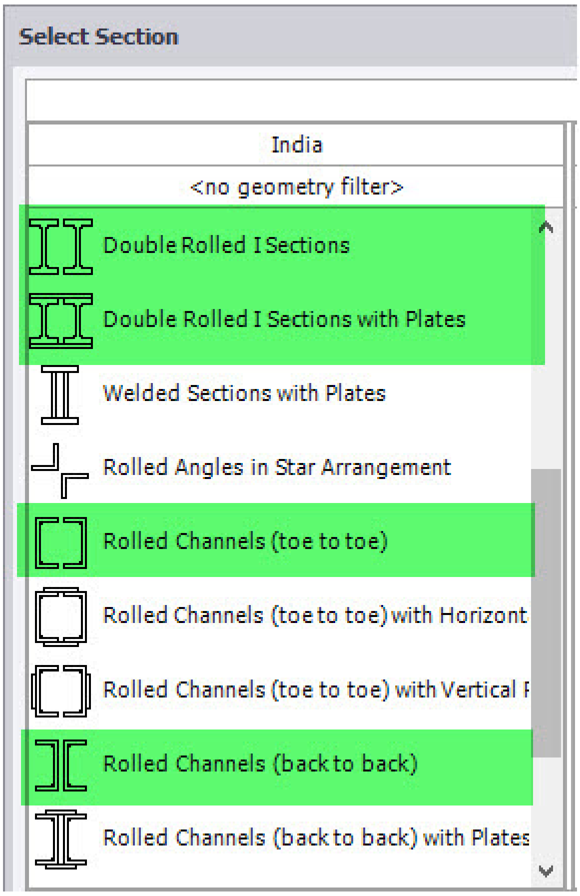
Minimum design force used.

- Rounding Increment - separate rounding increment options are available for Brace/Beam forces and Foundation Reactions for Force and Moment.
  - When enabled, the force value from analysis is rounded up to the next value of the increment considering the sign of the force - e.g. for a force increment of 5.0 kN; a force of 10.438 kN will be rounded to 15.0 kN; a force of -6.224 kN will be rounded to -10.0 kN...and so on.
  - Note that the Moment increment applies to both major and minor axis moments but not torsion (M_x) which is not rounded.



Steel Design - Compound Section Design - Indian Head Code

Following customer requests, a number of steel compound sections can now be designed to the Indian design code. Such sections can be user-defined using the Compound section tool in Home > Materials > Sections > Manage Sections.

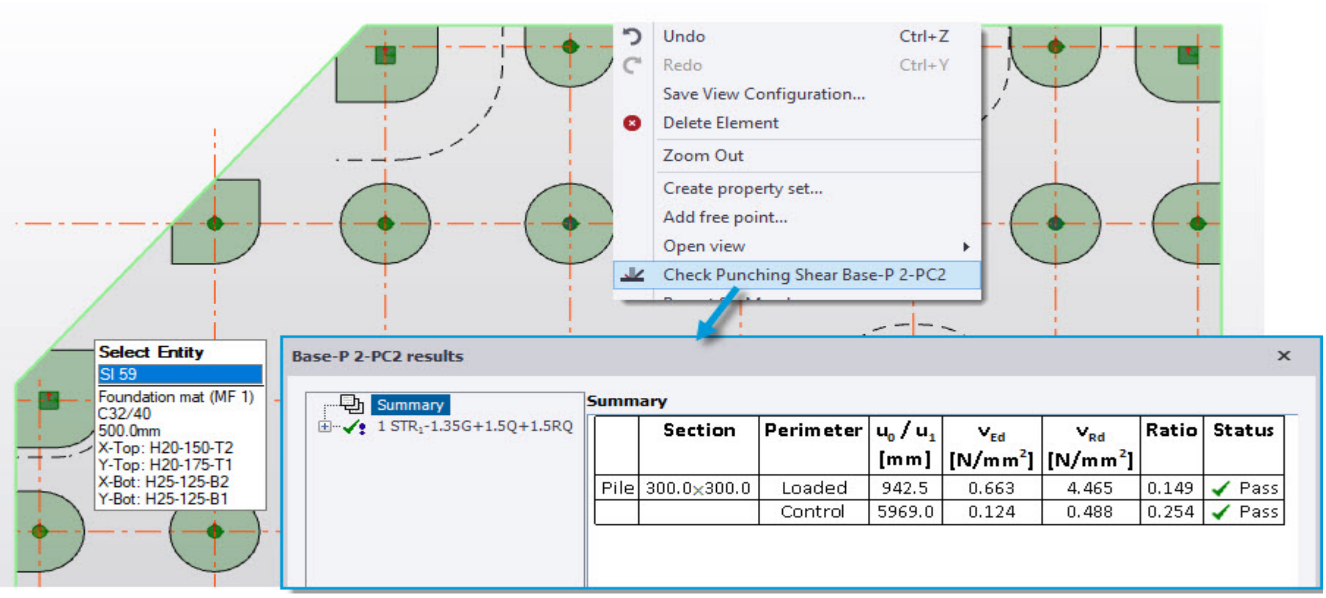


- The section shapes supported by the new design are:
  - Rolled Channels back to back
  - Rolled Channels toe to toe
  - Doubled rolled I sections
  - Doubled rolled I sections with plates
- The scope of design of these compound sections includes both beams and columns and auto-design. For auto-design, the user can create their own section Order Lists for compound sections once they have added these to their database.
- In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:
  - The design is for non-composite sections only.
  - Slender Sections are beyond scope.
  - High Shear case with minor axis moment is beyond scope.
  - Design of the lacing or battening system is beyond scope.
  - Only parallel flange sections can be used.

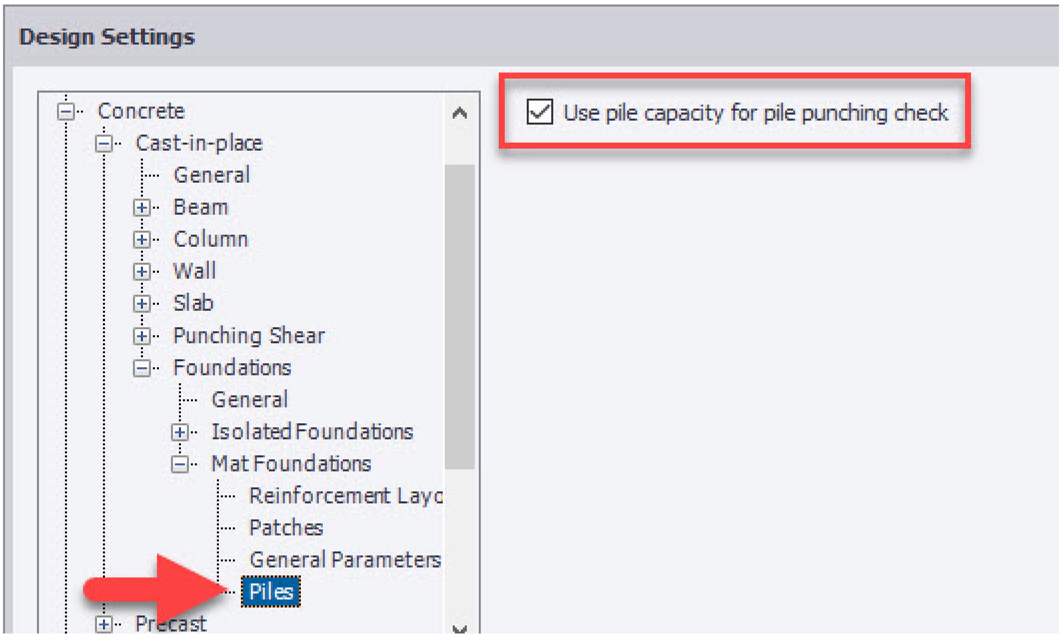
Concrete Design - Pile Punching Checks on Mat Foundations - Eurocode and US Head Codes

The scope of mat foundation slab design is now expanded to including pile punching checks for the Eurocode and US Head Codes.

- The process of creating, designing and reporting punching checks of piles is essentially the same as that for columns:
    - The punching check object can be created in both 2D and 3D views - both by selecting individual piles and by drawing a selecting line/ window to create checks for multiple piles in a single operation. Just as for columns, the check location and tension reinforcement surface are automatically set.
    - Pile punching check is independent of mat foundation design.
    - All punching checks can be designed via the "Design Punching Shear" button in the Punching Shear group of the Foundations tab of the ribbon. Design details of individual checks can also be opened via the context menu as shown in the picture below.



- Key aspects of the check performed are:
    - Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
    - The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.
    - The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).
    - Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
    - There is an additional new pile-specific Design setting to use the pile capacity in the punching check in Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles as shown below (default Off).



Release notes: Tekla Structural Designer 2020 SP3

This release will upgrade your Tekla Structural Designer installation to version number 20.0.3.28 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than release 2020 SP2 (version 20.0.2.33 released May 2020), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing

- No new license is required for this version.
- **Tekla Structural License Service** - the latest Service Pack for this was released in July 2020. This is available in Tekla Downloads and should be installed on all clients for optimum functionality.
    - For more information see the Release Notes page for the Tekla License Service update July 2020 (v3.1.3.4).
- **License Server Version** - for Server licensing, the latest version of the Tekla Structural License Service 3.1.xxxx (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

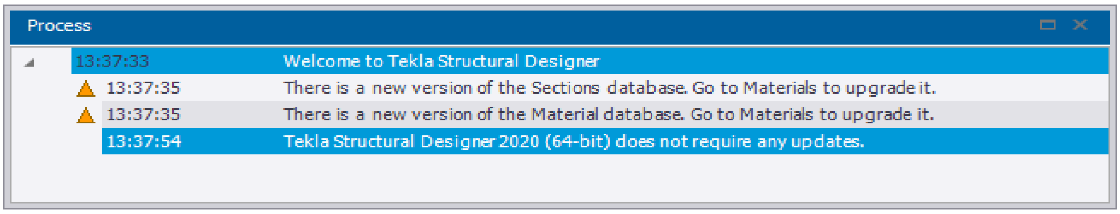
Installation

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) or later to be installed.
- **Integration**
    - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate Tekla Structural Designer 2020 SP3 with Tekla Portal Frame Designer and/or Tekla Connection Designer, you must use Tekla Portal Frame Designer and/or Tekla Connection Designer release 20 or later (note that these require activation of 2020 release PAK's for these products). For optimum function we recommend you install Service Pack 1 for these products, which is available from the same date as this release. Installations for these can be obtained from Tekla Download Service.
    - **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.0) was released on 1st July 2020 and is available in Tekla Downloads. If you are now using Autodesk Revit©

2021, you can install this to perform BIM integration with Tekla Structural Designer.

- All fixes and enhancements included in this release are also included in updates for the Integrators specific to the following currently supported Revit© versions; 2020 (Integrator version 6.01); 2019 (Integrator version 5.02); 2018 (Integrator version 4.03). For more information see the Tekla Structural Designer Integrator July 2020 updates Release Notes. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - Some Databases are updated in this release. Hence for an existing installation a message informing you that new databases are available will be displayed in the Process Window when this release is run, as shown below.



  - Please ensure you upgrade your local databases as follows;
    - Open “Materials” from the Home Ribbon, select the “Sections” page in the list of options, click the “Upgrade” button if it is displayed, then click this button again in the subsequent “Upgrade Database” dialog. Repeat this process for all the other databases (Material, Reinforcement, Decking...etc) to ensure all your databases are up to date.

General & Modeling

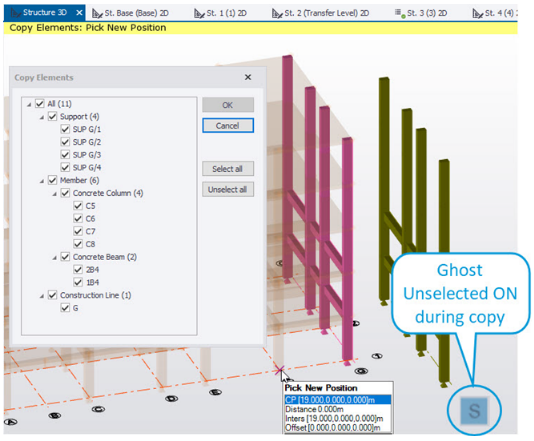
- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of such an issue is listed directly below here:
  - [TSD-6927] - Databases - Decking and Connectors - an exception would occur on closing the Materials dialog after adding user defined Decking and/ or Connector data for certain more recently added Countries, such

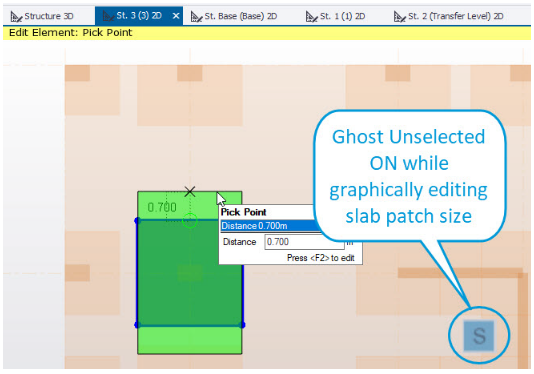
as Finland. This issue is fixed in this release and such user defined data can now be added for the affected countries.

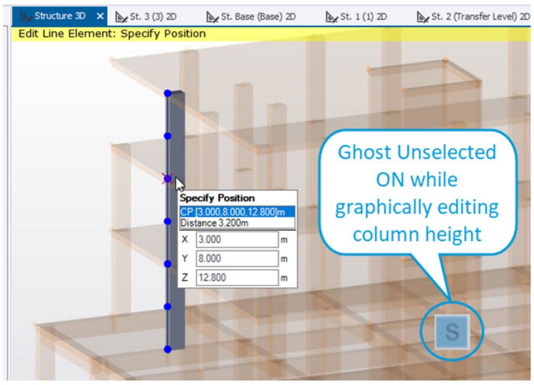
- Note that the fix requires that you upgrade your local Decking and Connector databases after installing this release - see the **Installation** section above for more directions.

- [TSD-6916] - Ghost Unselected - use of the powerful new Ghost Unselected toggle (at bottom right of the view) is extended to a number of additional operations in this release. For more about this new feature see also Release notes: Tekla Structural Designer 2020 SP2 (page 208).

  - As illustrated below, Ghost Unselected can now be used during the following operations; Copy, Move, Mirror and also Edit geometry - i.e. for example when graphically adjusting the node positions of a selected member, or of a slab panel or patch etc.



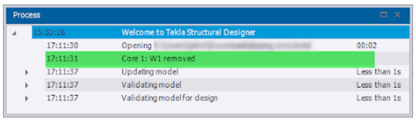




- [TSD-7364] - Licensing - in some circumstances, when a valid license could not be detected, it would not be possible to perform a manual save which could result in a loss of work. Now in this circumstance the user will be able to perform one manual save, removing the potential for lost work while re-establishing the license.
  - Note that if you are using an LT license, as before, when the limits of the license are exceeded, saving of the model will not be allowed. For more about this topic please see the article What is Tekla Structural Designer LT?
- [TSD-6900] - Cores - walls with a sloping top and/or bottom edge can no longer be selected for inclusion in cores, as they are beyond the scope of the current core forces calculations. While such walls could be added to

cores in previous releases, this would cause a program error when trying to view core results.

- For existing models with this circumstance, the problem wall(s) will be removed from the core(s) when opened in this release. A message to this effect listing the removed wall(s) will be issued in the Process window as shown below.



- [TSD-7037] - Delete - Slabs - when selecting multiple slab items for deletion, due to a recent change in release 2020, the parent slab item would also be selected as default which could lead to the inadvertent deletion of all items of a slab. This is changed in this release so that only the selected slab items are populated in the list of entities to be deleted.

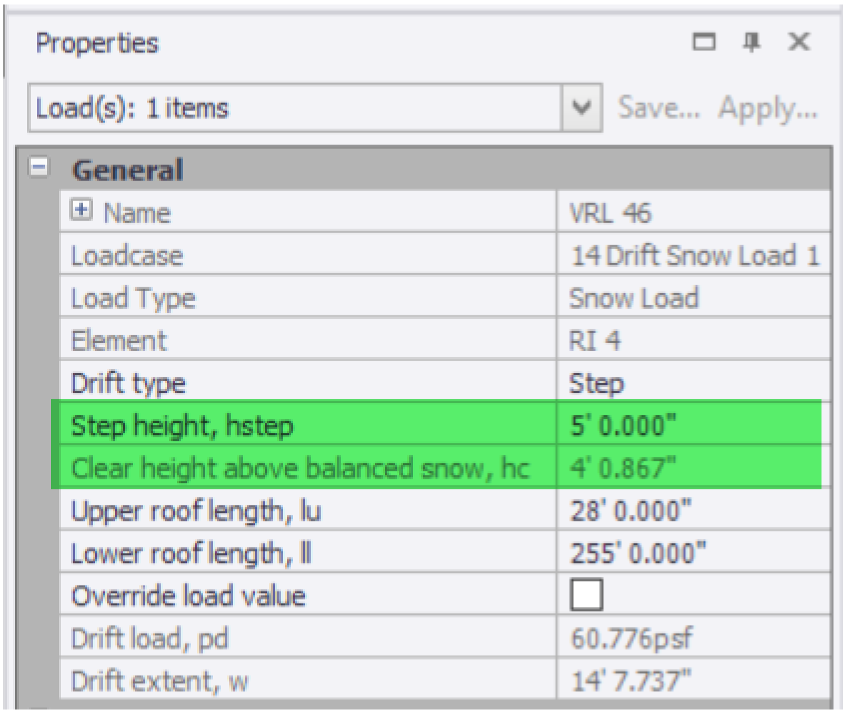
Integration

- [TSD-6910] - Structural BIM Integration - **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.0) was released on 1st July 2020 and is available in Tekla Downloads. If you are now using Autodesk Revit© 2021, you can install this to perform BIM integration with Tekla Structural Designer.
  - All fixes and enhancements included in this release are also included in updates for the Integrators specific to the following currently supported Revit© versions; 2020 (Integrator version 6.01); 2019 (Integrator version 5.02); 2018 (Integrator version 4.03). For more information about this see the Tekla Structural Designer Integrator July 2020 updates Release Notes. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

Loading

- [TSD-4790] - Snow Loading - US Head Code - in some circumstances, for Step and Parapet type drift loads applied using the Snow Wizard > Local Drift Snow command, the automatically calculated drift load p_d and drift extent w could be incorrect. The issue affected all years of the ASCE7 Snow Loading Action code available in Design Codes.
  - The error would occur when the drift height was > the step/parapet height and had two effects:
    - The maximum drift load p_d was double-counted

    - The drift width ***w*** would be incorrect and generally too low due to the use of the total parapet / step height for h_c, rather than the clear height above balanced snow load.
    - We note that the double-counting of p_d would generally counteract the reduction in ***w***, producing too much rather than too little total drift load.
- The engineer could work around the issue by enabling the “Override load value” option in the Drift Load properties and entering their own correct calculated values of drift load and extent. The issue is fixed in this release. Note that for existing files the drift values are automatically re-calculated and corrected when opened in this release.
- As shown in the picture below, as part of the fix the Drift Load properties are adjusted; the (user defined) Step/ Parapet height is now labelled “hstep”/”hpara” (formerly incorrectly labelled “hc”) and the calculated value of h_c is now reported beneath this.



- [TSD-6697] - Swedish NA (EKS11) - Eurocode based loading has been updated to reflect the changes brought by the latest version of the mandatory provisions and general recommendations on the application of European design standards for Sweden EKS 11. The new design in TSD replaces the design to the previous version, which can now be considered withdrawn from the program. The update affects both combinations from the Load Combination Generator and loads from the Wind Wizard:

- Load Combinations Generator - variable actions are no longer considered in combinations of actions based on equation 6.10a per EKS 11.
- Wind Wizard Loads:
  - Wind peak velocity - calculation of this is changed to include the peak factor k_p per EKS 11.
  - The peak factor k_p is now displayed in the Wind Data Report, appearing in the section titled 'Site Details' between the existing items 'Default Height for Internal Pressure, z_i' and 'Average height of roof tops of upwind buildings, h_ave'.

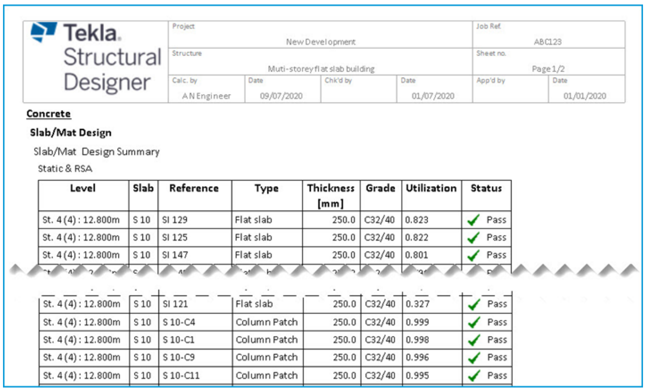
Design

US Head Code

- [TSD-7092] - Concrete Design - Beams - the calculation of the minimum beam depth h_{min} is corrected to follow the provisions of Table 9.5(a) of ACI 318-2008 and 2011 and Section 9.3.1 of ACI 318-2014. Previously, for all grades of reinforcement, the calculation incorrectly used a factor of f_y / $f_{y,mod}$. This is corrected in this release as follows:
  - For f_y = 60,000 psi; $h_{min} = h_t$
  - For f_y other than 60,000 psi; $h_{min} = h_t *f_{y,mod}$
    - Where $f_{y,mod} = (0.4 + f_y / 100,000)$

Reports and Drawings

- Reports:
  - [TSD-6906] - Concrete Design - Slabs - a concise new report content item for Slab/Mat Design Summary is now available under Model report > Concrete > Slab/Mat Design/. As shown in the picture below, this replicates in the report the Review Data > Design Summary table for Slabs/Mats which was added in the previous 2020 SP2 release. For more on this new feature see Release notes: Tekla Structural Designer 2020 SP2 (page 213).



Project			Job Ref.
New Development			ABC123
Structure			Sheet no.
Muti-storey flat slab building			Page 1/2

Calc. by	Date	Chk'd by	Date	App'd by	Date
AN Engineer	09/07/2020		01/07/2020		01/01/2020

Concrete

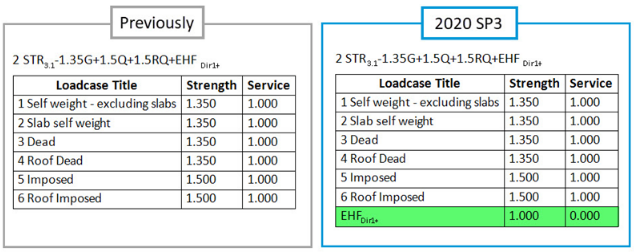
Slab/Mat Design

Slab/Mat Design Summary

Static & RSA

Level	Slab	Reference	Type	Thickness [mm]	Grade	Utilization	Status
St. 4 (4) : 12.800m	S 10	Sl 129	Flat slab	250.0	C32/40	0.823	Pass
St. 4 (4) : 12.800m	S 10	Sl 125	Flat slab	250.0	C32/40	0.822	Pass
St. 4 (4) : 12.800m	S 10	Sl 147	Flat slab	250.0	C32/40	0.801	Pass
St. 4 (4) : 12.800m	S 10	Sl 121	Flat slab	250.0	C32/40	0.327	Pass
St. 4 (4) : 12.800m	S 10	S 10-C4	Column Patch	250.0	C32/40	0.999	Pass
St. 4 (4) : 12.800m	S 10	S 10-C1	Column Patch	250.0	C32/40	0.998	Pass
St. 4 (4) : 12.800m	S 10	S 10-C9	Column Patch	250.0	C32/40	0.996	Pass
St. 4 (4) : 12.800m	S 10	S 10-C11	Column Patch	250.0	C32/40	0.995	Pass

- [TSD-7160] - Analysis Diagrams - 2D - when 2D contour diagrams - e.g. such as Mdx Top - were included in reports, the title displayed under the diagram could be incorrect. The actual effect being displayed would however be correctly shown in the top left of the diagram. This is now corrected.
- [TSD-3898] - Combinations - the report tables of load combinations listing the constituent load cases and their factors now include the Equivalent Horizontal Forces (EHF) - also termed Notional Loads (NL) and Notional Horizontal Loads (NHL) in the US and BS codes respectively - case(s) and factors where appropriate, as shown in the picture below.



Previously

2 $STR_{3.1}$-1.35G+1.5Q+1.5RQ+EHF $_{Dir1+}$

Loadcase Title	Strength	Service
1 Self weight - excluding slabs	1.350	1.000
2 Slab self weight	1.350	1.000
3 Dead	1.350	1.000
4 Roof Dead	1.350	1.000
5 Imposed	1.500	1.000
6 Roof Imposed	1.500	1.000

2020 SP3

2 $STR_{3.1}$-1.35G+1.5Q+1.5RQ+EHF $_{Dir1+}$

Loadcase Title	Strength	Service
1 Self weight - excluding slabs	1.350	1.000
2 Slab self weight	1.350	1.000
3 Dead	1.350	1.000
4 Roof Dead	1.350	1.000
5 Imposed	1.500	1.000
6 Roof Imposed	1.500	1.000
EHF_{Dir1+}	1.000	0.000

- Drawings
  - [TSD-6945] - All Drawings - Units - in some circumstances, DXF drawings produced by the program would appear to use Imperial units despite the model being set to Metric units. This is now corrected.
  - [TSD-7106] - Foundation Reactions - the reactions for supports which referenced a frame, rather than the foundation level, would be incorrectly excluded from the drawing. This is now corrected.

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Release notes: Tekla Structural Designer 2020 SP2

This release will upgrade your Tekla Structural Designer installation to version number 20.0.2.33 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than release 2020 SP1 (version 20.0.1.54), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing & Installation

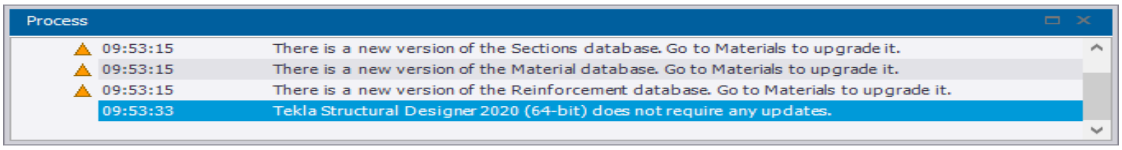
Licensing:

- No new license is required for this version.
- **License Server Version** - for Server licensing, version 3.00.0001 or later of the Tekla Structural License Service (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Server Licensing will not function correctly if this is not the case. The latest version of the License Service can be obtained from Tekla Downloads.
  - For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation:

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) to be installed.
- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - For an existing installation, if your databases are out of date, a message informing you that new databases are available will be displayed

in the Process Window as shown below when Tekla Structural Designer is first opened.



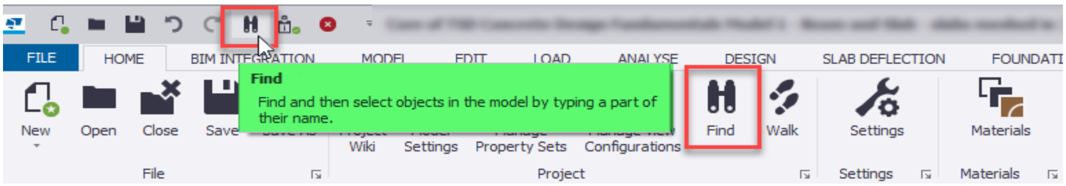
- Please ensure you upgrade your local databases as follows;
  - Open “Materials” from the Home Ribbon, select the “Sections” page in the list of options, click the “Upgrade” button, then click this button again in the subsequent “Upgrade Database” dialog. Repeat this process for all the other databases (Material, Reinforcement...etc) to ensure all your databases are up to date.

Issues with Associated Bulletins

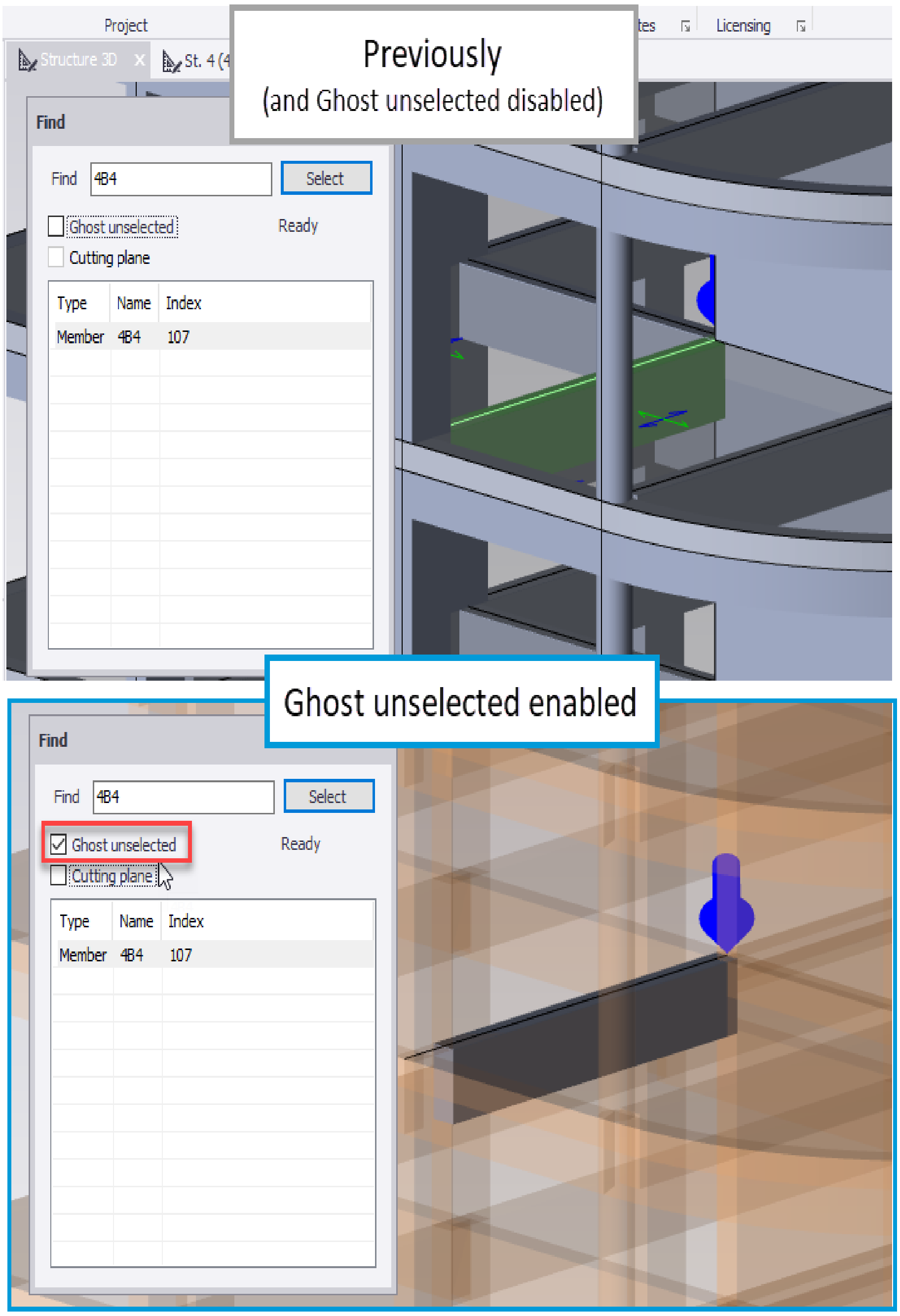
- [TSD-7014] - Concrete Column Design - US Head Code - this issue relates to the design of concrete columns to the 2014 edition of ACI 318 (years 2008 and 2011 are not affected). The issue affects the slenderness limit of section 6.2.5 for braced columns and the calculation of the uniform moment factor C_m for columns classified as slender, and could result in an unconservative design in some circumstances. For more information please see Product Bulletin Product Bulletin PBTSD-2005-1.
  - The issue is fixed in this release.
- [TSD-7035] - Modeling - Shear Only Walls - this issue relates to the new feature introduced in TSD 2020 for ‘Shear only walls’. The current modeling formulation, as the name suggests, is restricted to shear resistance and shear stiffness behaviour only. This has the limitation of failing to report the vertical ‘push-pull’ effects at the ends of the panel that would be expected to be present to resist overturning. For more information please see Product Bulletin PBTSD-2005-2.
  - Due to the potential for mis-use and mis-understanding, the option to use shear only walls is accordingly removed in this release.
  - For existing models containing shear only walls a validation error will now be issued and the analysis and design conditions are set to out of date. In order to run analysis or design, any shear only walls must then be removed, changed to a valid wall type or replaced with a bracing system as suggested in the Product Bulletin.

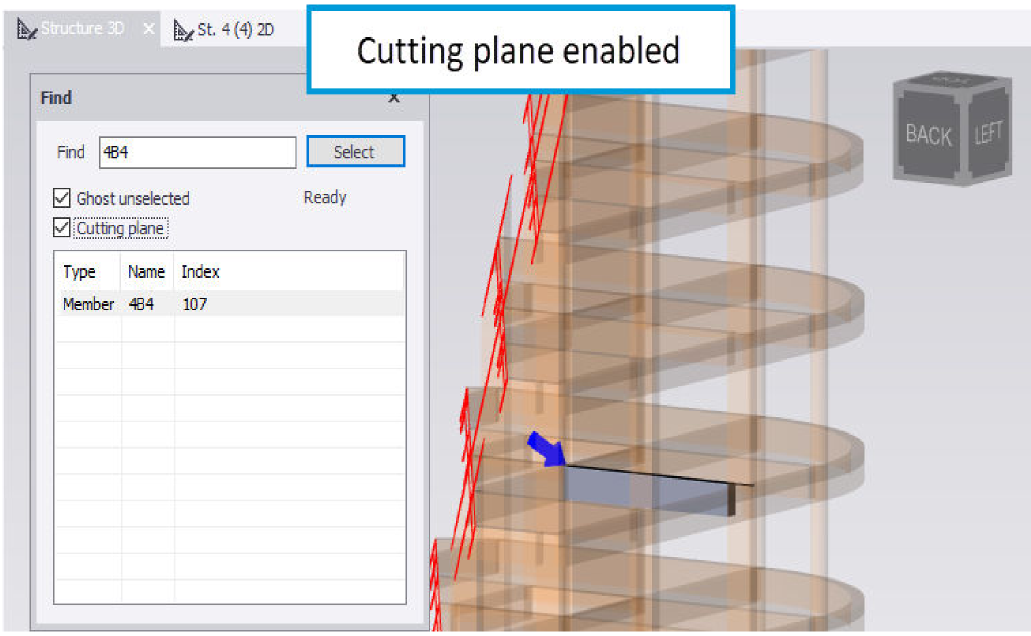
General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability.
- [TSD-5522] - Enhanced Find and Select - the Find feature and working with Selections has been significantly enhanced as detailed below:

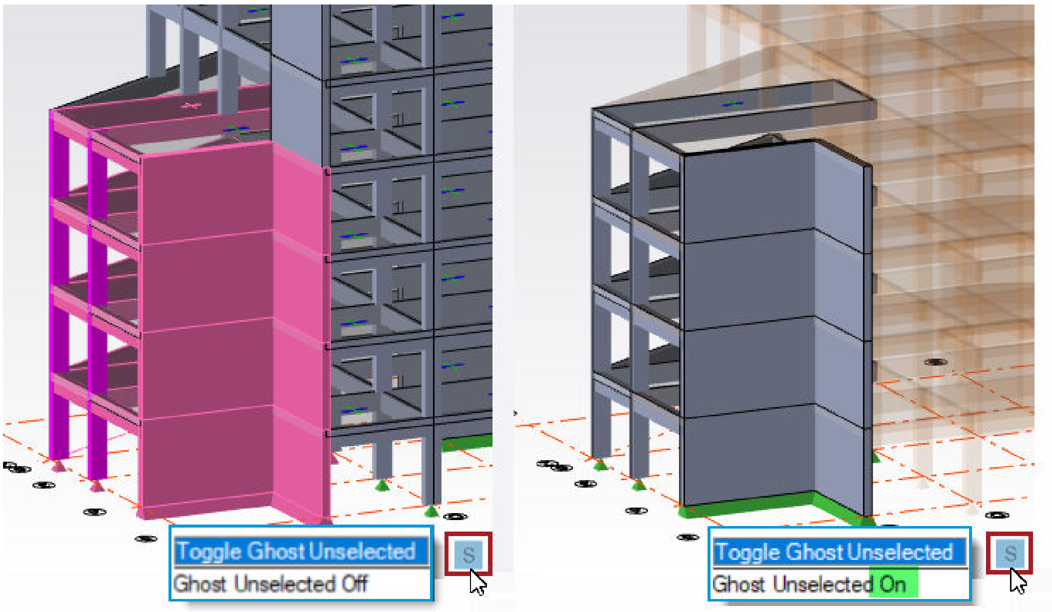


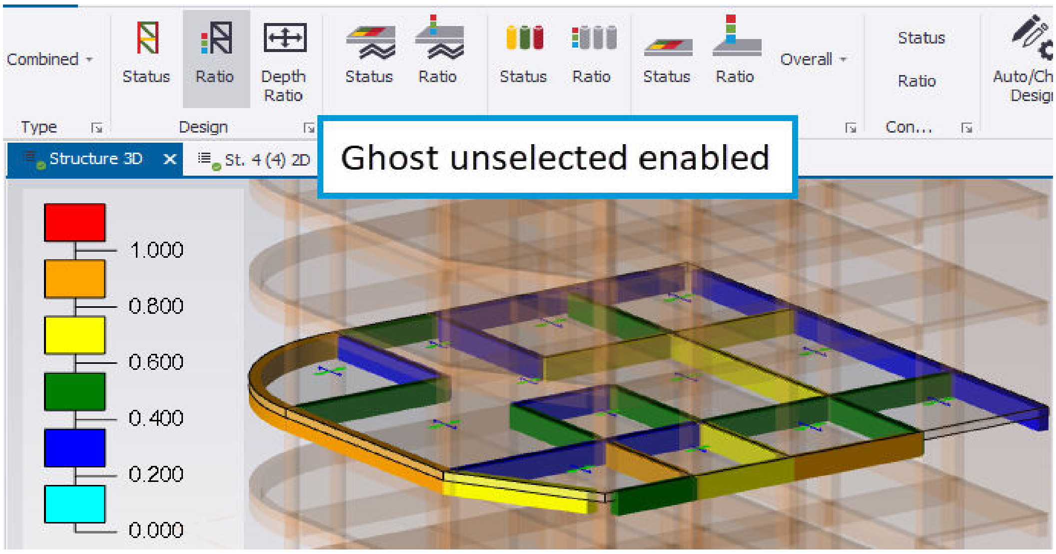
  - The Find command is now added to the Quick Access Toolbar by default as shown in the picture above.
  - Other dialogs and View windows are now accessible while the Find dialog is open. Thus, for example, the view can now be manipulated by panning, rotating etc and/ or object references checked in various locations all during an active Find operation.
  - The following new options are added
    - "Ghost unselected" (enabled by default) - when enabled, everything other than the selected item is now switched to the Ghosted view (for more on this feature see the video [Ghosted Structure view](#)) , making the selected item extremely clear and easy to locate.
      - When the Find dialog is closed, the selected item remains selected and the view reverts to normal (un-ghosted)
    - “Cutting plane” (not enabled by default) - when enabled, an automatic cutting plane perpendicular to the current view direction is activated, rendering the part of the model containing the selected object even more clear.
      - The cutting plane can then be changed simply by altering the view direction then disabling and reenabling the cutting plane option.
  - The object location arrow, that previously displayed to highlight the selected item only in some cases, now applies to all object types.
  - When the Find command is run again, the Find dialog now re-opens with the last found object listed and selected.


Previously
(and Ghost unselected disabled)
Project
Licensing
Structure 3D
Find
Find
4B4
Select
Ghost unselected
Ready
Cutting plane
Type
Name
Index
Member
4B4
107
Ghost unselected enabled
Find
Find
4B4
Select
Ghost unselected
Ready
Cutting plane
Type
Name
Index
Member
4B4
107



- Ghost Unselected Toggle - in addition to the above, the Ghosting of unselected objects is extended to all views - of all or part of the structure - via the new “S” toggle button at the bottom right of the view window.
    - As illustrated in the pictures below, when toggled On (at right), all unselected objects are shown as ghosted and only selected objects have solid shading but without the distinct opaque selection colours. The selection can be altered as the engineer requires in this view, making editing and working with selections even easier.
        - This can prove especially useful in the Review View where otherwise the selection colours would obscure other colour-coding such as pass/fail status and utilization ratio etc. This facilitates reviewing the status of parts of the structure, such as member design groups, especially in larger models.
        - The new control can be used in a view of part of the structure - for example a floor or sub structure view - to further focus the desired selection operation. 2D views - such as of Levels and Frames - must first be toggled to 3D ‘mode’ to enable this option.
    - When toggled Off (on the left) behaviour is as previously; all objects have regular shading, with selected objects assigned the distinct opaque selection colours.


Toggle Ghost Unselected
Ghost Unselected Off
S
Toggle Ghost Unselected
Ghost Unselected On
S



- [TSD-6192] - Review View - Copy Properties - following customer feedback, the default attribute for the Copy Properties Show/Alter State attribute is changed to “All properties” where formerly it was “Deflection Limits”.

Integration

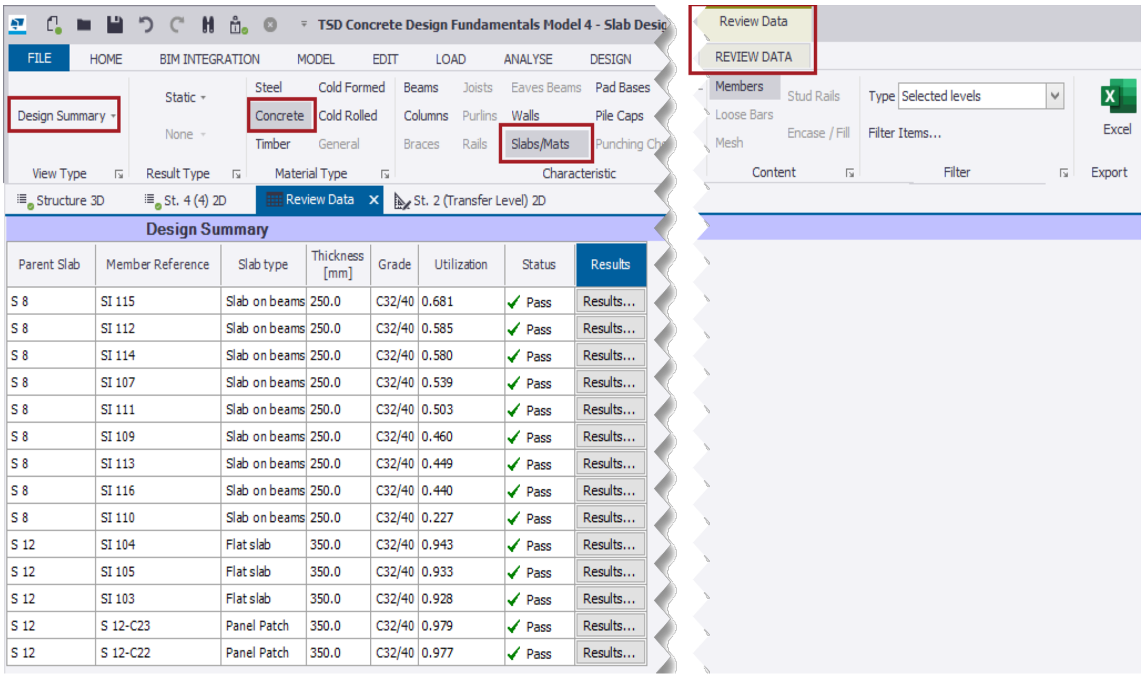
- [TSD-6193] - Steel Connections - IDEA Statica - export of steel connections to IDEA StatiCa has now been enabled for models set to the British Standards (BS) Head Code. Note the following:
    - Connection design in IDEA StatiCa can only be performed to Eurocode EC3 and not BS 5950.
    - The elastic modulus in such cases is exported as 205,000 MPa which is the BS value rather than the EC3 value of 210,000 MPa.

Design - General

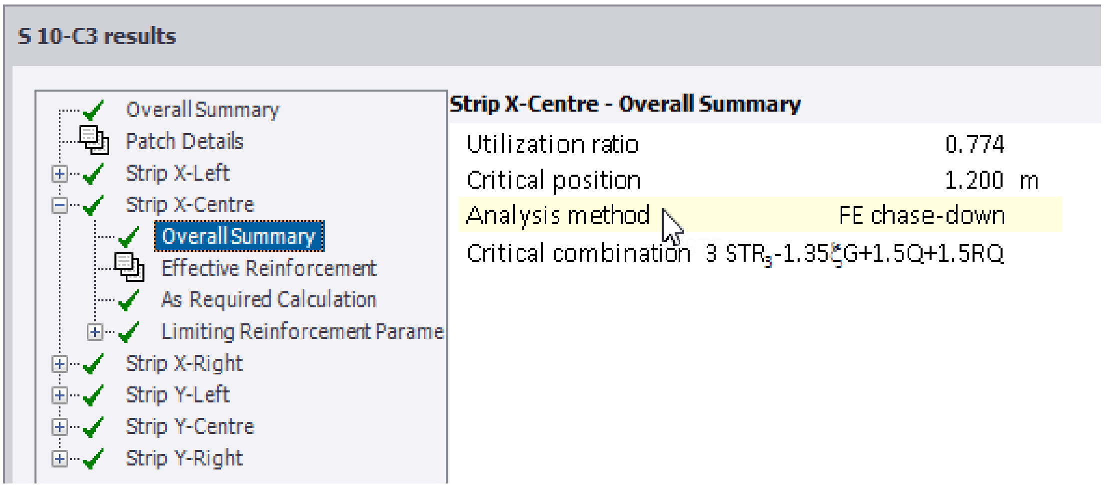
- Concrete Design - Slabs - All Head Codes:
    - [TSD-1229] - Review View > Design Summary - a brand new Design Summary for Slabs & Mats is added to the Review View design result tables, giving a concise tabular view of slab design results for all slab

and mat types. This can be selected as shown in the picture below and, like other tabular design details, can be exported directly to Excel.

- All slab items and patches associated with the slab are listed together. By default, slab and patch items are listed in order of utilization ratio with slab items listed first followed by patches, then mats and their patches.
- The table can be sorted differently simply by clicking on a column header (with the exception of the Grade and Result columns)
- The summary table reflects the critical results from all the available RSA and Static combinations.



- [TSD-6872] - Analysis Method - the design details now state the Analysis method from which the critical design moment originates, as shown in the picture below.



Reports and Drawings

- [TSD-4253] - Tabular Data & Reports - Member Design Summary - user-defined span/ stack names are now included in the Tabular Data design summary and associated report item where previously the span/ stack number only was reported. This applies to all member types and materials.
- [TSD-6749] - Wall Schedule Drawing - improved tolerance is added to prevent detection of very slight section differences along the wall height which would commonly be ignored and could formerly result in unnecessary section duplication and notes regarding the cross section not being consistent in the drawing.

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Release notes: Tekla Structural Designer 2020 SP1

This release will upgrade your Tekla Structural Designer installation to version number 20.0.1.54 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than release 2020 first release (version 20.0.0.129), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

Licensing & Installation

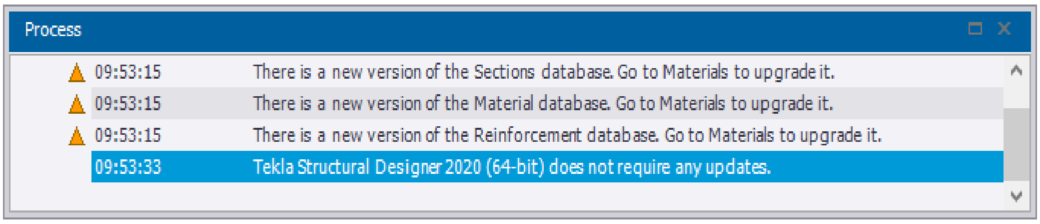
Licensing:

- No new license is required for this version.
- **Tekla Structural License Service** - a Service Pack for this was released on 1st April 2020. This addressed the issue of the “Automatic” license method not correctly checking for a Sentinel RMS Server license when all Online licenses were in use. This is available in Tekla Downloads and should be installed on all clients for optimum functionality.
  - For more information see: https://teklastructuraldesigner.support.tekla.com/support-article/3050337
- **License Server Version** - for Server licensing, the latest version of the Tekla Structural License Service 3.00.0001 (incorporating Sentinel RMS 9.5) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2020 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation:

- This service pack requires Tekla Structural Designer 2020 first release (version 20.0.0.129) to be installed.
- **Integration**
  - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate Tekla Structural Designer 2020 SP1 with Tekla Portal Frame Designer and/or Tekla Connection Designer, you must use Tekla Portal Frame Designer and/or Tekla Connection Designer release 20 or later (note that these require activation of 2020 release PAK’s for these products). For optimum function we recommend you install Service Pack 1 for these products, which is available from the same date as this release. Installations for these can be obtained from Tekla Download Service.
- **Previous Versions and file compatibility** - Files from all previous versions can be opened in this release however note that, once saved, they may not open in an older version. If you wish to retain this option we therefore recommend using the File > Save As... option to save a new version of the file and retain the original.
- **Databases** - For an existing installation, if your databases are out of date, a message informing you that new databases are available will be displayed

in the Process Window as shown below when Tekla Structural Designer is first opened.



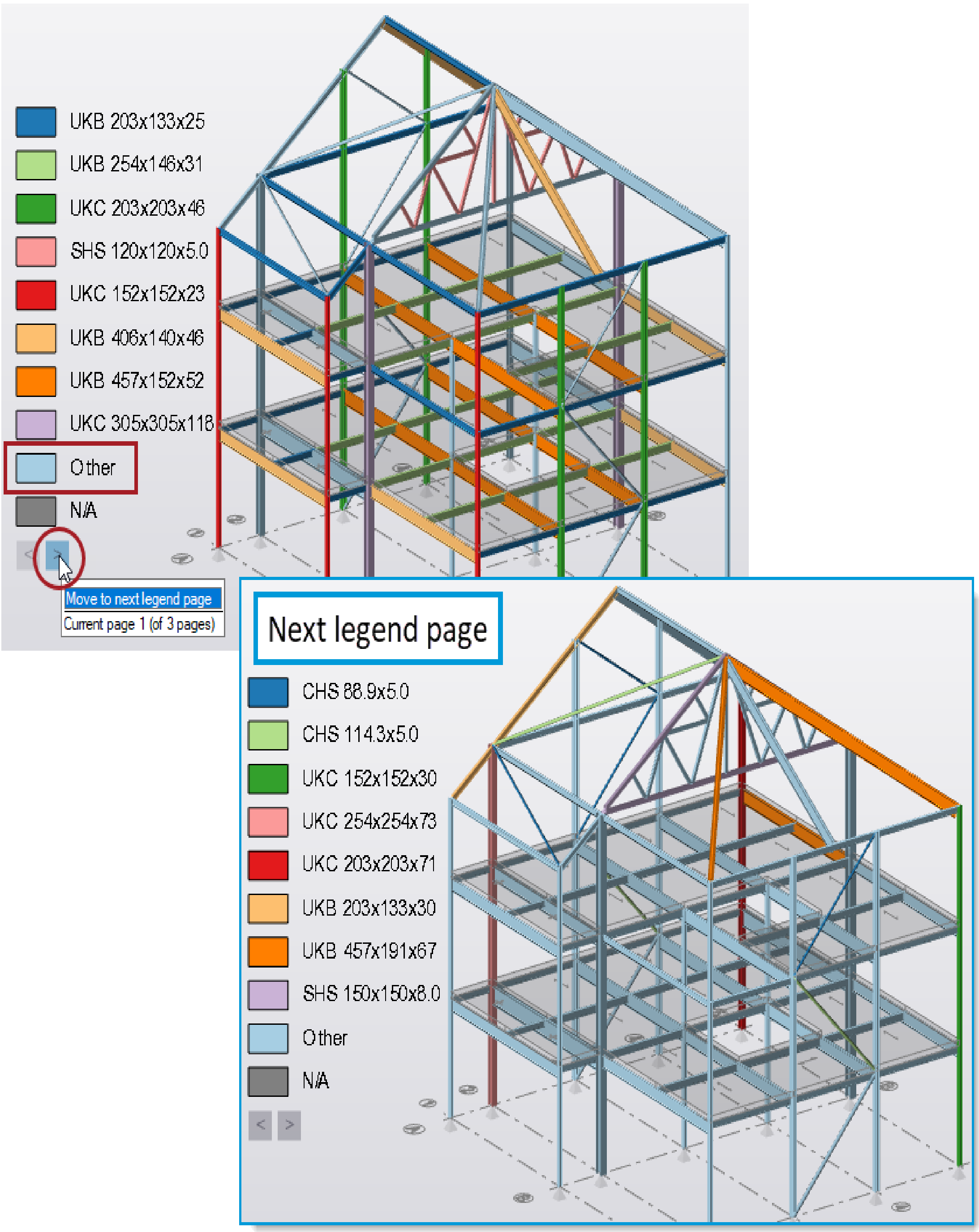
- Please ensure you upgrade your local databases as follows;
  - Open “Materials” from the Home Ribbon, select the “Sections” page in the list of options, click the “Upgrade” button, then click this button again in the subsequent “Upgrade Database” dialog. Repeat this process for all the other databases (Material, Reinforcement...etc) to ensure all your databases are up to date.

Issues with Associated Bulletins

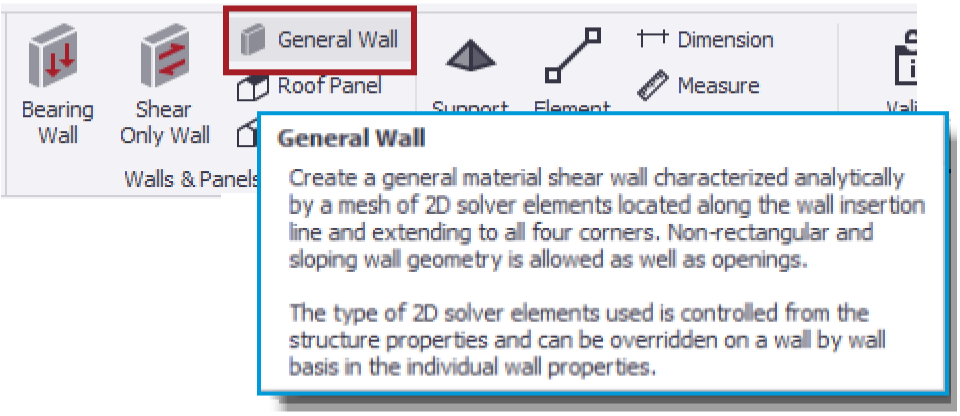
- None of the issues addressed in this service pack have associated bulletins.

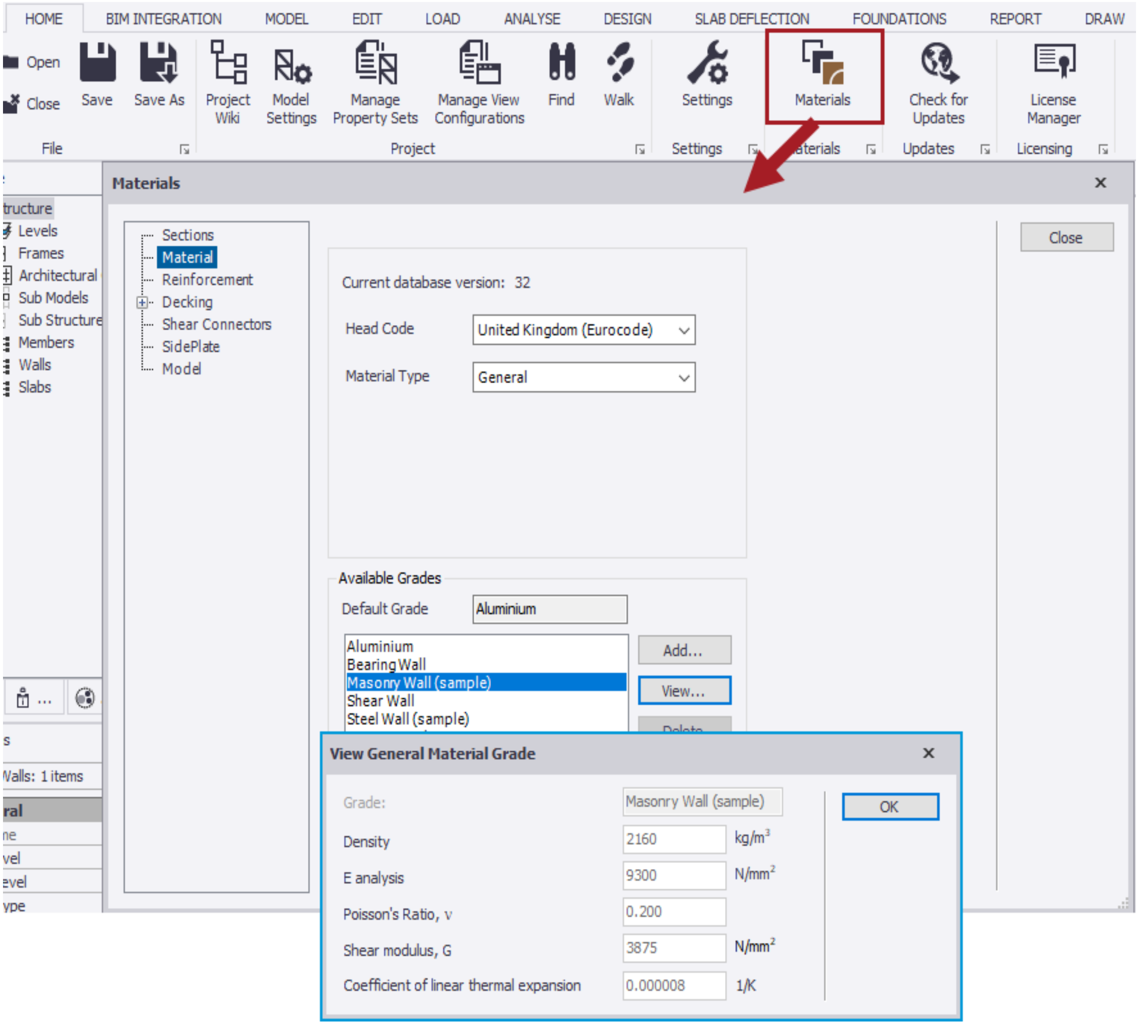
General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of such an issue is listed directly below here:
  - [TSD-6704] - Property Sets - Using the Apply... button at the top of the Properties Window to apply an existing property set to a selected member could in certain circumstances produce a crash. This has now been corrected. Note that this issue applies only to version 2020 first release in which this issue was introduced.
- [TSD-422] - Review View - Show/Alter State - The legend of the Show/Alter state operation of the Review regime has been enhanced so that it now allows multiple pages and will effectively list, for the selected mode, all the different states/attributes from all relevant objects in view. This is a significant enhancement for cases where the maximum number legend items (10) is exceeded, which previously limited functionality.
  - As shown in the picture below, where > 10 items/states exist (e.g. section sizes for the Section Attribute), forward/ back buttons now appear at the bottom of the legend; click the forward > button to access the next legend page. On subsequent pages the legend colours differentiate objects classed as ‘Other’ on the previous page(s).


UKB 203x133x25
UKB 254x146x31
UKC 203x203x46
SHS 120x120x5.0
UKC 152x152x23
UKB 406x140x46
UKB 457x152x52
UKC 305x305x118
Other
N/A
Move to next legend page
Current page 1 (of 3 pages)
Next legend page
CHS 88.9x5.0
CHS 114.3x5.0
UKC 152x152x30
UKC 254x254x73
UKC 203x203x71
UKB 203x133x30
UKB 457x191x67
SHS 150x150x8.0
Other
N/A

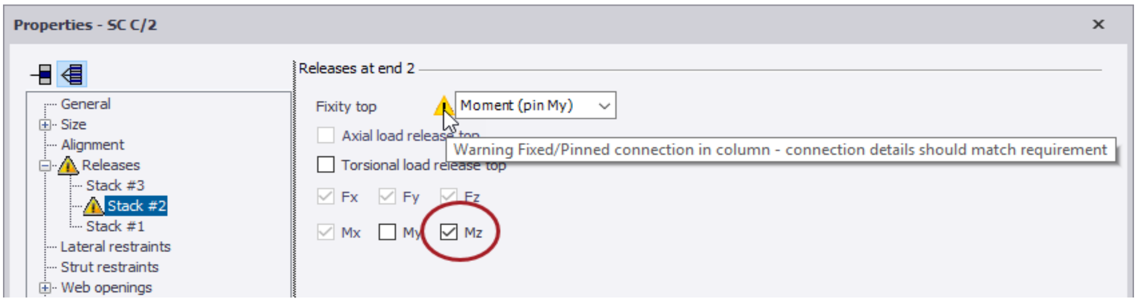
    - This new functionality applies to the following Review View > Show/Alter State Attributes:
        - Section (Review mode)
        - Material Grade (Review mode)
        - SFRS (Review mode)
        - Size Constraints (Review mode)
        - UDA
- [TSD-5113] - General Walls - following customer requests, a new wall type “General Wall” has been added to the modeling options. This uses the same analytical model as that of meshed concrete walls (for more on this see the Help Topic How meshed walls are represented in solver models (page 750)), however the material type is General and any of the materials types defined in this category can be used.
    - Key aspects of the new General Walls are:
        - The wall properties type of “Meshed Shear Wall” is the same as that used for meshed concrete walls - thus the distinction between meshed Concrete and General wall is controlled solely by the Material Type.
            - Mid-pier walls can only be Concrete.
        - They are distinguished graphically in Scenes by use of a specific default light blue colour different to the grey of concrete walls.
        - Geometric rules are as for meshed concrete walls; they can be vertical or sloped and openings are allowed.
        - The same graphical and tabular data results are available for them as for concrete meshed walls, including 2D contours (other than As Req), 2D Wall Lines and Result Lines.
        - New Analysis Modification factors are added for them, accessed via Analyze > Settings > Modification Factors > General > Wall
        - They cannot be included in cores, are not designed and have no reinforcement or seismic properties.
    - Some new sample general materials for walls are included in the latest database version (note this requires the Material Database to be updated to the latest version).
    - As for meshed concrete walls, the behaviour of the 2D mesh elements - and hence General Walls - is isotropic. We do not advocate their use for any particular wall type or building/ structure - this and the material properties used for them are the responsibility of the engineer.


General Wall
Roof Panel
Bearing Wall
Shear Only Wall
Walls & Panels
Dimension
Measure
General Wall
Create a general material shear wall characterized analytically by a mesh of 2D solver elements located along the wall insertion line and extending to all four corners. Non-rectangular and sloping wall geometry is allowed as well as openings.
The type of 2D solver elements used is controlled from the structure properties and can be overridden on a wall by wall basis in the individual wall properties.

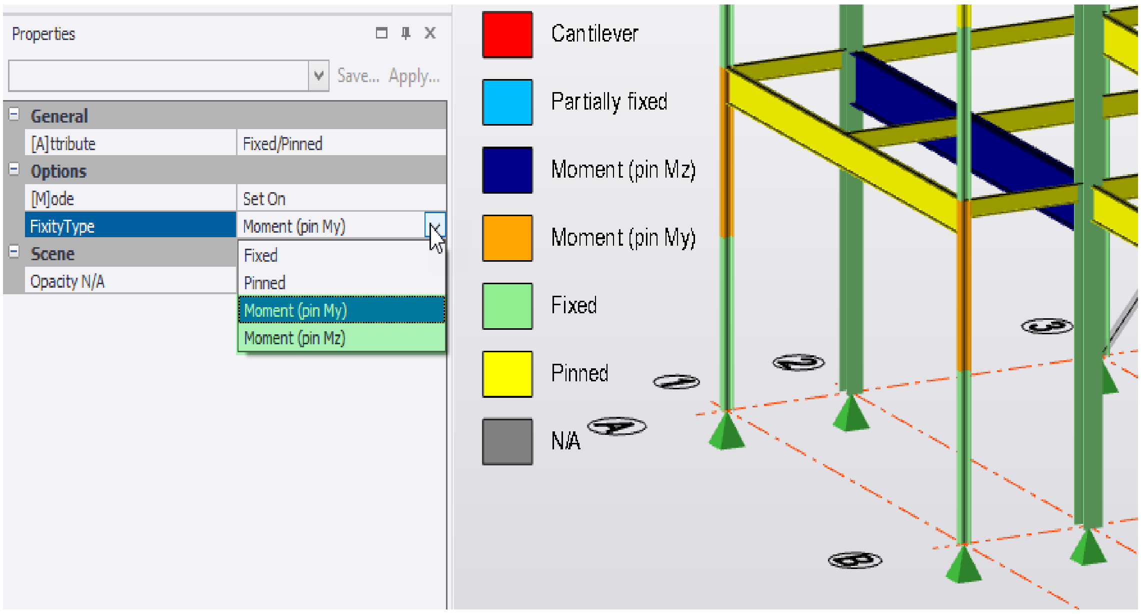


- [TSD-1365] - Columns - Releases - following customer feedback, for continuous columns composed of two or more stacks, the ability is added to pin columns in one direction (My or Mz) whilst the other direction remains fixed at the top and bottom of stacks within a column. While such a release condition is less usual, it is now allowed for columns of all materials.

- Note that, when such a release is applied, a Warning status will be issued as shown in the picture below. This is intentional and does not prevent analysis or design - as the Warning tooltip states, it is to prompt the engineer to consider the connection details at this location in the column.



- Review View > Show/Alter state Fixed/Pinned command is also enhanced to enable rapid review and application of these releases (for beams as well as columns); new “Moment (pin My)” and “Moment (pin Mz)” options are added to the Fixity Types and legend as shown in the picture below.



- [TSD-6110] - Steel Compound Sections - the plate width for the double I section with plates compound section may now be less than the combined width of the I section plus any gap.
- [TSD-6709] - Connection Resistance Data - when upgrading the connection resistance database from earlier versions, zero values in the database (which previously indicated a connection type as being not applicable), would be incorrectly transferred, leading to connection types being listed that were not applicable for the selected Head Code. This issue applies only to the release 2020 first-release and is now corrected.

Integration

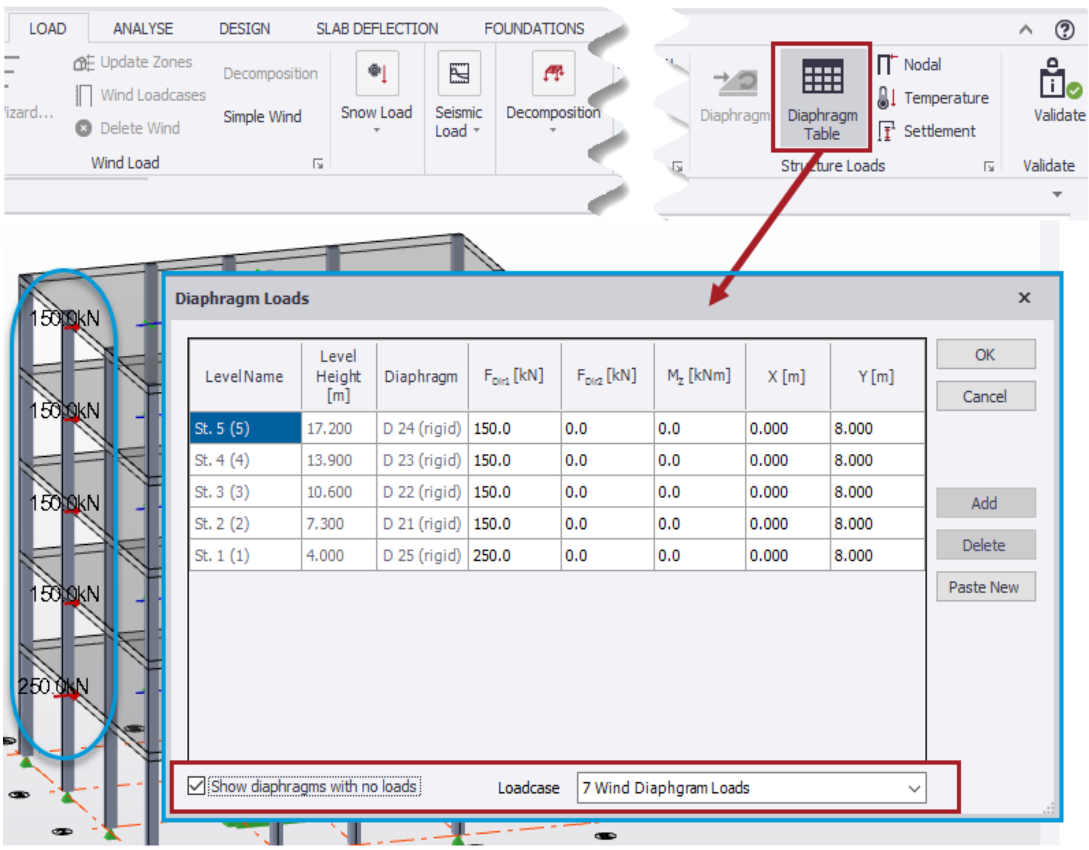
- [TSD-6230, 6232] - API - the API version has been updated to enable the extraction of internal Member forces both directly from the Solver and Loading analysis. Full details of the API, including worked examples, will be available shortly in the Tekla Developer Center.

Performance

- [TSD-5437] - Reports - Enhancements have been made which significantly reduce peak RAM demands during report creation, this may improve general operation speed for some structures.

Loading

- [TSD-5678] - New Diaphragm Loads Table - building on the new Diaphragm Loads feature of the recent 2020 release - for the application of externally determined level loads (for example from Wind Tunnel testing of large/ unusual structures) - in this release a Diaphragm Loads Table is added for these as illustrated in the picture below. The Table allows rapid tabular definition, editing and verification of Diaphragm loads.
    - All Diaphragm loads added using the existing Diaphragm Load command (active in 2D views) are listed in the table. Note that multiple loads can be applied to a single diaphragm, and that by default only diaphragms with loads applied are listed.
    - Loads can also be added manually in the table to existing diaphragms with no current loads as follows:
        - Check on the “Show diaphragms with no loads” check box to list all diaphragms with no loads currently applied.
        - Select the Loadcase you wish to add loads to from the Loadcase list box
        - Select the row for a diaphragm then click the “Add” button - the cells of the row are then activated and the Load and X and Y coordinate values can be input.



- Data can also be pasted into the table from an external application for example Excel - ensure that the data is in the same format and order and includes a Z (Level height) column as shown in the picture below (this is not the same order as in the diaphragm loads table).
    - Note that pasted data replaces any existing loads in the current loadcase, it does not merge loading. Also note that the level height does not need to be exact, during the import process the closest diaphragm will be identified and the height will be adjusted (with warnings if the adjustment is excessive).

Fx	Fy	Mz	X	Y	Z
200	0	50	0	8	17.2
200	0	50	0	8	13.9
200	0	50	0	8	10.6
200	0	50	0	8	7.3
400	0	100	0	8	4

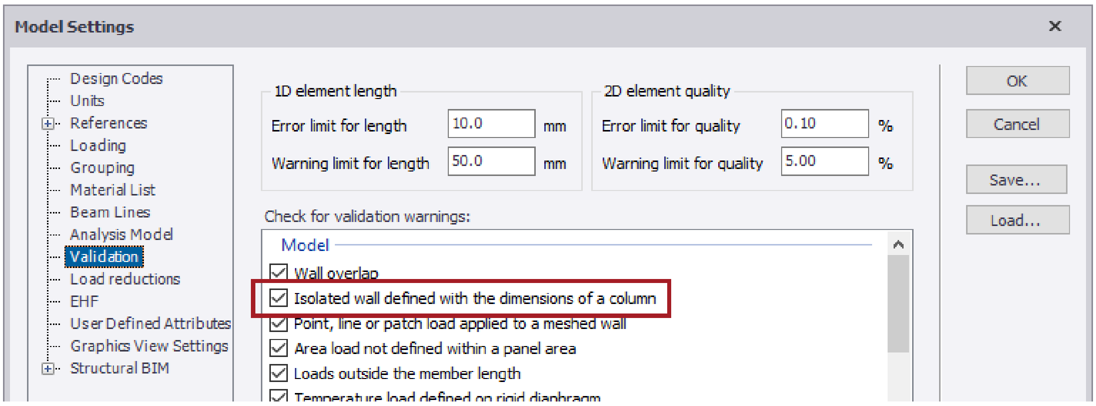
- Load Combination Generator:
  - [TSD-6347] - Eurocode - GEO Combinations - to help prevent the automatic creation of unnecessary combinations, new intelligence is added to the combination generator for the case where the foundation bearing capacity calculation is set to use the presumed bearing capacity method of EN 1997-1:2004, cl. 6.5.2.4 (Design Settings > Concrete > Cast-in-place > Foundations > General > "Use presumed bearing capacity (...)" checked on) and hence GEO combinations are not usually required. In this case the GEO combinations option is no longer checked on by default in the Combination generator. The engineer can still activate these for inclusion should they so wish.
  - [TSD-6341] - Eurocode - EQU Combinations - since these combinations are less commonly required and for the same reason as described in the above issue, the EQU combinations option is no longer checked on by default in the Combination Generator. The engineer can still activate these for inclusion should they so wish.
    - Note related issue in the **Design - Eurocode** section - EQU combination results are not considered currently for any member design within Tekla Structural Designer, nor are there any EQU limit state checks performed based on them.

Validation

- [TSD-6040] - Concrete Walls - All Head Codes - a new validation warning is now issued for isolated walls defined with “the dimensions of a column” (with L ≤ 4W where L is the wall length and W the thickness) and not forming part of a core wall. The warning message states “Isolated wall defined with dimensions of a column” and the tooltip "If the wall is not part of an interconnected concrete core then it should be defined and designed as a column".
  - At some ratio of L to W a concrete member would be considered to be more a column than a wall. Logically, some level of engineering judgement applies in the distinction and when - i.e. at what ratio - it should be made. Eurocode EC2 Cl 5.3.1(7) states a member is a column when the ratio “does not exceed 4”. Hence this ratio is adopted for the

warning limit and we thought it prudent it be applied for all Head Codes.

- The warning can be acted upon or ignored at the engineer's discretion. It can also be enabled/ disabled via Home - Model settings - Validation - checkbox for "Isolated wall defined with the dimensions of a column" as shown in the picture below.

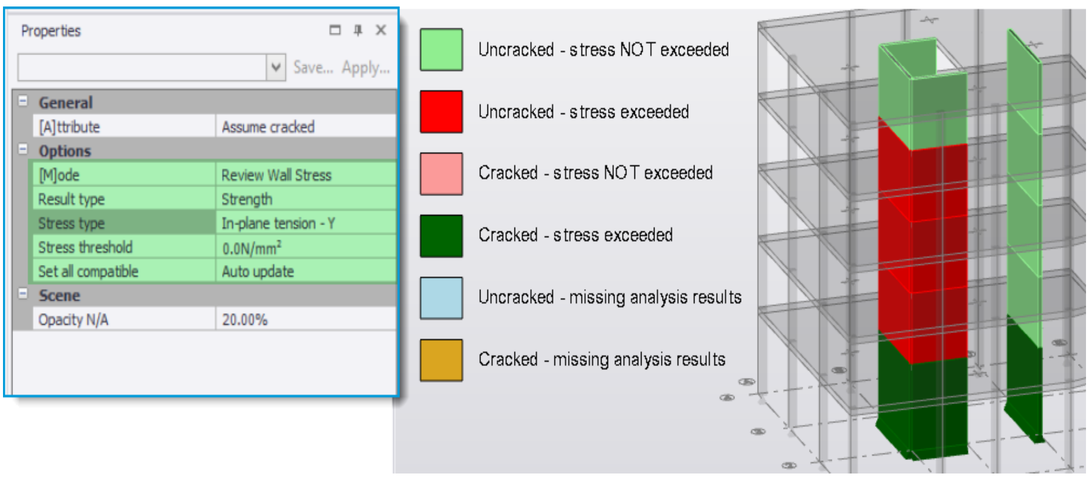


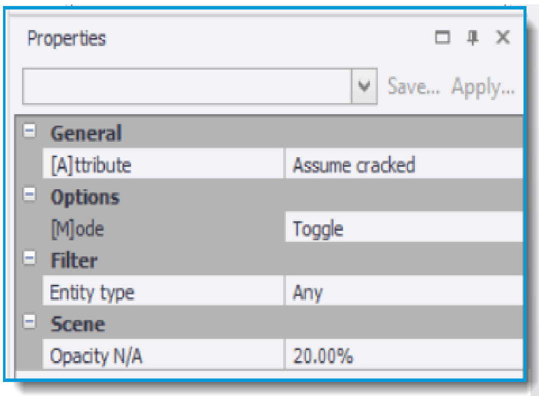
Analysis & Results

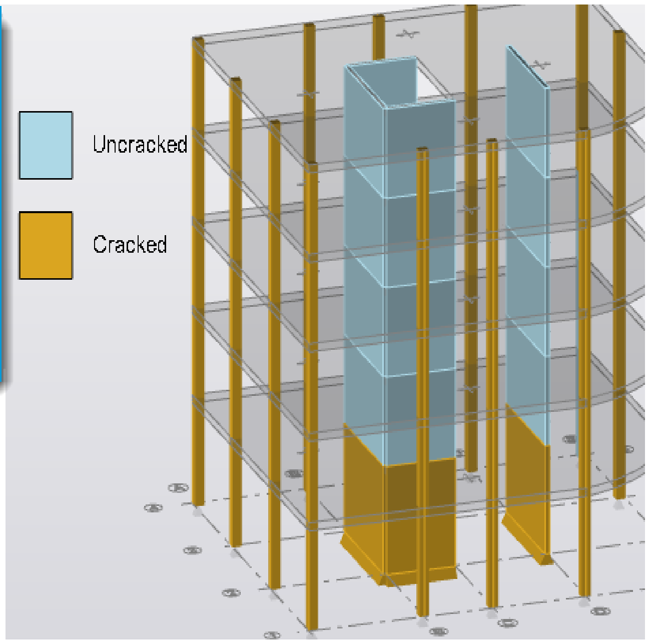
- [TSD-6480] - Solver model - improvements are made to the solver model for haunched beams which could result in a zero length analysis element error for some geometries.
- [TSD-6373] - Chase-down Analysis - an analysis process error would occur during the calculation of chasedown combinations when slab mesh elements in a semi-rigid diaphragm were connected at the supported end of a meshed shear wall. The analysis model is enhanced to cater for this circumstance and the error no longer occurs.
- [TSD-6544] - Analyze Ribbon - the following minor changes are made to the ribbon to make its function more clear:
    - "Solver" group is renamed to "Data".
    - "Tabular Data" button renamed to "Model & Results"
    - The Meshing group (with buttons Mesh Slabs and Update Wall Beams) now only appears when a graphical view is active.

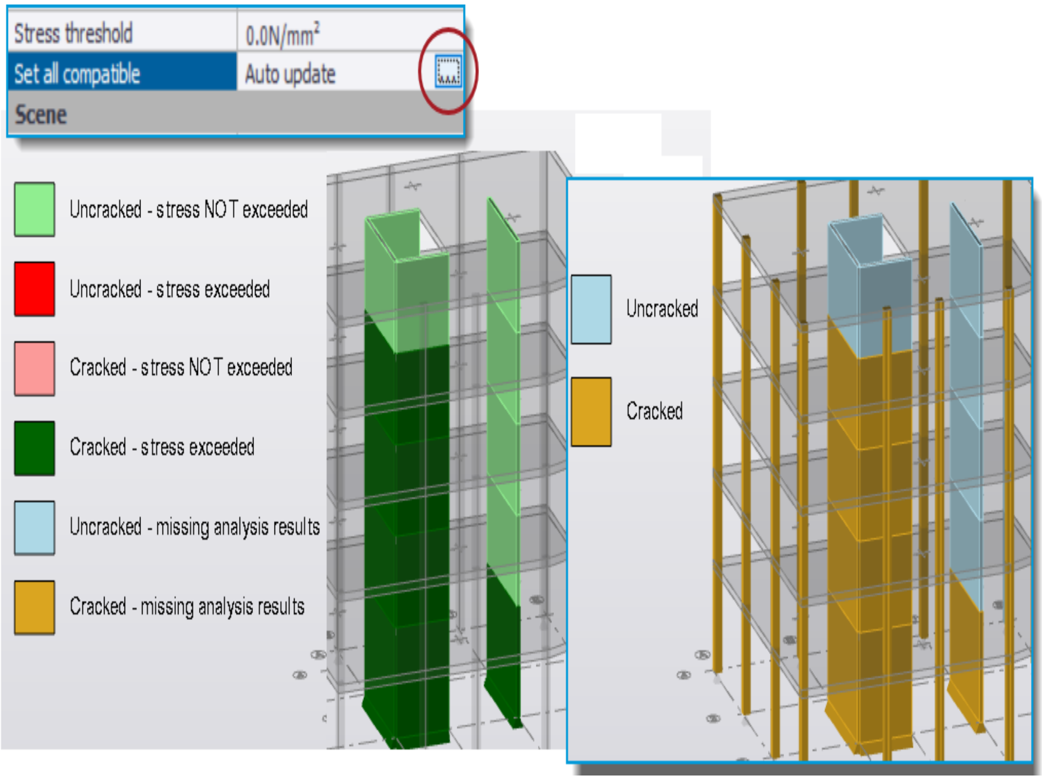
Design - General

- [TSD-5107, 5109] - Concrete Walls - Cracked State - as illustrated in the pictures below, the Review View > Show/Alter State > Assume cracked command is significantly enhanced with a powerful new “Review Wall Stress” mode. This builds on the enhancements made in recent releases for the assessment and control of the cracked status of concrete walls; the addition of In-plane stress 2D result values and the Assume cracked Attribute to Review View > Show/Alter State.
    - The Review Wall Stress mode controls allow the engineer to set the Result and Stress type and stress threshold value to be used for the wall panel cracked status determination - the view is updated immediately following changes to these.
        - Note that a (tension) stress threshold of zero as shown in the picture below would be conservative.
    - Using this feature the engineer can rapidly assess the cracked status of wall panels relative to the current option settings. Panels where the cracked status is correct based on the current settings are shown in shades of green colours, panels where the status is not correct are shown in shades of red.
    - There is also a powerful Auto update feature which automatically sets the assume cracked status from the graphical check results; as illustrated below, this is activated via the [...] button at the right of the Set all compatible > Auto update Properties cell.




Properties
Save... Apply...
General
[A]ttribute
Assume cracked
Options
[M]ode
Toggle
Filter
Entity type
Any
Scene
Opacity N/A
20.00%


Uncracked
Cracked



For more information please see: Concrete member cracked or uncracked status (page 1286)

- [TSD-2341] - Concrete Seismic Design - US and Indian Head Codes - following the enhancement in previous releases to allow columns to be assigned to an SFRS in both directions, the Seismic Force Resisting System framing members check is enhanced to cater for all different SFRS Type / Direction combinations between the two allowed SFRS directions.

Design - Eurocode

- [TSD-6341-5] - EQU Combinations - the results of EQU combinations are no longer included in any member design.
  - Eurocode EQU combinations were added to the Combination Generator in an earlier release, principally for industrial structures in which they are used for checking the equilibrium of the structure as a rigid body,

    but not for member design. However, in previous releases, results of these combinations were incorrectly used in member design.

    - Note the related issue in the **Loading** section - EQU combinations are no longer active by default in the Combination Generator.

  - This applies to the following circumstances and member types (which were formerly checked for EQU combinations); concrete members, walls and slabs; Steel/Cold formed members of all designed characteristics.

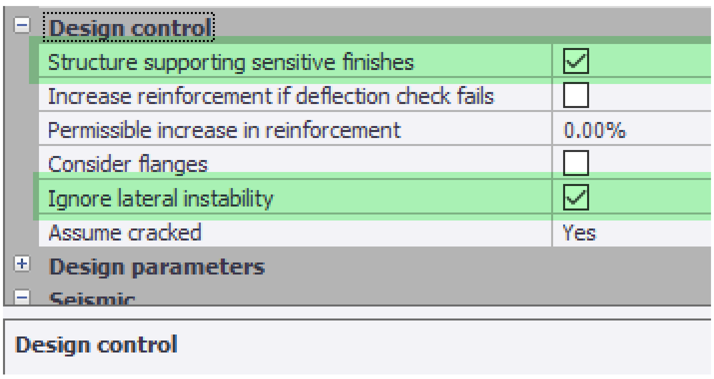
- Concrete Design - Beams:

  - [TSD-1377] - Longitudinal Reinforcement - the method to calculate the minimum area of longitudinal reinforcement, as required by EN 1992-1 clause 9.2.1.4, has been altered so that the requisite applies directly to the required areas of reinforcement instead of applying to the design moments as previously. This change ensures that the minimum is checked against an overall area of reinforcement in the design region without any further additions. As a result engineers may, in cases where min reinforcement requirements govern, obtain a more efficient longitudinal reinforcement design.

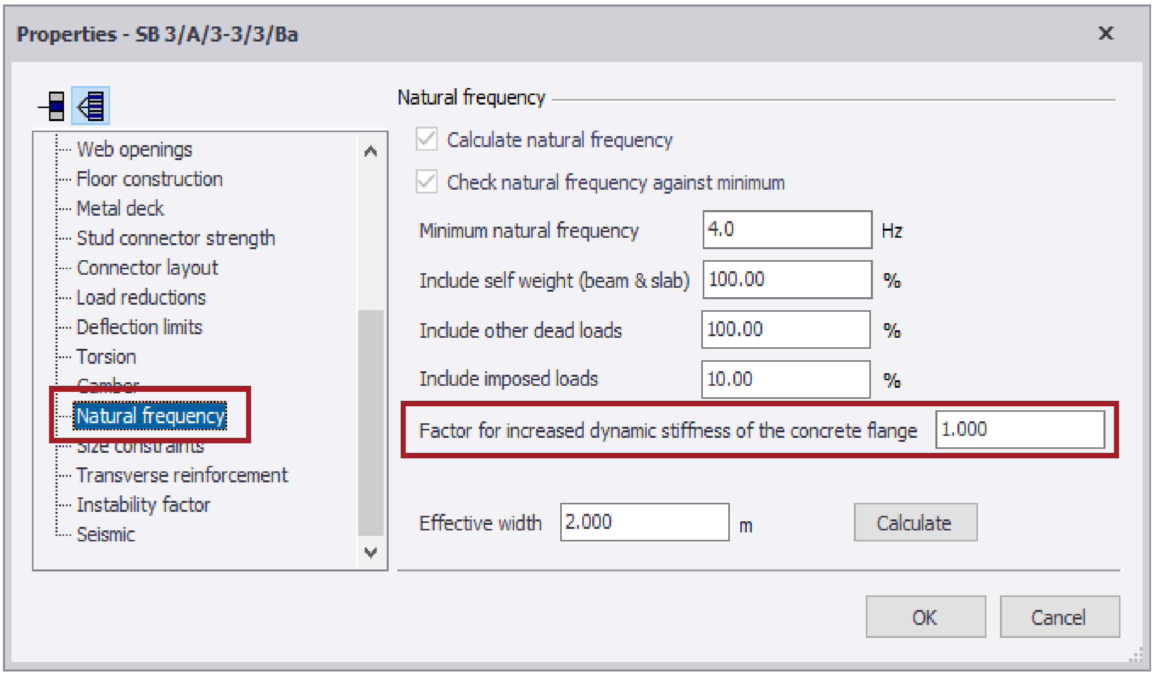
  - Design Control - Allowing engineers to exercise their judgement , the following new "Design control" options are added (available for individual spans of continuous beams):

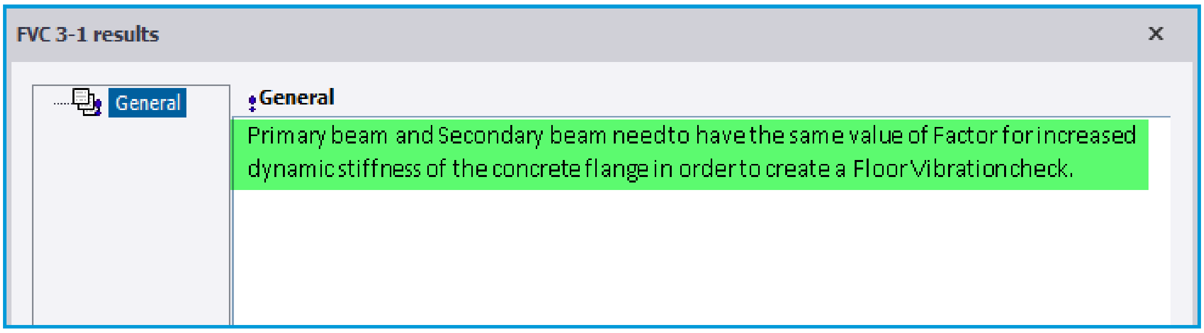
    - [TSD-1801] - new option to consider or ignore lateral instability for slender spans to EC2 clause 5.9(1) (off by default). When the option is checked on - as shown in the picture below - the slender span check is excluded from design.

    - [TSD-1893] - Deflection Check - new option "Structure supporting sensitive finishes" is now available under "Design Control" (on by default) to control the value of the f_2 parameter used in the deflection check. When unchecked f_2 will be taken as 1.0.



- Floor Vibration Check to SCI P354:
  - [TSD-5668] - the “Factor for increased dynamic stiffness of the concrete flange” value set for the beams specified in the check is now used by the Floor Vibration check to determine the dynamic modular ratio of the slab (rather than a fixed value of 1.100 as previously). For more information about this see the TUA article Beam and Floor Vibration Checks - Dynamic Modular Ratio and SCI AD 433.
    - The “Factor for increased dynamic stiffness of the concrete flange” is set in the Natural Frequency group of the Beam properties as shown in the picture below. The value must be the same for both the primary and secondary beams of the check. The check is not performed and a warning now issued if this is not the case, as shown below.

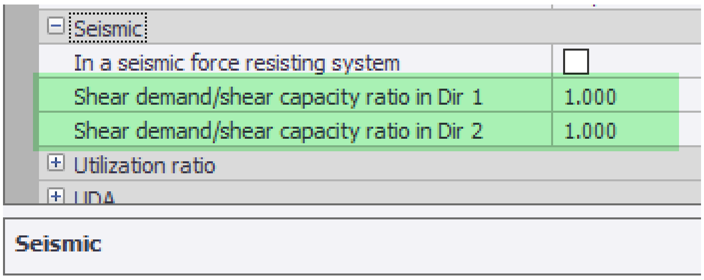




- [TSD-4234] - Fabsec beams with openings are now considered without error in the SCI Floor Vibration Check to the SCI P354 Handbook.

Design - US

- Seismic Drift:
  - [TSD-4134] - Shear demand / capacity ratio (beta) - the engineer can now directly define the shear demand / capacity ratio (beta) for columns and walls. Previously this was fixed to a value of 1.0 which could be over-conservative. The option to specify the value is added to the “Seismic” group of member properties, and can be set separately for 'Direction 1' and 'Direction 2' and for each stack/panel, as shown in the picture below.
    - The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.



  - [TSD-4234] - Seismic Stability Coefficient - the calculation of the seismic stability coefficient, theta, is modified to take advantage of Clause 12.8.7 when 2nd order analysis is used in design. This change applies to all years of the ASCE7 code and the changes are summarized as follows:
    - Modified theta calculation following Clause 12.8.7 when 2nd order analysis is set for design (Design > Settings > Analysis)
    - Where First order analysis is set for design, a warning is issued to use 2nd order analysis even if theta is greater than theta max.
    - The drift ratio check and status are updated to follow Clause 12.12.1.1 when the seismic design category is D,E or F and it is not a fixed structure.
  - [TSD-4136] - Review Data - following customer feedback, the Seismic Drift results table in Review data (and the associated Report item) has been improved; the table now reports the drift ratio and theta ratio respectively for Dir 1 & Dir 2 where previously it reported absolute values of drift and theta for Dir 1 & Dir 2.
- Seismic Design:
  - [TSD-6623] - SFRS Direction - Beams - when beams are selected as a part of an SFRS system, the SFRS direction is now assigned

automatically based on the modelling direction of the beam. The automatically assigned direction can be reviewed and edited by the engineer as required. Automatic assignment works as follows:

    - Applies to both concrete and steel beams.
    - Beams that are skewed in plan w.r.t. the building directions are assigned a direction as follows: where the angle with Dir 1 is < = to 45 degrees, Dir 1is assigned, otherwise Dir 2 is assigned.
    - The default SFRS type assigned is SMF (again this can be reviewed and edited by the engineer as required).

- [TSD-6494] - Concrete Walls - some conservatism is removed from the calculation of minimum vertical reinforcement in walls designed to ACI 318 (all available years). For walls with a height to length ratio (h_w/l_w) ≤ 0.5, the program now considers the minimum allowed ratio of vertical reinforcement equal to that for horizontal reinforcement - as prescribed by the design code - rather than using a single equation in all cases where h_w/l_w < 2.5.
- [TSD-5444, 6650, 6612] - Steel Joists - an erroneous Fail status and message regarding “Non-uniform load...” could be issued when uplift occurred for combination service factors but not strength factors. This is corrected such that the uplift condition for strength and service combination factors are checked and reported separately. In addition, the load types are reported separately for both strength and service in all combinations.

Reports and Drawings

- Reports:
    - [TSD-5678] - Material List - Westok Cellular Beams - the gross weight of cellular sections (as reported in the Material List) is calculated as the average of the two Tee sections to give the same value as that in the Westok Cellbeam application.
    - [TSD-6115] - Analysis Diagrams - the labeling of diagrams added to reports has been enhanced to include the applied model filter.
    - [TSD-6365] - Analysis Result Reports - analysis result reports for envelopes which included pattern load combinations did not include the results for the pattern combinations. Note that this issue would only have applied where envelopes were created and for reports with these envelope filters applied. The issue did not affect the analysis or design procedures.
- Drawings - Concrete Beam Detail:
    - [TSD-6537] - Side bars - for some cases where a reinforced concrete (RC) beam was connected to another RC beam and the connection was collinear, the side bar anchorage in the detail drawing would be excessive. This is now improved.

- [TSD-6551] - Cross-sections - in some circumstances the cross-sections could overlap with the elevation. This is now fixed.

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

2 Get started with Tekla Structural Designer

Tekla Structural Designer is an integrated model-based 3D tool for analysis and design (of both concrete and steel members) in multi-material structures.

Features include interactive modeling, automated structural analysis and design, drawing, and report creation.

Multiple design codes are supported:

- ACI/AISC
- Eurocodes
- British Standards
- Indian Standards
- Australian Standards

2.1 Philosophy

The aim is to allow you to rapidly build your model, apply loads, and design the model for an appropriate set of design forces.

On a day to day basis, you do not need to be involved with the underlying analysis models to achieve this. Instead, you can focus on the design results.

To make this possible, Tekla Structural Designer automatically creates and analyzes multiple solver models, each one being based on a different but widely accepted approach.

By designing for the forces from all the solver models, you can be confident that each scenario has been catered for.

2.2 Tekla Structural Designer way of working

Tekla Structural Designer differs slightly from traditional modeling, analysis and design process.

Traditional modeling, analysis and design process

The traditional modeling, analysis and design process can be summarized in the following phases:

1. Provide a way to input or describe the model.
2. Analyze the model.
3. Design the model.
4. Produce calculations.
5. Produce drawings.

Process in Tekla Structural Designer

In Tekla Structural Designer, the analysis and design phases are merged into a single process. As a result, the workflow is as follows:

1. Input the geometry and loads

   A key requirement these days is BIM integration, or the ability to be able to transfer data from one application to another. Tekla Structural Designer has tools to automatically import model data from Neutral Files and from 3D DXF to facilitate BIM integration.

   Of course, you can also build the model directly.

   Once the physical model has been created, the next step is loading it. Tekla Structural Designer allows you to apply a wide range of loads in a flexible system of load cases. The system contains, among other things, a wind load generator available to automatically create wind load cases. You can also generate load combinations automatically.

   You should also consider pattern loading. You can create patterned beam loads automatically, and create patterned slab loads manually for design of slabs.

2. Analyze and design the model

   Tekla Structural Designer automatically performs the analyses required to enable member design to proceed. This means that analysis and design are combined into a single automated process. The only exception to this rule is slab design.

3. Produce reports

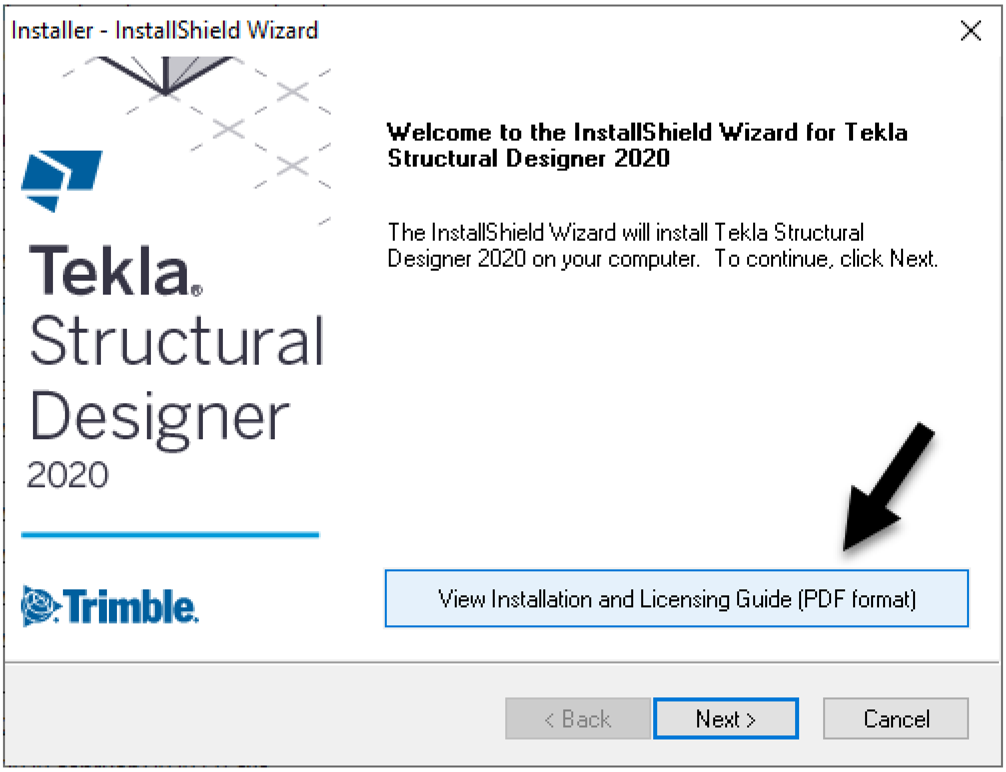
   You can create a wide range of calculations. You can also tailor the calculations extensively to meet your requirements.

4. Produce drawings

   You can produce beam and column detail drawings, and member schedules.

2.3 Install and license Tekla Structural Designer

When you install Tekla Structural Designer, you will be asked to select your license method from a range of options. These are fully explained in the "Tekla Engineering Software - Installation and Licensing Guide 2020" which can be viewed during the installation process by clicking the button shown below.



To install Tekla Structural Designer:

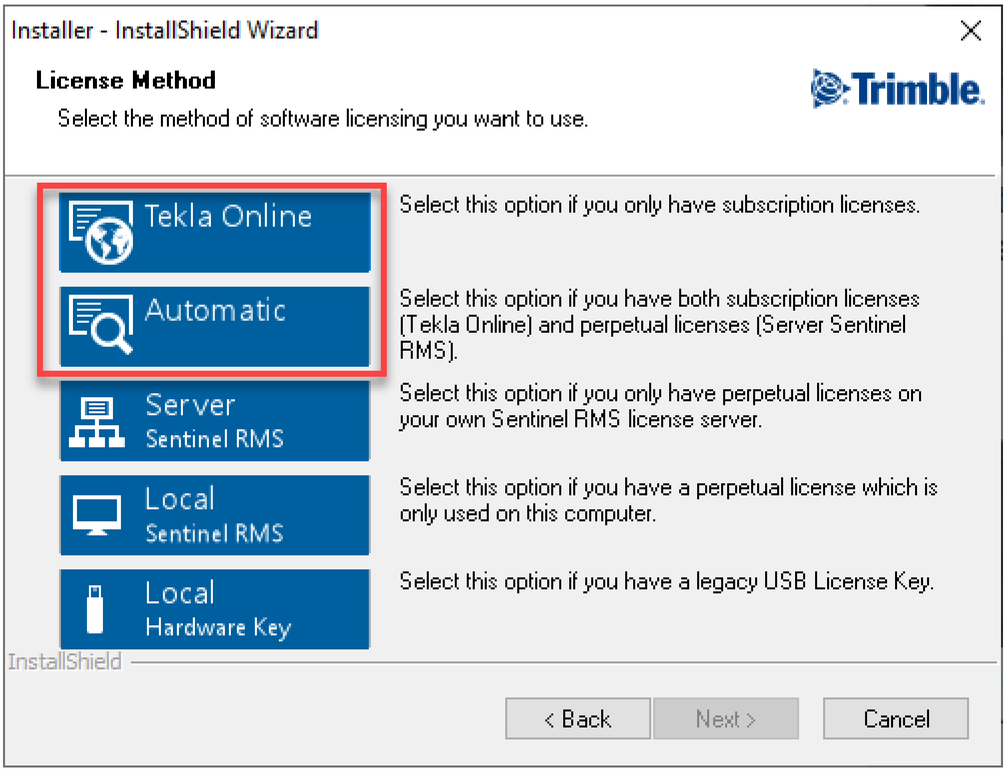
1. Download the installation file from Tekla Downloads to your computer.
2. Double-click the installation file to run the installation.
3. Follow the steps in the installation wizard to complete the installation.

If you are upgrading an existing Tekla Structural Designer version, you will also need to upgrade your local databases, see: Install the upgrade (page 91).

Licensing & installation information specific to Tekla Structural Designer 2020

- **Upgrade Licenses** - Tekla Structural Designer 2020 will require the activation of a new license. You should already be in possession of your Product Activation Key (PAK) as these are usually distributed prior to the software release. Please contact your local Service Department now if you do not have your PAK. To minimise any down time we advise that your PAK is activated BEFORE installing Tekla Structural Designer 2020.

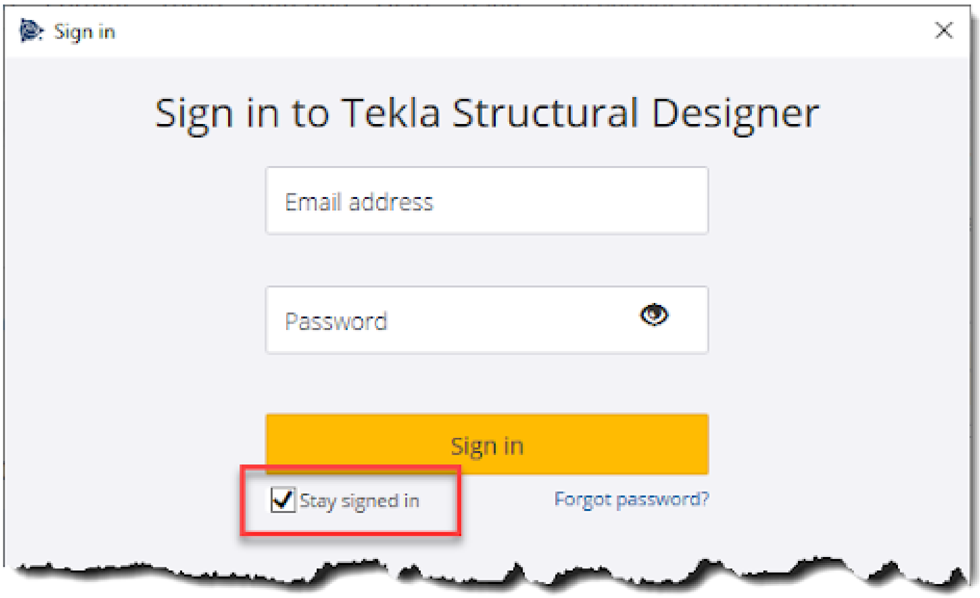
- **License Server Version** - the Tekla Structural License Service **must** be updated to the new version 3.00.0001 (incorporating Sentinel RMS 9.5) to be compatible with this release. Licensing will not function correctly if this update is not performed! Please see this article for more information about this, and see System Requirements for specific version details.
- **Subscription Licenses & Tekla Online Licensing** - if you have just purchased your license(s) they will use the new **Tekla Online** License Method. Select this option when installing the program as shown in the picture below.

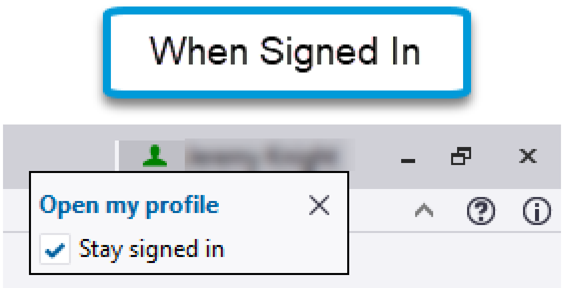


Please note the following:

- In order to use a **Tekla Online license** you will need:
  - A verified **Trimble Identity Account**. For more on this see the TUA Article Trimble Identity for Tekla Online services
  - To be a member of a **Tekla Online Organization**. For more on this see the TUA Articles Managing your Tekla Online services Organization and How do I join a Tekla Online Service Organization?
  - To have an Online License assigned to you by your Organization's Administrator.

- To be **Signed In** in the program. When the Tekla Online license method is set, you will be prompted to sign into the program when you start it as shown in the picture below - ensure you check on the “Stay signed in” option.





Installation - no previous installation of Structural Designer is required. To aid with transition, this release will install alongside existing versions and does not overwrite them.

- **Integration**
  - **Tekla Structures** - improvements to integration with Teka Structures requires version 2020 of Tekla Structures. See the Tekla Structures 2020 Release Notes > Other interoperability improvements for more information.
  - **Tekla Portal Frame and Connection Designer 20** - if you wish to integrate Tekla Structural Designer 2020 with Tekla Portal Frame Designer and/or Tekla Connection Designer you must install Tekla Portal Frame Designer 20 and/or Tekla Connection Designer 20 available from Tekla Download Service (note that these too will also need a new license).

Further help and update information

Tekla User Assistance

The Tekla User Assistance services are 24/7 online support channels for fast self-service, where you can find all the product guides, additional knowledge-base articles and instructional videos.

Tekla Discussion Forum

The Tekla Discussion Forum is the place to meet other users and discuss topics related to Tekla products. You can ask questions, contribute to the community by sharing your knowledge or get answers from support personnel.

Helpdesk

The Tekla Helpdesks support your daily operations so that our systems function as expected and any problems are solved as quickly as possible.

Software Update Service

The Software Update Service allows you to get the latest improvements to Tekla Structural Designer as soon as they are available. Providing you have an active internet connection you will be notified by Tekla whenever a new update is available.

The Software Update Service is enabled via Settings> General> Update Service on the Home ribbon.

You can also check for updates at any time via Check for Updates on the Home ribbon.

Previous versions

You can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

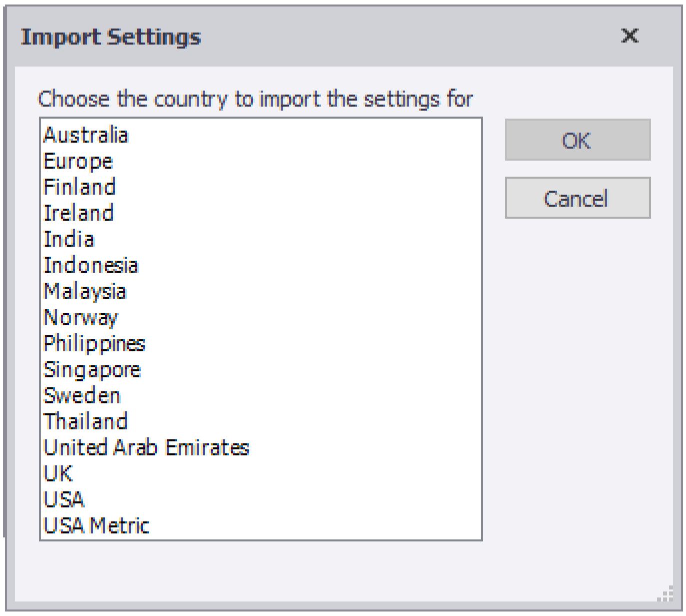
2.4 Start Tekla Structural Designer

When you start Tekla Structural Designer for the first time, you are asked to choose a country or region. Based on your selection various region specific settings such as design codes, units and default section sizes are configured. These settings, which can be changed to meet your company requirements, are applied when you start each new project.

Choose the country for your settings

1. Start Tekla Structural Designer by selecting it from the Windows Start menu or by double-clicking the desktop icon.

   A dialog box where you choose your settings appears.

2. Select a country that fits the region where your project is done.
3. Click **OK** The Tekla Structural Designer interface appears, with settings configured for the selected country.

Check or change your settings

You can check or change the settings that get applied to new projects at any time.

1. On the **Home** ribbon, click **Settings** The Settings dialog opens, displaying the settings that have been configured for the selected settings set.
2. Change the settings as needed.

NOTE Any changes that you make only apply to *new* projects, *existing* projects are unaffected. For more information about changing settings, see: Manage settings sets (page 1000).

Start a new project

1. On the **Home** toolbar, click **New**

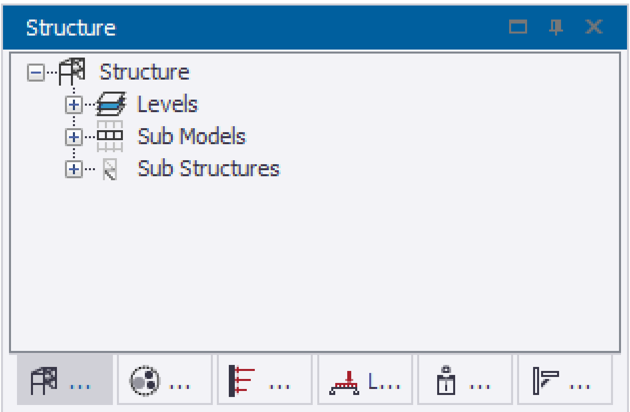
   NOTE Once familiar with the program, some users might prefer to set up a template (page 246) to use as the basis for new projects - particularly when there is a lot of commonality between projects.

   Your new project opens and you will see:

   - Two scene views (page 250): **Structure 3D** and **St. Base (Base) 2D**
   - A tabbed (page 251)
   - A (page 252)

2. Click the **Structure** tab under the **Project Workspace**

   A **Structure** tree is displayed in the **Project Workspace**.



3. In the **Structure** tree, click **Structure**

   The **Properties** window displays the **Structure** properties.

4. In the **Structure** tree, click **Levels**

   The **Properties** window changes to display **Levels** properties - at this point there is only only level in the model.

   NOTE The units in which the levels are displayed, (along with other settings), have been copied into the model from the settings set that was active when the project was started. These are known as 'model settings'. Model settings apply to the current project only and can be edited to suit the model as required.

What to do next:

- Check or change model settings in your project (see below).
- Try setting out some grid lines (page 371) in the **St. Base (Base) 2D** view.
- Next, double-click **Levels** in the **Structure** tree to create some levels (page 367).
- With grids in place and levels established, you can then open views of the other levels (page 276).
- You are now ready to begin creating the model (page 366).

Check or change model settings in your project

You can check or change the model settings in the current project at any time.

1. On the **Home** ribbon, click **Model Settings**

   The **Model Settings** dialog opens, displaying the model settings that have been copied in to your project from the active settings set.

2. Change the model settings as needed.

NOTE Any changes that you make only apply to the *current* project, *new* projects are unaffected. For more information about changing model settings, see: Apply and manage model settings (page 991).

Modify project details

The **Project Wiki** allows you to modify project details, view revision history, and record revisions.

Modify project details and view revision history

You set the initial project details as you create a new project. If you want to change the project details later, or view other project parameters, see the following instructions.

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.

   In the **Project Wiki** dialog box, you can:

- Manage the parameters that are included in the output reports on the **Project Summary** page.
- Track revision history on the **Revisions** page.
- View the time at which each revision was started and last saved on the **Sessions** page.
- View the changes associated with each revision on the **Changes** page.
- View statistics related to the model size on the **Metrics** page and its sub-pages.

2. On the **Project Summary** page, modify the project details according to your needs.
3. Click **OK**.

Record revisions

1. Save the project before recording revisions.
2. In the file menu, click **Start Revision**.

   The **Start Revision** dialog box opens.
3. Type the revision ID and add additional notes related to the revision according to your needs.
4. In order to keep a record of the changes made in the revision, select the **Track Changes** option.
5. Click **OK**.
6. Continue to develop the model and save the model under a new name.
7. Repeat steps 2–5 as required.

Apply revision ID as an attribute for each modified element

1. On the **Home** tab, click **Project Wiki**.
2. Click **Changes**.
3. Select an appropriate UDA from the Revision Attribute droplist.
4. Either:.
   a. Click **Apply automatically** in order for the revision ID to be automatically recorded against the selected UDA on each change, or
   b. Click **Apply** to manually record the revision ID against the selected UDA for the currently elements listed in the dialog.
5. Click **OK**.

Use templates in new projects

If you are creating projects that share a common start point, you can use templates to avoid having to repeat inputting information.

In order to use a template, you must first create one.

A template can contain as much or as little information on the model as you consider applicable to serve as the start point for subsequent models.

For example, you can create a template that only contains a simple grid and the height of the first floor construction level.

See also

Create a new template (page 246)

Create a new project based on a template (page 246)

Create a new template

You can create a new template to simplify creating new models that have the same starting point. For more information, see the instructions below.

1. Start a new project (page 243)
2. Create the model data that you want to include in the template.
3. On the **Home** tab, click **Save As**.

   The **Save As** dialog opens.
4. Change **Save as type** to **Tekla Structural Designer template file (*.tsmdt)**.
5. Name the template and ensure that it is saved in the right location.
6. Click **Save**.

See also

Create a new project based on a template (page 246)

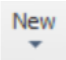
Create a new project based on a template

You can use an existing template as the base of your project. For more information, see the following paragraphs.

NOTE Before creating a new project, ensure that an appropriate settings set is active.

For more information, see Use settings sets.

---

1. On the **Home** tab, click 

.

   A list opens.
2. In the list, select the required template.

   The template opens.
3. On the **Home** tab, click  **Save As**.

   The **Save As** dialog opens.
4. Change the **Save as type** to **Tekla Structural Designer project files (*.tsmd;*.cscbd)**.
5. Name the file and ensure that it is saved in the right location.
6. Click **Save**.
7. Add the necessary building objects and loads in order to complete the project.

See also

Create a new template (page 246)

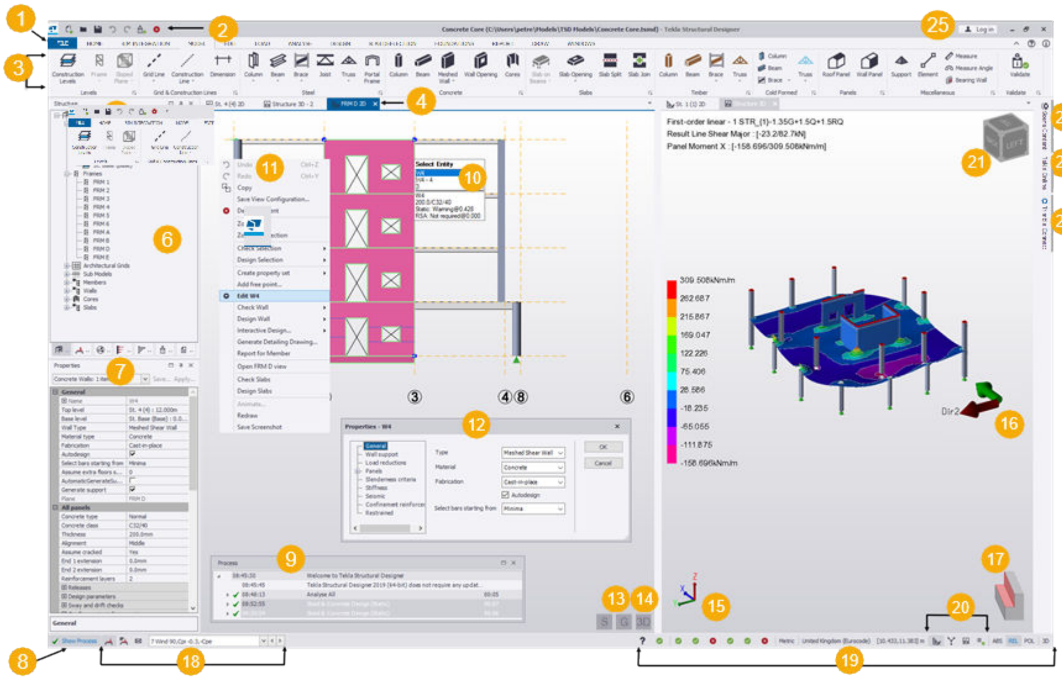
2.5 Get familiar with the user interface

Before creating a new project, we recommend you take a moment to familiarize yourself with the user interface as explained in the following topics:

- Interface components (page 247)
- How to use the project workspace (page 261)
- How to manage scene views, view modes and scene content (page 275)
- How to hide, re-display and move windows (page 289)
- Keyboard shortcuts (page 291)

Interface components

The main components of the Tekla Structural Designer user interface are labeled in the image below:



1. File menu

The **File** menu contains those commands that you can use to perform file-related operations.

Button	Description
 **New**	Creates a new blank project from scratch.
 **Open**	Opens an existing project.
 **Save**	Saves the currently open project.
 **Save As**	Saves the currently open project with a new name, or as a template.

Button	Description
Save Model Only	Saves smaller model-only files without the analysis results. The model-only files can easily be shared amongst the project team.
Close	Closes the currently open project.
Print	Prints the currently open project.
Start Revision	Records changes to this revision of the project.
Send As Email	Creates a new email with the project attached.
Contact Support	Automatically uploads your model file and sends this, together with your issue description and key information (needed by Support to assist you) about your installation and system, to your Local Support Team as defined in your Tekla Online Profile.
Exit	Prompts you to save any open project, and then closes Tekla Structural Designer.

2. Quick access toolbar

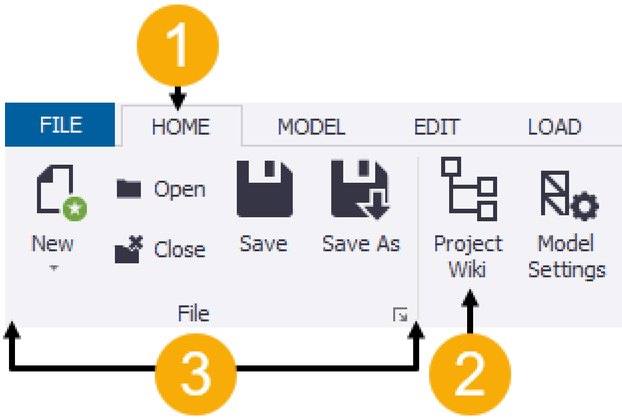
The **Quick Access** toolbar displays commonly used commands, listed below:

- **New**
- **Open**
- **Save**

- **Undo**
- **Redo**
- (page 2187)
- (page 2201)
- **Delete**

3. Ribbon

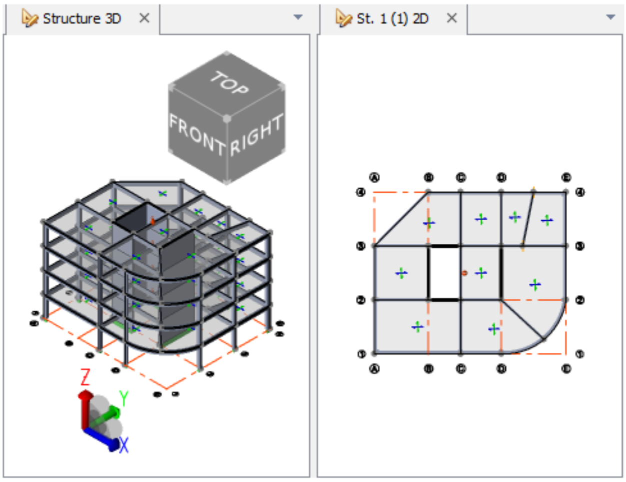
The ribbon consists of a number of toolbars, placed on various tabs. Each toolbar contains related commands organized into logical groups.



1. Toolbar tab
2. Command
3. Group

4. Scene views

2D views, 3D views and solver views, of the model, sub models, frames, construction levels and individual members can be displayed in tabbed windows.



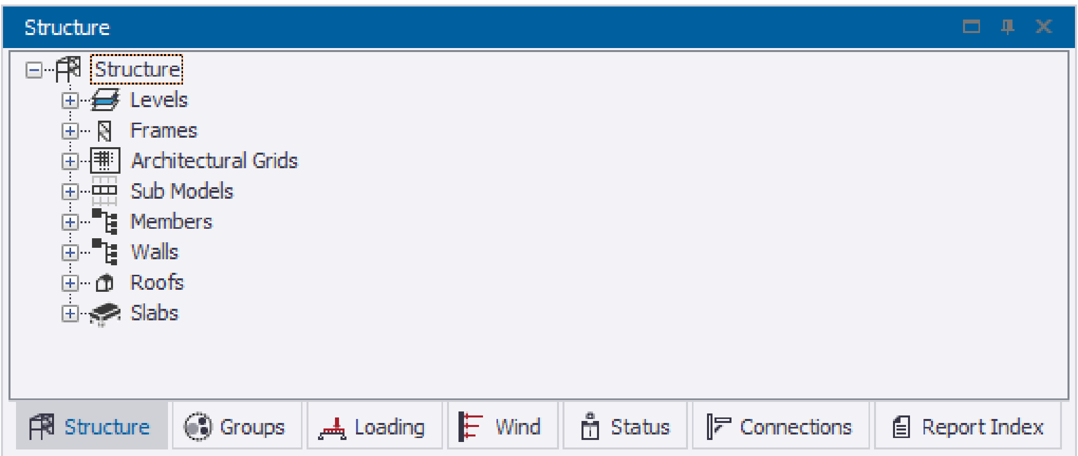
For more information, see: How to manage scene views, view regimes and scene content (page 275)

5. Structure tree

The **Structure** tree refers to the hierarchy of information displayed on the Structure tab in the **Project Workspace** . Similar 'trees' are displayed for the other tabs also.

6. Project Workspace

The **Project Workspace** is a central location of organizing the entire model into a hierarchical structure.

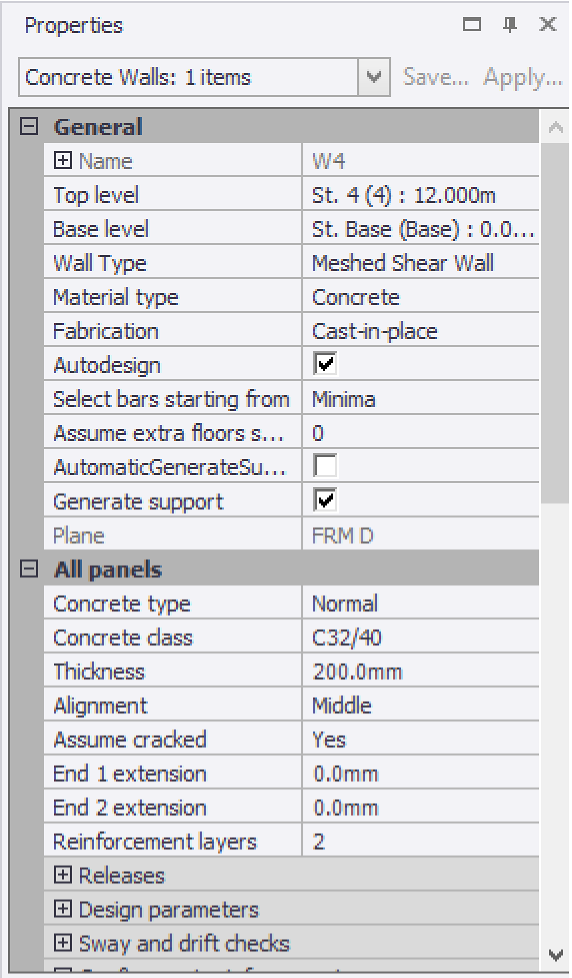


It provides access to a range of functions and is a key tool in creating and controlling your model.

For more information, see: How to use the project workspace (page 261)

7. Properties window

The **Properties** window facilitates the input, review and editing of model properties.



The **Properties** window is used to:

- input data when a command is run from a toolbar.
- review or edit existing properties when individual or multiple items are selected in the active scene view.
- review or edit properties when a branch of the **Properties** window is selected.

The properties displayed vary according to the selection. You can edit all of the properties, unless they are greyed out.

A key feature of the **Properties** window is that it enables the editing of multiple items at the same time. Existing properties of selected items are only displayed when if are identical for all selected items. If the properties differ, the property field is left empty.

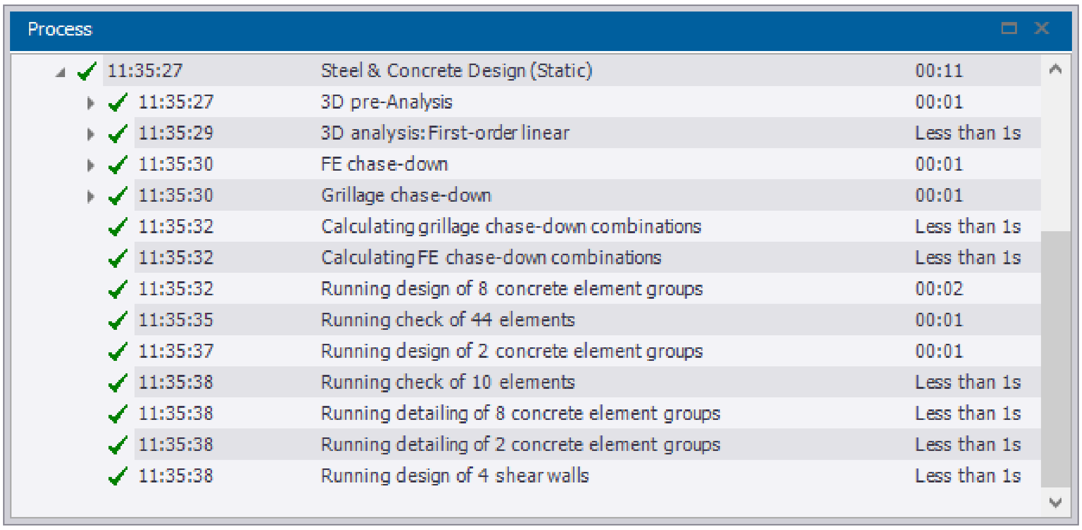
By default, the **Properties** window is docked at the bottom left of the main window, but you can reposition it if required.

8. Show Process button

Click to display the **Process Window**, described below.

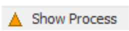
9. Process Window

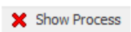
Initially, the **Process Window** is minimized. It can be displayed by clicking the **Show Process** button, located at the left end of the **Status bar**.



When you analyze or design the model, each step of the process is logged and displayed in the window.

When the window is minimized in the **Status bar**, any detected warning and errors are flagged. See the flags below:

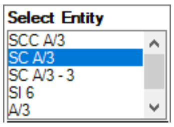
- 

- 

Such warnings and errors should always be fully investigated, as they may have an adverse affect the design.

10. Select Entity tooltip

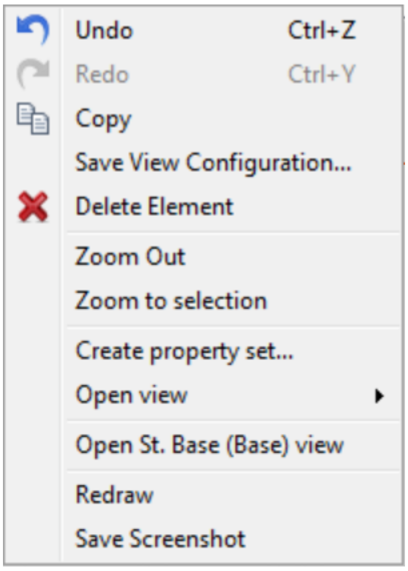
The program is automatically in 'select mode' when no other commands are being performed. In this mode you can hover the cursor over an entity and its name will be displayed in the **Select Entity** tooltip.



When the correct entity is displayed, click the entity to select it. If several entities are displayed, you can select one by using the Tab key or Up/Down arrow keys.

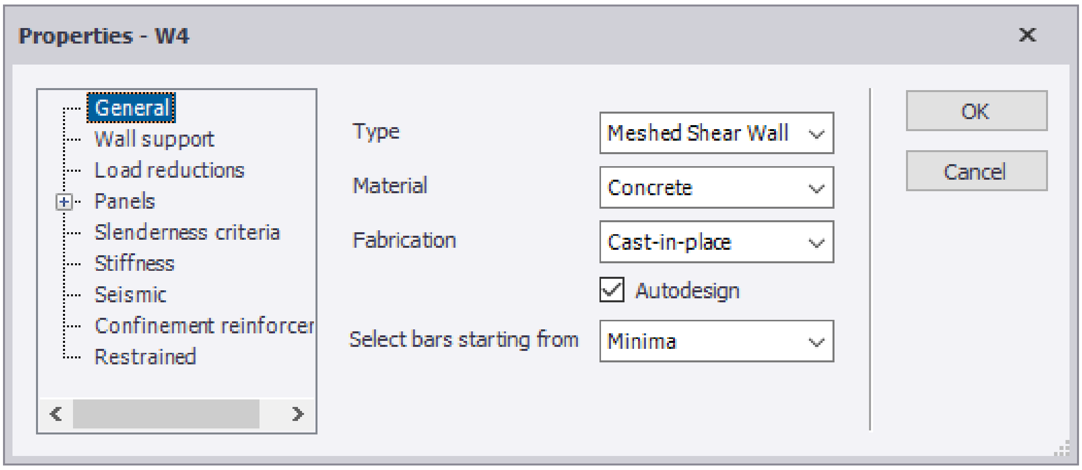
11. Context menu

You can right-click anywhere within a 2D or 3D view to display a menu that is context-sensitive to the item that is currently highlighted.



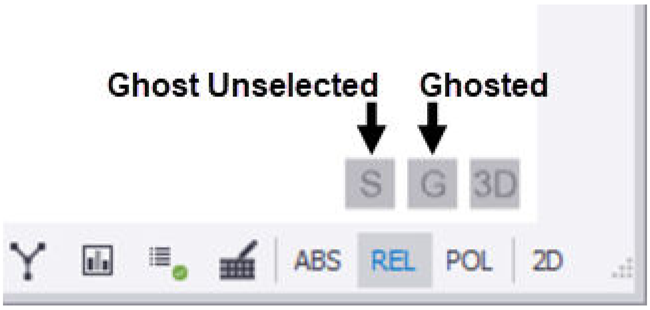
12. Properties dialog

You can use the **Properties** dialog box for viewing and editing parameters associated with an individual model object.



The **Properties** dialog box is displayed by right-clicking an object in the graphical display and selecting the **Edit** option from the context menu that appears.

13. Ghost Unselected and Ghosted toggle buttons



- **Ghost Unselected** button is displayed in all 2D and 3D Views. It is used to toggle the display of selected and unselected objects, making it easier to focus on a particular subset of objects within the model.

  See: Use Ghost Unselected to focus on the selection (page 357)

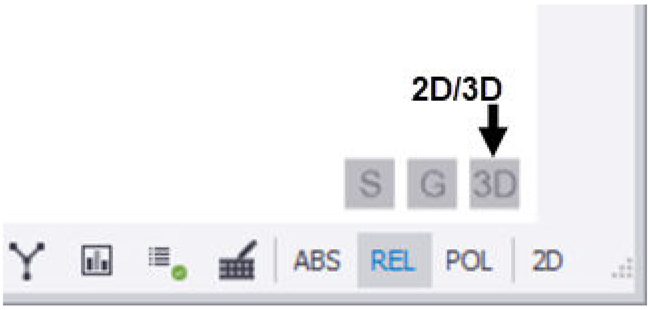
- **Ghosted** button is displayed in Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button). It is used to toggle the display of a ghosted view allowing you to see the current view in the context of the whole structure.

  See: Use Ghosted to see the view in the context of the whole model (page 1034)

Related video

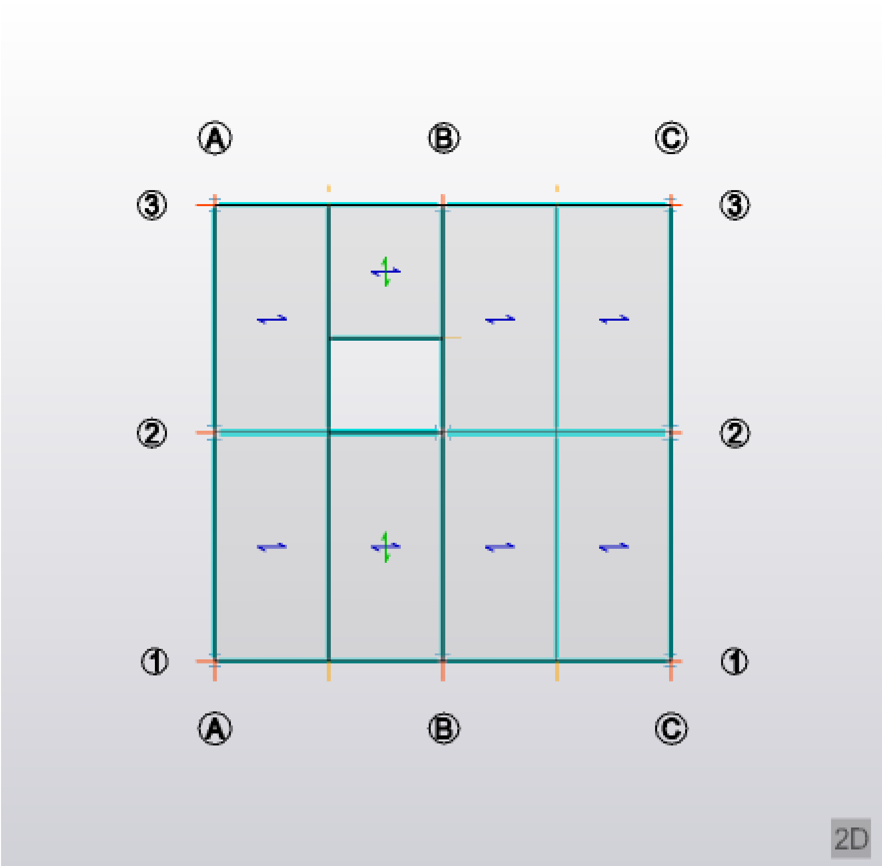
[Ghosted Structure view](#)

14. 2D/3D toggle button

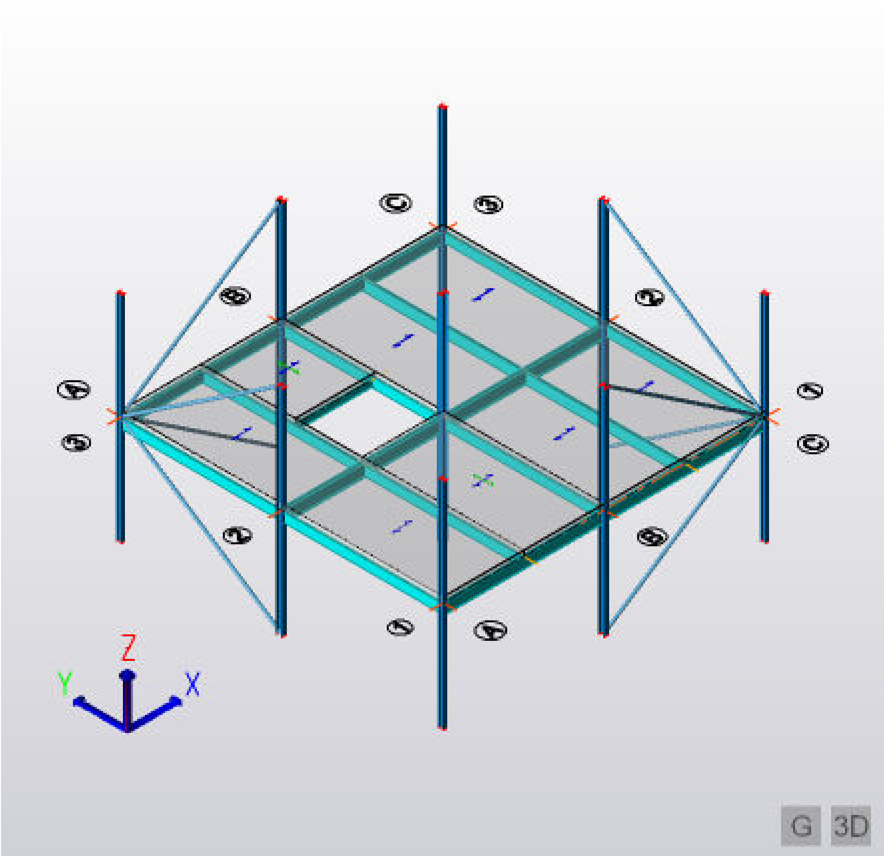


2D/3D button is displayed in 2D Scene Views only. It is used to toggle the view between 2D and a 3D isometric projection.

Example of a level view in 2D:



The same level view when toggled to 3D:



15. Global XYZ axes

The global XYZ axis system within which all other systems exist.

16. Building directions

The principle axes of the building can be rotated about global Z if required.

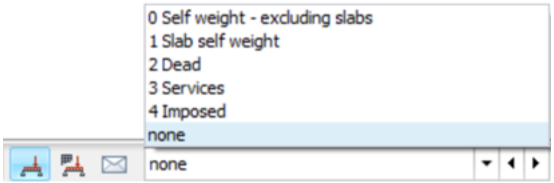
The building direction axes can be switched on/off (and their labelling contriolled) in Structure Properties (page 2067).

17. Cutting planes

The **Cutting Planes** symbol is displayed in the bottom right corner of the scene view when any of the cutting planes are currently active.

18. Loading List

The **Loading** list is permanently docked at the bottom edge of the main window.



The **Loading** list has two main functions:

- Selecting a specific load case to add loads into
- Selecting a specific load case, combination, or envelope to view the results for.

When viewing results, first click the **Loadcase**, **Combination**, or **Envelope** button according to your needs. The, choose the specific load case, combination, or envelope name in the list.

19. Status bar

The **Status bar** is permanently docked at the bottom edge of the main window. The **Status bar** performs a number of different functions.

The **Status bar** provides feedback by indicating:

1. The analysis and design validity
2. The units system
3. The design code
4. The coordinates of the cursor relative to the global origin in 2D views
5. The view regime buttons that allow you to switch the view regime applied to the active scene view, see **View Regime buttons** below.
6. Coordinate display that allows you to set the display method for coordinate tooltips:
    - **ABS** (Absolute)
    - **REL** (Relative)
    - **POL** (Polar)
7. **2D**/**3D** toggle button
    - **2D**: the content of the 2D view is displayed in plan.

- **3D**: the content of the 2D view is displayed in isometric.

20. View regime buttons

The view regime buttons allow you to switch the view regime applied to the active scene view:

-  **Structural View** shows the geometry and loading of the structure.
-  **Solver View** shows the analysis model.
-  **Results View** shows the analysis results.
-  **Wind View** shows the wind model.
- **Review View** graphically examines the model properties or status.
-  **Slab Deflections View** shows the slab delection analysis results.

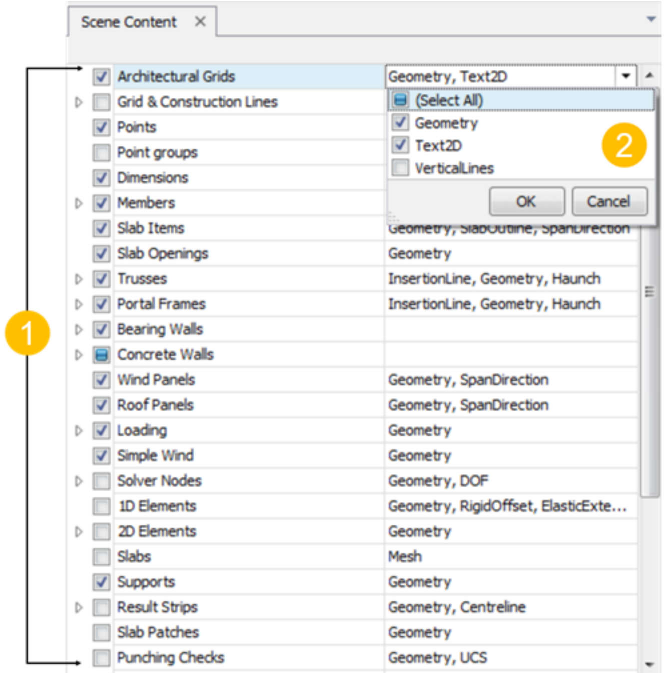
21. ViewCube

In 3D views, you can click a vertex, edge, or face of the **ViewCube** to rotate the model to a preset view.

For more information, see: Zoom, pan, rotate and walk through scene views (page 348)

22. Scene Content

The **Scene Content** sidebar is used to control the displayed content in the 2D and 3D Scene Views.



The sidebar contains:

1. Entity categories (with check boxes)

   Some categories have arrow symbols to their side, indicating sub categories. Click the arrow symbol in order to see the sub categories.

   The check box controls whether the entity category and its associated information is displayed. You can check the entities that you want to display and clear the ones that you do not.

2. Entity information controls

   Entity information controls list the information in each category that will be displayed. When clicked, they expand to lists that allow you to select the information you want to display in the model view.

For more information, see: Manage scene content information (page 280)

23. Tekla Online side pane

The Tekla Online side pane is used to access Tekla Online services provided for users of Tekla Structural Designer, for example Tekla User Assistance and Tekla Discussion Forum.

24. Trimble Connect side pane

The Trimble Connect side pane is used to access the the Trimble Connect project collaboration tool.

For more information, see: Working collaboratively with Trimble Connect (page 310)

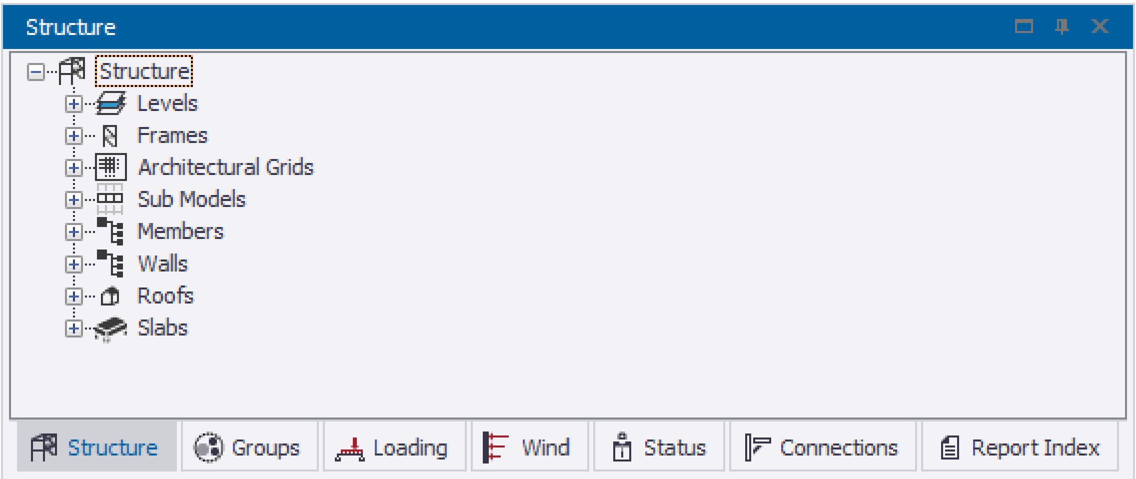
25. Sign in

Signing in to your Trimble Identity gives you access to a greater number of online resourses.

You can sign in to one service and then browse to another online service without a need to log in again. Find the services landing page here: http://www.tekla.com/services

How to use the project workspace

The **Project Workspace** is the central control area for your model.



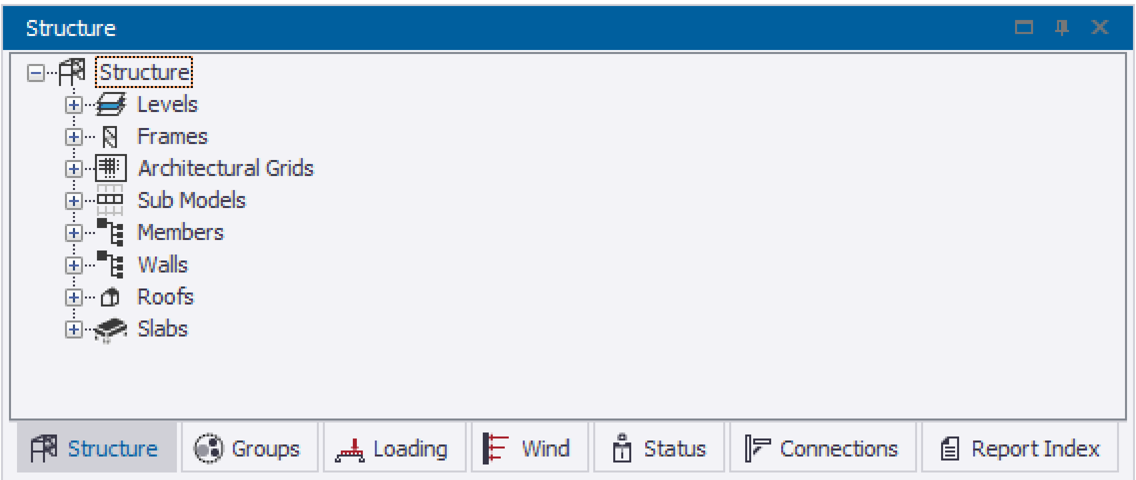
Its tabs are used to access a range of functions:

Icon	Tab	Content
	Structure	Structure (page 262) tree: for opening scene views, viewing and modifying model properties, controlling grids and sub models.
	Groups	Groups (page 267) tree: for organizing members into design and detailing groups and for displaying UDAs.
	Loading	Loading (page 270) tree: for viewing load status.

Icon	Tab	Content
	Wind Model	Wind Model (page 271) tree: for viewing and modifying wind model properties
	Status	Validation (page 272) tree: for highlighting validation issues and other aspects of the model status, i.e. • Model Geometry • Wind Model • Meshing • Decomposition • Solver • BIM • Design
	Connections	Connections (page 273) tree: for viewing and designing steel connections
	Report Index	Report Index (page 955): contains bookmarks that can be used to move around the report when a report view is displayed.

View and modify model properties in the Project Workspace

The **Structure** tree organizes the model geometry in a hierarchical way. It also allows you to view and modify model properties.



When you open a new model, the **Structure** tree contains two sub-branches: **Levels** and **Sub Models**. As the model geometry develops new branches are added, as shown above.

When a branch or sub-branch is selected, the common properties of the selected branch are viewed in the **Properties** window. You can then modify the common properties in the **Properties** window.

View and modify structure properties

Structure properties, or building parameters, control the principal direction and default meshing properties of the building.

1. In the **Structure** tree, click **Structure**.

   The structure properties are viewed in the **Properties** window.
2. Modify the structure properties according to your needs.

View and modify construction level properties

- In the **Structure** tree, do one of the following:

To	Do this
View and modify the properties common to all construction levels	1. Click **Levels**. The construction level properties common to all levels are viewed in the **Properties** window. 2. Modify the properties according to your needs.
View and modify the parameters of a single construction level	1. Click the **+** sign next to **Levels**. All construction levels in the model are viewed. 2. Click the desired construction level. The level properties are viewed in the **Properties** window. 3. Modify the level parameters according to your needs.

View and modify frame properties

The **Frames** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Frames**.

   All frames in your model are viewed.

2. Click the desired frame.

   The frame properties are viewed in the **Properties** window.

3. Modify the frame properties according to your needs.

View and modify the slope properties

The **Slopes** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Slopes**.

   All slopes in your model are viewed.

2. Click the desired slope.

   The slope properties are viewed in the **Properties** window.

3. Modify the slope properties according to your needs.

View and modify grid properties

- In the **Structure** tree, do one of the following:

To	Do this
Modify the color, name, or visibility of a grid	1. Click the **+** sign next to **Architectural Grids**. All architectural grids in your model are viewed. 2. Click the desired grid. The grid properties are viewed in the **Properties** window. 3. Modify the color, name and visibility of the grid according to your needs.
Renumber all grids	1. Right-click **Architectural Grids**. 2. In the context menu, click **Renumber**.

View and modify sub model properties

1. In the **Structure** tree, click the **+** sign next to **Sub Models**.

   The sub models in your model are viewed.

2. Click the desired sub model.

   The sub model properties are viewed in the **Properties** window.

3. Modify the properties according to your needs.

TIP To open the **Sub Models** dialog dialog, double-click **Sub Models**.

View and modify sub structure properties

See: Manage sub structures (page 1031)

View and modify member properties

Tekla Structural Designer classifies members by material and type, and further by fabrication and by shape. Members are classified by material and type, then further classified by fabrication and then by shape.

- In the **Structure** tree, do one of the following:

To	Do this
View and modify common properties for members of a particular type and fabrication	1. Click the **+** sign next to **Members**. The existing member types are viewed. 2. Click the **+** sign next to a member type. The existing fabrication types are viewed. 3. Click a fabrication type. Common properties of all members of the fabrication type are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify common properties for members of a particular type, fabrication and shape	1. Expand the **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the desired fabrication type. The shapes of the fabrication type are viewed. 3. Click a shape. The common properties of all members of the selected shape are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify the properties of an individual member	1. Expand the **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the shape type. The member references of the shape are viewed.

	3. Click a member reference. The properties of the member are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View a member in a new window, select it in visible views, delete it, or modify it in the **Properties** dialog box	1. Expand the **Members** branch and its sub branches by clicking them. 2. Right-click a member reference. 3. In the context menu, select the desired option.

View and modify slab properties

- In the **Structure** tree, do one of the following:

To	Do this
View and modify properties common to all slabs	1. Click **Slabs**. The common properties to all slabs are viewed in the **Properties** window. 2. Modify the properties according to your needs.
View and modify the properties of a parent slab	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the desired parent slab. The properties of the parent slab are viewed in the **Properties** window. 3. Modify the properties according to your needs.
Modify the properties of a parent slab, or delete it	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Right-click a parent slab. 3. In the context menu, select if you want to modify the parent slab in the, or delete it.
View and modify the properties of a slab item	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the **+** sign next to a parent slab. All the slab items within the parent slab are viewed. 3. Click a slab item. The properties of the slab item are viewed in the **Properties** window.

	4. Modify the properties according to your needs.

View and modify wall or roof panel properties

1. In the **Structure** tree, click the **+** sign next to **Walls** or **Roofs**.

   The currently defined walls and roof panels are viewed.

2. Click the desired wall or roof panel.

   The properties of the selected wall or roof panel are viewed in the **Properties** window.

3. Modify the properties according to your needs.

View and modify result strip properties

1. In the **Structure** tree, click the **+** sign next to **Result Strips**.

   All the currently exiting result strips are viewed.

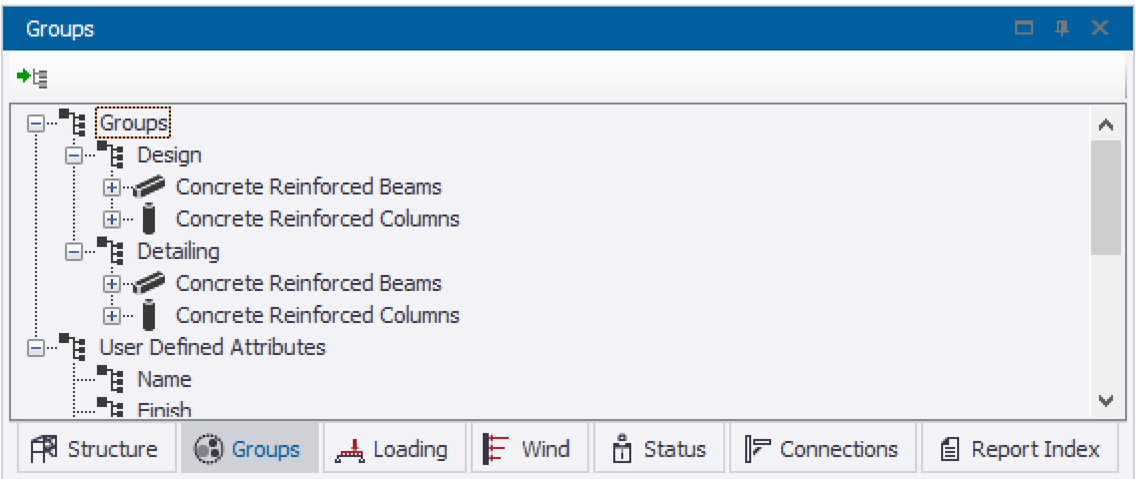
2. Click the desired result strip.

   The properties of the selected result strip are viewed in the **Properties** window.

3. Modify the properties according to your needs.

Manage groups in the Project Workspace

The **Groups** tree organizes the members in your model into design groups. It also allows you to manage design groups according to your needs.



If you have defined concrete members, they are also organized into further groups for detailing purposes. The application of grouping is most useful when handling concrete structures.

User defined attributes (UDAs) are also listed, allowing you to see which members have been assigned specific UDAs.

NOTE You can right-click on a group name (or UDA) in order to select all members in the group, (or with that UDA), simultaneously in the visible views.

Regroup a specific member type

1. In the **Groups** tree, right-click the member type you want to re-group.
2. In the context menu, select **Regroup Members**.

   The selected member type is regrouped.

Re-group all member types

WARNING Re-grouping all member types undoes any manual grouping you may have done.

- Click the **Re-group ALL Model Members** button on the top left corner of the **Groups** tree.

Add group

1. In the **Groups** tree, select the appropriate branch within which you want to manually create a new group.
2. In the context menu, select **Add Group**.

   A new group empty group is created.

Set as default group

When a new empty group is set as default, if you create a new member it will be placed in that group in preference to another empty group.

1. In the **Groups** tree, select an empty group.
2. In the context menu, select **Set As Default Group**.

   An asterix is placed next to the group name to indicate that it is the default group for new members.

Split a member group into smaller groups

1. In the **Groups** tree, right-click the name of the group that you want to split.

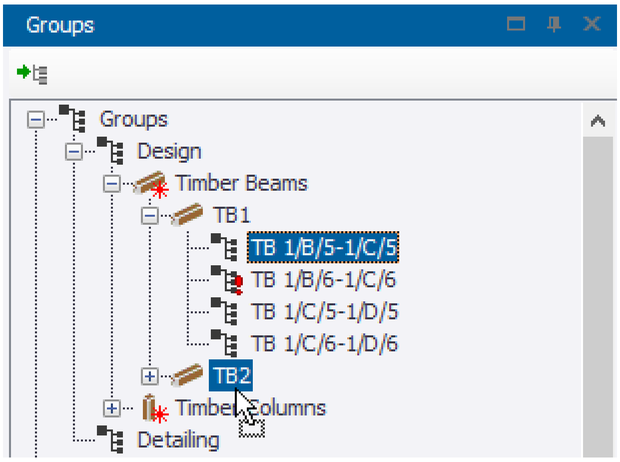
2. In the context menu, select **Split Group**.

   All members of the group are ungrouped and placed into individual groups.

Move members from one group to another

1. In the **Groups** tree, open the group that contains the member(s) to be moved.
2. Click the first member name to select it, then if you want to add further members to the selection, hold down **Ctrl** while clicking on their names.
3. Drag the selection over the group name where you want to move it.

   Provided that the selection meets the geometric criteria to belong to the group, a small rectangle will be displayed alongside the cursor.



NOTE You must drag over the **group name** as shown above, rather than over the group content for the rectangle to appear.

4. Release the mouse button.

TIP You can also change the group to which members belong by selecting them and then updating the group name in the **Properties** window.

Rename a member group

1. In the **Groups** tree, right-click the group that you want to rename.
2. In the context menu, select **Rename Group**.
3. Type the new name of the group and press Enter.

Modify the group name defaults for future projects

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**, and select the settings set that you want to edit.
3. Go to the **Grouping** page.
4. Review and modify the default group names according to your needs.
5. To save the changes, click **OK**.

Check a member group

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Check Group**.

   The selected member group is checked.

Check a member group using Tekla Tedds

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Check Group**.

   The selected member group is checked using Tekla Tedds.

Design a member group

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Design Group**.

   The selected member group is designed.

Remove a member group

1. In the **Groups** tree, right-click the name of the group you want to remove.
2. In the context menu, select **Remove Group**.

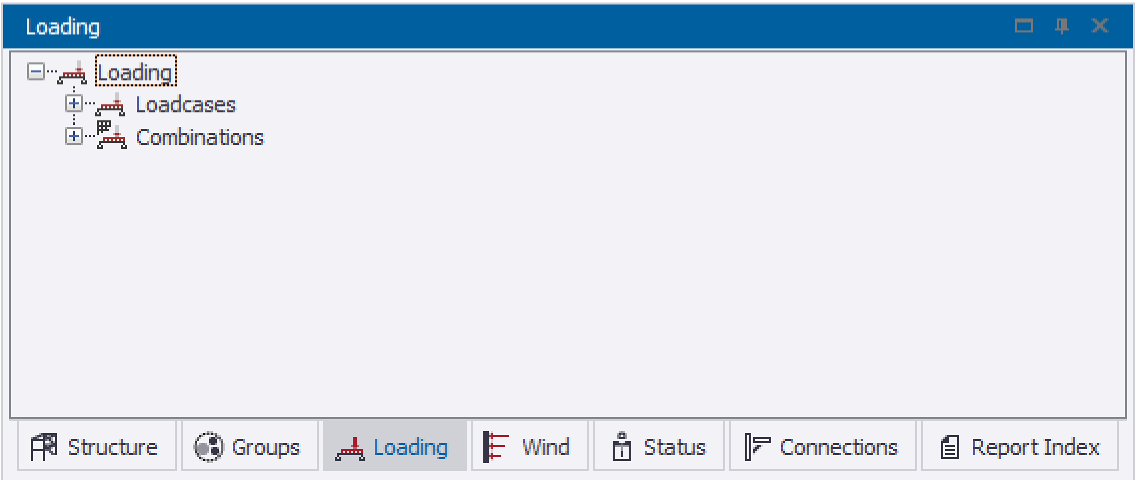
   All members are ungrouped and placed into individual groups.

See also

Create and manage user-defined attributes (page 1026)

View load status in the Project Workspace

The **Loading** tab lists the individual loads that have been applied in each load case. After analysis it sums the applied loads in each combination and checks these against the sum of reactions.



See the status options in the following table:

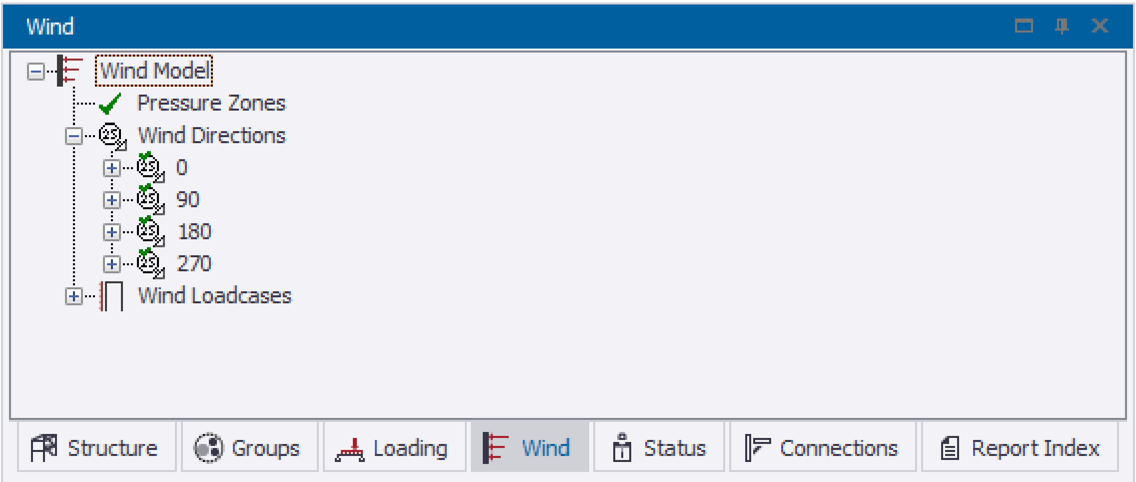
Icon	Meaning
✓	**Total Reaction** is in equilibrium with the **Total Load on Structure**.
✗	**Total Reaction** is not in equilibrium with the **Total Load on Structure**.
?	**Total Reaction** is unavailable.

1. In the **Project Workspace**, go to the **Loading** tab.
2. Click the desired load case.

   The properties of the load case are viewed in the **Properties** window.

View and modify wind properties in the Project Workspace

The **Wind Model** tree is used after completing the wind modeling process to display wind direction views, modify wind directions, and modify wind load cases.



View a wind direction view

1. In the **Wind Model** tree, click **Wind Directions**.
2. Right-click the desired wind direction.
3. In the context menu, select **Open View**.

Modify wind direction properties

1. In the **Wind Model** tree, click **Wind Directions**.
2. Click the desired wind direction.

   The properties of the wind direction are viewed in the **Properties** window.
3. Modify the properties according to your needs.

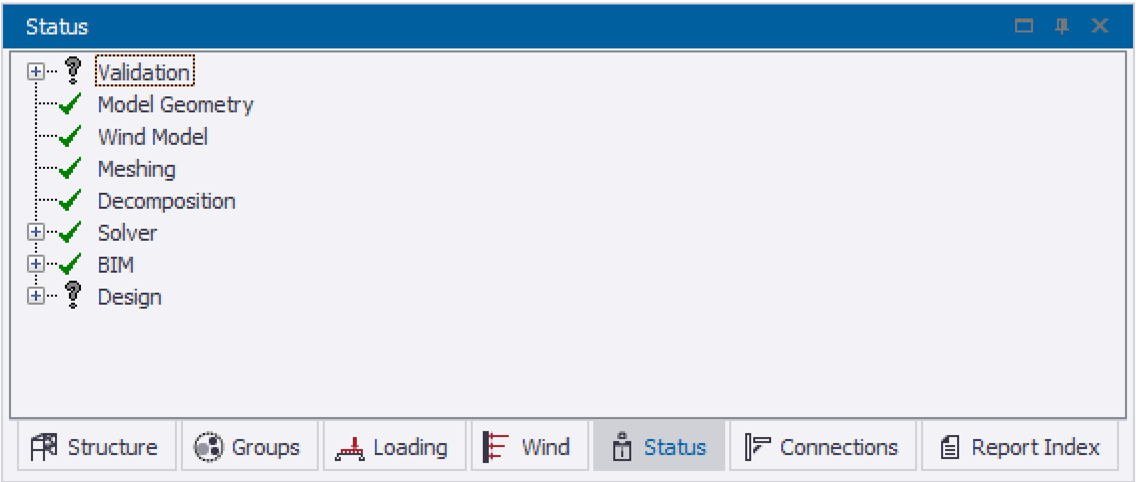
View and modify wind loadcases

1. In the **Wind Model** tree, click **Wind Loadcases**.
2. Click the desired wind loadcase.

   The properties of the wind direction are viewed in the **Properties** window.
3. Modify the properties according to your needs.

View validation status in the Project Workspace

The **Status** tree is used to review the validation messages and other model status indicators.



You can review:

- The validation status for the model

  For more information, see Check the model (page 512)

- The validation status for the analysis

  The analysis model validation is performed automatically when you analyze the model.

- The decomposition status

  The **Decomposition** branch shows if load decomposition has been successfully completed.

- The solver status

  The **Solver** branch lists the analyses that have been performed.

- The BIM validation

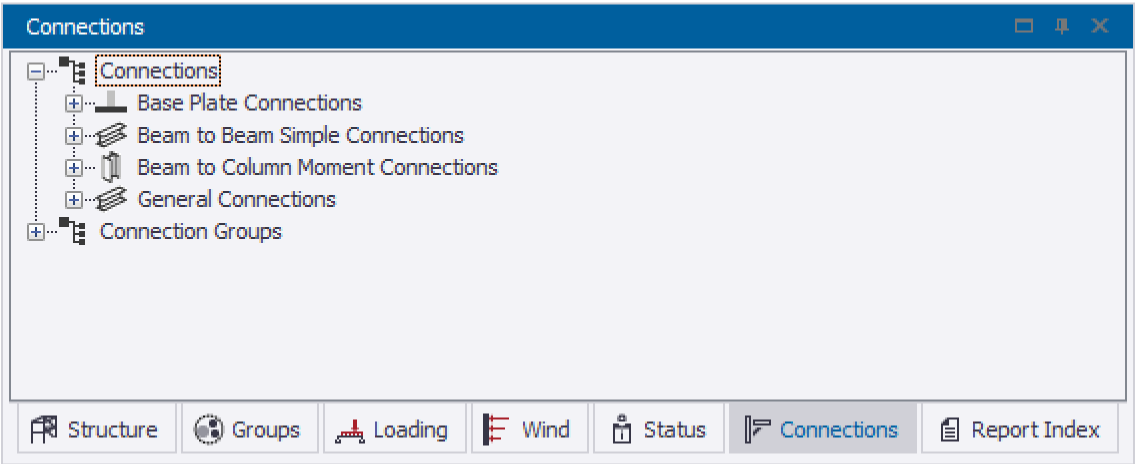
  If the model has been imported or exported, the **BIM** branch lists any warnings and errors that relate to the import or export process.

NOTE

- ✓ indicates that the analysis or process has been successfully completed.
- ✖ indicates that the analysis or process has resulted in an error.
- ⚠ indicates that the analysis or process has resulted in a warning.

Manage and design connections in the Project Workspace

The **Connections** tree organizes the model's steel connections in a hierarchical way, by Type (i.e. Type/Reference), and by Group (i.e. Type/Section size/Reference).



When opened for the first time, or if the model has been edited in a way that affects the connection configurations within it, you must run **Update Connections** in order to determine the valid connections.

This applies a set of rules (page 822) to create and organize connections into the following Types:

- Base plate connections
- Beam to Beam Moment Connections
- Beam to Beam Simple Connections
- Beam to Column Moment Connections
- General Connections

Valid connections are displayed in the scene views and are also listed in the **Connections** tree

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 822) to determine all valid steel connections in the model. The resulting connections are listed in the

Connections Tree and are also shown by bounding boxes in the Scene Views.

NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Edit connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Edit...** from the context menu. A dialog opens allowing the connection to be edited.

Select connection in visible views

- In the **Connections** tree, right click on the appropriate connection reference and choose **Select in visible views** from the context menu. The connection is selected if it exists in the active scene view.

Design connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

Export connection to Tekla Connection Designer

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

See also

Check steel connections (page 809)

How to manage scene views, view regimes and scene content

Scene views

A scene view displays the model or a part of it in a tabbed window. It might display: the entire structure, a sub structure, or an individual member in 3D; a specific level, frame, or plane in 2D. A separate load analysis view is also available for displaying force and moment diagrams for individual members.

Click the following links to learn how to manage and manipulate scene views:

- Open, close and save scene views (page 276)
- Create and modify scene view tab groups (page 279)

View regimes

For each scene view, you should select a view regime from the **Status bar** appropriate to the task being performed:

Icon	View regime	Used for
	Structural View	Creating the structure geometry and applying loads
	Solver View	Displaying the analysis model
	Results View	Viewing analysis results
	Wind View	Viewing the wind model
	Review View	Examining the design status or reviewing specific model properties
	Slab Deflections View	Viewing slab deflection analysis results

Click the following link to learn how to switch between view regimes:

- Change the view regime (page 280)

Scene content

You control the level of information displayed in each scene view by switching items on/off in the **Scene Content** dialog.

Click the following links to find out more about scene content:

- Manage scene content information (page 280)
- Scene Content entity categories (page 282)

Open, close and save scene views

You can display multiple different scene views simultaneously as tabbed windows within the main window. See the following instructions to open, close and save scene views.

Open 3D views

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Open a 3D view of the entire structure	• Double-click **Structure**.

Open a 3D solver view of the entire structure	a. Right-click **Structure**. b. In the context menu, select **Open solver view**.
Open a 3D view of a sub model	a. Click the **+** sign next to **Sub Models**. The existing sub models are viewed. b. Double-click a sub model.
Open a 3D solver view of a sub model	a. Click the **+** sign next to **Sub Models**. The existing sub models are viewed. b. Right-click a sub model. c. In the context menu, select **Open solver view**.
Open a 3D view of a sub structure	a. Click the **+** sign next to **Sub Structures**. The existing sub structures are viewed. b. Double-click a sub structure.
Open a 3D solver view of a sub structure	a. Click the **+** sign next to **Sub Structures**. The existing sub structures are viewed. b. Right-click a sub structures. c. In the context menu, select **Open solver view**.
Open a 3D view of a single member from within another view	a. Hover the mouse over the desired member to highlight it. b. Right-click the member. c. In the context menu, select **Open [element name] view**.
Open a 3D view of a single member in the **Project Workspace**	a. Click the **+** sign next to **Members**. b. Expand the required sub branches similarly. c. Right-click the required member. d. In the context menu, select **Open view**.

Open 2D views

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Open a 2D view of a construction level	a. Click the **+** sign next to **Levels**. The existing construction levels are viewed. b. Double-click the desired level.

Open a 2D solver view of a construction level	a. Click the **+** sign next to **Levels**. The existing construction levels are viewed. b. Right-click the desired level. c. In the context menu, select **Open solver view**.
Open a 2D view of a frame	a. Click the **+** sign next to **Frames**. The existing frames are viewed. b. Double-click the desired frame.
Open a 2D solver view of a frame	a. Click the **+** sign next to **Frames**. The existing frames are viewed. b. Right-click the desired frame. c. In the context menu, select **Open solver view**.
Open a 2D view of a sloped plane	**NOTE** Before you can view a 2D view of a sloped plane, you must create a sloped plane in your model. a. Click the **+** sign next to **Slopes**. The existing slopes are viewed. b. Double-click the desired slope.
Open a 2D solver view of a sloped plane	**NOTE** Before you can view a 2D view of a sloped plane, you must create a sloped plane in your model. a. Click the **+** sign next to **Slopes** The existing frames are viewed. b. Right-click the desired slope. c. In the context menu, select **Open solver view**.

Save, open and delete view configurations

Once a scene view has been adjusted to show a specific area of the model, you can save the scene view to a view configuration.

NOTE You can use view configurations in two ways:

- Include them as views in model reports.

  In this case, the view configurations retain the scene content settings that were in place when you saved the view configuration.

- Re-open them in a new scene view at a subsequent time.

  In this case, the view configurations adopt the scene content settings that are currently in place in the scene view.

- According to your needs, do one of the following:

To	Do this
Save a view configuration	1. Right-click anywhere in the view. 2. In the context menu, select **Save View Configuration...** The **View name** dialog box opens. 3. Name the view. 4. Click **OK**.
Open a saved view configuration	1. On the **Home** tab, click **Manage View Configurations**. 2. Select the desired view configuration. 3. Click **Open View**. 4. Click **OK**.
Delete a view configuration	1. On the **Home** tab, click **Manage View Configurations**. 2. Select the desired view configuration. 3. Click **Delete**. 4. Click **OK**.

Close views

- Click **x** on the top right corner of the view tab.

Create and modify scene view tab groups

When you have created multiple scene views, only the active view is visible by default. However, it is often useful to display views side by side or one below another. You can do this by creating tab groups.

Create a new tab group from an existing view tab

1. Right-click an existing view tab.
2. In the context menu, select **New Horizontal Tab Group** or **New Vertical Tab Group** according to your needs.

Create a new tab group using the docking control

1. Start dragging a view tab.

   A docking control appears in the middle of the view.

2. Do one of the following:

To	Do this
Create a new vertical tab group	• Drag the view over the left or right button of the docking control and release the mouse.
Create a new horizontal tab group	• Drag the view over the top or bottom button of the docking control and release the mouse.

Move a view from one tab group to another

1. Right-click the view tab.
2. In the context menu, select **Move to Next Tab Group**.

Change the view regime

- In the **Status bar** at the bottom of the main window, click the desired view regime button.
  - **Structural View** to show the geometry and loading of the structure.
  - **Solver View** to show the analysis model.
  - **Results View** to show the analysis results.
  - **Wind View** to show the wind model.
  - **Review View** to graphically examine the model properties or status.
  - **Slab Deflections View** to show the slab delection analysis results.

Manage scene content information

Different entity types have different levels of information associated with them. You can select how much of this information is displayed in each of the different scene views and view regimes. Scene content selections are saved independently with each scene view.

For example, in a **Solver View** regime, it is generally sufficient to represent beams by their insertion lines. However, in a **Structural View** regime, you are likely to also include their geometric outlines. In either of the views, you may also choose to display their direction arrows and possibly their reference texts also.

View the Scene Content window

1. Do one of the following:

If	Do this
Scene Content is set to **Auto Hide**	The **Scene Content** tab is docked on the edge of the main window. • Click the tab to expand the **Scene Content** window.
The **Scene Content** has been closed	• On the **Windows** tab, click **Scene Content**.

Select items in the Scene Content window

- In the **Scene Content** window, do one of the following:

To	Do this
View the sub categories of an entity category	• Click the arrow sign on the left side of the category name.
View a category in the model view	• Select the check box on the left side of the category name.
Hide a category in the model view	**NOTE** If you hide an entity category, you can no longer perform some commands that affect that entity category. For example, if you hide slab items, you cannot define slab or area loads on a floor because there are no slab panels to select. • Clear the check box on the left side of the category name.
Adjust the information viewed in the model view	1. Click the cell on the right side of the category name. A list of possible details opens. 2. Select the details you want to view. 3. Click **OK**.

Reinstate the default Scene Content selections

You can discard your current selections and reinstate the default **Scene Content** selections at any time.

See the following instructions.

1. Close the current view.
2. Re-open the view using the **Structure** tree.

See also

Open, close and save scene views (page 276)

Scene content entity categories

The following paragraphs describe some of the most important entity categories of **Scene Content**.

Architectural Grids

Allows you to show or hide architectural grids in 3D views. When selected, architectural grids are displayed at the lowest level of the model.

Grid & Construction Lines

Allows you to show or hide grid and construction lines at individual levels in both 2D and 3D views.

You can control the levels at which grids are or are not displayed in the **Properties** window for each level. Grids are only displayed in 3D views if the **Show grids in the 3D view** option is selected and in 2D views if the **Show grids in plane view** option is selected.

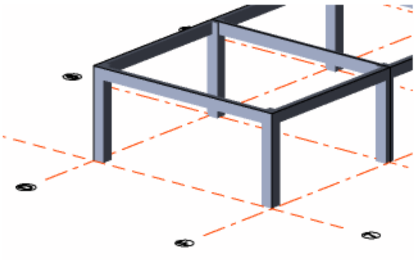
Members

Allow you to manage how each member type is displayed. For each of the mentioned elements, you can display the following information:

Geometry

Displays the faces of the member by shading them according to the member type.

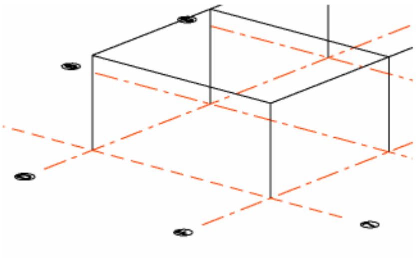
The following image displays a concrete beam and column example with the **Geometry** and **Architectural Grids** options selected.



InsertionLine

Displays a solid line between the start and end node of the member.

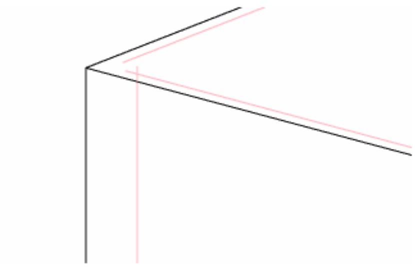
The following image displays the concrete beam and column example with the **InsertionLine** and **Architectural Grids** options selected.



LoadingLine

Displays a solid line through the center of the member. Any member loads are applied along the line.

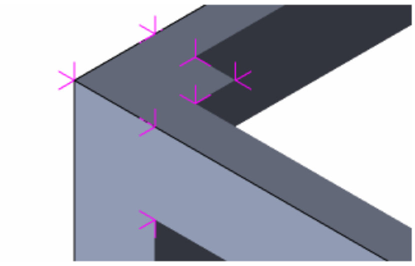
The following image displays the concrete beam and column example zoomed with the **InsertionLine** and **LoadingLine** options selected.



Normals

Displays the normal directions at each corner of the member.

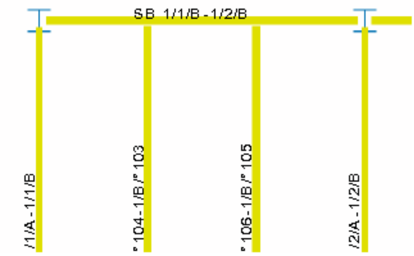
The following image displays the concrete beam and column example zoomed with the **Geometry** and **Normals** options selected.



Text

Displays the member name.

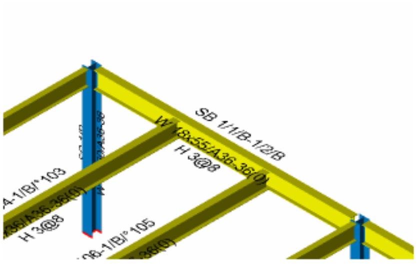
The following image displays a composite steel beam example with the **Geometry** and **Text** options selected.



Text2D

Displays the member name, section and class in the 2D plane of the member.

The following image displays the composite beam example with the **Geometry** and **Text2D** options selected.

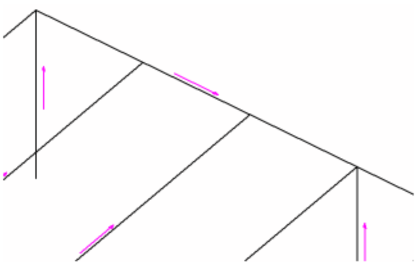


Direction

Displays the direction from end 1 to end 2 of the member.

If the direction is incorrect, go to the **Edit** tab and click **Reverse**.

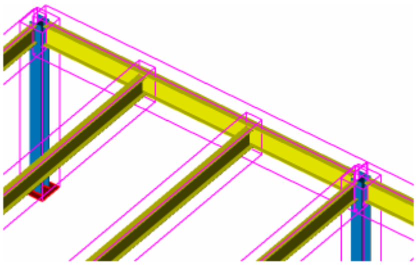
The following image shows the composite steel beam example with the **InsertionLine** and **Direction** options selected.



BoundingBox

Displays the bounding box of sections. Selecting the option may simplify right-clicking sections of a small cross section.

The following image displays the composite steel beam example with **Geometry** and **BoundingBox** selected.



Decking

Displays the strip of decking that is connected to the member.

The following image displays the composite steel beam example with **Geometry** and **Decking** selected.

Plan

NOTE	The **Plan** entity category is only available in 2D views.

The initial display for 2D views is configured for modeling purposes, so by default, it does not display all the information that will be output when drawings are created. However, you can select the **Plan** entity category to overlay one of the following planar drawing categories on the 2D view:

- General arrangement
- Beam end forces
- Foundation layout
- Loading plan
- Slab/mat layout

You can also select the individual drawing layers to be overlaid. The **Plan** category can be very useful, as it allows you to display information that would otherwise not be available whilst modeling.

To use the **Plan** category, do the following:

1. Open a 2D view.
2. In **Scene Content**, select **Plan**.
3. In the list on the right side of the **Plan** category, you can select the drawing category.
4. Click the arrow on the left side of the **Plan** category to view the different drawing layers that you can optionally display.

   Only the layers that exist in the previously mentioned planar drawing categories can be displayed. The most important layers are:

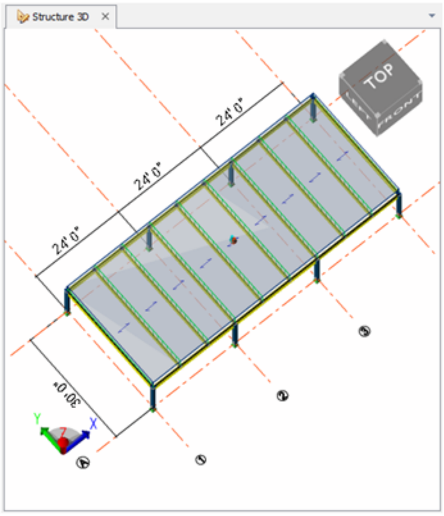
Name	Description
General	Allows you to display grids, construction lines, and dimensions as they would appear in drawings.
Members	Allows you to display the various different member types labeled as they would appear in drawings.
Walls	Allows you to display concrete walls labeled as they would appear in drawings.
Slabs/Mats	Allows you to display concrete slabs labeled as they would appear in drawings.
Other	Allows you to display various other items that can be output to drawings. **NOTE** It is not yet possible to overlay beam end forces or foundation reactions on a scene view. However, you can display them in general arrangement drawings.
Tables	Allows you to display tables of information that can be output to drawings.
Connections	Allows you to display connection names, attributes, and reactions as they would appear in drawings.
Beam End Forces	Allows you to display beam end forces as they would appear in drawings.

   By default, the layers that are initially selected match those in the first layer configuration for the drawing category in **Model Settings** --> **Drawings** --> **Layer Configurations**.
5. Select or clear the layers according to your needs.

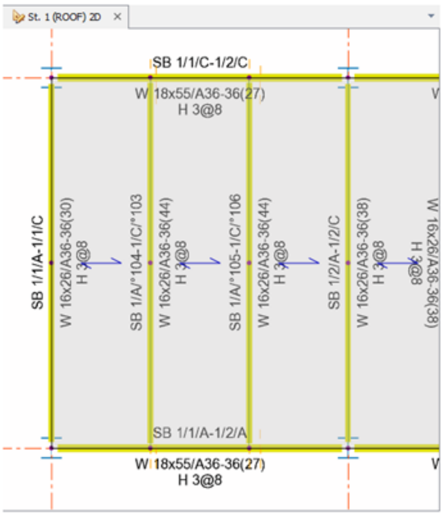
   TIP Layer content can be further controlled in **Model Settings** --> **Drawings** --> **Layer Configurations**.

Example

To illustrate how the **Plan** category would typically be used, consider the following composite beam example designed to the AISC 360 ASD head code:



In the following floor view, the standard beam labeling for modeling is applied. The labeling consists of the beam name, section, grade, number of connectors, and transverse reinforcement.

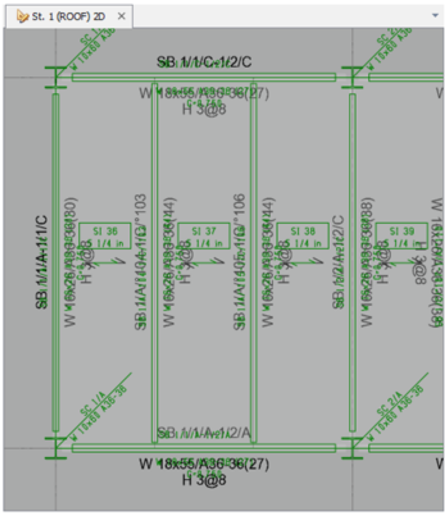


When you produce drawings, additional design information would probably also be conveyed. The information can contain, for example, the amount beam camber required. The camber is included in one of the drawing layers, so you can include it in the 2D scene view, as long as you know the layer it belongs to.

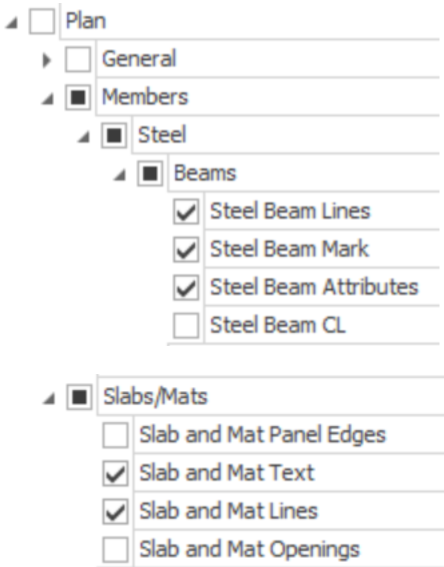
To include the camber, do the following:

1. Open a 2D scene view containing the beams.
2. In **Scene Content**, select **Plan**.

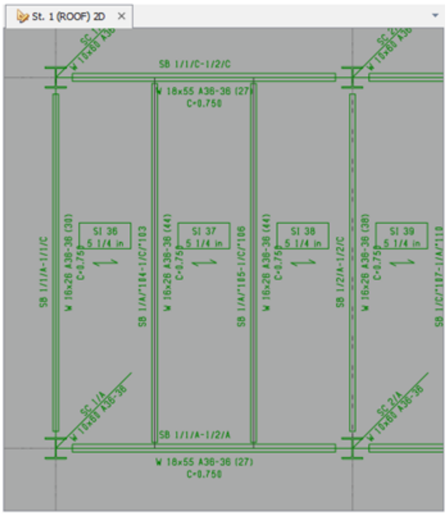
Depending on the current scene content selection, there may be a substantial duplication of axes, members, labeling, and so on.



3. Expand the **Plan** category and do the following:
   a. Clear the **Members** and **Slabs/Mats** options.
   b. Select options according to the following images:



The scene view should now appear less cluttered, as shown in the following image.



4. On the **Draw** tab, click **Settings** and adjust the drawing settings as follows:

   a. Go to **Options** --> **Planar Drawings** --> **General Arrangement**.

   b. Go to the **Beams** tab.

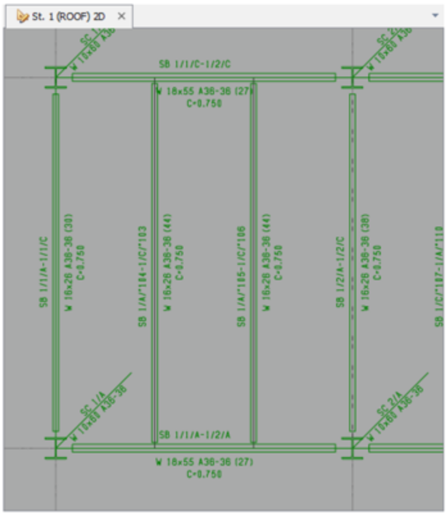
   c. Ensure that the **Camber** option is selected.

   d. Go to the **Slabs/Mats** tab.

   e. Clear all **Panel Labelling** options and the **Include panel span direction symbol** option.

   f. Click **OK**.

   The drawing is updated to match the new drawing options.



How to hide, re-display and move windows

The**Properties** window, the , **Scene Content**, and each of the **Project Workspace** tabs are displayed in windows that can be resized and repositioned, or docked to an edge of another window.

Auto-hide a window

To increase the area available for graphical display, you can choose to auto-hide any of the windows.

- In the desired window, click **Auto Hide**.

  The window immediately contracts to a tab.
- Click the window tab to expand it.

  You can only expand a window if it is docked against an edge of the main window.

Close a window

- Click **Close** at the top right corner of the window.

Re-display a closed window

1. Click the **Windows** tab.
2. In the **View** group, click the window name.

   If the window is already displayed, its control is highlighted in the **View** group.

Move a window

- Select the title bar of the window, and drag the window to its new location.

  If you place the selected window over an edge of the main window, or over a divider within the main window, the window docks to that edge or divider.

Dock a window as a tabbed page in another window

- Select the title bar of the window, and drag the window over the title bar or the tab group of the other window.

Open a tabbed page in another window

- Select the tabbed page, and drag the page to its new location.

  The tabbed page opens in a new window.

Dock a window using the docking control

1. Select the title bar of the window, and start dragging the window.

   The window docking control appears in the middle of the main window.

2. Drag the window over the docking control button that indicates the desired location.

Dock a window to a tabbed page of the current window

1. Select the title bar of the window, and drag the window over the central button of the docking control.
2. Release the left mouse button.

Keyboard shortcuts

Keyboard shortcuts in Tekla Structural Designer are described in the tables below.

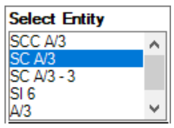
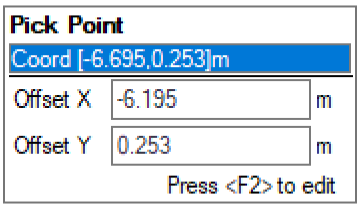
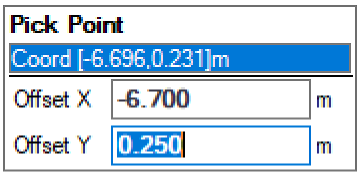
General keyboard shortcuts

Keys that perform general functions in Tekla Structural Designer are listed below.

Key	Function
F1	Displays the Tekla Structural Designer Help.
Ctrl	Holding down Ctrl while selecting entities adds the new entities to the current selection.
Esc	Cancels the current command.
Ctrl F	Opens the Find dialog
Ctrl O	Open a file
Ctrl N	Starts a new file

Keyboard shortcuts in 2D and 3D Views

The following keys function as described below in the Structural View and other graphical views.

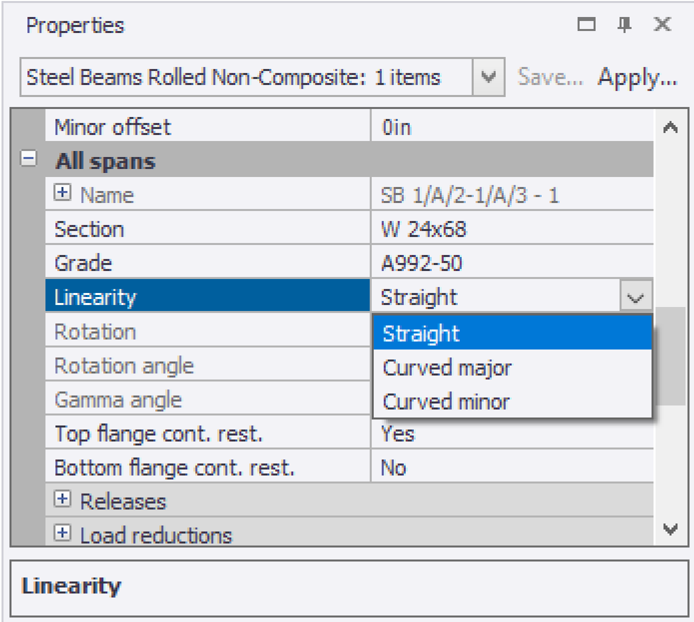
Key	Function
ZA	Zooms to the extents of the model.
ZS	Zooms to the selected entities. **NOTE** Prior to using this shortcut, at least one entity must be selected in order for the command to zoom to it.
Z1	Zooms to 1m (3ft) area around the cursor.
Z2	Zooms to 2m (6ft) area around the cursor.
Z3	Zooms to 3m (9ft) area around the cursor.
Up/Down arrow keys	Allows you to scroll between entities in the **Select Entity** tooltip when Tekla Structural Designer has detected multiple entities. 
Ctrl	Holding down Ctrl while selecting entities adds the new entities to the current selection.
F2	Whenever you are prompted to *pick a point*, the **Data Entry** tooltip displays the co-ordinates at the cursor position.  Pressing **F2** enables keyboard input of the exact co-ordinates required. 

Key	Function
F8	Switches between displaying and hiding the **ViewCube** in 3D views. 

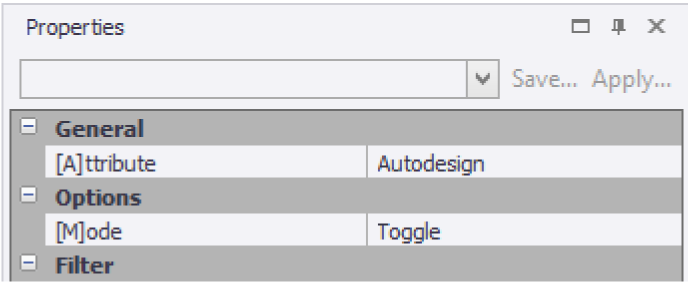
Keyboard shortcuts in Properties windows

Keys that perform specific functions in any **Properties** window are described below.

Key	Function
Up/Down arrow keys	Move between different properties and navigate within combo-box drop-down lists.
Tab	Tab - Enter or leave (and save) property editing mode.
Space	Space - Toggle check-boxes and expand combo-boxes.

<table>
<tr><th>Key</th><th>Function</th></tr>
<tr><td></td><td>
</td></tr>
<tr><td>Enter</td><td>Select and save property editing.</td></tr>
</table>

When a **Review View** is active, the following **Properties** window shortcuts are also available for **[A]ttibute** and **[M]ode**.



<table>
<tr><th>Key</th><th>Function</th></tr>
<tr><td>A</td><td>Press A to cycle through attributes in the [A]ttibute list.

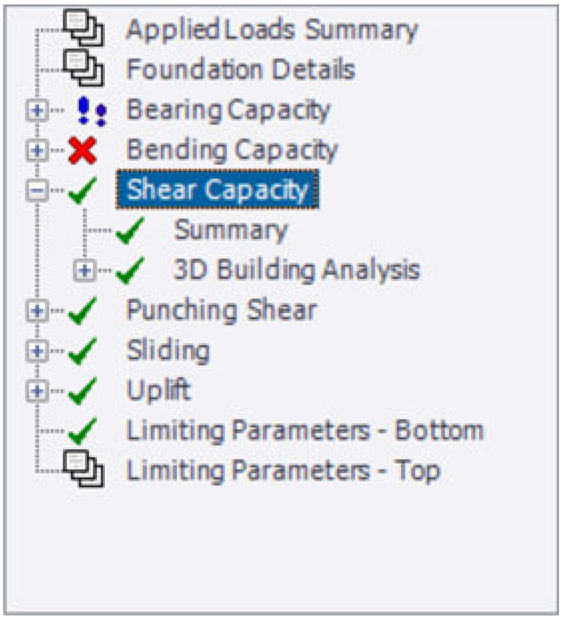
TIP If this doesn't work it is probably because the focus is in the Properties window. Click the Review View to makes it active and try again.</td></tr>
</table>

Key	Function
M	Press **M** to cycle through modes in the **[M]ode** list. **TIP** If this doesn't work it is probably because the focus is in the **Properties** window. Click the **Review View** to makes it active and try again.

Keyboard shortcuts in tree structures

Keys that perform specific functions in any tree structure are described below.

<table>
<tr><th>Key</th><th>Function</th></tr>
<tr><td>Right/Left arrow keys</td><td>Expand and collapse headings.

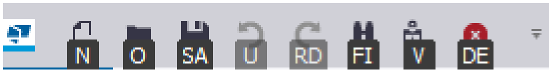


</td></tr>
</table>

Keyboard shortcuts to the Quick Access Toolbar

The Quick Access Toolbar is normally displayed as below.

If you press the **Alt** key letters are superimposed to enable each command to be accessed directly from the keyboard.



Command	Shortcut
New	**Alt N** (or, you can also use **Ctrl N**)
Open	**Alt O** (or, you can also use **Ctrl O**)
Save	**Alt SA**
Undo	**Alt U**
Redo	**Alt RD**
(page 2187)	**Alt FI** (or, you can also use **Ctrl F**)
(page 2201)	**Alt V**
Delete	**Alt DE**

Keyboard shortcuts to ribbon commands

All commands on the ribbon toolbars can be accessed from the keyboard after pressing the **Alt** key.

1. Press **Alt** then the appropriate letters (summarized in the table below) to open the required ribbon toolbar.

2. Press the appropriate letter(s) for the command you want to run.

Example: Press **Alt HFN** to open the **Home** toolbar and run the **Find** command shown above.

Toolbar	Shortcut
File	Press **Alt AP** then the letter(s) shown against the required command.
Home	Press **Alt H** then the letter(s) shown against the required command.
BIM Integration	Press **Alt B** then the letter(s) shown against the required command.
Model	Press **Alt M** then the letter(s) shown against the required command. .
Edit	Press **Alt E** then the letter(s) shown against the required command.
Load	Press **Alt LO** then the letter(s) shown against the required command.
Analyze	Press **Alt AN** then the letter(s) shown against the required command.
Design	Press **Alt DE** then the letter(s) shown against the required command.
Slab Deflection	Press **Alt SL** then the letter(s) shown against the required command.
Foundations	Press **Alt FO** then the letter(s) shown against the required command.
Report	Press **Alt RE** then the letter(s) shown against the required command.
Draw	Press **Alt DR** then the letter(s) shown against the required command.

Toolbar	Shortcut
Windows	Press **Alt W** then the letter(s) shown against the required command.
Loading Analysis	Press **Alt LA** then the letter(s) shown against the required command.
Results	Press **Alt RS** then the letter(s) shown against the required command.
Review	Press **Alt RV** then the letter(s) shown against the required command.
Review Data	Press **Alt RV** then the letter(s) shown against the required command.

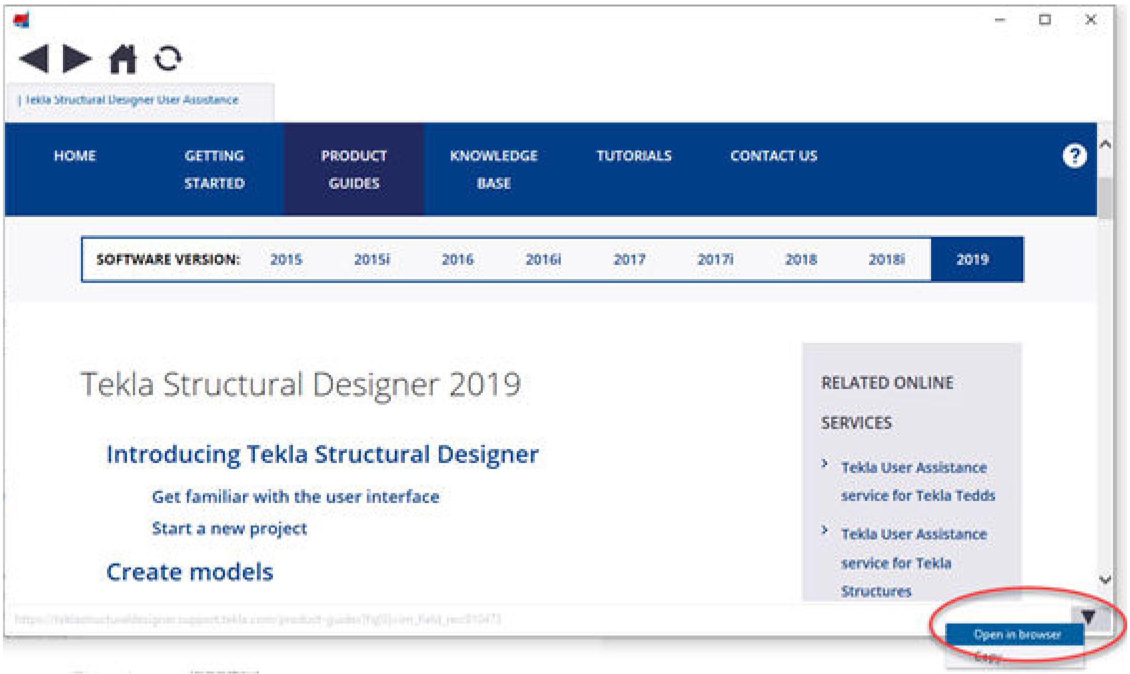
2.6 NOTE: Steps to take if the Help Viewer appears to be inactive

Please note that, since Help is displayed within a Tekla Structural Designer dialog, it will become inactive when another dialog is opened.

Should this occur, the following workarounds are suggested:

- Simply close the other dialog to continue.
- Alternatively you can access Tekla Online Help (TUA) for Structural Designer directly within any browser.

The Help can also be opened in a browser from any page by clicking the 'arrow' button at the bottom of the page as shown below:



Tekla Structural Designer User Assistance
HOME
GETTING STARTED
PRODUCT GUIDES
KNOWLEDGE BASE
TUTORIALS
CONTACT US
SOFTWARE VERSION: 2015 2015i 2016 2016i 2017 2017i 2018 2018i 2019
Tekla Structural Designer 2019
Introducing Tekla Structural Designer
Get familiar with the user interface
Start a new project
Create models
RELATED ONLINE SERVICES
Tekla User Assistance service for Tekla Tedds
Tekla User Assistance service for Tekla Structures
Open in browser

3 BIM integration

To simplify your work, Tekla Structural Designer allows you to both export data to different file formats, and import files to Tekla Structural Designer.

Click the following links to find out more:

BIM fundamentals (Playlist)

3.1 Import model data

Tekla Structural Designer allows you to import model data from other applications.

Click the following links to find out more about importing a model into Tekla Structural Designer:

- Import a project from a Structural BIM Import file (page 301)
- Import a project from a TEL file (page 302)
- Import data from a 3D DXF file (page 308)

Import a project from a Structural BIM Import file

1. On the **BIM Integration** toolbar, click **Structural BIM Import**.

   The **BIM Integration** wizard opens.
2. Use the **[...]** button to browse and select the appropriate CXL file.

   The remainder of the fields are populated with the settings read directly from the file.
3. Click **Next** to continue.

   The **Import Options** page shows the settings related to the file, it will indicate if this is a first time import or an update, and allows the user to choose Metric or Imperial for the units, a design code and the default level type. The level type can later be adjusted in the construction levels dialog if required.
4. Click **Next** to continue.

   The **Relocate Import Model** page allows for models to be moved to around the datum position in Tekla Structural Designer from a real world co-ordinate. The extent of the model can be seen from the information displayed on the right hand side of the dialog.
5. Click **Next** to continue.

   The **Integration Filter** will be displayed. The five options shown here allow you to verify if grids, levels etc. are to be imported. Please note that the settings held in the Delete Existing... and four Update... columns should not be used when performing a First Time Import and are only used when synchronizing changes into an existing Tekla Structural Designer model.
6. Click **Next** to continue.

   Mapping of materials to recognised grades will be shown now. Here you can view the different materials recognised in the incoming CXL file and the options to alter materials with a more preferred grade.

7. Click **Next** to continue (if available).

   Following on from Material mapping, the dialog for Decking mapping will be shown (if applicable). Again this allows users to view the decking types being imported and have the option of overwriting the details if required.

8. Click **Finish**

   After the file has completed processing, the model will be displayed within the Structural 3D view and you can then proceed with validation of the structure before applying any analytical information.

BIM fundamentals (Playlist)

Integration with Tekla Structures (Playlist)

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration

Import a project from a TEL file

Before importing a project from a TEL file, you are advised to be aware of the restrictions. Then, you can proceed to follow the detailed instructions to perform the import.

Restrictions

When importing TEL files, note the following points:

Round tripping	Does not exist in TEL file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.

Data that is imported	• Project summary (new models only)
	  • Project name, engineer, etc.
	• Support conditions
	  • Any associated UCS
	  • Spring supports, including linear and non-linear
	NOTE In S-Frame, for non-linear spring the default is F_max = 0. This does not

mean that the spring has zero capacity, but F_max is simply ignored.

This is not the case in Tekla Structural Designer. In Tekla Structural Designer, F_max =0 means the spring has zero capacity. Hence for models with compression-only springs imported from S-Frame, all nonlinear spring supports with F_max = 0 will need editing, or analysis will fail.

- 1D elements
  - Imported as analysis elements. However, contiguous elements are not merged into members (straight or curved).
  - Replicated by additional relatively stiff 1D elements.

    NOTE Automatic supports are not created, for example, under columns.

- Panels - area load only
  - Created as roof or wall panels without openings
- Panels - shell: tri, quad, or mixed
  - Created as meshed concrete walls (vertical planes only) or 2-way spanning slab items
  - Material properties are mapped manually during the import.
  - Thickness
- Panels - rigid or independent diaphragm
  - Created as 1-way spanning slab items.

	- Material properties are mapped manually during the import. - Thickness - Panels - holes - Created as slab or wall openings. **NOTE** The holes must be rectangular for walls and rectangular or circular for slabs. Otherwise, Tekla Structural Designer will generate a warning. - Loadcases (linear only) - Nodal loads - Settlement loads 1D element loads, including uniform temperature loads - Area panel loads, not including uniform temperature loads - Combinations
Exclusions	- Units: the Tekla Structural Designer model units are not changed to match the TEL file units. However, values are converted to the Tekla Structural Designer model units. - There is no special handling for 2D files. 2D files are imported in the same plane as they are defined in S-Frame (the X-Y plane), and default constraints are not imported. - Default constraints are not imported, and no warning is generated. Default constraints are supports applied to all nodes without exception internally during analysis. Default constraints are not displayed in the S-Frame interface. For models

marked as 2D, default constraints restrict displacement to the X-Y plane and are as follows: Fz, My and Mz fixed. Default constraints may be manually applied in 3D models, and the S-Frame model can be examined to confirm their nature. The import does not replicate default constraints. To ensure equivalence, default constraints must be applied manually to all nodes, either in the S-Frame model prior to import, or in Tekla Structural Designer subsequently.

- The import does not create any physical members. S-Frame physical members are treated like any other 1D element and imported as a single analysis element. In particular, Tekla Structural Designer does not merge contiguous elements into members (straight or curved) or identify columns, beams, and so on. No warnings are generated.

  In addition, for S-Frame physical models please note the following:

  - Intermediate nodes that do not form the ends of other elements are not imported. If such nodes have supports applied, the model will not be equivalent and should be adjusted to ensure equivalence.
  - If physical members have tapered sections, the sections should be sub-divided in S-Frame before importing to Tekla Structural Designer to produce an equivalent model.
  - Alternatively, the S-Frame model can be converted to an analytical model in S-Frame prior to import, using the S-Frame command for this.

- Staged construction data is not imported, and no warning is generated. Typically, the entire model is imported, representing the last stage in which the model is complete. Otherwise, turn the **Staged Construction** setting in S-Frame off prior to import. This will remove all stages but the last one, and issue a warning to this effect. The model is then non-staged, and so should be valid for import.
- Although the following can be modeled as single objects in Tekla Structural Designer, no attempt is made to import them as single objects from collections of S-Frame objects:
  - Mid-pier walls
  - Trusses
  - Portal frames
- Inactive elements are imported as inactive analysis elements of the beam type.

  NOTE Inactive elements are quite likely to originate from tension-only cross bracing in Fastrak Building Designer models. In this situation, we recommend that you click the warning to identify the relevant part of the model, delete both "braces", and create new braces using the specific X-brace pair.

- Wall and strip integration lines are not imported. A warning is issued to this effect.
- Tapered sections: a 1D element is imported. However, no tapered section dimension data is imported. A warning is issued to this effect.

- Prestress data for 1D & 2D elements is not imported.
- Percentage fixity data for 1D elements is not imported.
- Non-linear spring data by graph: 1D elements and supports are imported, but the spring stiffnesses are set to 0. A warning is issued to this effect.
- Non-structural alignment and offsets (cardinal point data) are ignored, and warnings are issued if they are non-zero.
- Panels: general diaphragms, mat foundations, membranes and plates are excluded, and a warning is issued.
- Diaphragm panel node exclusions are ignored.
- 2D elements are excluded, and a warning is issued.
- Meshing properties for panels are excluded without warning.
- Shear Walls: only quadrilaterals can be created.
- Diaphragm constraints are excluded, and a warning is issued to this effect. Any diaphragm constraints must be replicated in Tekla Structural Designer to ensure equivalence.
- Slaved nodes are excluded, and a warning is issued.
- Lumped mass is excluded.
- Groups are excluded.
- Notional load factors: NHF and EHF are added to combinations, with the sign indicating positive or negative for each direction. However, the actual value is ignored. The standard notional load calculation method and default strength factor are used.

	• Non-zero gravitational factors for Global X & Y are excluded, and a warning is issued. • Thermal gradient loads for 1D and 2D elements are excluded. • Moving loads are excluded. • Time history loads are excluded. • 2D elements are excluded, and a warning is issued. • RSA data is excluded.

Instructions

The model is imported. If there are any associated warning messages, these can be reviewed from the BIM branch of the Status tree in the Project Workspace.

1. On the **BIM Integration** tab, click **'TEL' File Import**.

   The **'TEL' File Import** button is not active unless you have a Tekla Structural Designer document open.

   The **BIM Integration** wizard opens.
2. Click **...** to browse the .TEL file, and click **Next**.
3. If necessary, adjust the location and rotation of the import model, and click **Next**.
4. Select an appropriate material type and grade for each material using the lists.
5. Click **Finish**.

The model is now imported. Any associated warning messaged can be reviewed in the **BIM** sub-branch of the **Status tree**.

Import data from a 3D DXF file

See the following paragraphs for the restrictions of 3D DXF import, and the instructions on importing data from a 3D DXF file.

Restrictions

Note the following points when importing 3D DXF files:

• Round tripping does not exist in TEL file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.

- Analysis elements are created from line segments in the selected layers of the DXF file as follows:
  - All lines in these layers become 1D analysis elements.
  - Arcs and circles in the selected layers are excluded without a warning.
  - Blocks are not handled, and no warnings are issued.
  - Polylines in the selected layers are excluded without a warning.
  - 3D solids in the selected layers are excluded without any warning.
  - 2D faces for 3D objects in the selected layers are excluded without a warning.
  - All ends of lines in the selected layers become nodes.
  - Nodes are not introduced in the intersections of crossing lines.
  - The following elements are not included in the import:
    - Reading of any text
    - Intelligence on "through members"
    - Gridlines and construction lines
    - 2D elements
    - Supports
    - Section properties
    - Materials
    - Loads
    - Combinations

Instructions

1. On the **BIM Integration** tab, click **3D DXF Import**.

   The **3D DXF Import** can only be selected in a 3D view.

   The **Open** dialog opens.
2. Browse to the required .dxf file.
3. Click **Open**.

   The **DXF Import Wizard** opens.
4. Select the layers and colors that you want to import.
5. If necessary, apply offsets and rotate the model before data is imported.
6. Click **OK**.

Any line segments found in the selected layers are now imported as analysis elements.

3.2 Working collaboratively with Trimble Connect

Trimble Connect is a project collaboration tool allowing project stakeholders access to reliable, up-to-date project information. It is available as a cloud-based platform (Trimble Connect Web) and a Windows application (Trimble Connect for Windows). Projects are synchronised between the Windows app and the cloud.

NOTE To learn more about Trimble Connect, see:

- https://connect.trimble.com/
- https://trimbleconnect.support.tekla.com/

NOTE You need to have a Trimble Identity before you can start using Trimble Connect

Trimble Connect Project Explorer is used within Tekla Structural Designer to control the flow of information between the open model and a Trimble Connect project.

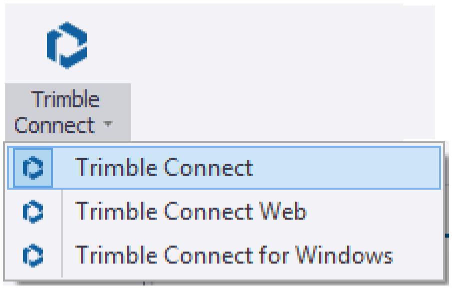
With Trimble Connect Project Explorer you can:

- link a Tekla Structural Designer model to a Trimble Connect project
- create and rename folders in a Trimble Connect project
- view a file list and rename files in a Trimble Connect project
- upload an IFC of the model to a Trimble Connect project
- upload drawings
- upload reports

NOTE A Tekla Structural Designer model can only be linked to a Trimble Connect project from the Trimble Connect Project Explorer and not from the web or Windows apps. The model must be linked before any information can move between Tekla Structural Designer and Trimble Connect.

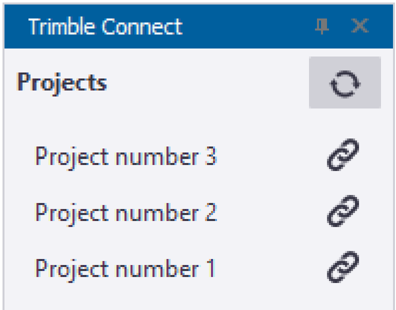
Launch Trimble Connect Project Explorer

1. Log in using your Trimble Identity, (if not already signed in).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select **Trimble Connect**



Trimble Connect Project Explorer opens, either as a docked window, or as a tab on the right of the interface.

Available projects are listed with link icons as below.



TIP If the project in which you want to collaborate is not shown, try clicking the refresh button to synchronise with the cloud. If it still not shown you would need somebody with the appropriate permissions to create it, or grant you access, using Trimble Connect Web, or Trimble Connect for Windows.

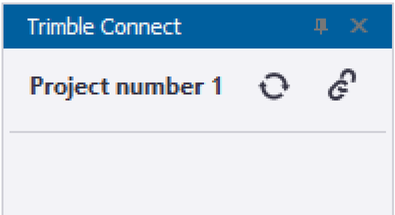
TIP If a 'Cannot find linked project' message is displayed, this indicates either that you do not have permission to view the project; or, that the project to which the model was previously linked has been deleted - in this situation you would need to click 'Unlink' before you are able to link the model with another project.

Link or unlink a project

1. To link to a project:

1. Click  next to the project you want to link to.

   When the model is linked, the project name appears at the top of the window and the icon changes as shown below.

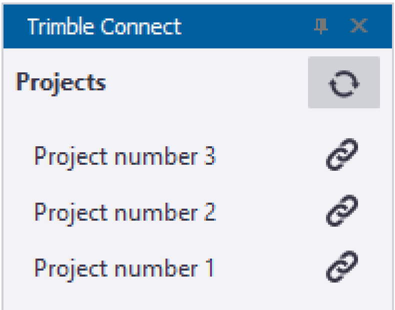


Any folders or files in the Trimble Connect project that you have *read* access to are also displayed.

2. To unlink from a project:

2. Click  to unlink the project.

   The full list of available projects is displayed once more.



Create folders, rename folders, rename files

When you link to a Trimble Connect project the existing project folder structure is displayed. You can add to this, if required.

NOTE You can only create but not delete folders in Trimble Connect Project Explorer. Folders can only be deleted in Trimble Connect Web, or Trimble Connect for Windows.

You can only rename folders and files to which you have *write* access.

---

1. To create a folder:
1. Right click in the Trimble Connect Project Explorer window.
2. Select **Create folder** and enter the folder name.
2. To create a subfolder:
3. Right click on an existing folder.
4. Click Create folder and enter the folder name.

3. To rename a file or folder:
5. Right click on the file/folder.
6. Select **Rename**

Upload an IFC file of a model

1. Right-click on the folder where you want to create the IFC.
2. Select **Upload New IFC file**
3. Follow the steps in the Export to IFC wizard to create the ifc file.
    a. Adjust the location and rotation of the model, and click **Next**
    b. Specify the export names of material grades, and click **Next**
    c. Specify the ifc file name

    The ifc file will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the IFC model in Trimble Connect.

NOTE If the model is subsequently updated, you can upload a new revision of the ifc file. To do this, right-click on the ifc file name and select **Upload new version of IFC**.

Upload a multi-member drawing

1. Open a view displaying the part of the model that you want to include in the drawing.
2. On the **Draw** ribbon, click the drawing type.

    The **DXF Export Preferences** dialog box opens.
3. Select **Upload to Trimble Connect**

    NOTE The above option is only available after the model has been linked to a Trimble Connect project.

    A default drawing name for the upload is displayed.

    If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

    The **Create link between model member and uploaded drawing** option is not applicable to multi-member drawings, so this option cannot be selected.

4. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here will be true for the whole model.

Upload a single member drawing

1. Highlight the member for which you want to create a drawing.
2. Right click, then from the context menu select **Generate Drawing**

   The **DXF Export Preferences** dialog box opens.
3. Select **Upload to Trimble Connect**

   NOTE The above option is only available after the model has been linked to a Trimble Connect project.

   A default drawing name for the upload is displayed.

   If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.
4. To link the model member and drawing, click **Create link between model member and uploaded drawing**.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.
5. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here apply for the whole model.

Upload a model report

1. Open the required report in Tekla Structural Designer.
2. On the **Report** ribbon, click **PDF Upload**

NOTE Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

The **Upload Report** dialog box opens and a pdf name for the report is displayed.

If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

The **Create link between model member and uploaded report** option is not applicable to model reports, so this option cannot be selected.

3. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Upload a member report

1. Highlight the member for which you want to create a report.
2. Right click, then from the context menu select **Report for Member**

   The report is displayed in Tekla Structural Designer.
3. On the **Report** ribbon, click **PDF Upload**

   NOTE Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

   The **Upload Report** dialog box opens and a pdf name for the report is displayed.

   If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.
4. To link the model member and report, click **Create link between model member and uploaded report**.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.
5. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Link a drawing or report to an existing IFC

When drawings and reports are uploaded they are automatically linked to an existing IFC file as long as the appropriate option is selected (*Create link between model member and uploaded report / Create link between model member and uploaded drawing).*

It is important to **save** the Tekla Structural Designer model after exporting the IFC to Trimble Connect so that the link between the two is stored in the TSD model.

Linking reports

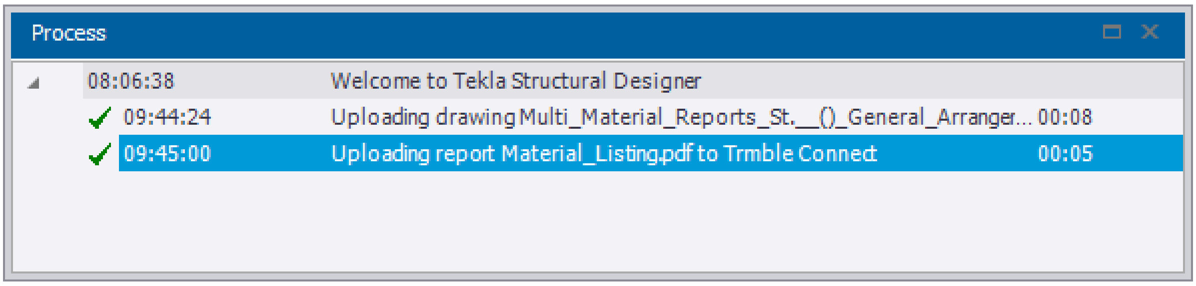
Linking of reports occurs for member reports only. If there are 3 IFC objects associated with one member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Linking drawings

Linking of drawings occurs for member drawings only. If there are 3 IFC objects associated with one TSD member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Check linking progress in the Process Window

The progress of the linking is logged in the process window. This will show the number of successful/unsuccessful links that have been created.



There are a few reasons why linking could be unsuccessful:

1. The member associated with the report/drawing does not exist in IFC model
2. Connection to the Trimble Connect project is lost
3. IFC model was not successfully uploaded

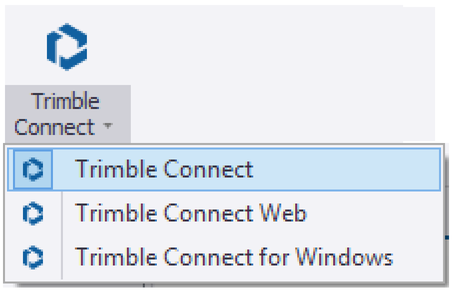
Open Trimble Connect to a model view for an IFC

1. In the Trimble Connect Project Explorer window, right-click on the IFC file to be viewed.
2. Select **Open in Trimble Connect Web**, or **Open in Trimble Connect for Windows** as required.

   Trimble Connect opens the linked project and displays a view of the selected ifc.

Open Trimble Connect

1. To open Trimble Connect from the ribbon:
1. If not already signed in, click log in (at the top right of the interface).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select select **Trimble Connect Web**, or **Trimble Connect for Windows** as required.



   - **Trimble Connect Web:** This launches the in-browser web app. If the open model is associated with a Trimble Connect project, the project itself is opened, otherwise a default Trimble Connect page is opened.
   - **Trimble Connect for Window:** This launches the Windows application. Again, if the open model is associated with a Trimble Connect project then that project is opened.
2. To open Trimble Connect from **Trimble Connect Project Explorer**:
4. Right-click on a folder in the Trimble Connect Project Explorer window.
5. Select **Open in Trimble Connect Web**, or **Open in Trimble Connect for Windows** as required.

   The in-browser web or Windows app is launched and opened at the selected folder.

3.3 Export to Trimble applications

Click the following links to find out more about exporting to other Trimble applications:

- Export a model to Tekla Structures (page 318)
- Export to Tekla Connection Designer (page 318)
- Export to Tekla Portal Frame Designer (page 319)
- Export to Tekla Tedds (page 324)

Export a model to Tekla Structures

To export a model to Tekla Structures, see the following instructions.

1. Create your model in the usual way.
2. On the **BIM Integration** tab, click **Tekla Structures Export**.

   The **BIM Integration** wizard opens.
3. On the first page, click **Next**.
4. Adjust the location and rotation of the model, and click **Next**.
5. Select the items that are included in the model, and click **Next**.
6. Specify the export names of material grades, and click **Next**.
7. Specify the file name and location.
8. Select whether the file is exported for the first time, or whether you want to update an existing model.
9. Click **Finish**.
10. Start Tekla Structures and open the file to see your project.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 2069). This allows individual levels to be:

- only included if the level is a floor
- always included
- never included

Integration with Tekla Structures (Playlist)

Export to Tekla Connection Designer

Before exporting connections to Tekla Connection Designer, you are advised to be aware of the limitations and also the recommended workflows for the different connection types.

NOTE Only valid base plate, column splice and moment connections listed in the Project Workspace Connections (page 273) tree can be exported.

To export a single connection

NOTE Export requires a Tekla Connection Designer licence.

1. Right-click the connection in a Scene View.
2. Choose Export *(connection name)* to Tekla Connection Designer from the right-click menu.

   The Tekla Connection Designer application opens to allow the selected connection to be designed.

To export multiple connections

NOTE Export requires a Tekla Connection Designer licence.

1. Select the connections to be exported in a Scene View.
2. On the **BIM Integration** tab, click **TCD Export**.

   The Tekla Connection Designer application opens to allow the selected connections to be designed.

To return connection data from Tekla Connection Designer

Connection data from Tekla Connection Designer can be returned to Tekla Structural Designer as follows:

1. In Tekla Connection Designer click Connection> Return Connection to Tekla Structural Designer

   NOTE In the current release, Moment Connection data from Tekla Connection Designer is only partially returned. (e.g. Bolt layouts and endplate dimensions are not returned).

Export to Tekla Portal Frame Designer

Export to Tekla Portal Frame Designer workflow

It is important to follow the correct definition, loading and design sequence when frames are to be transferred for design to Tekla Portal Frame Designer.

1. Define frames/loading etc inTekla Structural Designer.
2. Run Analyse All (Static).

   Analysis is necessary in order for the loading to be included in the export to Tekla Portal Frame Designer.
3. Export the frame to Tekla Portal Frame Designer to design it and check member stability.
4. Return revised sections/grades and haunch geometry back to the Tekla Structural Designer model.

NOTE Only section sizes, haunch dimensions and steel grades are returned to Tekla Structural Designer. Any other changes to the model data and loading are not returned.

How loading, restraints and supports are handled in the export

Loadcases and combinations

When you open the Loadcases dialog in Tekla Portal Frame Designer the first four loadcases are automatically created and named as follows:

- Self Weight
- Frame Dead Load
- Frame Service Load
- Frame Imposed Load

The loading in these loadcases will be taken from Tekla Structural Designer loadcases (providing the loadcase names have been specified exactly as per the below table):

Tekla Structural Designer Loadcase	Tekla Portal Frame Designer Loadcase
Self weight - excluding slabs	Self weight
Dead	Frame Dead Load
Services	Frame Service Load
Imposed	Frame Imposed Load

Every other Tekla Structural Designer loadcase that applies in-plane loading to the frame gets exported to Tekla Portal Frame Designer retaining it's original name.

If a particular Tekla Structural Designer loadcase does not apply any in-plane load to the frame being exported, the loadcase is not exported and it is also removed from the Tekla Portal Frame Designer load combinations.

If the Tekla Structural Designer model does not contain loadcases named "Dead", "Services" or "Imposed"; the "Frame Dead Load", "Frame Service Load" and "Frame Imposed Load" loadcases are created regardless, (defaulting to the loading values specified in the Tekla Portal Frame Designer Preferences). However, since these loadcases are not included in any load combinations they will have no affect on the design.

If the Tekla Structural Designer model contains loadcases named "Dead", "Services" or "Imposed" which do not apply any in-plane load to the frame being exported; then in the same way as any other loadcase that does not apply any in-plane load, the loadcase is removed from the Tekla Portal Frame Designer load combinations. (In this situation the "Frame Dead Load", "Frame Service Load" or "Frame Imposed Load" loadcase is still created regardless (defaulting to the loading value specified in the Tekla Portal Frame Designer Preferences), but once again since it does not get included in any load combinations it will have no affect on the design.

Once loadcases that do not apply in-plane load have been removed from a load combination, if the resulting combination is empty, or if it only contains self weight, the combination itself is removed.

Lateral Restraints

The Tekla Structural Designer lateral restraints are exported, strut restraints are ignored.

NOTE The below mappings assume default rotations of the rafters and columns. (Changing the rotation by 180 degrees flips the restrained flange in Tekla Portal Frame Designer.)

Rafter Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Top Flange	Bottom Flange	
X	X	Torsional
X	-	Outer
-	X	Inner
-	-	No restraint

Column Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Face A	Face C	
X	X	Torsional
X	-	Outer
-	X	Inner
-	-	No restraint

NOTE The rafter top flange and column face A can be identified graphically, for details expand the Axis systems topic and then refer to the **Object Orientation** section within it.

Supports

The Fy, Fz and Mx Tekla Structural Designer support properties are used to configure the Tekla Portal Frame Designer base fixities as follows

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Fy Fixed	Horiz. Stiffness (ULS, SLS, Stability): Restrained	
Fz Fixed	Vertic. Stiffness (ULS, SLS, Stability): Restrained	
Mx Fixed	ULS:	Rot. 100%, Cap. 100%
	SLS:	Restrained
	Stability:	Restrained
Fy Free-Release	Horiz. Stiffness (ULS, SLS, Stability): 0kN/m	
Fz Free-Release	Vertic. Stiffness (ULS, SLS, Stability): 0kN/m	
Mx Free-Release	ULS:	Rot. Free, Cap. 1%
	SLS:	Free
	Stability:	Free
Fy Free-Spring Linear: XkN/m	Horiz. Stiffness (ULS, SLS, Stability): XkN/m	
Fz Free-Spring Linear: XkN/m	Vertic. Stiffness (ULS, SLS, Stability): XkN/m	
Mx Free-Spring Linear: XkNm/rad	ULS:	Rot. XkNm/rad, Cap. 1%
	SLS:	XkNm/rad
	Stability:	XkNm/rad
Fy Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Horiz. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	
Fz Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Vertic. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Mx Free-Spring Non Linear: -ve=XkNm/rad, +ve YkNm/rad	ULS:	Rot. Max (X, Y) kNm/rad, Cap. 1%
	SLS:	Max (X, Y) kNm/rad
	Stability:	Max (X, Y) kNm/rad
Mx Free-Nominally Pinned: 10%	ULS:	Rot. Free, Cap. 1%
	SLS:	20%
	Stability:	10%
Mx Free-Nominally Pinned: X%	ULS:	Rot. X%, Cap. 1%
	SLS:	X%
	Stability:	X%
Mx Free-Nominally Fixed: X%	ULS:	Rot. X%, Cap. 100%
	SLS:	X%
	Stability:	X%

To export a single frame

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Right-click the frame in a Scene View.
2. Choose Export *(frame name)* to Tekla Portal Frame Designer from the right-click menu.

   The Tekla Portal Frame Designer application opens to allow the selected frame to be designed.

To export multiple frames

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Select the frames to be exported in a Scene View.
2. On the **Home** tab, click **TPFD Export**.

   The Tekla Portal Frame Designer application opens to allow the selected frames to be designed.

To return revised sections from Tekla Portal Frame Designer

If the section sizes have been revised in Tekla Portal Frame Designer they can be returned to Tekla Structural Designer as follows:

1. In Tekla Portal Frame Designer click **Portal Frame**> **Return sections to model**

Export to Tekla Tedds

Once they have been designed using Tekla Tedds, precast and/or timber members can then be exported to Tedds in order to output the calculations.

NOTE	Export requires a Tekla Tedds licence.

Understanding each of the export options

Command	Description
Export to Tekla Tedds > Model	Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member in the model. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Member	This option only appears for a highlighted member if the option to design using groups is not active. Launches Tekla Tedds and: • Creates a new Tedds Project containing a single document for the member. • The data exported is taken from the last design run (either Member or Selection) associated with the member. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Group	This option only appears for a highlighted member if the option to design using groups is active. Launches Tekla Tedds and:

Command	Description
	• Creates a new Tedds Project containing a single document for the group. • The data exported is taken from the last design run (either Member, Group, or Selection) associated with the group. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Selection	This option only appears if one or more members are selected. Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member or group in the selection. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > <*Substructure name*>	Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member in the sub structure. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.

To export all timber and precast members

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Export using Tekla Tedds > Model**

   Tekla Tedds opens to allow the calculations for the all the exported members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a single member

1. Right-click the member in a Scene View.
2. Choose **Export to Tekla Tedds > Member** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted member to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export multiple members

1. Select the precast and/or timber members to be exported in a Scene View.
2. Right-click and choose **Export to Tekla Tedds > Selection** from the context menu.

   Tekla Tedds opens to allow the calculations for the selected members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a group

1. Right-click any member of the group in a Scene View.
2. Choose **Export to Tekla Tedds > Group** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted group to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a substructure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to export.
3. In the context menu, select **Export using Tekla Tedds > *<Sub structure name>***

Tekla Tedds opens to allow the calculations for the substructure to be output or recalculated.

NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

3.4 Export to and import from other applications

Tekla Structural Designer allows you to import and export data to different file formats.

Click the following links to find out more about importing to and exporting from other applications:

- Export a model to Autodesk Revit Structure (page 327)
- Export a model to IFC (page 328)
- Export to and import from Westok Cellbeam (page 329)
- Export to and import from FBEAM (page 330)
- Export a model to ADAPT (page 335)
- Export a model to STAAD (page 339)
- Export a model to Autodesk Robot Structural Analysis (page 340)
- Export a model to the cloud (page 341)
- Export to One Click LCA (page 341)
- Export to IDEA StatiCa Connection Design (page 344)

Export a model to Autodesk Revit Structure

To export a model to Revit Structure, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Autodesk Revit Export**.

   The **BIM Integration** wizard opens.
3. Adjust the location and rotation of the model, and click **Next**.
4. Select the items that are included in the model, and click **Next**.
5. Specify the export names of material grades, and click **Next**.
6. Specify the file name and location.
7. Select whether the file is exported for the first time, or whether you want to update an existing model.

8. Click **Finish**.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 2069). This allows individual levels to be:

- only included if the level is a floor
- always included
- never included

NOTE To open the exported file in Revit, you will need to have installed the Tekla Structural Designer Integrator for Autodesk Revit. This is available for download from the following link: https://download.tekla.com/tekla-structural-designer/for-businesses/all-downloads?field_package_reference_tid_selective=5713

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration

Export a model to IFC

To export your model to IFC, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **IFC Export**.

   The **BIM Integration** wizard opens at the **Relocate Export Model** page.

   The **Relocate Export Model** page allows for models to be moved from the datum position in Tekla Structural Designer to a real world co-ordinate.
3. Move and rotate the export model if required, and click **Next**.
4. On the **Integration Filter** page, select the items that are included in the export model, and click **Next**.
5. On the **IFC Export File** page, select whether the file is exported for the first time, or whether you want to update an existing model. Then specify the file format, name and location.
6. Click **Finish**.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 2069). This allows individual levels to be:

- only included if the level is a floor

- always included
- never included

Export to and import from Westok Cellbeam

To export and import beams to and from Westok Cellbeam, see the following restrictions and instructions.

RESTRICTION Cellbeam import and export are only available for BS and Eurocode head codes.

Export to Cellbeam

1. On the **BIM Integration** tab, click **Cellbeam Export**.

   The **Export Westok Beams** dialog box opens. Any Westok Cellular beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to Westok Cellbeam, and click **Next**.
3. Select the export format and name the file.
4. Click **Next**.
5. Ensure that you have selected the design combinations whose results you want to use in the Westok beam design.
6. Click **Next**.
7. Select the folder where you want to place the Westok files, and click **Export**.
8. Click **Finish**.

Import from Cellbeam

NOTE In order to import Westok beams into your model, you must have previously created the beams in your Tekla Structural Designer model, and have exported them to Westok design. When you do this, each beam is given a unique identifier. This means that when you import the results from the Westok file, Tekla Structural Designer knows to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **Cellbeam Import**.

   The **Import Westok Beams** dialog box opens.
2. Click **Add...**

3. Select the files that you want to import.
4. If necessary, change the file names.
5. Click **Open**.
6. Click **Import**.

   Provided that Tekla Structural Designer recognizes the identifier, the details in the files are imported to the associated beam in your model.

Export to and import from FBEAM

To export to and import from FBEAM, see the following instructions.

TIP Create substructures for your FABSEC® beams to make it easier to review and manage the FBEAM workflow.

Overview

FBEAM is software from Fabsec for the design of composite and non-composite plain and cellular beams.

The workflow for taking FABSEC® beams from a Tekla Structural Designer model and designing then within FBEAM is as follows:

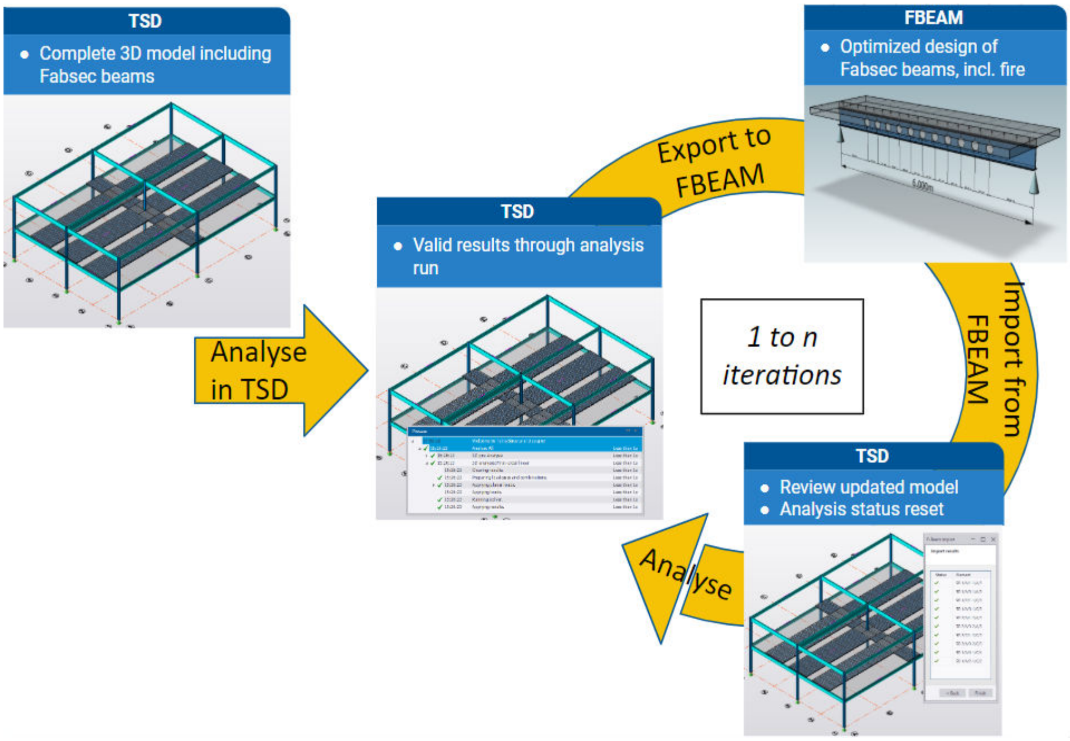
- Select the FABSEC® beams in your Tekla Structural Designer model and click **FBEAM Export**.

  This generates an XML file containing all the required properties required for each member.
- You can then launch FBEAM and open the project you wish to import the beams into.
- You then click the import button which starts FBEAM's import work flow. This will allow you to load the previously created XML file and select a subset or all of the beams in the file to import.

  Once the process is complete FBEAM will have created all of the members and their associated properties.
- You can then design the FABSEC® beams and specify fire protection coatings.
- Once complete, click the export button in FBEAM.

  This generates another XML file that can be imported back into Tekla Structural Designer.
- In Tekla Structural Designer, (with the model that was used to export the beams open), click the **FBEAM Import** to start the import workflow. This will then update each of the previously exported FABSEC® beams with the properties contained within the FBEAM generated XML file.

The above workflow can be summarized as follows:



Limitations

This following limitations apply to the import/export of FABSEC® beams:

1. Consider the case where multiple combinations are exported and a loadcase of a particular load type e.g Dead appears in one combination but not all. In FBEAM the analysis will include all loadcases of the same type in each combination. This can overestimate the load significantly.

2. When Wind loadcases are exported to the 'Additional Load Set' for a particular beam, Tekla Structural Designer searches for the Wind loadcase that has the largest positive end force and exports that value to the 'Additional Load Set' and gives it a 'Type', 'Wind' in FBEAM. Similarly for the maximum negative value in Tekla Structural Designer to the 'Type', 'Wind Up' in FBEAM. This loadcase will be used in the design of that beam for all positive wind combinations and all negative wind combinations respectively. Whether a wind combination is considered negative is determined from the start and end values of the distributed (Wind) load applied to that beam (or point loads). If both are negative then the wind combination is negative (uplift).
3. The special loadcase in Tekla Structural Designer 'Self weight - excluding slabs' (i.e. beam self weight) is assumed always to be included in design combinations. Consequently, the setting in FBEAM, 'Include the beam self weight' is always switched on.
4. In any combination if any load type e.g. Dead or Imposed appears more than once with different load factors, only one value of load factor is exported. This situation is unusual but might occur for imposed loads for example when different psi factors might apply.
5. Any load types or direction not supported by FBEAM are not passed through. This includes minor axis loading, axial loading, trapezoidal loads and torsion.
6. Imposed loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer to represent live loads during construction e.g. due to 'heaping'. These are exported to FBEAM as Load Type 'CL'. They are given a load factor that will be the maximum from all imposed loadcases included in the Construction Stage combination although typically only one is included and with a load factor of 1.5.
7. Beam self weight loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer. Normally these would be represented by the Load Type ' D' in FBEAM. However, in order to avoid taking into account all of the dead loads, including those at composite stage, the Construction Stage load combination in FBEAM is populated with the beam self weight as a 'cladding' load, 'C'.
8. The Slab Wet loads exported from Tekla Structural Designer should set the 'Construction Stage Floor type' in FBEAM to 'CL' - typically this will ensure that a load factor of 1.5 is used in the design combination for Construction Stage. For EC this is correct but for BS it should be set to 'D'. This setting is not accessible to the export from Tekla Structural Designer and so must be changed manually if BS 5950 design is to be used.
9. For composite beams when the wet weight of the concrete is included in the Construction Stage combination in Tekla Structural Designer, the results in FBEAM will only be correct in the following circumstances:

a. Automatic Loading is checked on in the loadcase dialogue when Slab Dry is selected as the load Type,

b. Automatic Loading is checked on in the loadcase dialogue when Slab Wet is selected as the load Type,

c. At least on combination includes the Slab Dry loadcase,

d. The Construction Stage combination includes the Slab Wet loadcase.

10. Beams can be rotated in Tekla Structural Designer but in FBEAM are always assumed to have their web vertical i.e. unrotated. Consequently any beam rotation is not passed through from Tekla Structural Designer to FBEAM and any loads in the global axis system that are applied to a rotated beam are passed through to FBEAM as if they are applied through the vertical web of the beam.

11. When several beams on the same floor are exported from Tekla Structural Designer there can be several instance of 'Floor' in FBEAM. Each of these could be assigned a different size of 'Mesh'. However, only one of these will be imported into Tekla Structural Designer. It is recommended that either only one value of mesh is set for these beams or that any required changes to the slab mesh are made in Tekla Structural Designer and not FBEAM.

12. In composite design, transverse reinforcement whether in the slab or as part of the beam data is exported and imported as an area only. In Tekla Structural Designer the area is derived from the bar size and spacing (or mesh size). In the case of bars, the export transfers the area and in FBEAM the default bar size will be shown with the exported area. For import the area in FBEAM is processed by Tekla Structural Designer to find a bar size and spacing that has an area greater than (or equal to) the area held in FBEAM. Furthermore when 'loose bars' are used in the slab (unusual) these are added to any from the beam data and transferred as one value to FBEAM. Clearly on import these cannot be separated. This approach has an impact on round tripping that the user must be aware of and make appropriate adjustments post import or export.

13. Partial length shear connector layout with empty segments cannot be created in Tekla Structural Designer. If cases like these are detected when importing from FBEAM the partial shear connector layout will be extended for the full length of the beam in Tekla Structural Designer.

14. Web opening stiffeners applied only to one side of a beam are always set to the right hand side when importing into FBEAM. When importing from FBEAM to Tekla Structural Designer, the existing stiffeners in Tekla Structural Designer keep their side regardless of how they were modeled in FBEAM.

Export to FBEAM

Before exporting your FABSEC® beams, you should ensure the analysis has been performed.

1. On the **BIM Integration** tab, click **FBEAM Export**.

   The **FBEAM Export** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to FBEAM, and click **Next**.
3. Select the combinations you want the beams designed for, and click **Next**.
4. Select the location to save to, name the file, and click **Next**.

   The export status of the selected beams is displayed.
5. Click **Finish**.

You can now go in to FBEAM and import the file that you have just created into an existing or new project, so that the beams can be designed.

Import from FBEAM

NOTE In order to import FABSEC® beams, you must have previously created them in your Tekla Structural Designer model, and have exported them to FBEAM. When you do this, each beam is given a unique identifier. After the beams have been designed in FBEAM you must then choose the Tekla Structural Designer XML option to export them to a file. This enables Tekla Structural Designer to know to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **FBEAM Import**.

   The **FBEAM Import** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the file that you want to import.
3. Click **Next**

   A list of the FABSEC® beams in the file is displayed from which you can choose the beams to be imported.
4. Select the beams that you want to import.

   You can only select beams to import that correspond to existing beams in the Tekla Structural Designer model.
5. Click **Next**

   The import status of the selected beams is displayed. If any errors have occurred, hovering the cursor over the error displays the reason in a tooltip.

6. Click **Finish**.

   Provided that Tekla Structural Designer recognizes the beams, the details in the file are imported to the associated beam properties, (including in particular the beam UDA properties) in your model.

Review the imported beams

When FABSEC® beams are imported, User Defined Attributes are added to the beam properties to record import status and fire design details. These can be reviewed graphically.

To review the FABSEC® beam UDAs graphically:

1. If necessary, change the view regime (page 280) to a **Review View**.
2. On the **Review** tab, click **UDA**.
3. In the **Properties** window, with **[M]ode** set as **Review**, change the **Attribute** to **Import Status**.

   Each FABSEC® beam is color-coded to represent the import status that applies.

Export a model to ADAPT

Before exporting to ADAPT, read the following limitations. Then, you can proceed to follow the detailed instructions to perform the export.

Limitations

The scope of the ADAPT Export is constrained by the fact that some geometries, configurations and properties possible in Tekla Structural Designer are beyond the scope of ADAPT. This imposes some unavoidable limitations on the Export at this time, for example:

- Only concrete and steel materials are exported.
- Only horizontal beams and slabs are exported.
- Roof and wall panels are not supported.
- For concrete walls, the following are not supported (all of which may be in a Tekla Structural Designer); wall openings, non-rectangular walls, wall extension, walls with top ends not at a level.

NOTE Both loads as well as reactions are linked to their reference member/object in Tekla Structural Designer. This means that any loads or reactions applied to a member that are beyond scope of the export will not be exported either.

Details of what gets exported and what is beyond scope are as follows:

Units	• Units are SI in the export file
Levels	• Number of levels Included • Level height Included
Gridlines	• Beyond scope (Adapt import)
Materials	• Concrete: Normal weight Included • Structural Steel Included • Concrete: Lightweight Beyond scope (Adapt import) • Mild Steel (Rebar) Beyond Scope (Tekla Structural Designer export) • Timber Beyond Scope (Tekla Structural Designer export) • Prestressing Beyond Scope (Tekla Structural Designer export) • Other / generic Beyond Scope (Tekla Structural Designer export)
Load combinations	• All active combinations are included apart from Seismic Inertia which is beyond scope
Loadcases	• All loadcases are included apart from EHF/NHF which is beyond scope
Beams	• Regular and irregular sections are Included • Beam at level Included • Export by span Included • Horizontal offsets Included • Vertical offsets Included • Tekla Structural Designer Labelling Included • Beam not at level Beyond Scope (Tekla Structural Designer export) • End trimming at column interface Beyond Scope (Tekla Structural Designer export) • Sloping beams Beyond scope (Adapt import)

	• Axially rotated beams Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Columns	• Regular and irregular sections are Included • Export by stack Included • Axially rotated columns Included • Tekla Structural Designer labeling Included • Tilted columns Included • Multi-stack columns Included • Columns with holes Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Walls	• Concrete meshed walls Included • Concrete mid-pier walls Included • Export by stack Included • Tekla Structural Designer labeling Included • No internal wall members Included • Multi-stack walls Included • Bearing walls Included (exc. reactions) • Wall alignments Included • Slopping walls Beyond scope (Adapt import) • Wall openings Beyond scope (Adapt import) • Wall extensions Beyond Scope (Tekla Structural Designer export) • Non-rectangular walls Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Slabs	• Concrete slab on beams Included

	• Concrete flat slabs Included • Export by panel Included • Tekla Structural Designer labeling Included • Vertical offsets Included • Slab overhangs Included • Non-concrete slabs Beyond Scope (Tekla Structural Designer export) • Slopping slabs Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Drop panels	• Concrete slab drops Included • Trimming at slab edges Included • Reinforcement Beyond scope (Adapt import)
Openings	• Slab rectangular openings Included • Slab circular openings Included
Braces	• Beyond Scope (Tekla Structural Designer Export)
Loading	• Self weight Included • User applied Force Included • User applied Moment Included • Level Point loads Included • Horizontal Line loads Included • Level area loads Included • Offsets from reference level Beyond scope (Adapt import) • Line and Area Loads applied to horizontal plane Beyond scope (Adapt import) • Vertical line loads applied as point loads at levels
Releases	• Translational releases Beyond Scope (Tekla Structural Designer Export)

	• Torsional releases Beyond Scope (Tekla Structural Designer Export)
Stiffness modifiers	• Beyond Scope (Tekla Structural Designer Export)

Instructions

1. On the **BIM Integration** toolbar, click **ADAPT Export**.

   This shows a dialog which gives the name of the ADAPT import file which Tekla Structural Designer will create. You can change the name and location of the file if necessary.
2. Once the file details are correct click **Save** to create the ADAPT import file.
3. Launch ADAPT and import the file to see your project.

Export a model to STAAD

To export a model to STAAD, see the following limitations and instructions.

Limitations

The scope of the STAAD Export is constrained by the limitations of the STAAD Text File (.std) format.

Note that:

- Elastic extensions cannot be defined in STAAD. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference STAAD library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference STAAD library materials.
- Objects such as Shear Only Walls have no equivalent native object in STAAD and some of the internal data such as panel geometry and spring properties end up lost during conversion.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **STAAD Export**.

   The **Save As** dialog opens.
3. If necessary, change the name and location of the file.

4. Click **Save**.
5. Start STAAD and open the file.

Export a model to Autodesk Robot Structural Analysis

Before you export a Tekla Structural Designer model to Autodesk Robot Structural Analysis, read the limitations of Robot Export in the following paragraphs. Once you have read the limitations, you can proceed to the detailed instructions on how to export models from Tekla Structural Designer to Autodesk Robot Structural Analysis.

Limitations

The scope of the Robot Export is constrained by the limitations of the Robot Text File (STR) Format.

Note that:

- The model exported is the one used for 1st order linear static analysis (see <Linear analysis of structures containing material non-linearity> to remove non-linearity from the exported model.)
- 1D element springs are totally omitted from the export.
- Elastic extensions cannot be defined in Robot. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference Robot library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference Robot library materials.
- Loads in projection are converted to equivalent loads in Robot.
- Robot will not import the material properties for timber from the STR file. When the file is opened in Robot, you can delete the G value and adjust the values manually after import.
- It is not possible to define part-length distributed torsional loads in the STR file. Therefore, they are converted to equivalent full-length distributed torsional loads.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Robot Export**.

   The **Save As** dialog opens.

3. If necessary, change the name and location of the file.
4. Click **Save**.
5. Start Robot and open the file.

Export a model to the cloud

To export a Tekla Structural Designer model to the cloud, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Cloud Export**.

   The **Save As** dialog opens.
3. If necessary, change the file name and location.
4. Click **Save**.

Export to One Click LCA

One Click LCA is third party software for performing a life cycle assessment of your structure and optimizing its embodied carbon.

From Tekla Structural Designer you can either compile an *offline* report, which is a compact summary of materials in the structure that can be edited if required before being manually imported into One Click LCA, or you can upload the report data directly to One Click LCA *online*.

Overview

Scope

Tekla Structural Designer includes material information in tabular data tables, in general the scope of the exported data matches this. This gives a lot of the most commonly requested material quantity information, but is not considered to be exhaustive:

- Data available in Tekla Structural Designer but not exported
  - Shear studs on composite beams
- Data NOT available in Tekla Structural Designer (and therefore not exported)
  - Reinforcement in any slab other than cast-in-place reinforced concrete slabs
  - Bearing Walls

    - Shear Only Walls
    - Transverse shear reinforcement required by composite beams
    - surface area of timber members, cold rolled members, general material members
    - DELTABEAM® mass - a zero value is exported to help ensure allowance is added manually
    - analysis elements
- Exported Data not otherwise available in Tekla Structural Designer
    - Precast slab screed/topping area
    - Composite slab decking type / area
    - material in gable and parapet posts
    - surface area of defined “roof panels” and “wall panels”

The above summary is not guaranteed to be exhaustive, it is important to consider “what is missing”? Are all the significant physical items you defined in Tekla Structural Designer being covered by the export?

Additional information

The following notes may assist with initial use of the export:

- Grouping in One Click LCA

  On import there is a phase where it is suggested that lots of the exported data items can be grouped together on the basis of a shared “IFCMATERIAL”. The sections below give examples where you may consider the value of doing this:

    - composite / non-composite beams

      the mass of these beam types are exported separately.

      you may wish to apply higher carbon factors to composite beams to allow for shear studs and transverse shear reinforcement.

    - steel beams of all different fabrication types

      the mass of steel members of different fabrication types are exported separately.

      e.g. rolled, plated, Westok, FABSEC®, DELTABEAM®

      you may wish to apply different carbon factors to these different types of member

    - surface areas

      the surface areas of slabs and many beams are all exported separately.

      if you have no requirement to use any surface area information then it will be easier to leave these grouped for simpler deletion.

- Slabs
  - The data for each slab is exported separately (a slab may contain many slab-items).

    In simple situations this will mean one group of data for each level, but if multiple slabs are defined in a level you will get separate data for each.
  - Data for different types of slab is also kept separate

    e.g. Cast-in-place reinforced slabs, post-tensioned slabs, precast slabs, composite slabs, various other general material slabs

    For reinforced concrete slabs this allows a more detailed understanding of reinforcement per slab.
  - Limitation - as noted above reinforcement data is not available/ exported for other slab types, this will need to be allowed for in one of two ways:
    - Use the surface area information to add an allowance per m2
    - Adjust the carbon factor or concrete/topping in the slabs to allow for reinforcement content

    In either case this will mean avoiding grouping of the relevant data items.
- “roof panels” and “wall panels” surface areas
  - Roof and wall surface area data is included in case it assists with the inclusion of cladding materials.

Feedback

- Please send us feedback if you feel that:
  - Important data is missing
  - Included data should be totalled/presented differently - perhaps there should be options to do things in different ways (possibly at different stages of design?)
  - There should be options to filter out certain data types because you don’t always want to use them (for example surface area info)

Show report

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Click **Show report** The report opens in Excel, (or whatever software is associated with .xlsx files).

The resulting materials file can then be edited if required before being manually imported into **One Click LCA**, or it can be passed on to a third party and imported into **One Click LCA**.

Show online results

NOTE Online export requires a One Click LCA license.

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Enter your One Click LCA user name and password, then click **Log in**.
3. Click **Show online results**. The material data from the Tekla Structural Designer model is uploaded to One Click LCA.
4. Proceed through the One Click LCA wizard:
   a. Create/Choose project
   b. Consider groupings
   c. Review / apply mapping

   On completion the amount of embodied carbon in the model is determined.

Export to IDEA StatiCa Connection Design

Before exporting connections to IDEA StatiCa, you are advised to be aware of the limitations. Then, you can proceed to follow the detailed instructions to perform the export.

NOTE Only valid connections listed in the Project Workspace Connections (page 273) tree can be exported.

Limitations

NOTE The link is compatible with the latest versions of IDEA StatiCa 9.1 and 10.0; it is also compatible with IDEA StatiCa 10.1 (build 10.1.113), but may not work with earlier builds of 10.1.

The following geometrical limitations should be noted.

- Connection checks to BS 5950 are not supported.
- For connections identified as 'Moment Connections' using Update connections in Tekla Structural Designer any out of plane members are not exported.

- For any connections identified in Tekla Structural Designer the brace members are not passed through to IDEA.
- For any connection type, haunches are not exported.
- Tekla Structural Designer uses ToS and the wire model is at this level too. The export to IDEA depicts the ToS so that the 3D graphic looks correct. However, the wire model typically connects at offsets of half the beam depth. This introduces additional forces/moments due to eccentricity of line of action etc. This line of action of the force set can be adjusted in IDEA.

Instructions for the export to IDEA StatiCa

NOTE Export requires an IDEA StatiCa licence.

1. Right-click the connection in a Scene View.
2. Choose Export Connection to IDEA StatiCa from the right-click menu, selecting the analysis results to use at the same time.
3. Specify the file name and location for this connection.

   The IDEA StatiCa application opens to allow the connection to be designed, the following data having been transferred:

   - Connection geometry
   - Section profile & material grade
   - Connection forces for each active solved Tekla Structural Designer combination.

   NOTE The Tekla Structural Designer model remains frozen until you close the IDEA StatiCa connection.

4. Add bolts, stiffeners etc as required and design the connection.

When you close the connection Tekla Structural Designer becomes active once more; the connection file is automatically embedded in the model.

Review of IDEA connections designed in Tekla Structural Designer

1. Within Tekla Structural Designer click Review>UDA
2. In the Properties Window set Mode to Review and Attribute to IDEA StatiCa file

   Joints associated with IDEA StatiCa are shown

3. Use the right-click context menu to open previously exported connections back to IDEA StatiCa.

Related video

Integration with IDEA StatiCa Connection design

4 Create models

To get started with modeling in Tekla Structural Designer see:

- Get to know Tekla Structural Designer basic working methods (page 347), for a few first principles
- Create the model (page 366), to learn how to create and modify model objects

Once you are comfortable creating objects, see:

- Edit the model (page 494), to get familiar with the model editing commands
- Check the model (page 512), for validation and measuring commands
- BIM integration (page 300), to learn how to exchange model data between applications

See also

Create and design foundations (page 836)

BIM integration (page 300)

4.1 Get to know Tekla Structural Designer basic working methods

Before creating real models, get to know some basic methods and techniques that will help you to work efficiently.

We recommend you familiarize yourself with how to:

- work with grids and construction lines (page 380)
- define construction levels (page 367)
- zoom and rotate (page 348) the model
- create (page 366), select (page 350) and edit the properties of (page 360) entities

- re-position entities (page 361) by moving nodes or edges
- copy, move and mirror (page 494) part of, or all of the model

Zoom, pan, rotate and walk through the model

You can use the mouse to manually zoom, pan, or rotate the model. Additionally, you can rotate the model by using the **ViewCube**, and walk through your model in 3D views.

Zoom in and out, or zoom extents

- According to your needs, do one of the following:

To	Do this
Zoom in or out	• Scroll in or out with the middle mouse button.
Zoom to the center of the visible objects	• Right-click anywhere within the view. • In the context menu, select **Zoom Out**. **NOTE** You can also use the keyboard shortcut **ZA** to zoom extents.

Pan the view

- Hold down the middle mouse button and drag the view according to your needs.

Rotate the view manually

You can use the **ViewCube** to adjust the view. However, if none of the standard **ViewCube** are appropriate, you can also rotate the model manually.

- Hold down the right mouse button and drag the view according to your needs.

Adjust the view with the ViewCube

1. Move the mouse pointer over the **ViewCube**.
2. According to your needs, do one of the following:

To	Do this
Display one of the eight isometric views	• Click the required vertex on the **ViewCube**.

<table>
<tr><td></td><td>

TIP If the required vertex is not visible, click one of the other vertices to spin the **ViewCube** until you can see the required vertex.</td></tr>
<tr><td>Display one of the twelve edge views</td><td>

- Click the required edge on the **ViewCube**.

TIP If the required edge is not visible, spin the **ViewCube** by clicking the vertex that is adjacent to the required edge.</td></tr>
<tr><td>Display one of the six face views</td><td>

- Click the required face on the **ViewCube**.

TIP If the required face is not visible, spin the **ViewCube** by clicking the vertex adjacent to the required face.

NOTE When a face view is displayed, the additional **ViewCube** controls appear:

- If the required face is not visible in a face view, roll the **ViewCube** by</td></tr>
</table>

	clicking one of the triangular controls.

Walk through the model in a 3D view

- On the **Home** tab, click **Walk**.

NOTE In the **Walk** mode:

- To move forward, backward, left or right, use the arrow keys.
- To move up and down, use the **Q** and **Z** keys.
- To rotate the view, hold down the right mouse button and drag the view.
- To exit **Walk** mode, press **Esc**.

Display a 2D view in 3D

1. In a 2D view, click the **2D/3D** toggle button, located on the bottom right corner of the window.

Select entities

Many Tekla Structural Designer commands require you to make selections. You can make single and area selections from any 2D or 3D view. You can also make selections from the **Project Workspace**.

TIP Use (page 255) to display everything *not* in the selection as ghosted, this makes it easier to focus on the selection.

TIP If planar objects, such as slabs, roofs and area loads, obscure what you want to select, you can hide them using the (page 259) window.

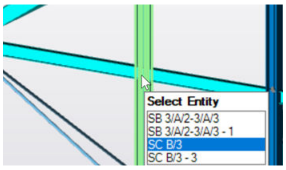
Use the different methods listed below to select the entities.

Select single entities

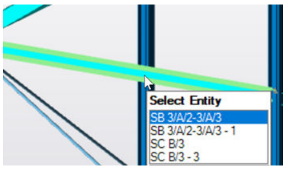
1. In a 2D or 3D view, move the mouse pointer over the desired entity.

   If the entity is the only one at the location, it becomes highlighted and its name is shown in the **Select Entity** tooltip.

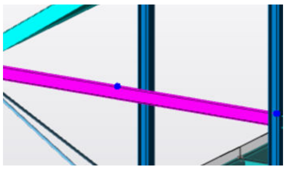
   If several entities are located at the same location, their names are all listed in the **Select Entity** tooltip.



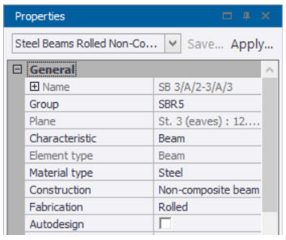
Use the up and down keys to scroll through the list to reach the desired entity.



2. When the desired entity is highlighted in the **Select Entity** tooltip, press **Enter** to select it.



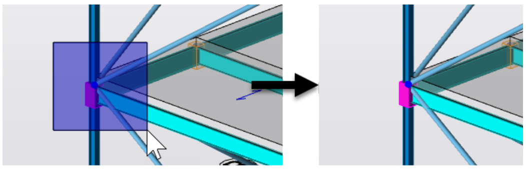
The selected entity is then displayed in the *active selection color*, the nodes for positioning it are exposed, and the entity properties are displayed in the **Properties** window.



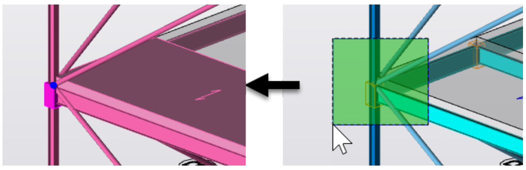
Select multiple entities using area selection

1. You can select multiple objects using area selection. The dragging direction affects the selection of objects.

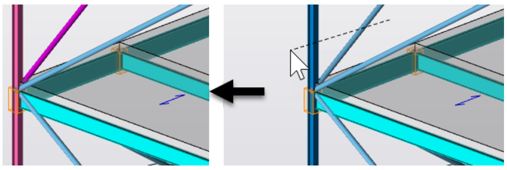
   a. To select all entities that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



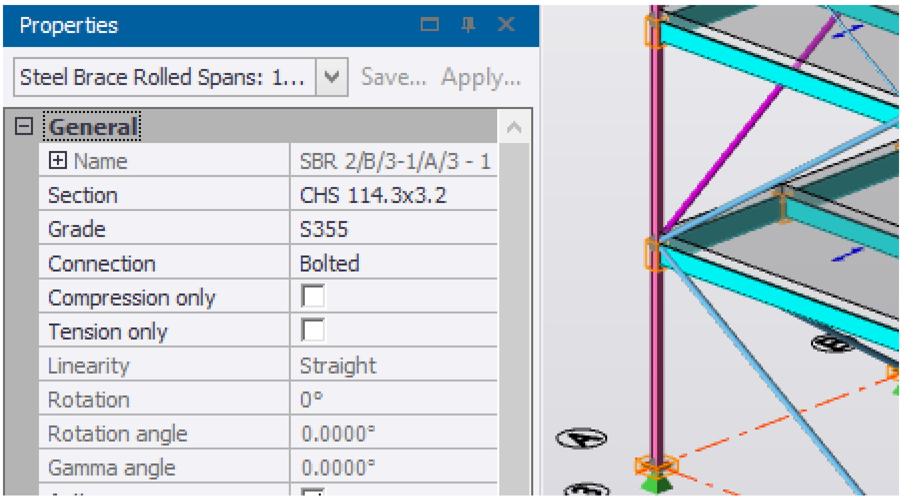
b. To select all entities that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.



c. To select all objects that are at least partly crossed by a straight line, hold down the **shift key**, then hold down the left mouse button and drag the mouse to draw the line.



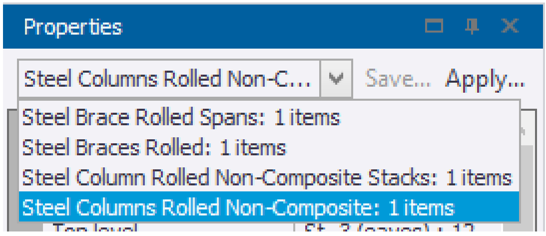
After selection, one entity type is displayed in the *active selection color* and has its properties shown in the **Properties** window.

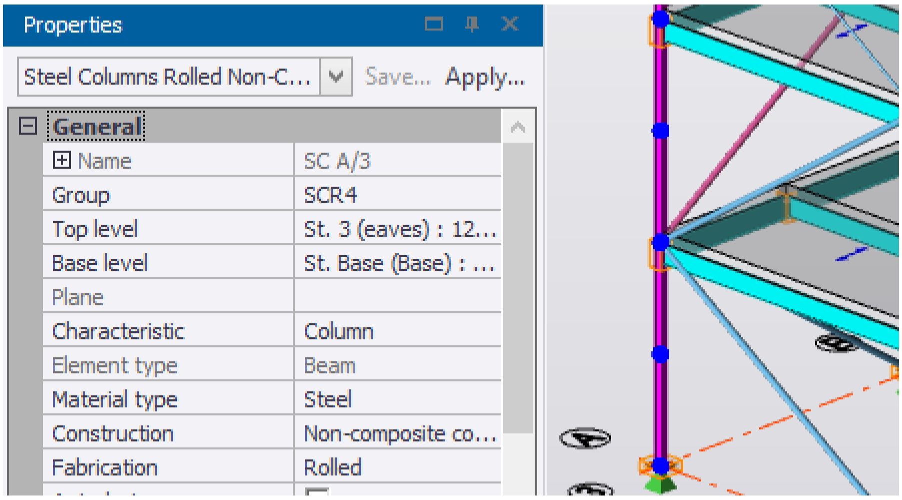


Any other entity types in the selection are displayed in the *local selection color*.

NOTE You can change the *active* and *local* selection colors by clicking **Home > Settings > Scene > Colors > Selection**.

2. To display the properties of another entity type in the selection, choose it from the droplist at the top of the **Properties** window.





NOTE If there is only a single item in the active selection, all its properties are shown; if there are multiple items in the active selection, only common properties are shown.

Select using Find

1. Click **Find** on either the **Home** tab, or the **Quick Access** toolbar.

   The **Find** dialog opens.

2. In the **Find** box type any of the following, in whole, or in part:

   a. An object 'Type'

   b. An object 'Name'

   c. An object 'Index'

   As you type, any objects that match the criteria are listed in the table below.

3. When the object(s) that you require are shown, click to highlight them as follows:
   a. Single click to highlight a single object
   b. Ctrl+click to select multiple non-sequential rows
   c. Shift+click to select multiple sequential rows Once the object(s) are highlighted, **Select** becomes available.
4. Click **Select** to locate the object in the active view, zoom in, and display its properties in the **Properties** window.

NOTE If the object doesn't exist in the active view a new view containing the object will become active.

TIP Sometimes when clicking **Select** the object being located is obscured from view by other objects. By leaving (page 255) checked this is avoided, as all objects other than those being found are made semi-transparent.

NOTE If not using Ghost Unselected, you can change the color used to highlight the found entity by clicking **Home > Settings > Scene > Colors > Selection** and changing the **Global - User** color.

Select from the Project Workspace

1. Do one of the following:

To	Do this
Select individual members from the **Structure** tree	a. Expand the **Members** branch and any appropriate sub branches until you can click the required member. b. Right-click the member name. c. In the context menu, select **Select in visible views**. The member's properties are viewed in the **Properties** window.
Select member groups or individual members from the **Groups** tree	a. Expand the appropriate branches until you can click the required group or member name. b. Right-click the member or group name. c. In the context menu, select **Select in visible views**. The member's or group's properties are viewed in the **Properties** window.

Select nodes

Sometimes you need to select only the nodes of an entity, for example when moving a beam.

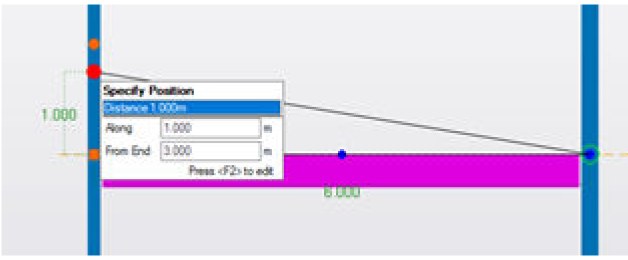
1. In a 2D or 3D view, select the entity.

The entity is selected and any nodes associated with the entity are displayed.

2. Move the mouse pointer over the desired node.

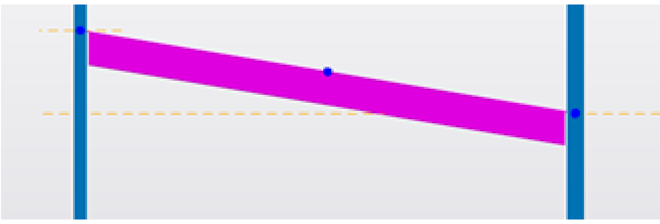
The node becomes highlighted and appears in the **Select Entity** tooltip.

3. Press **Enter**.



You will now be prompted to pick a point to move the node to.

4. Click at a new location to reposition the node.



The entity is updated accordingly.

See also: Re-position entities by moving nodes or edges (page 361)

Modify the selection

You can add or remove entities from the current selection.

1. Do one of the following:

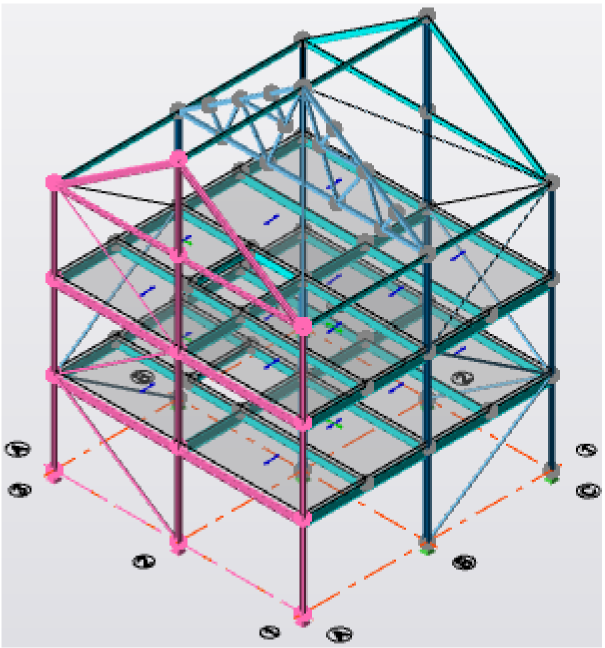
To	Do this
Add an entity to the current selection	a. Hold down the **Ctrl** key. b. Click the entity that you want to add to the selection. The properties of the newly selected entity are added to the **Properties** window.
Remove an entity from the current selection	• Click the entity you want to de-select. The entity's properties are removed from the **Properties** window.
Remove multiple entities	a. Do one of the following: • To de-select only the entities that are encompassed by the box, move the mouse pointer to the left corner of the imaginary box that will encompass the entities. • To de-select the entities that are encompassed by the box and that it crosses, move the mouse pointer to the right corner of the imaginary box that will encompass the entities. b. Hold down the **Ctrl** key and drag the mouse pointer to the opposite corner of the box. A rectangle on the screen shows the area that you are selecting. c. Release the mouse button and the **Ctrl** key. The entities are removed from the **Properties** window.
Remove a beam/column/wall and simultaneously remove its constituent spans/stacks/panels	If a continuous beam and its constituent spans are selected, they can all be deselected with a

	single mouse-click while holding down the **Ctrl** key. The same method can be applied to a continuous column and its constituent stacks, or a wall and its constituent panels.
Remove all entities	• Press **Esc**. The information displayed in the **Properties** window is cleared.

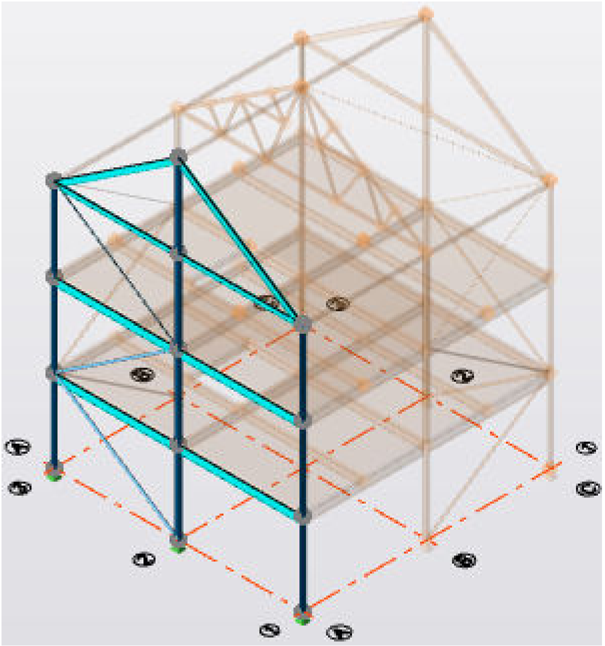
Use Ghost Unselected to focus on the selection

You can toggle **Ghost Unselected** on and off using the S button in the bottom corner of the view.

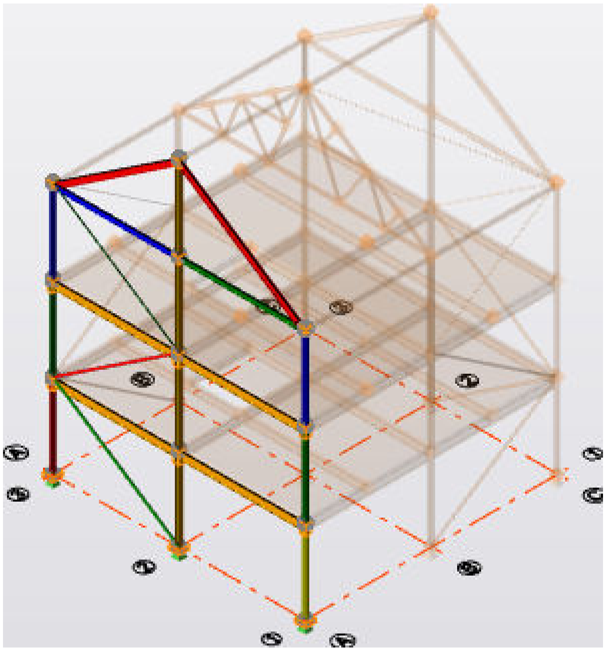
- **Ghost Unselected off**: selected entities are displayed in the *active selection color*, this can make it hard to distinguish between individual entities in the selection.



- **Ghost Unselected on**: selected entities are displayed in their nomal colors (appropriate to the Scene View that is active), unselected objects are semi-transparent. If nothing is selected, everything is displayed in its normal color.
    - In a Structure View, the selection would be displayed as:



- In a Review View showing Design Status, the selection would be displayed as:



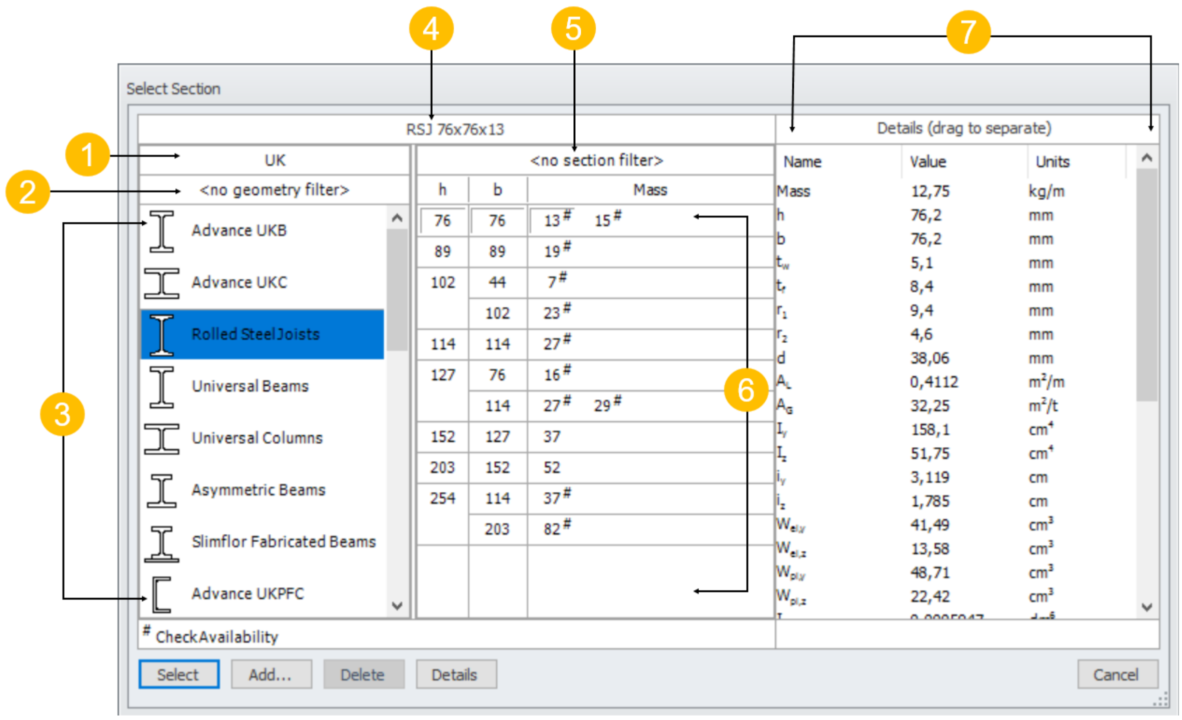
Select a section in the Select Section dialog box

Select Section dialog box allows you to select sections to use them in your model. You can also add and delete user-defined sections in the database by using the dialog, if necessary.

To access the **Select Section** dialog box:

1. On the **Model** tab, select the member you want to design.
2. In the , click the arrow on the right side of the **Section** field.
3. Click **<New\Edit...>**.

   The **Select Section** dialog box opens.



1. **Country** list
2. **Geometry filter**
3. **Page** pane
4. Selected section
5. **Section filter**
6. **Item** pane
7. **Details** pane

1. In the **Country** list, select the desired country.
2. In the **Page** pane on the left, select the desired geometry.

NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.

3. In the **Item** pane on the right, select the desired section size.
4. Click **Select**.

Edit entity properties

Any selection of one or more entities can be edited using the **Properties** window. Individual **model objects** can also have their properties edited using the **Properties** dialog box.

Edit properties using the Properties window

All entities can have their properties edited using the **Properties** window

1. Select the entity either in the graphical display or in the **Structure** tree.

   For more information, see Selection methods (page 350).
2. In the **Properties** window, modify the properties according to your needs.

Edit properties using the Properties dialog box

Individual **model objects** can have their properties edited using the **Properties** dialog box

1. Hover the mouse pointer over the object that you want to edit.
2. In the **Select Entity** tooltip, select the object.

   For more information, see Selection methods (page 350).
3. Right-click the object.
4. In the context menu, select **Edit**.

   The **Properties** dialog box opens.
5. Modify the properties according to your needs.
6. Click **OK**.

Edit properties of multiple entities

NOTE You can view and modify the common properties of multiple entities in the **Properties** window.

Note that:

- If the value of a certain item is not identical for all the selected entities, the cell appears blank in the **Properties** window.
- If you modify the blank cell, Tekla Structural Designer applies the new setting to all the selected entities.
- If you have selected multiple entities of different types, the property information is displayed separately for each type. Use the list at the top of the **Properties** window for moving between types.

1. Select the entities in the graphical display.

   For more information, see Selection methods (page 350).
2. If you have selected entities of different types, select the desired type in the list on top of the **Properties** window.
3. In the **Properties** window, modify the properties according to your needs.
4. Repeat steps 2–3 for entities of different types.

Re-position entities by moving nodes or edges

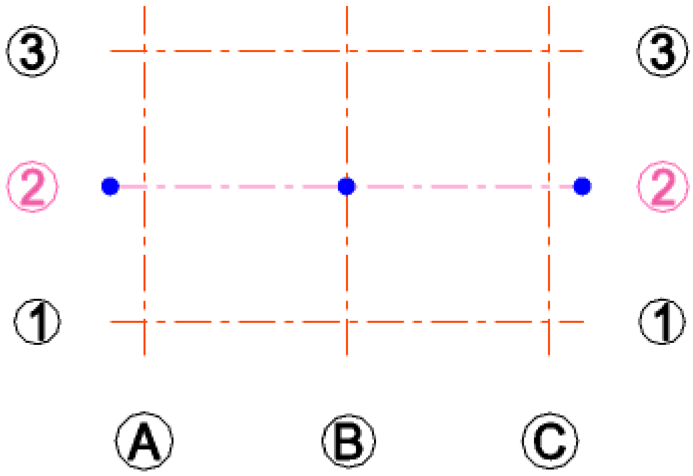
After selecting an entity you can re-position it by moving nodes or edges.

Modify one end of a grid or construction line

Grid and construction lines can only be modified in 2D views.

1. Select the grid or construction line.

The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

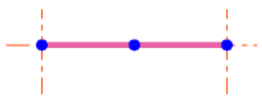
   Tekla Structural Designer redraws the line, moving the end node to the selected point.

Move a grid or construction line

Grid and construction lines can only be moved in 2D views.

1. Select the grid or construction line.

   The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

   Tekla Structural Designer redraws the line, moving the end node to the selected point.

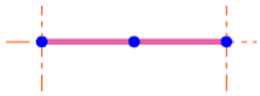
Modify one end of a member

To move an entire member, see: Move objects (page 495).

To move just one end of a member, proceed as follows:

1. Select the member in a 2D or 3D view.

   The nodes at each end and at the mid-point become visible.

2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

   ---

3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the member, moving the end node to the selected point.

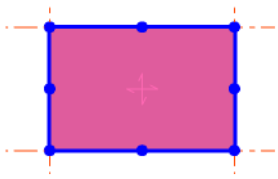
Modify slab items and panels by moving a node

To move entire slab items and panels, see: Move objects (page 495).

To move a slab/panel node, proceed as follows:

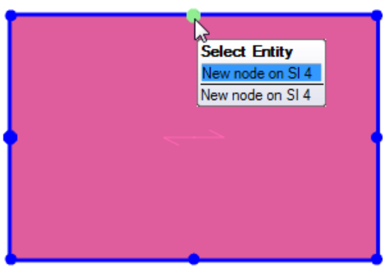
1. Select the slab item or panel that you want to modify in a 2D or 3D view.

   The nodes at each vertex and at the mid-point of each edge become visible.

2. Click a node to select it.

   You can select either corner nodes, or nodes at the mid-points of the edge.

3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

NOTE When you reposition nodes, note that:

- The new node position must be in the same plane as the slab item or panel.
- When you move a node at the mid-point of an edge, it becomes a new corner node. Tekla Structural Designer automatically creates two new mid-point nodes.
- When you move a corner node directly over an adjacent corner node, Tekla Structural Designer deletes the original corner node.

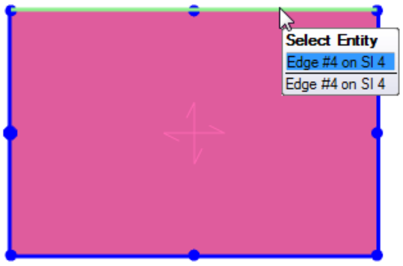
Modify slab items by moving an edge

To move entire slab items and panels, see: Move objects (page 495).

To move a slab/panel edge, proceed as follows:

1. Select the slab item that you want to modify in a 2D or 3D view.
2. Click an edge to select it.

   Ensure that you have selected an edge, and not a node.



3. Move the mouse pointer over an existing construction point or intersection.
4. Click to reposition the edge so that it passes through the selected point.

NOTE The new edge position must be in the same plane as the slab item.

Modify walls by moving a node

To move an entire wall, see: Move objects (page 495).

To move a wall node, proceed as follows:

1. Select the wall that you want to modify in a 2D or 3D view.

   The nodes at each vertex become visible.

2. Click a node to select it.
3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

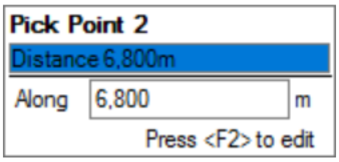
   NOTE The new node position must be in the same plane as the wall.

Tips for basic tasks

Here we provide useful hints and tips that help you use the Tekla Structural Designer user interface and its basic features more efficiently.

Use the tooltip for input in a command

When you need to specify a distance or pick a point, a tooltip appears, displaying the current value.



1. Do one the following:
   - Move the mouse pointer to the required point and click the point to use it.
   - Press **F2** to type an exact number, and press **Enter**.

Undo a command

1. In the **Quick Access** toolbar, click  **Undo**.

Cancel a command or go back to the previous prompt

Yellow command prompts are viewed on the top of the active view to guide you through each command. To react to command prompts, see the following instructions.

- Do one of the following:

To	Do this
Cancel a command	• Press **Esc** at the first command prompt.
Go back to the previous command prompt	• Press **Esc** at a subsequent command prompt.

TIP Selecting another command also cancels the current command.

Example

The parallel grid line command consists of four steps. Hence, four consecutive command prompts are displayed.

1. Add Parallel Grid/Construction Line: Select Reference Line <press ESC to cancel>
2. Add Parallel Grid/Construction Line: Specify Distance **<press ESC to cancel>**
3. Add Parallel Grid/Construction Line: Pick Point 1 **<press ESC to cancel>**
4. Add Parallel Grid/Construction Line: Pick Point 2 **<press ESC to cancel>**

The grid line is created after the fourth prompt. After that, prompt 2 is automatically redisplayed.

At this point, you can either continue to create additional lines parallel to the original, or go back to the first prompt by pressing **Esc**.

Then, you can either select a new reference line and continue again, or cancel the command by pressing **Esc** again.

4.2 Create the model

When you model in Tekla Structural Designer, you create and work with different types of model object. In most cases, a model object represents a member type that will exist in the real building or structure, or that will be closely related to it. You will also work with modeling aids such as grids and construction lines, that represent information that is only relevant when you are creating the model.

You can use the following modeling aids:

- Construction levels (page 367)
- Grids (page 370), construction lines (page 380) and free points
- Frames and slopes
- Dimensions (page 388)

- DXF reference drawings (page 377)

You can create the following physical member/object types:

- Beams, columns and braces (page 388)
- Walls and cores (page 425)
- Slabs and decks (page 448)
- Trusses and steel joists
- Portal frames
- Cold rolled members

The following panels/objects are useful for applying loads to the model:

- Wall panels and roof panels
- Ancillaries (page 474)
- Inactive members (page 482)

You can also use the following general members/objects when required:

- Supports (page 490)
- Analysis elements (page 493)

Create and manage construction levels

In the **Construction Levels** dialog box, you can define and modify the levels that you need to construct your model. The levels can be floors, roofs, or intermediate levels required to define specific items.

Open the Construction Levels dialog

- Do one of the following:
    - On the **Model** tab, click **Construction Levels**.
    - In the **Structure** tree, double-click **Levels**.

Insert a single construction level

1. In the **Construction Levels** dialog box, select an existing level.
2. According to your needs, click either **Insert Above** or **Insert Below**.
3. If necessary, change the level type:
    - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).

- If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).

4. If necessary, change the level name.
5. For the new level, do one of the following:
   - Specify the height above the base in the **Level** field.

     The inter-storey spacing is calculated automatically.
   - Specify the inter-storey spacing in the **Spacing** field.

     The level is calculated automatically.

   NOTE A default height is calculated for the new level based on the spacings of any existing levels immediately above or below it. Either accept the height, or adjust it as required.

   ---

6. If you want Tekla Structural Designer to treat the level as a floor, select the **Floor** option.

Insert multiple construction levels

Multiple levels can be inserted, either above the current top level, or below the current bottom level.

1. In the **Construction Levels** dialog box, select an existing level.
2. According to your needs, click **New on Top** or **New at the Bottom**
3. For each construction level, type the level spacing and click **OK**.

   TIP If you have new levels at 12' 6" , 18' 6" , 27' 6" , 36' 6" and 27" above the current level, you can specify the levels as 12' 6" , 18' 6" , 3*9'.

   ---

4. If necessary, change the level names.
5. If necessary, change the level type:
   - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).
   - If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).

Make a level an identical copy of another level

If needed, you can determine a level as an identical copy of another level. In this case, all changes made to either the source level or the identical level are automatically applied to both levels. To make a level an identical copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.

3. Select the level that you want the current level to be identical to.
4. Click **OK**.

Make a level an independent copy of another level

If you determine a level as an independent copy of another level, any changes made to each level only apply to that level. To make a level an independent copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.
3. Select the level that you want the current level to be identical to.
4. Click **OK**.
5. Reopen the **Construction Levels** dialog box.
6. At the same level, click the arrow in the **Source** column again.
7. Set the level as **-unique-**.
8. Click **OK**.

Modify the properties of a construction level

You can modify some properties of construction levels in the **Construction Levels** dialog box dialogs, but other parameters can only be edited in the **Properties** window. For more information, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to **Levels**.
2. Click the name of the construction level that you want to modify.

   The properties of the level are viewed in the **Properties** window.
3. In the **Properties** window, modify the level properties according to your needs.

Delete construction levels

To delete construction levels, see the following instructions.

WARNING	Deleting a construction level completely deletes the entire level and all its associated details, such as beams, members, or slab. You cannot recover the level once you have deleted it.

1. In the **Construction Levels** dialog box, select the level that you want to remove.
2. Click **Delete**.
3. Click **OK**.

NOTE You can also delete construction levels using the **Structure** tree. Note that :

a. Click the **+** sign next to **Levels**.

b. Right-click the level that you want to remove.

c. In the context menu, select **Delete**.

Create and manage architectural grids and grid lines

Architectural grids

An architectural grid is a collection of grid lines that lie in a horizontal plane. Each architectural grid is only displayed in the 3D View and at the lowest level in the structure. Vertical lines can be projected from each grid intersection of the architectural grid. The vertical lines extend to the topmost level at which the architectural grid has been applied.

You can assign a color for each architectural grid, and set to display grid line names and vertical lines in the architectural grid properties.

NOTE Even when architectural grid properties are checked, they are only displayed provided that the **Architectural Grids** --> **Text2D** and **Architectural Grids** --> **VerticalLines** options are selected in (page 259).

Grid lines

Each grid line is associated with an architectural grid, and is only created at a given level, provided the architectural grid has been applied at that level.

Grid lines that have been created at a given level can optionally be shown or not shown in both the 2D and 3D views using the level properties.

NOTE Even when the show/hide options are checked the lines are only displayed provided that the relevant **Grid & Construction Lines** options are also checked in (page 259).

TIP Even when you have activated the display of grid & construction lines in **Scene Content** for a particular 2D or 3D view, you may still find that the grid lines are not displayed on particular levels.

In this situation, do the following:

1. In the **Structure** tree, open the **Levels** branch.

2. Click the level at which you want to view the grids.

   The level properties are viewed in the **Properties** window .

3. In the **Properties** window:

   - To make grids visible in a 2D view, select the **Show grids in plane view** option.
   - To make grids visible in a 3D view, select the **Show grids in the 3D view** option.

If the grids are still not visible in the 3D view at certain levels, ensure that grids are applied at the levels in question. For further information, see Apply an architectural grid to a specific level (page 377).

Create grid lines

Tekla Structural Designer allows you to create various types of grid lines according to your needs. For detailed instructions, see the following paragraphs.

Create a single grid line

NOTE If you want to create a series of grid lines which form a regular or irregular, rectangular or radial grid line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.

---

1. Ensure that the 2D view containing the construction level on which you want to create your grid line is active.
2. Go to the **Model** tab.
3. In the **Grid Line** list, select **Grid Line**.
4. Select the point where you want the grid line to start.

   TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.

   ---

5. Select the point where you want the grid line to end.

   TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the (page 258).

   - To switch the display, click one of the other buttons.

NOTE The grid line does not extend to infinity. Do ensure that the grid line is sufficiently long to meet your needs.

Tekla Structural Designer creates the grid line between the selected points.

Create parallel grid lines

This option creates a grid line parallel to an existing one, but of a different length. To use this option, you must have at least one existing grid line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Parallel**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid lines will be parallel.
5. Move the mouse pointer to the distance where you want to place the parallel grid line.

   TIP The tooltip views the distance of the dotted line from the initial grid line you selected in step 3. Press **F2** to type the exact distance.

   For more accuracy, you can zoom in.

6. Click to locate the new grid line.
7. Click the points where you want the new grid line to start and end.

   The construction line is now created.

8. Move the mouse pointer and place other grid lines with respect to the line you selected in step 3, or press **Esc** to end grid placement.

   TIP To create parallel grid lines of the same length as the selected gridline, select the **Parallel (quick)** command.

   When you create quick parallel grid lines, you can press **F2** to define the distance between successive pairs of grid lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

   For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular grid lines

To use this option, you must have at least one existing grid line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Perpendicular**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid line will be perpendicular.
5. Move the new grid line to the desired position and click to locate it.

   TIP The tooltip views the perpendicular distance of the dotted line from the middle of the grid line you selected. Press **F2** to type the exact distance.

6. Click the points where you want the new grid line to start and end.
7. Move the mouse pointer and place other grid lines with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Create a rectangular grid line system

1. Ensure that the 2D view representing the construction level on which you want to create your rectangular grid line system is active.
2. On the **Model** tab, in the **Grid Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.
6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
7. Click **Next**.
8. Select if you want to create grid lines in the X direction only, in the Y direction only, or in both directions.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. On the following pages, define the layout of grids for the bays in the X and Y directions.

NOTE You can define either a regular or an irregular grid layout:

- For a regular grid, define the number of bays you want to create and the bay centers.
- For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).

You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

- For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

12. Click **Next**.
13. Define the rotation of the grid either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction. This is helpful if you are creating a grid line system that is not orthogonal.

14. Click **Next**.
15. Specify the angle between the axes with respect to either the X or Y axis system.
16. Click **Finish**.

Create a radial grid line system

1. Ensure that the 2D view representing the construction level on which you want to create your radial grid line system is active.
2. On the **Model** tab, in the **Grid Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.
3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.

6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
7. Click **Next**.
8. Select whether you want to create the radial lines only, the arcs only, or both of these.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. Define the layout of the arcs that you will create.

   NOTE You can define either a regular or an irregular grid layout:

   - For a regular grid, define the number of bays you want to create and the bay centers.
   - For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

   You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

   - For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

   You can also select whether the arc grid lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other grid lines created as part of this process.

12. Click **Next**.
13. Define the layout of radial grid segments that you want to achieve.

   Again, you can specify the reference of the first grid line, and how you want to increment the reference to give the references of the other radial lines that you create.
14. Click **Next**.
15. Define the rotation of the grid either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

   TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction.

16. Click **Finish**.

Create a grid arc

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Arc**.
3. In the **Properties** window, type a name for the grid line.
4. Select the point which lies at the center of the grid arc which you want to create.

   The tooltip views the mouse pointer's exact coordinates.
5. Click the points where you want the grid arc to start and end.
6. Move the mouse pointer and place other grid arcs with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Number and renumber grids

Each grid that you create is automatically numbered. To set the initial number or letter for naming grids, or to renumber all grids, see the following instructions.

Set the initial number or letter used for grids

1. On the **Home** tab, click **Model Settings**.
2. Go to **References** --> **General**.
3. In the **Grid Line Naming** section, set the initial values and the naming style.
4. Click **OK**.

Renumber all grids

1. In the **Structure** tree, right-click **Architectural Grids**.
2. In the context menu, select **Renumber**.

   All grids in the model are renumbered in sequence.

See also

Change the name of a grid line or grid arc (page 376)

Change the name of a grid line or grid arc

To change the name of a single grid line or grid arc, see the following instructions.

1. Select the grid line or grid arc that you want to name.

2. In the **Properties** window, modify the name according to your needs.

See also

Number and renumber grids (page 376)

Change the name or color of an architectural grid (page 377)

Apply an architectural grid to a specific level

To view an existing architectural grid at a specific level of your model, you must apply the grid to the level. See detailed instructions in the following paragraphs.

1. In the **Structure** tree, click **Architectural Grids**.
2. Right-click the architectural grid name.
3. In the context menu, select **Edit...**
4. Ensure that the levels where you want the architectural grid to be applied are selected.

See also

Create and manage architectural grids and grid lines (page 370)

Change the name or color of an architectural grid

To modify the properties of an existing architectural grid, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to **Architectural Grids**.
2. Select the architectural grid whose name you want to change.
3. In the **Properties** window, modify the architectural grid's properties according to your needs.

See also

Change the name of a grid line or grid arc (page 376)

Import grids from a DXF file or a shadow of the DXF file

If necessary, you can import grids from an existing DXF file to your Tekla Structural Designer document. In addition, you can use a DXF file as a shadow, or a base that helps you in creating objects in Tekla Structural Designer.

NOTE The DXF file that you use must be available before you start the import process, either sent to you, or created by you.

1. Open a 2D view of a construction level, and go to the **Model** tab
2. In the **Grid Line** list, select **Import DXF**.

   An **Open** dialog opens.
3. Browse to the file that you want to import, select the file, and click **Open**.

   The **DXF Import Wizard** opens.
4. On the first page, manage the layers and colors that you want to import.
5. Select if you want to import the architectural grids from the file, or import the file as a shadow.
   - If you import the architectural grids, all elements in the selected layers of the DXF file are mapped to the Tekla Structural Designer grid lines. You must therefore ensure that you switch off all the layers in the DXF file, apart from the layers in which the grids have been defined.
   - • If you import the DXF file as a shadow, Tekla Structural Designer imports the .dxf file but does not create any Tekla Structural Designer objects. You can use the intersection points and other elements as the source on which to add the Tekla Structural Designer objects you require.
6. If necessary, adjust the scale and offsets for the DXF file.
7. Click **Next**.
8. Select how the grids are created:
   - To create separate named grids for each layer, select **By layer**.
   - To create separate named grids for each color in the DXF file, select **By color**.
   - To create a single merged grid containing all layers and colors, select **Merged**.
9. Ensure that each level to which you want to import the grid layout is selected.
10. Click **Finish**.

See also

Create and manage architectural grids and grid lines (page 370)

Change the name of a grid line or grid arc (page 376)

Change the name or color of an existing architectural grid (page 377)

Extend, move, or rotate grid lines and arcs

You can modify grid lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate grid lines, or move them in a perpendicular direction. As for grid arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION	You can only move grid lines and arcs in 2D views.

Stretch, shorten, or rotate a grid line

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The node moves to its new position.

Move a grid line to a perpendicular direction

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click the center node.
3. Click the location where you want to move the node.

   The grid line moves to its new position.

Stretch or shorten a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc.
3. Click the location where you want to move the node.

   The arc moves to its new location.

Create and manage construction lines

Construction lines serve the same purpose as architectural and construction lines. The only difference is that they do not display a construction bubble. To create construction lines, see the following instructions.

Create a single construction line

NOTE If you want to create a series of construction lines which form a regular or irregular, rectangular or radial construction line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.

1. Ensure that the 2D view representing the construction level on which you want to create your construction line is active.
2. Go to the **Model** tab.
3. In the **Construction Line** list, select **Construction Line**.
4. Select the point where you want the construction line to start.

   TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.
5. Select the point where you want the construction line to end.

   TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the (page 258).

   - To switch the display, click one of the other buttons.

   NOTE The construction line does not extend to infinity. Do ensure that the construction line is sufficiently long to meet your needs.

Tekla Structural Designer creates the construction line between the selected points.

Create parallel construction lines

This option creates a construction line parallel to an existing one, but of a different length. To use this option, you must have at least one existing construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Parallel**.
3. Select the construction line to which your new construction lines will be parallel.
4. Move the mouse pointer to the distance where you want to place the parallel construction line.

   TIP The tooltip views the distance of the dotted line from the initial construction line you selected in step 3. Press **F2** to type the exact distance.

   For more accuracy, you can zoom in.

   ---

5. Click to locate the new construction line.
6. Click the points where you want the new construction line to start and end.

   The construction line is now created.
7. Move the mouse pointer and place other construction lines with respect to the line you selected in step 3, or press **Esc** to end construction placement.

   TIP To create parallel construction lines of the same length as the selected construction line, select the **Parallel (quick)** command.

   When you create quick parallel construction lines, you can press **F2** to define the distance between successive pairs of construction lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

   For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular construction lines

To use this option, you must have at least one existing grid or construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Perpendicular**.
3. Select the construction line to which your new construction line will be perpendicular.
4. Move the new construction line to the desired position and click to locate it.

   TIP The tooltip views the perpendicular distance of the dotted line from the middle of the construction line you selected. Press **F2** to type the exact distance.

   ---

5. Click the points where you want the new construction line to start and end.
6. Move the mouse pointer and place other construction lines with respect to the line you selected in step 4, or press **Esc** to end construction placement.

Create a rectangular construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your rectangular construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
4. Click **Next**.
5. Select if you want to create construction lines in the X direction only, in the Y direction only, or in both directions.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. On the following pages, define the layout of constructions for the bays in the X and Y directions.

   NOTE You can define either a regular or an irregular construction layout:

- For a regular construction, define the number of bays you want to create and the bay centers.
- For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).

---

9. Click **Next**.
10. Define the rotation of the construction either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction. This is helpful if you are creating a construction line system that is not orthogonal.

---

11. Click **Next**.
12. Specify the angle between the axes with respect to either the X or Y axis system.
13. Click **Finish**.

Create a radial construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your radial construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
4. Click **Next**.
5. Select whether you want to create the radial lines only, the arcs only, or both of these.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. Define the layout of the arcs that you will create.

NOTE You can define either a regular or an irregular construction layout:

- For a regular construction, define the number of bays you want to create and the bay centers.
- For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

You can also select whether the arc construction lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other construction lines created as part of this process.

9. Click **Next**.
10. Define the layout of radial construction segments that you want to achieve.

    Again, you can specify the reference of the first construction line, and how you want to increment the reference to give the references of the other radial lines that you create.
11. Click **Next**.
12. Define the rotation of the construction either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction.
13. Click **Finish**.

Create construction arcs

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Arc**.
3. Select the point which lies at the center of the construction arc which you want to create.

   The tooltip views the mouse pointer's exact coordinates.
4. Click the points where you want the construction arc to start and end.

5. Move the mouse pointer and place other construction arcs with respect to the line you selected in step 3, or press **Esc** to end construction placement.

Extend, move, or rotate construction lines and arcs

You can modify construction lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate construction lines, or move them in a perpendicular direction. As for construction arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION	You can only move construction lines and arcs in 2D views.

Stretch, shorten, or rotate a construction line

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The node moves to its new position.

Move a construction line to a perpendicular direction

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.
2. Click the center node.
3. Click the location where you want to move the node.

   The construction line moves to its new position.

Stretch or shorten a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.

2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.

2. Click the center node of the arc.
3. Click the location where you want to move the node.

   The arc moves to its new location.

Create frames and slopes

Frames and slopes are different 2D views of the model that can help you in creating your model. To create a frame or a slope, see the following instructions.

Create a frame

A frame is a 2D View of the model, created in a vertical plane defined by an existing grid line. Since only the members that lie within the plane of the frame are displayed, a frame view can be particularly useful for defining bracing.

1. Obtain a 3D view of your model where you can see the base grid line associated with the frame that you want to create.
2. On the **Model** tab, click **Frame**.
3. Position the mouse pointer over the grid line for the frame that you want to create.
4. Click to create the frame.

   Tekla Structural Designer creates a frame view for the selected grid line.

TIP To open a frame view:

a. Click the **+** sign on the left side of **Frames** in the **Structure** tree.

b. Double-click the name of the frame whose view you want to open.

Create a slope

A slope is a 2D View of the model, created in a sloped plane. You can define a slope by selecting 3 existing grid points. Since only those members that lie within the plane of the slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps.

NOTE Before creating a slope, note that:

- In order to create a slope, you need to be able to click three existing (not co-linear) grid points that lie in the plane of the slope.
- Grid points are formed at grid line intersections on construction levels. Therefore, if the points required to define the slope do not currently exist, you may need to insert new construction levels and grid lines to form them.
- A sloped plane must be entirely contained within a single sub model because the sub model determines the mesh parameters to be applied.
- In a sloped plane, positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

1. Obtain a 3D view of your model where you can see three grip points that define the sloped plane.

   TIP If you cannot see the grid line intersections on a particular construction level in the 3D view:

   a. In the **Structure** tree, select the level.

   b. In the , select the **Show grids in the 3D view** option.

      If you wish, you can clear the **Show grids in the 3D view** option for other levels to simplify the display.

2. On the **Model** tab, click **Sloped Plane**.
3. Click the three points which define the sloped plane.

   Tekla Structural Designer creates sloped plane view.

   TIP To open a slope view:

   a. Click the **+** sign on the left side of **Slopes** in the **Structure** tree.

b. Double-click the name of the slope whose view you want to open.

Create dimensions

Dimensions allow you to show distances between the appropriate points in your structure. The dimension lines are included on any drawings you create. To create dimensions, see the following instructions.

Create a single dimension

1. On the **Model** tab, click **Dimension**.
2. Click the grid point at the start of the dimension.
3. Click the grid point at the end of the dimension.

   Tekla Structural Designer shows a line between the selected points.
4. Move the line to the point where you want the dimension line to lie.

   Choose a point that is easily visible and does not conflict with the rest of your model's details.
5. Click to create the dimension.

Create beams, columns and braces

These topics introduce you to the methods of creating beams columns and braces (in any material).

We recommend you familiarize yourself with how to:

- Create beams (page 401)
- Create columns (page 388)
- Create braces (page 418)

Create columns

This section focuses on the operations required to create columns (in any material).

- Specify the column type and section size (page 389)
- Create a single column or series of columns (page 391)
- Create inclined columns and cranked columns (page 393)
- Create gable posts or parapet posts (page 394)

- Align a column to a specific angle or an angled... (page 395)
- Modify the position of columns and column stacks (page 395)

The following topics are relevant to steel columns only:

- Setting out steel and cold formed columns (page 396)
- Create plated or compound section steel columns (page 397)
- Specify a column splice (page 398)
- Create web openings (page 414)

Concrete columns specifically have an automatic alignment facility:

- Specify concrete column alignment relative to the grid (page 399)
- Modify the column alignment or specify offsets (page 400)

See also

Column properties (page 2089)

Specify the column type and section size

Before you can place a column you must first specify the column type and an initial section size.

Specify the type of column

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Steel Column** type.
Specify a plated or compound steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Plated** column type.
Specify a concrete filled steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Concrete Filled** column type.
Specify a concrete encased steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Concrete Encased** column type. 3. In the**Properties** window, click **Encasing section**. 4. Click the arrow on the right side of **Encasing section**. 5. In the list that appears, select **<New\Edit...>** 6. Enter the size of encasing concrete section.

Specify a cast-in-place, or precast concrete column	• 1. Click the arrow under **Concrete Column**. 2. In the list that appears, select **Concrete Column** for cast-in-place, or **Precast**.
Specify a timber column	Click **Timber Column**.
Specify a cold formed column	In the **Cold Formed** group, click **Column**.

Specify the size of steel, cold formed and timber columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 358) opens.
4. Select the desired section size, and click **Select**.

   TIP To define a custom section, click **Add...**

5. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 358) opens.
4. In the (page 358), choose the required compound section type from the left hand pane.
5. Select the desired section size, and click **Select**.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

6. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
4. Select the column shape.
5. Define the dimensions of the column.

   NOTE Only click **Add** if you want to create a hollow column.

   Then, to define the void:

   a. In the tabular part of the dialog, select the shape and dimensions of the void.

   b. Leave the minor and major offsets as 0.0 to position the void centrally in the column, or adjust as necessary to create an offset.
6. Click **OK**
7. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Create a single column or series of columns

With Tekla Structural Designer, you can create various kinds of column. You can model columns in 2D views, frame views, and structure views.

Create columns in a level view

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.
2. Select the column type and size (page 389).
3. In the **Properties** window , adjust the base level and top level of the column, and other column properties, if necessary.
4. Do one of the following:

To	Do this
Create a single column	a. Click the point where you want to place the column. b. Press **Esc** to finish creating columns.

Create a series of columns	a. Move the mouse pointer to one corner of an imaginary box that will encompass the grid intersection points where you want to create columns. b. Hold down the left mouse button. c. Drag the mouse pointer to the diametrically opposite corner of the box. d. Release the mouse button.

Create a single column in a frame or structure view

NOTE In order to define a column in a frame view or a structure view, you must have already created the construction levels between which the column will run, and the grid points between which it will lie.

1. Select the column type and size (page 389).
2. Click the start point of the column.
3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create plated or compound section steel columns

1. On the **Model** tab, click the arrow under **Steel Column**.
2. In the list that appears, select **Plated**.
3. In the **Properties** window, click the arrow next to the **Section** property.
4. In the list that appears, select **<New\Edit...>**

   The **Select Section** dialog box opens.
5. Select the desired section type and section size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

   d. Select the section in the **Select Section** dialog box.

6. In the **Properties** window, adjust the remaining properties according to your needs.

7. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

> **TIP** In order to apply a plated or compound section an existing steel column, do the following:
>
> a. In the **Properties** window, set **Fabrication** to **Plated**.
>
> b. Click the arrow next to the **Section** property.
>
> c. In the list, either select a compound section that has already been applied in the model, or select **<New\Edit...>** to add a new one.

Create concrete filled or encased concrete columns

1. On the **Model** tab, click the arrow under **Steel Column**.
2. In the list that appears, select either **Concrete Filled** or **Concrete Encased**.

   The column adopts the properties that are currently displayed in the **Properties** window for creating steel columns.
3. In the **Properties** window, adjust the properties as necessary.
4. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

> **NOTE** You can now modify the **Section** property and specify a different section size above the space position.

See also

Create gable posts or parapet posts (page 394)

Create inclined columns and cranked columns

Tekla Structural Designer allows you to create inclined and cranked steel and concrete columns.

Create an inclined column

An inclined column is any column that is not truly vertical. In order to define an inclined column, you must have defined the construction levels between which the column will run, and the grid points between which it will lie.

> **NOTE** An inclined column can only be created in a frame view or a structure view.

1. Select the column type and size. (page 389)
2. In a frame or structure view, Click the start point of the column.

3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create a cranked column

In order to define a cranked column, you must have created the construction levels between which the column will run and the grid points between which it will lie.

NOTE An inclined column can only be created in a frame view or a structure view.

---

1. Select the column type and size. (page 389)
2. Click the start point of the column.
3. Hold down the **Ctrl** key and click the next node of the column.
4. Repeat step 3 until you have selected the second to last node of the column.
5. Release the **Ctrl** key and click the end point of the column.

   Tekla Structural Designer creates the column.

Create gable posts or parapet posts

Steel parapet posts are single span members with fixed end connections. Their specific purpose is to act as a means to transfer load from wind wall panels into columns: the decomposed load from the panel is applied as a point load, and the moment at the node connects the parapet post to the column. To create parapet or gable posts posts in Tekla Structural Designer, see the following instructions.

NOTE Parapet posts are not designed in Tekla Structural Designer.

---

1. On the **Model** tab, click any element type (for example,  **Steel Column**).
2. In the **Properties** window, set the **Characteristic** to **Gable post** or **Parapet post**.

   The properties in the **Properties** window are updated to a type that is appropriate for the chosen post.
3. In the **Properties** window, adjust the properties according to your needs.

   NOTE For gable posts:

   - If an axial load release is required at the top, consider carefully which wind load deflection parameters are required.

   ---

4. Click the start point of the post.

NOTE If you are using a point along a member, do the following:

a. Click the member to see its points.

b. Either click the point that you want to use, or type the distance to the point from the start of the member.

5. Click the end point of the post.

   Tekla Structural Designer creates the post.

Align a column to a specific angle or an angled grid line

You can align a column to a specific angle or a grid line either before creating the column or by editing an existing column in the **Properties** window.

1. Do one of the following:
   - To create a new column, on the **Model** tab, select the desired column type.
   - To align an existing column to a specific angle, in the model, click the desired column.
2. In the **Properties** window, click the **Rotation** property.
3. In the list, select one of the following options:
   - The **0°**, **90°**, **180°**, and **-90°** options align the column to the global axes.
   - The **Angle** option aligns the column to the exact rotation angle you specify.
   - The **Define** option aligns the column to the angle of any grid line you select.

Related information

Related video

Align columns to grids

Modify the position of columns and column stacks

You may sometimes need to modify the position of columns and column stacks in your model. To do so, see the following instructions.

Move a column

See: Move objects (page 495).

Modify the position of a single column stack

You can change the position of a column stack in either a frame view or a structure view.

1. In (page 247), ensure that the **Grid & Construction Lines** options is selected.
2. Select the column stack that you want to move.
3. Click one end node for the column stack to be moved.

   Ensure the node is highlighted in the **Select Entity** tooltip when selecting.
4. Click a grid or construction point where you want to move the selected end node.

   The entire column is redrawn, with the selected node moved to the new position.
5. Select the next end node.
6. Click a grid or construction point where you want to move the selected end node.

   The column is redrawn once more, with the selected node moved to the new position.

Setting out steel and cold formed columns

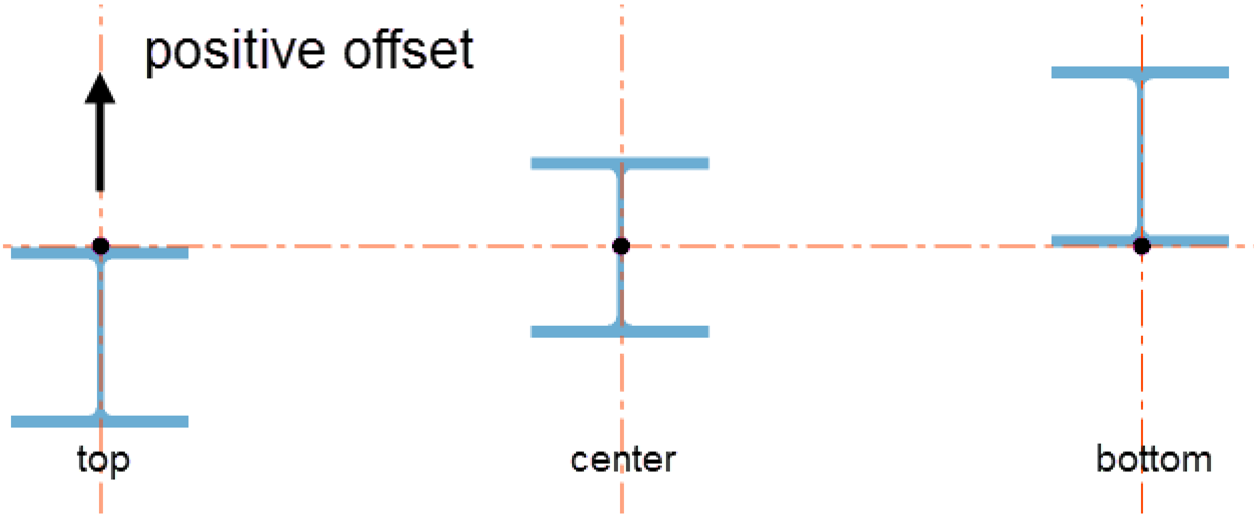
With Tekla Structural Designer, you can create various different types of steel and cold formed columns, including series of columns, inclined and cranked columns, gable or parapet posts, and plated, concrete filled or concrete encased columns. In addition, Tekla Structural Designer allows you to specify a column splice, align a column to a specific angle, or modify the position of existing columns.

Steel column alignment

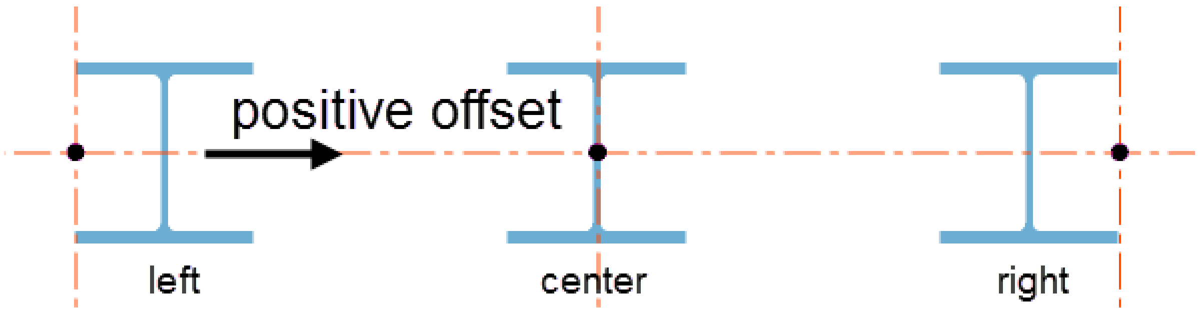
You can create a single steel column over several storey heights. These kind of columns are referred to as column stacks. The columns can start and finish at any level. Different column sections can be defined in each stack, provided that a splice is defined at the change point.

Each steel column stack is placed on an insertion line between points, but its geometry is drawn to reflect the major and minor snap points and any offsets specified in the column properties.

Major offset:



Minor offset:



NOTE Steel column alignment snap points and offsets have no effect on the positioning of 1D solver elements in the solver models. The solver elements are always inserted along the steel column insertion lines.

This is different to the approach adopted for concrete column solver elements, in which the alignment snap points and offsets are structurally significant.

Create plated or compound section steel columns

Along with many other types of steel columns, Tekla Structural Designer allows you to create plated or compound section steel columns in your models.

Create plated or compound section columns

1. On the **Model** tab, click the arrow under **Steel Column**.
2. In the list that appears, select **Plated**.

3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**
   b. Type the sections, plate dimensions, and gaps according to your needs.
   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.
   d. Select the section in the **Select Section** dialog box.

   ---
6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the column.
8. Click the end point of the column.

   Tekla Structural Designer creates the column.
9. Do one of the following:
   - To create more similar column, click the start point of the new column.
   - To stop creating columns, press **Esc**.

Apply a plated or compound section to an existing steel column

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.
3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add, modify and delete user-defined sections (page 1004)

Specify a column splice

You can add splices at the base of each column stack (except for stack 1) as required. The splice offset is used to locate each splice at a practical distance above the floor level.

1. Select the desired columns.

   The properties of the columns are displayed in the **Properties** window.
2. Expand the properties of the column stack within which you want to create the splice.
3. Under **Release**, select the **Splice** option.
4. If necessary, modify the splice offset.

   NOTE The **Section** property is now editable, so that you can specify a different section size above the splice position, if necessary.

See also

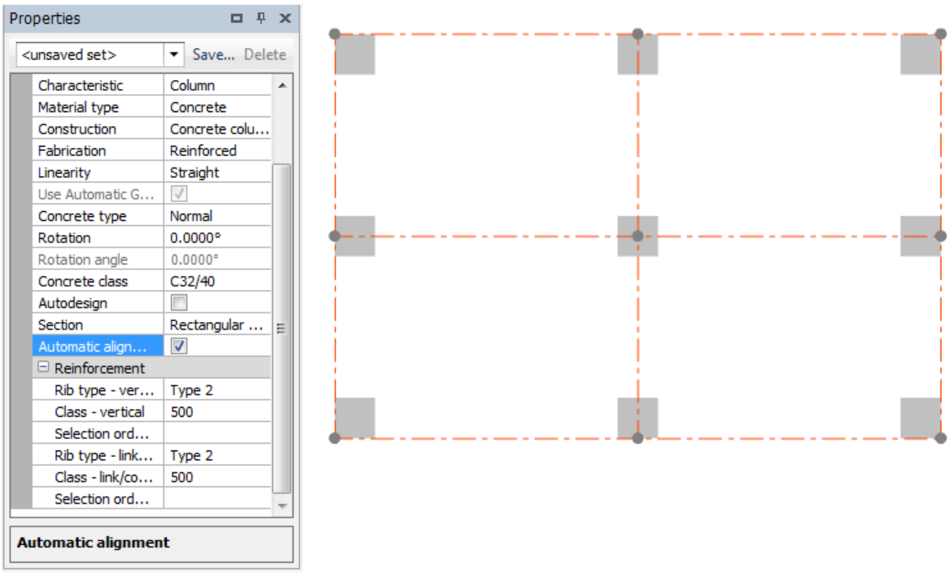
Specify the column type and section size (page 389)

Specify concrete column alignment relative to the grid

The initial placement of each column relative to the grid depends on whether the **Automatic alignment** option is selected in the concrete column properties.

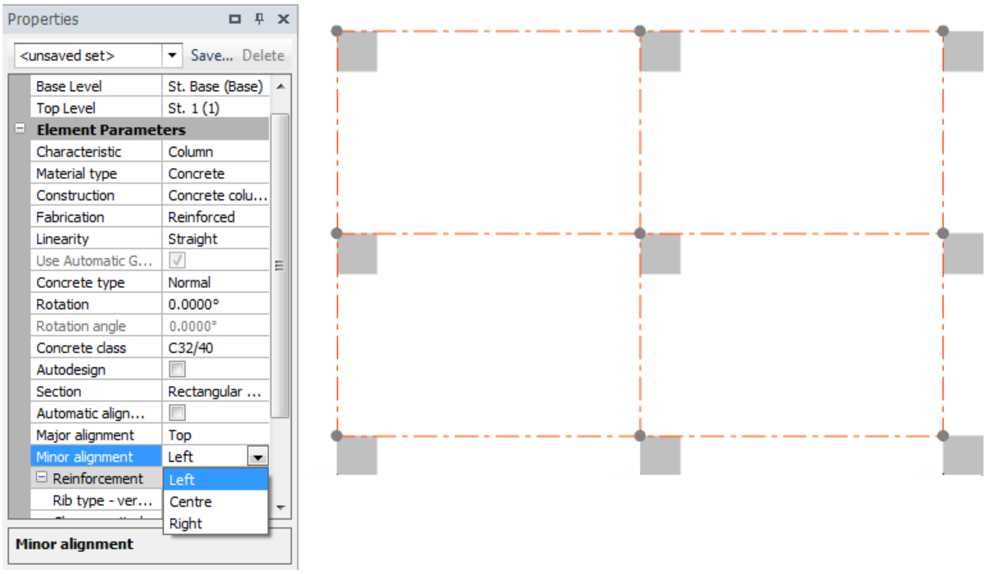
If **Automatic alignment** is selected:

Tekla Structural Designer aligns the columns on the perimeter of the grid with their faces flush to the perimeter, and the internal columns centrally on the grid.



If **Automatic alignment** is not selected:

Tekla Structural Designer aligns the columns according to the **Major alignment** and **Minor alignment** options in the concrete column properties.



- To switch the alignment option, in the **Properties** window, select or clear **Automatic alignment**.

See also

[Align a column to a specific angle or an angled grid line (page 395)]

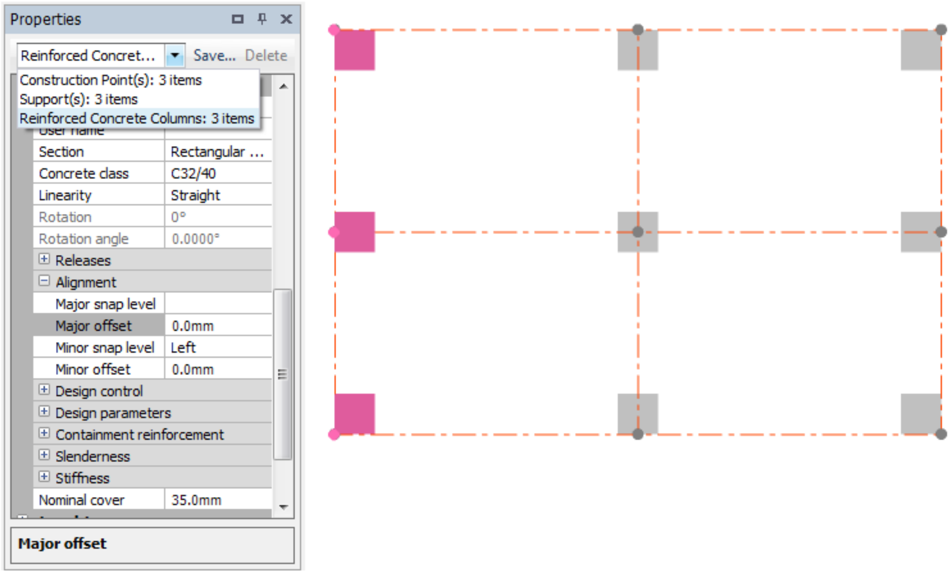
[Modify concrete column alignment or specify offsets (page 400)]

Modify concrete column alignment or specify offsets

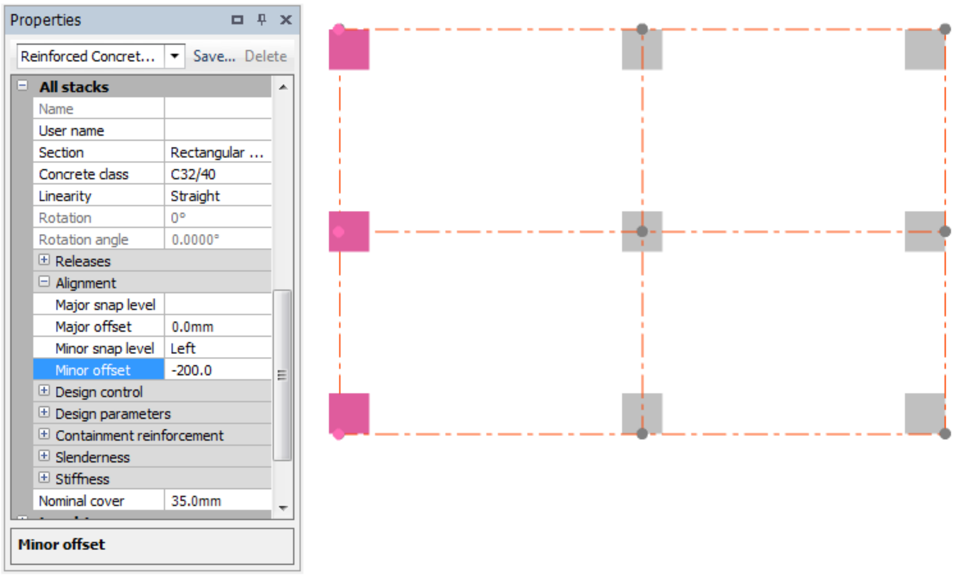
Once you have created columns, you can adjust their alignments and specify their offsets, if necessary. You can adjust the alignment of a single column in the **Properties** window or the **Properties** dialog box, whereas multiple columns can only be realigned in the **Properties** window.

The following example illustrates editing the alignment in the **Properties** window.

1. Select the columns that you want to offset.
2. Ensure that the column properties are viewed in the **Properties** window.



3. Modify the **Major offset** and **Minor offset** values according to your needs.



Tekla Structural Designer moves the columns relative to the major and minor snap levels by the value that you specified.

See also

Specify concrete column alignment relative to the grid (page 399)

Create beams

This section focuses on the operations required to create beams (in any material).

- Specify the beam type and section size (page 402)

- Create single-span beams (page 404)
- Create continuous beams (page 405)
- Create curved beams (page 406)
- Modify the position of beams (page 407)

The following topics are relevant to steel beams only:

- Setting out steel and cold formed beams (page 407)
- Create plated or compound section steel beams (page 409)
- Create Westok cellular, Westok plated or FABSEC® beams (page 410)
- DELTABEAM® (page 411)
- Create web openings (page 414)
- Add haunches to steel beams (page 417)

See also:

Beam properties (page 2073)

Member global offsets (page 422)

Specify the beam type and section size

Before you can place a beam you must first specify the beam type and an initial section size.

Specify the type of beam

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel beam	1. Click the arrow under **Steel Beam**. 2. In the list that appears, select the desired beam type.
Specify a plated, Westok, FABSEC®, or DELTABEAM®	1. Click the arrow under **Steel Beam**. 2. In the list, select the required beam type.
Specify a cast-in-place, or precast concrete beam	• 1. Click the arrow under **Concrete Beam**. 2. In the list that appears, select **Beam** for cast-in-place, or **Precast**.
Specify a timber beam	Click **Timber Beam**.
Specify a cold formed beam	In the **Cold Formed** group, click **Beam**.

Specify the size of steel, cold formed and timber beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 358) opens.
4. Select the desired section size, and click **Select**.

   TIP To define a custom section, click **Add...**

5. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 358) opens.
4. In the (page 358), choose the required compound section type from the left hand pane.
5. Select the desired section size, and click **Select**.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**
   b. Type the sections, plate dimensions, and gaps according to your needs.
   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.
   d. Select the section in the **Select Section** dialog box.

6. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.

3. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
4. Select the beam shape.
5. Define the dimensions of the beam.
6. Click **OK**
7. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Create single-span beams

Tekla Structural Designer allows you to create concrete, steel, and timber single-span beams. For detailed instructions, see the following paragraphs.

Create a single-span beam

1. Specify the beam type and section size (page 402)
2. In the **Properties** window, adjust the beam properties according to your needs.
3. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
4. Click the end point of the beam.

   NOTE If Tekla Structural Designer prompts you to pick another point, and you do not want to create a continuous beam, do one of the following:

   - Click again the end point again.
   - Press **Enter**.

Create a series of single-span beams

RESTRICTION In order to a series of single-span beams using this method, the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level.

1. Select the beam type and size (page 402)
2. In the **Properties** window, adjust the beam properties according to your needs.

3. Move the mouse pointer to one corner of an imaginary box that will encompass the columns between which you want to create beams.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the diametrically opposite corner of the box.

   Ensure that the box encompasses all of the columns between which you want to create beams.
6. Release the mouse button.

   Tekla Structural Designer creates the beams between each adjacent pair of columns within the area you selected.

See also

Create continuous beams (page 405)

Create curved beams (page 406)

Create continuous beams

You can create continuous beams in your model by selecting the **Continuous** option before creating the beam. For more information, see the following instructions.

RESTRICTION You cannot create continuous beams that are curved either horizontally or vertically.

---

1. Specify the beam type and section size (page 402)

   You can create steel, concrete, timber, or cold rolled beams.
2. In the **Properties** window, ensure that the **Continuous** option is selected.
3. In the **Properties** window, adjust the remaining properties according to your needs.
4. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---
5. Click the next node of the beam.
6. Repeat step 5 until you have clicked the second to last node of the beam.
7. Click the end point of the beam.

   Tekla Structural Designer prompts you to pick another point.
8. To create the beam, do one of the following:

- Click the end point again.
- Press **Enter**.

Tekla Structural Designer creates the continuous beam.

See also

Create single span beams (page 404)

Create curved beams (page 406)

Create curved beams

Tekla Structural Designer allows you to create curved steel, concrete, and cold formed beams in your models, if necessary. To model curved beams, see the following instructions.

1. Select the beam type and size. (page 402)
2. In the **Properties** window, ensure that **Linearity** is set to one of the following:
   - **Curved major**, if you want the beam to curve vertically.
   - **Curved minor**, if you want the beam to curve horizontally.

   TIP You can control the direction in which horizontally curved beams curve.

   When you place the beam, you select its start point and its end point. The curve of the beam always lies on the left side of the line from the start point to the end point of the beam.

3. Still in the **Properties** window, specify an appropriate chord height value to define the curve.

   TIP You can reverse the curve direction of a vertically curved beam by using a negative chord height value.

   NOTE Horizontally curved beams always use the chord height defined in the property set. Therefore, they do not curve automatically to fit on any curved grid line that you may have defined.

4. In the **Properties** window, adjust the remaining properties according to your needs.
5. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

6. Click the end point of the beam.

   Tekla Structural Designer creates the beam.

   NOTE If you are using a 2D view when defining beams that curve vertically, you cannot see the beam on the graphical display.

   Change to a 3D view to see the beams.

See also

Create single span beams (page 404)

Create continuous beams (page 406)

Modify the position of beams

If necessary, you can modify the position of an individual beam in your model by moving one of its end nodes. You can modify the position of a beam in both 2D and 3D views.

To move an entire beam, see: Move objects (page 495).

1. Select the desired beam.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

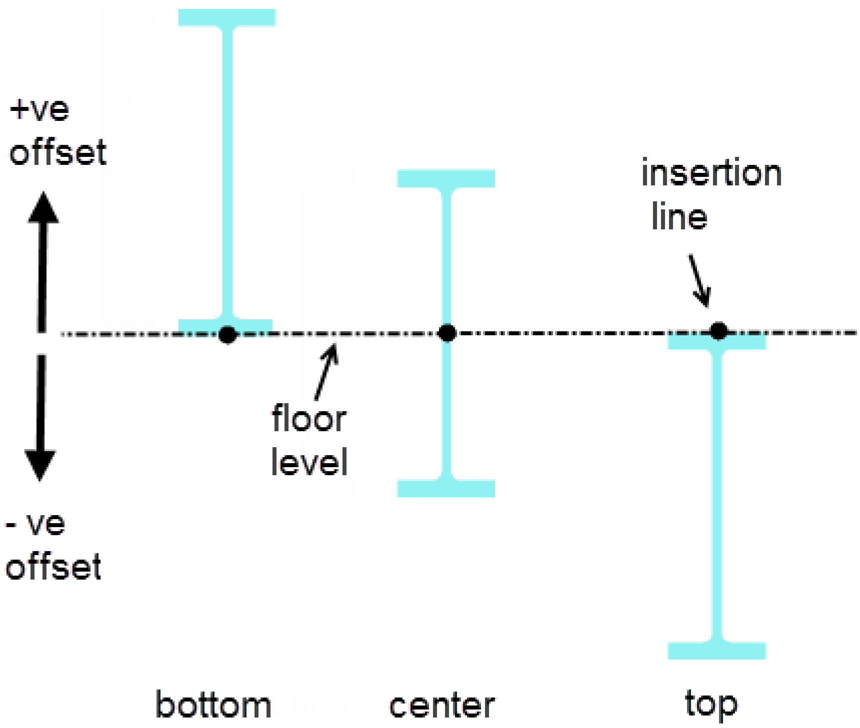
Setting out steel and cold formed beams

You can define steel beams as single span, or continuous over multiple spans. Even if you create a continuous beam, you can still define different beam sections in each span, if necessary.

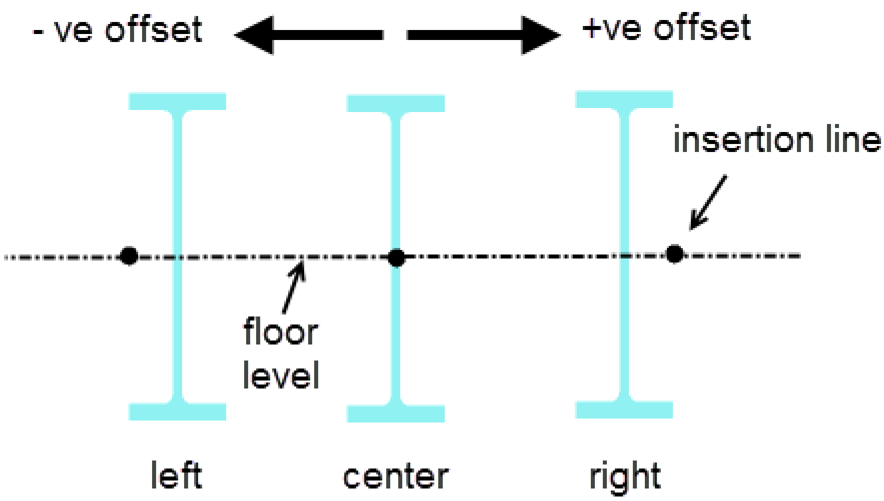
Each steel beam span is placed on an insertion line between points, but its geometry is drawn to reflect:

- the major and minor snap points and any offsets specified in the beam properties

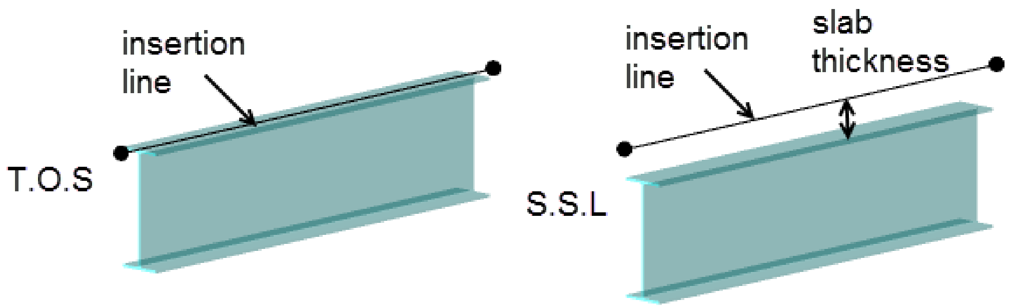
  Major snap and offset:



Minor snap and offset:



- the level type specified in the **Construction Levels** dialog box dialog:



  - When the type is set to **T.O.S.** (top of steel), each beam is displayed according to the alignment snap points and offsets specified.
  - When the level type is set to **S.S.L.** (structural slab level), each beam is also lowered by the slab thickness specified in the **Construction Levels** dialog box.

NOTE Steel beam alignment snap points and offsets and the construction level type have no effect on the positioning of 1D solver elements in solver models. This is a different approach to that adopted for concrete beam solver elements.

Create plated or compound section steel beams

Along with many other types of steel beams, Tekla Structural Designer allows you to create plated or compound section steel beams in your models.

Create plated or compound section beams

1. On the **Model** tab, click the arrow under **Steel Beam**.
2. In the list that appears, select **Plated**.
3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

---

6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the beam.
8. Click the end point of the beam.

   Tekla Structural Designer creates the beam.
9. Do one of the following:
   - To create more similar beams, click the start point of the new beam.
   - To stop creating beams, press **Esc**.

Apply a plated or compound section to an existing steel beam

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.
3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add, modify and delete user-defined sections (page 1004)

Create Westok cellular, Westok plated or FABSEC® beams

Tekla Structural Designer also allows you to create Westok Cellular, Westok Plated or FABSEC® beams, if that is necessary in your model. For more information, see the following instructions.

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select the desired beam type.

   The beam adopts the properties that are currently displayed in the **Properties** window.
3. In the **Properties** window adjust the properties according to your needs.
4. Click the start point of the beam.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the beam.

TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

- Click the end point again.
- Press **Esc**.

Tekla Structural Designer creates the beam.

See also

Create single span beams (page 404)

Create plated or compound section steel beams (page 409)

Create continuous beams (page 405)

Create curved beams (page 406)

DELTABEAM®

DELTABEAM® is a proprietary slim floor system. While DELTABEAM® sections can be analysed in Tekla Structural Designer they are not designed.

NOTE DELTABEAMS® are manufactured by Peikko in Finland: www.peikko.com.

The Peikko Designer download, (English language version) is available from:

https://www.peikko.com/design-tools/

Workflow in Tekla Structural Designer

- Initial DELTABEAM® section sizes are selected from the database and applied in the model. Peikko Designer can be used if required to better determine suitable initial sizes.
- The model is analysed, and a member forces report is created. This report is then sent to Peiko.
- Peiko provide the detailed design and final analysis properties.
- The Tekla Structural Designer model is updated as required.
- If the properties have been revised, the self weight could be affected and the model should be reanalysed.
- A further iteration may or may not be required.

Create DELTABEAMS®

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select DELTABEAM®

   The beam will adopt the properties displayed in the **Properties** window.
3. In the **Properties** window select the section required.

   NOTE D Series internal beams and DR Series edge beams can be selected provided the database country is set to Finland, Norway, Sweden, or Europe. You can add user defined sections to the database for the same countries if required.

4. Select the construction as either composite, or non-composite, as required.
5. Adjust the remaining properties according to your needs.
6. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.
   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the beam.

   TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

   - Click the end point again.
   - Press **Esc**.

   Tekla Structural Designer creates the beam.

   NOTE If you are placing a DR series edge beam, the flange will be drawn to the *left* of the line from start point to end point (easily verified in a 3D scene view). Once the beam has been placed, if you want to switch it to the other side simply select **Reverse** on the **Edit** tab and then click on the beam.

See also:

- Create single-span beams (page 404)
- Create continuous beams (page 405)

Create a member forces report

If you haven't already got a suitable report style configured you will need to create one as follows:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. To add a report, click **Add** and type a name for the report in the **Active Style** field.

3. To add the member forces chapter to the report, expand the Analysis headung in 'Chapters and Options' and drag the Member Forces chapter to the Report Structure pane on the right.

4. Review the report structure and modify it (page 950) according to your needs.

5. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 951).

   TIP You can further limit the output of **Loadcases** and **Combinations** sub chapters by applying a loading filter (page 951).

   ---

6. Click **OK**.

7. Click **Show Report** to review the report and check that it is displaying the required level of data.

Once the report is configured to your needs, it can then be exported to Excel, PDF, Word, or Tedds.

See also:

- Create general arrangement drawings (page 972)

Add DELTABEAM® properties to the database

Having submitted your member forces report to Peiko, you might be provided with a new set of analysis properties that are not in the Tekla Structural Designer database. In this situation you would need to add them as follows:

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.

2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.

3. In the country list, select the database country.

4. In the **Page** pane on the left, select the appropriate DELTABEAM Series.

5. Click **Add...**

6. Enter values for each of the requested variables as supplied by Peiko.

7. Click **OK**.

The new section size is now displayed in the **Item** pane.

Related video

Easy modeling of Peikko Deltabeams

Create web openings in steel members

You can add square, rectangular, and circular web openings to beams and columns either by specified them manually, or by using a **Quick layout** option. FABSEC® beams can also have elongated openings specified.

RESTRICTION

1. Although web openings can be added to steel beams and columns, they are only considered in the design of:
   - Non-composite beams designed to US codes, BS codes, or Eurocodes
   - Composite beams designed to US codes, BS codes, or Eurocodes
2. Web openings are only valid in check design. If the auto design flag is switched on the openings will be removed.

Web opening creation methods

You add web openings via the **Web openings** page of the **Properties** dialog box.

The **Quick layout** method adds web openings to meet the geometric and proximity recommendations published by the SCI. With this method, Tekla Structural Designer creates web openings at the maximum depth and spaced at the minimum centers recommended for the section size.

With the **Quick layout** option cleared you can also define web openings manually using one of the following two methods:

- Clicking the **Add** button adds a new line to the web openings grid and allows you to define the geometric properties of the opening.
- Clicking the **Add...** button opens the **Web opening** dialog box where you can get more help and guidance when defining the opening.

Both manual methods use and signs to indicate faulty data when defining the opening parameters.

For more assistance, hover the mouse pointer over a or sign.

The Web opening dialog box

In the **Web opening** dialog box, you can use the following buttons to automatically define the position of the web opening:

- The **Center** button positions the opening on the center of the member.
- The **Auto** button positions the opening to meet the spacing recommendations by the SCI.

As web openings are defined, they are immediately visible in the diagram in the **Web opening** dialog box.

Tekla Structural Designer also performs design checks when you are defining web openings in the **Web opening** dialog box. The checked areas are end posts, web posts, web opening dimensions and tee dimensions.

The and signs in the **Web opening** dialog box help you to decide whether to make any adjustments to the opening parameters before Tekla Structural Designer checks the design.

NOTE The design checks carried out at the current stage are geometric checks only. Compliance with recommended limits is no guarantee that the opening will pass the subsequent engineering design checks.

Add web openings using the Quick layout option

Using the **Quick layout** method allows you to create maximum depth openings spaced at the minimum centers, appropriate to the section size.

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.
3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Select the **Quick layout** option.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. In the **Type** column, select the web opening type.

   Data for the first web opening is automatically created as follows:

   - $\mathbf{l_O}$: the length of the opening (applies to rectangular openings only)
   - $\mathbf{d_O}$: the depth of the opening
   - $\mathbf{L_{CR}}$: the distance from the setting out point to the center of the opening

- **L_C**: the distance from the end 1 on the member to the center of the opening
- **d_C**: the distance from the top of the member to the center of the opening
- **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
- **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

  Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.

8. If necessary, in the **Stiffening** column, select the location of stiffening.
   a. Type the details of the stiffeners manually as follows:
      - **d_S**: depth of the stiffener
      - **t_S**: thickness of the stiffener
      - **L_S**: length of the stiffener
      - **e_S**: the distance from the edge of the opening to the center of the stiffener
9. To create further openings from the selected setting out points, click **Add** and select the opening type.

   TIP You can use the **Label openings from** list to switch to a new setting out point for the next opening at any point.

10. When you have created the desired openings, click **OK**.

Add web openings manually

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.
3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Ensure that the **Quick layout** option is not selected.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. Click **Add...**
8. In the **Type** column, select the web opening type.

9. Specify the following details for the opening:
    - **l_O**: the length of the opening (applies to rectangular openings only)
    - **d_O**: the depth of the opening
    - **L_{CR}**: the distance from the setting out point to the center of the opening
    - **L_C**: the distance from the end 1 on the member to the center of the opening
    - **d_C**: the distance from the top of the member to the center of the opening

      ---

      TIP Click Center for dC to be automatically calculated to position the opening centrally in the section depth.

    - **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
    - **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

      Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.
10. If necessary, in the **Stiffening** column, select the location of stiffening.
    a. Type the details of the stiffeners manually as follows:
        - **d_S**: depth of the stiffener
        - **t_S**: thickness of the stiffener
        - **L_S**: length of the stiffener
        - **e_S**: the distance from the edge of the opening to the center of the stiffener
11. To create the opening, click **OK**.
12. To create further openings, do one of the following:
    - To create multiple copies of a selected opening, click **Copy...**
    - To create a single opening of a different size or spacing, click **Add...**

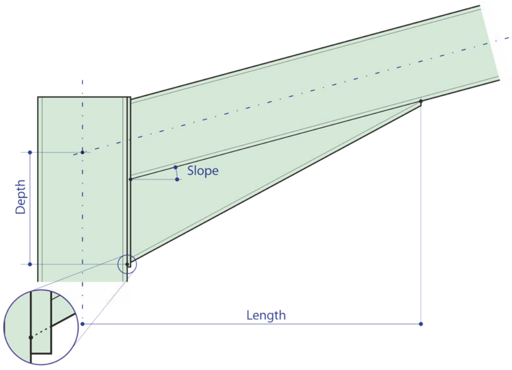
Add haunches to steel beams

1. Right-click the steel beam to which you want to add a haunch, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
2. Go to **Haunches**.
3. Select the appropriate **Create** option for the required haunch location.

4. In the **Haunch** dialog, define the haunch parameters as required.

   The haunch depth is measured from the beam centerline and length is measured horizontally from the sharp end of the haunch to the beam insertion point. In the case of an inclined beam the haunch length and depth would therefore be measured as shown below.



5. When you have defined the haunch parameters, click **OK**.

See also

Create single span beams (page 404)

Create braces

Tekla Structural Designer allows you to create single braces, or pairs of braces (in an X, K, V, or A configuration).

Braces are single span members with pinned end connections that are only able to resist axial compression and tension. Their purpose is to provide lateral stability to your structure.

TIP You can specify rigid frames to achieve the same effect as by using braces. If necessary, you can use both rigid frames and braces within a single structure.

Element loads cannot be applied directly to a brace itself and moments due to self weight loading are ignored.

NOTE Although it is possible to model A or V brace configurations using individual brace members instead of a brace pair, Tekla Structural Designer is then not able to calculate the notional loads or EHFs (Equivalent Horizontal Forces) correctly. In this case, elements of the vertical loads that are supported by the bracing system are lost, and are not included in the notional load or EHF calculations.

Setting out braces

Each steel brace is placed on an insertion line between points, with its geometry being drawn to reflect the major and minor snap points and any offsets specified in the brace properties.

NOTE The alignment snap points and offsets have no effect on the positioning of the solver elements created in the solver models, as these will be created directly between the insertion points.

---

1. Do one of the following:

To	Do this
Create a steel brace	a. On the **Model** tab, click the arrow under **Steel Brace**. b. In the list, select the desired brace type.
Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**. c. Select the desired brace type.
Create a timber brace	a. On the **Model** tab, click the arrow on the right side of **Timber Brace**. b. In the list, select the desired brace type.

2. In the **Properties** window, go to the **Section** property.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, click **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the desired section, and click **Select**.
6. In the **Properties** window, adjust the remaining properties according to your needs.

Specify the brace type and section size

Before creating a brace, you need to specify its type and section size. For more information, see the following instructions.

1. Do one of the following:

To	Do this
Create a steel brace	a. On the **Model** tab, click the arrow under **Steel Brace**. b. In the list, select the desired brace type.

Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**. c. Select the desired brace type.
Create a timber brace	a. On the **Model** tab, click the arrow on the right side of **Timber Brace**. b. In the list, select the desired brace type.

2. In the **Properties** window, go to the **Section** property.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, click **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the desired section, and click **Select**.
6. In the **Properties** window, adjust the remaining properties according to your needs.

Create a single brace

NOTE A and V Braces should be modeled using special tools. Although you can model the exact same A or V brace arrangement using individual brace members, Tekla Structural Designer cannot calculate the notional loads or EHFs (Equivalent Horizontal Forces) of the braces correctly. In this case, elements of the vertical loads that are supported by the bracing system are lost, and are not included in the notional load or EHF calculations.

1. Select the brace type and size.
2. Click the start point of the brace.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
3. Click the end point of the brace.

   Tekla Structural Designer creates the brace.

Create an X, K, V or A brace

1. Do one of the following:

To	Do this
Create a steel brace	• On the **Model** tab, click the arrow under **Steel Brace**.
Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**.
Create a timber brace	• On the **Model** tab, click the arrow on the right side of **Timber Brace**.

2. In the list, select the desired brace pattern.

   The braces adopt the properties that are currently displayed in the **Properties** window.

3. In the **Properties** window, adjust the properties according to your needs.
4. Click to identify a bottom corner of the bay that you want to brace.
5. Click to identify the opposite bottom corner of the bay.
6. Click to identify a top corner of the bay.
7. Click to identify the opposite top corner of the bay.

   Tekla Structural Designer creates the brace pattern within the selected area.

Specify a brace as tension only or compression only

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

Once a brace has been created, you can specify it as Tension only or Compression only in the **Properties** window.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed, they will be treated as linear elements.

1. Select the brace.
2. In the **Properties** window select either **Compression only**, or **Tension only** as required.

Modify the position of a brace

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

To move an entire brace, see: Move objects (page 495).

1. Select the brace.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the brace, moving the end node to the selected point.

See also

Brace properties (page 2085)

Member global offsets (page 422)

Member global offsets

In certain situations you may have a requirement to model different global (X,Y,Z) physical offsets at each end of a member.

In Tekla Structural Designer global offsets can be applied to steel, cold formed, cold rolled and timber member types, but with the following exceptions:

- columns (all materials),
- concrete beams
- analysis elements
- members with haunches
- curved members

NOTE Global offsets have no affect on the positioning of the 1D solver elements in the solver models; they have no impact on loading, analysis or design. Global offsets only affect the graphics and BIM integration.

Global offsets are applied to the member ends **before** any major/minor local offsets, the local offsets thus being applied to the whole length of the member in relation to the new line.

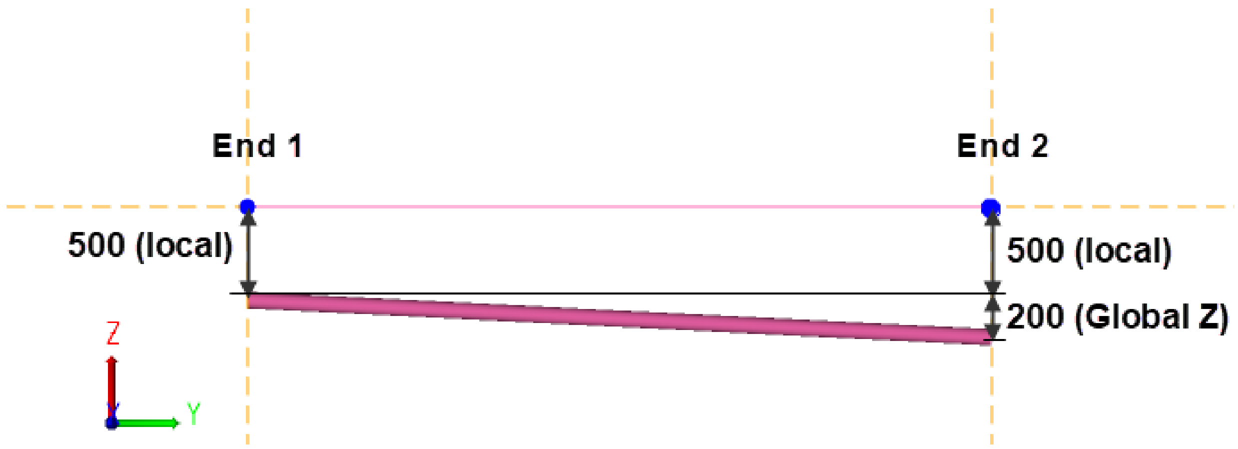
NOTE Global offsets are included in the BIM import/export to/from Tekla Structures

Global offset combined with major offset example

A horizontal brace member is specified with a Global End 2 offset of -200mm and a major (local) offset of -500mm, so that its alignment properties are as follows:

Alignment	
Global offset end 1	[0.0, 0.0, 0.0] mm
Global offset end 2	[0.0, 0.0, -200.0] mm
X	0.0mm
Y	0.0mm
Z	-200.0mm
Major snap level	Centre
Major offset	-500.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:

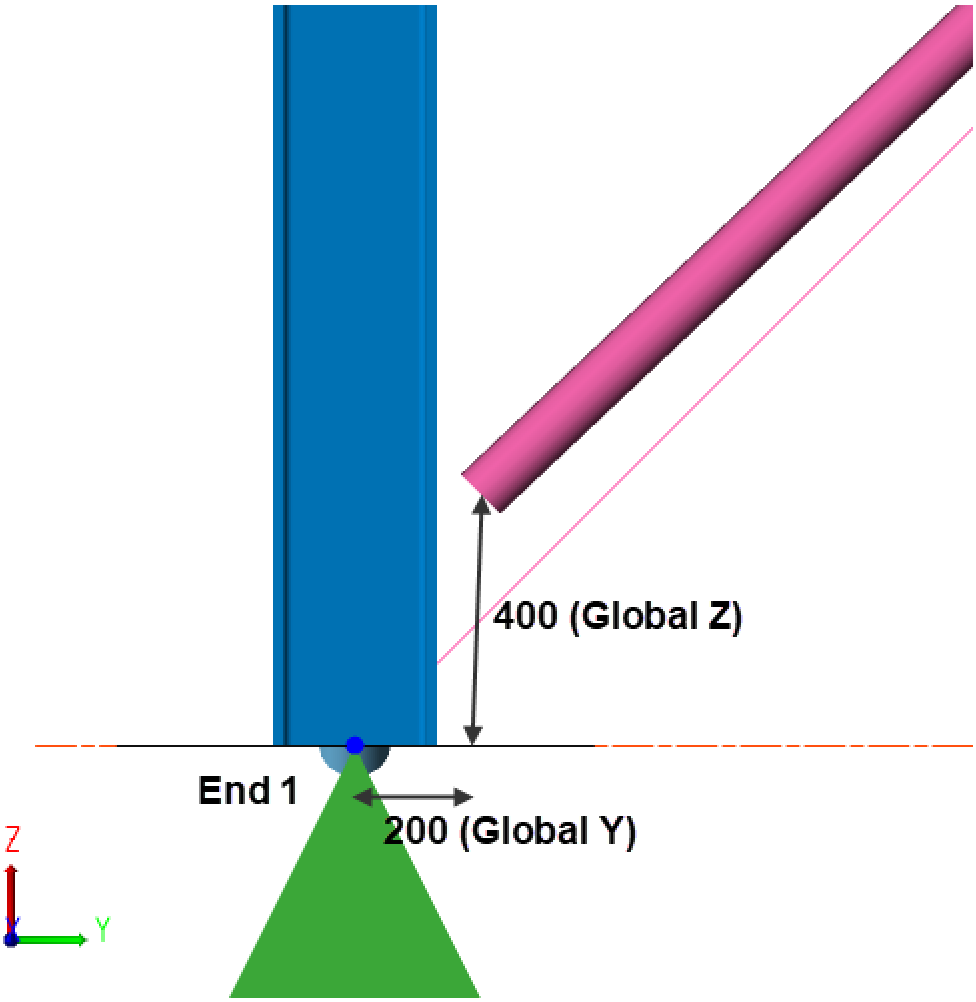


Global offset only example

A diagonal brace member is specified with a Global End 1 offsets of 200mm in Z and 400mm in Y, the Major and Minor (local) offsets are both 0mm, so that its alignment properties are as follows:

Alignment	
Global offset end 1	[0.0, 200.0, 400.0] mm
X	0.0mm
Y	200.0mm
Z	400.0mm
Global offset end 2	[0.0, 0.0, 0.0] mm
Major snap level	Centre
Major offset	0.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:



Global offsets

Create walls, cores and bearing walls

These topics introduce you to the methods of creating walls, cores, and bearing walls.

We recommend you familiarize yourself with how to:

- Create concrete walls (page 426)
- Create concrete cores (page 435) (from existing concrete walls, columns and beams)
- Create bearing walls (page 438)
- Create shear only walls (page 442)
- Create general walls (page 444)

Create concrete walls

This section focuses on the operations required to create concrete walls.

Overview of the concrete wall model

Tekla Structural Designer allows you to create both meshed and mid-pier concrete walls.

Geometric rules for meshed concrete walls

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 432) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- By default, meshed walls use the mesh parameters that are defined in **Structure Properties**. However, you can override these for an individual wall by checking **Override model's** in the Meshing section of the wall's properties. You are then able to apply a user-defined mesh to the wall.
- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Geometric rules for mid-pier concrete walls

- Mid-pier walls must be rectangular in a vertical plane.
- Wall openings are ignored.
- The alignment and offsets specified in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- Mid-pier walls can be included in cores.

Sub-division of mid-pier walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Continuous concrete walls

Tekla Structural Designer allows you to create walls that are several storeys high and that can start and finish at any level.

Although Tekla Structural Designer only creates one wall in this case, you can define different thicknesses for each wall panel. The panels are then set back on one or both faces, depending on the alignment that you have specified.

Creating a continuous wall this way, instead of defining a new wall on each floor, does not have any significance for analysis or design purposes. However, it is ultimately important for detailing purposes.

Analysis of concrete walls

The points that you use to place a concrete wall define the exact size and position of the wall's analysis model. The alignment and extension properties of the wall have no effect on the analysis model.

Tekla Structural Designer can adopt either an FE meshed or mid-pier wall analysis on a wall by wall basis.

Support

Provided that the **AutomaticGenerateSupport** option is selected, if no slab or other member exists beneath the wall when it is first created, Tekla Structural Designer automatically places a support underneath the wall.

Material Type

For walls with the **Material type** set to concrete (as opposed to general), there are then two types of **Fabrication** available:

- Cast-in-place
- Precast

For each fabrication type there are two **Concrete types** available:

- Normal
- Lightweight

The **Grade** lists all the available grades in the Materials database under the current head code for the selected Concrete type.

Releases

You can apply minor axis releases at the top and the bottom of each panel in order to model pinned connections to incoming slabs and members.

Door and window openings

You can only define door and window openings in meshed walls. Tekla Structural Designer does not allow you to create openings in mid-pier walls. For more information openings, and the alternative ways to model them, see: Concrete wall openings analysis model (page 432)

Purpose of concrete walls

Both meshed and mid-pier concrete walls introduce structural strength and stiffness to your model. However, they do not perform the same function as wall panels. This means that wind loads calculated by the Wind Wizard cannot be applied to your structure if the model does not contain wall panels.

In order to apply wind loads, you must create additional wall panels in the same physical locations as the concrete walls.

Create meshed or midpier concrete walls

You can create meshed or mid-pier concrete walls both in 2D views and frame or structure views. For detailed information on creating concrete walls, see the following instructions.

Create concrete walls in a level view

1. On the **Model** tab, click the arrow below  **Concrete Wall**.
2. In the list that appears, select the desired wall type.
    - Mid-pier Wall
    - Meshed Wall
3. Go to the **Properties** window.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

    TIP If you are using a point along a member, do the following:

    a. Click the member to view its points.

    b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

    ---

7. Click the end point of the wall.

    Tekla Structural Designer creates the wall.

Create concrete walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below **Concrete Wall**.
2. In the list that appears, select the desired wall type.
   - Mid-pier Wall
   - Meshed Wall
3. Go to the **Properties** window.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click the point where the base of the wall should start.
7. Click the point where the base of the wall should end.
8. Click the point where the top of the wall should start.
9. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

Concrete meshed and mid-pier wall properties (page 2098)

How meshed walls are represented in solver models (page 750)

How mid-pier walls are represented in solver models (page 755)

Create door or window openings (page 432)

Specify extensions and releases
Tekla Structural Designer allows you to trim new walls automatically, and trim or extend existing walls manually. In addition, you can specify minor axis releases to the top or bottom part of panels. For more information, see the following instructions.

See also

Create concrete walls (page 428)

Create door and window openings (page 432)

Concrete wall extensions

Specify extensions

- Do one of the following:

To	Do this
Automatically trim a new wall back to the face of existing columns and walls	1. In the **Properties** window, select the **AutomaticExtension** option. 2. Create the wall.
Manually trim or extend existing walls	1. Select the wall that you want to trim or extend. 2. Go to the **Properties** window. 3. In the **End 1 extension** or **End 2 extension** field, define the desired extension. **NOTE** A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from its insertion point.

Specify releases

1. Select the wall to be released.
2. In the **Properties** window, go to **Releases**.
3. In the **Minor Top** or **Minor Bottom** list, select the desired release:
   - **Fixed**
   - **Pinned**
   - **Continuous (incoming members pinned)**: only available for FE meshed walls

   TIP To specify a pinned connection to a supported slab, use an FE meshed wall and select the **Continuous (incoming members pinned)** option.

   We recommend this option because the **Pinned** option also releases the wall panel above from the wall panel below. This may result in a mechanism during the analysis.

Create and modify wall supports

If needed, you can set Tekla Structural Designer to automatically generate support for walls. In addition, you can modify the degree of freedom of the wall support, and remove an unnecessary wall support. For more information, see the following paragraphs.

Automatic support generation

The **AutomaticGenerateSupport** option in wall properties controls whether a support is automatically created at the wall base level.

When the **AutomaticGenerateSupport** option is selected:

- If there are members or slabs underneath the wall capable of providing support, no support is generated.
- If there are no members or slabs underneath the wall capable of providing support, a support is generated.

When the **AutomaticGenerateSupport** option is cleared:

- If the **Generate support** option is selected, a support is generated.
- If the **Generate support** option is not selected, no support is generated.

When a support is required, Tekla Structural Designer forms it as follows:

- Under a meshed wall, Tekla Structural Designer creates a line support.
- Under a mid-pier wall, Tekla Structural Designer creates a point support.
- Under a bearing wall, Tekla Structural Designer creates a series of point supports.

When a support is required, its degrees of freedom are as specified in wall properties, under **Wall support**.

See also

Create concrete walls (page 428)

Modify wall support fixity

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, expand **Wall support**.
2. Specify the degrees of freedom according to your needs.

    NOTE The discrete supports at each node are always angled in the global axis system, and not aligned with the wall major/minor axes.

    That is why you need to set both **Mx** and **My** to **Free**. This way, you can ensure that angled walls are pinned out of plane. This is not strictly necessary if the wall is aligned in global X or Y. In this case, you can set only **Mx** or **My** free, as appropriate.

    Similarly, set both **Mx** and **My** to **Fixed** in order to ensure that angled walls are fixed out of plane.

Remove a wall support

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, clear the **Generate support** option.

   NOTE The **Generate support** option is only available for editing if the **AutomaticGenerateSupport** option is inactive. When **AutomaticGenerateSupport** is selected, **Generate support** is automatically cleared if members are created underneath a wall to support it. Similarly, **Generate support** is automatically reselected, if the supporting members are deleted.

Create door or window openings

Tekla Structural Designer allows you to create openings for both doors and windows in existing walls. For detailed instructions, see the following paragraphs.

RESTRICTION	Openings are only active in meshed walls.

1. Open a frame view of the frame that contains the desired meshed wall.
2. On the **Model** tab, click  **Wall Opening**.
3. Do one of the following:
   - To define the opening relative to the bottom left corner of the wall, click the outline of an existing wall panel.
   - To define the opening relative to the selected node, click a specific node in the model.
4. Click the first corner of the opening or press **F2** to define its exact position.
5. Drag the mouse pointer to the opposite corner of the opening.
6. Click the opposite corner of the opening or press **F2** to define its exact position.

   Tekla Structural Designer creates the opening.

See also

Create meshed or midpier concrete walls (page 428)

Meshed wall openings analysis model (page 432)

Meshed wall openings analysis model

Limitations of wall openings

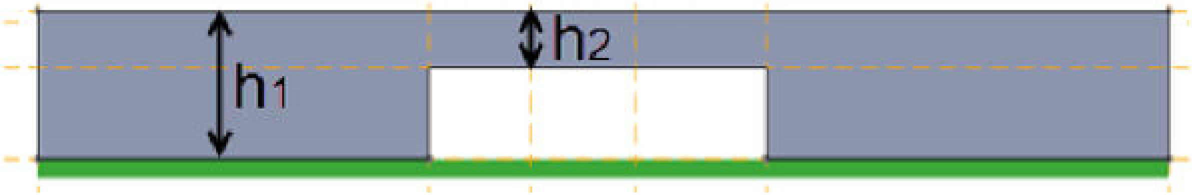
1. If you have specified a door or window opening in a wall panel you must model the wall using FE elements, otherwise a "Walls with openings have a mid-pier" validation error is displayed and the analysis will not proceed.

2. A "Panel contains openings - these are ignored in design" warning will always be issued when a wall containing openings is designed. When you encounter this warning, as well as taking stock of the design implications; you need also to consider if the analysis model is appropriate, as potentially it may not reflect your original intention. In certain situations the **Alternative model for wall openings** (described below) may prove to be a better solution.

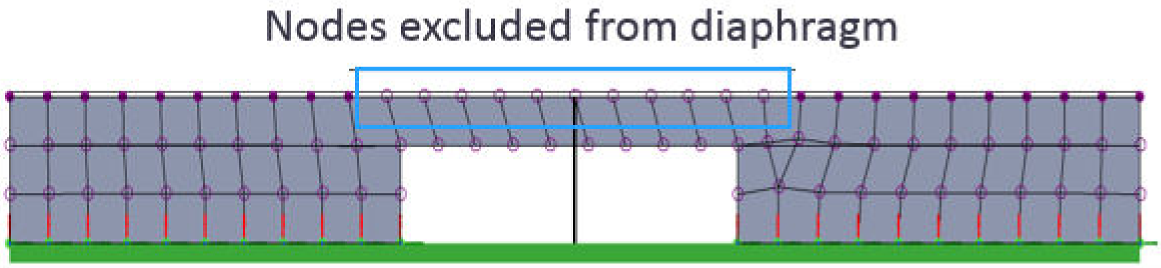
Analysis model applied to meshed wall panels with openings

If an opening is introduced in a meshed wall, the properties of the "lintel" wall beam directly above the opening are automatically adjusted in order to prevent the panel being unrealistically stiff. The adjustments that are applied are as follows:

• wall beam properties in the lintel use the lintel depth (h_2), rather than the panel depth (h_1)



• wall beam nodes in the lintel are removed from the slab diaphragm



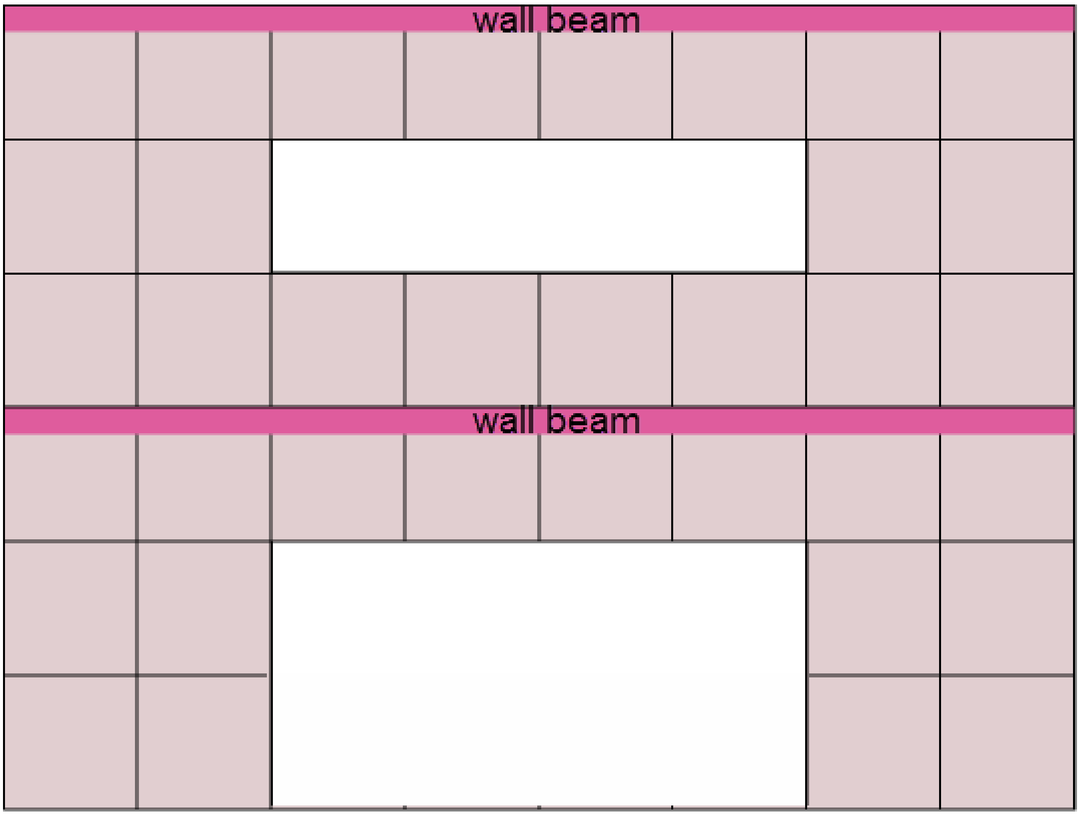
Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

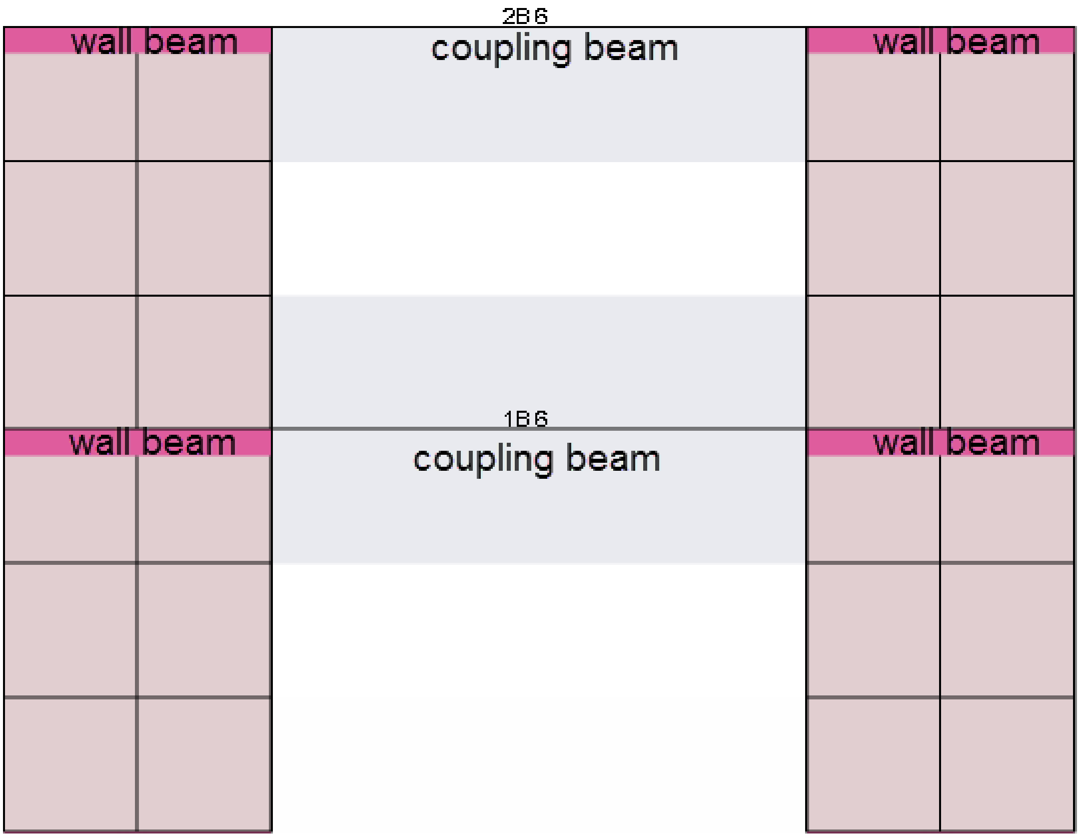
If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam ('coupling beam').

This method can be demonstrated by considering the below example, consisting of a two story wall with a large opening at each level.

If the openings were to be created as a window and door the resulting model would be as shown:



However, by separating the wall into discrete panels and inserting coupling beams you obtain an alternative model as below:



Such an idealisation enables the panels either side of the openings to be designed for their respective forces and enables the strips between the openings to be designed as beams.

Of course, this approach will require some additional detailing, but that would have been the case anyway had the openings been added and subsequently ignored by the design.

Create concrete cores

Tekla Structural Designer allows you to combine concrete wall panels, columns, and beams to form cores.

Overview of concrete cores

Concrete wall panels, columns, and beams can be combined to form cores for which overall analysis results are automatically calculated and available for review and output.

Such cores might then be used in a number of ways, for example:

- The engineer may wish to see/ output core forces to; better understand the distribution of lateral forces within the structure; for additional checks they wish to undertake; for overall core foundation forces or exclusive core reactions for cores on foundation mats.

- Cores provide a way of checking lateral forces per object type or SFRS type. This might be used when assessing buildings for some codes which require frames in dual systems to resist a minimum amount of seismic load.
- Adding all elements of the lateral system to a single core will give overall building vertical and lateral load, overturning moments or accumulated seismic torsion, in any direction.

Any number of walls (meshed and mid-pier), columns, and beams can be added to a core.

- Constituent objects of a core do not need to be physically attached.
- Once defined the core is listed in the structure tree and can be graphically selected as a single entity. When selected, its properties are displayed in the Property Window and Delete, Move, Copy and Mirror operations can be applied to it.

The directions of each core can be defined as any of; Dir 1/2 - Main Building Directions; Principal 1/2 - Major and minor local axis; Angle - (w.r.t.) Global Coordinate System.

The following core properties are automatically calculated/ displayed; Core section for each level; Core centroid and its coordinates above and below each level; a Core line with Local coordinate system (LCS) assigned and displayed at the centroid location; a Core support (for reporting purposes not analysis).

After analysis the following core results are available:

- **Core Line Forces** - 2D Integrated Results termed “Core Line” results for cores giving the following overall core forces; Axial Force, Major and Minor axis Moment and Shear, Torsion.
  - In the Results View via 2D Integrated Results > Core Lines, force diagrams of these results are displayed along the core line w.r.t. its LCS, with numerical values displayed at the top and bottom of each core stack and in the Tooltip when the result diagram is cursored over.
  - Results are available for all of; All Static Analyses including chase-down; RSA Seismic both for individual modes and Modal combination; Imposed/Live load reductions.
  - Tabular Results - Core results are also available in Tabular form via Analyze > Tabular Data and the associated Report option Analysis > “Core Line Forces”.

  NOTE Settings controls in the Member Forces and Wall Line Forces Report items allow you to optionally exclude columns/walls assigned to cores.

- **Core Reactions** - overall Reaction results are calculated for the core and can be viewed in the Results view via Reactions > Cores and output.

Create concrete core (assisted mode)

Cores are either created in **Assisted** or **Unassisted** mode. In assisted mode (which is the default), all potential members of a core are automatically highlighted when the cursor is moved over any of the constituent members, giving a preview of the core that can be created - a single click then adds all the highlighted walls to a new core.

1. On the **Model** tab, click **Cores**.
1. **Assisted Mode**
2. In the **Properties** window, select **Assisted**.
3. Select an appropriate wall panel or column and all highlighted members are added to the core.

   The mode switches to **Modify** to enable you to edit the core if required.
4. If no modifications are required, press Esc to finish.

   NOTE Assisted Mode detects overlapping concrete walls, columns and coupling beams* that may potentially form a core.

   *Beams for which the Construction property is set = “Coupling Beam”. Note that no additional design checks are performed for this construction type so currently it serves only to mark beams; for consideration by the assisted mode core function; for which additional design checks beyond those currently made in the program may be required.

Create concrete core (unassisted mode)

1. On the **Model** tab, click **Cores**.
2. In the **Properties** window, unselect **Assisted**.
3. Select either a wall panel or column as the first member of the core.
4. Select the next wall panel, column, or beam to be part of the core.

   NOTE You can drag the mouse pointer to select multiple members.
5. Continue selecting until all the members of the core have been chosen, and then press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Set the core axis direction

Analysis results are displayed with respect to the axis diection of each core. This can be aligned to the main building directions, the major and minor local axis, or to a specified angle.

1. Select the core.
2. In the **Properties** window, choose the rotation angle as required.

Modify concrete core

1. On the **Model** tab, click  **Cores**.
2. In the **Properties** window,
   a. Set **Mode** to **Modify**
   b. Select the **Active Core** as required.
   c. Set **Area Selection Mode** to **Set On** to add members to the core, or **Set Off** to remove members from the core.
3. Select a member to add or remove from the core.
4. Continue selecting or press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Dissociate concrete core

Existing cores can be disconnected if required.

1. Hover the mouse pointer over the core.
2. If a different entity is highlighted, press the <down arrow> cursor key until the required core reference is shown in the Select Entity tooltip.
3. Right click and select Dissociate Core.

Related video

[Core modeling]

Create bearing walls

Bearing walls provide resistance to vertical compressive loads (but not lateral loads) and support certain other member types. Unreinforced masonry walls, for example, can be modeled as bearing walls.

You can model bearing walls over several storey heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the bearing wall properties, and the selected insertion points.

NOTE Bearing walls do not perform the same function as wall panels. In other words, bearing walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the bearing walls.

Material type

Three **Material types** are available for bearing walls:

- Concrete
- Timber
- General

Concrete Bearing Walls

There are two **Concrete types** available:

- Normal
- Lightweight

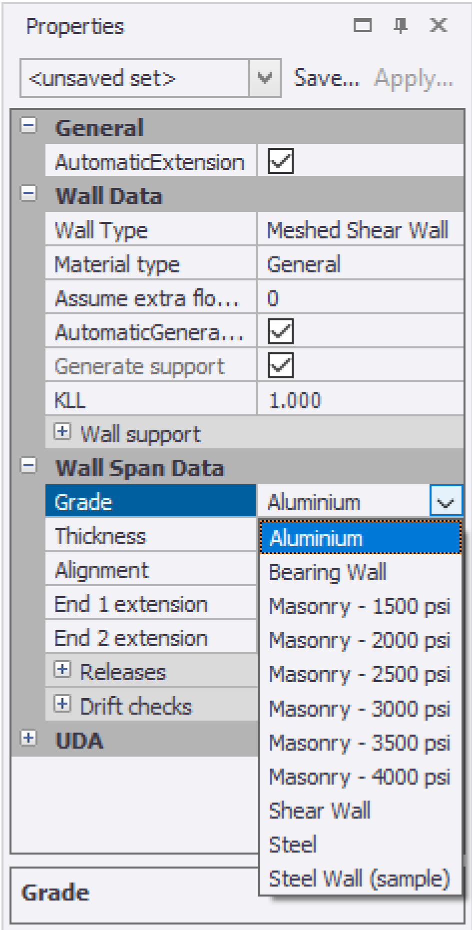
The **Grade** lists all the available grades in the Materials database under the current head code for the selected Concrete type.

Timber Bearing Walls

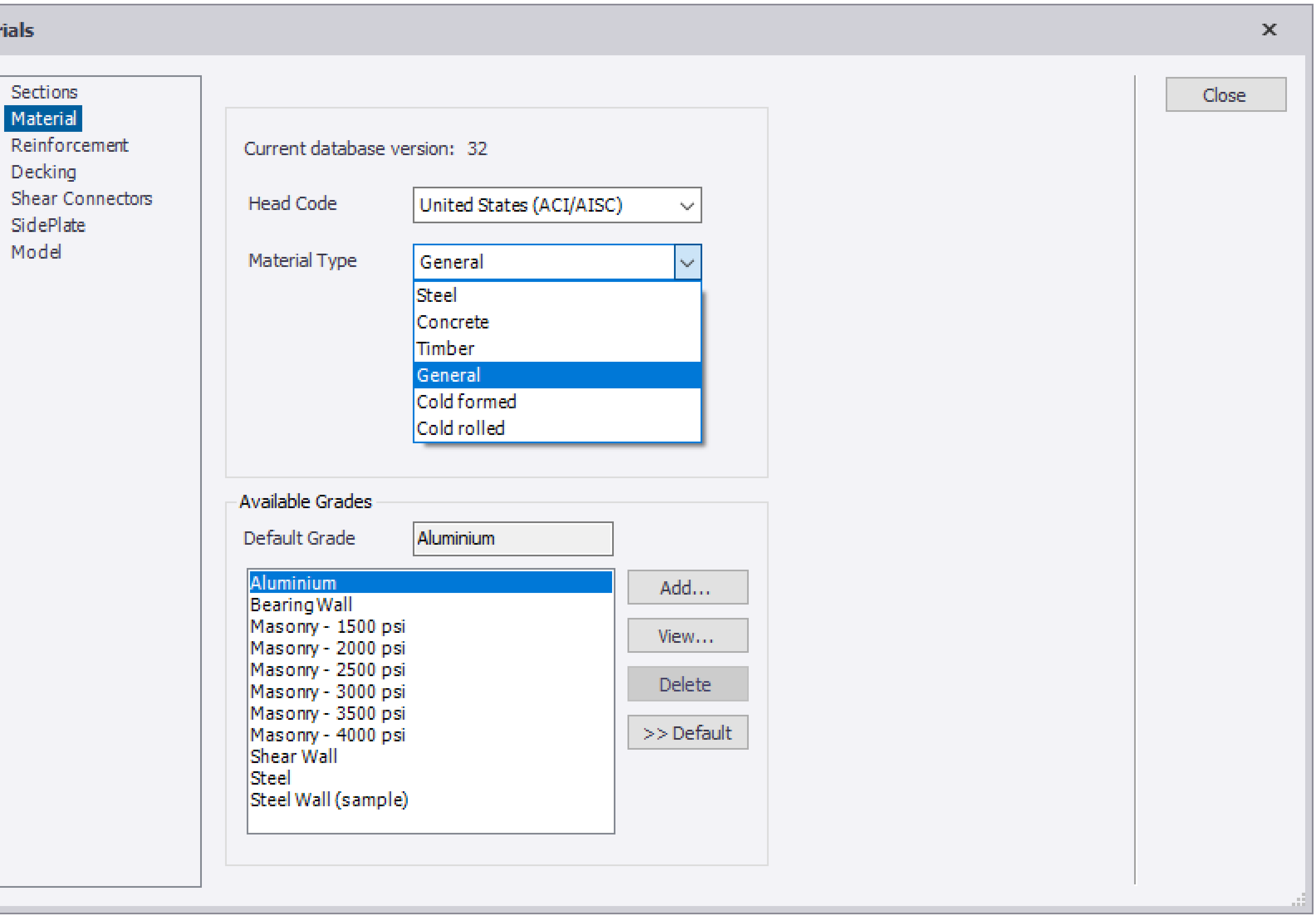
The **Grade** lists all the available grades in the Materials database under the current head code for Timber.

General Bearing Walls

The **Grade** lists all the available **General** materials in the Materials database under the current head code.



If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1016), taking care to first select the General material type as shown below.



Geometric rules

- Bearing walls can only be created as rectangular in a vertical plane.
- Wall openings are ignored.
- Bearing walls cannot be included in cores.

Create bearing walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust that the base level and top level of the wall.
4. Adjust thickness and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create bearing walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

---

1. On the **Model** tab, click **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. In the **Properties** window, adjust the thickness and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Bearing wall properties (page 2133)

How bearing walls are represented in solver models (page 761)

Create shear only walls

Shear only walls provide resistance to in-plane lateral loads only.

You can model shear only walls over several storey heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the shear only wall properties, and the selected insertion points.

NOTE Requirements for these walls are that they must be; within a single bay (i.e. do not overlap one or more columns); strictly rectangular; vertical and surrounded by columns and beams (other than at the bottom edge).

NOTE Shear only walls do not perform the same function as wall panels. In other words, shear only walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the shear only walls.

Material type

The chosen material will only affect the weight of the wall. Whilst Shear only walls are most suited to masonry, three **Material types** are available:

- Concrete
- Timber
- General

Geometric rules

- Shear only walls can only be created as rectangular in a vertical plane.
- They must be totally enclosed by members (concrete or steel), except at the base.
- Wall openings are ignored.
- Shear only walls cannot be included in cores.

Create shear only walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust the base level and top level of the wall.
4. Adjust stiffness, thickness, and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create shear only walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.

2. In the **Properties** window, adjust the stiffness, thickness, and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Shear only wall properties (page 2136)

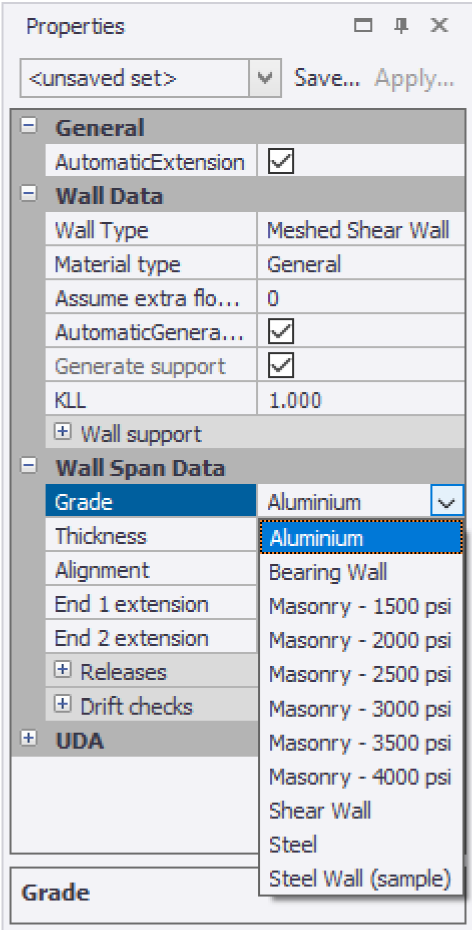
How shear only walls are represented in solver models (page 757)

Create general walls

You can create meshed general walls in level, frame, or structure views. For detailed information on creating general walls, see the following instructions.

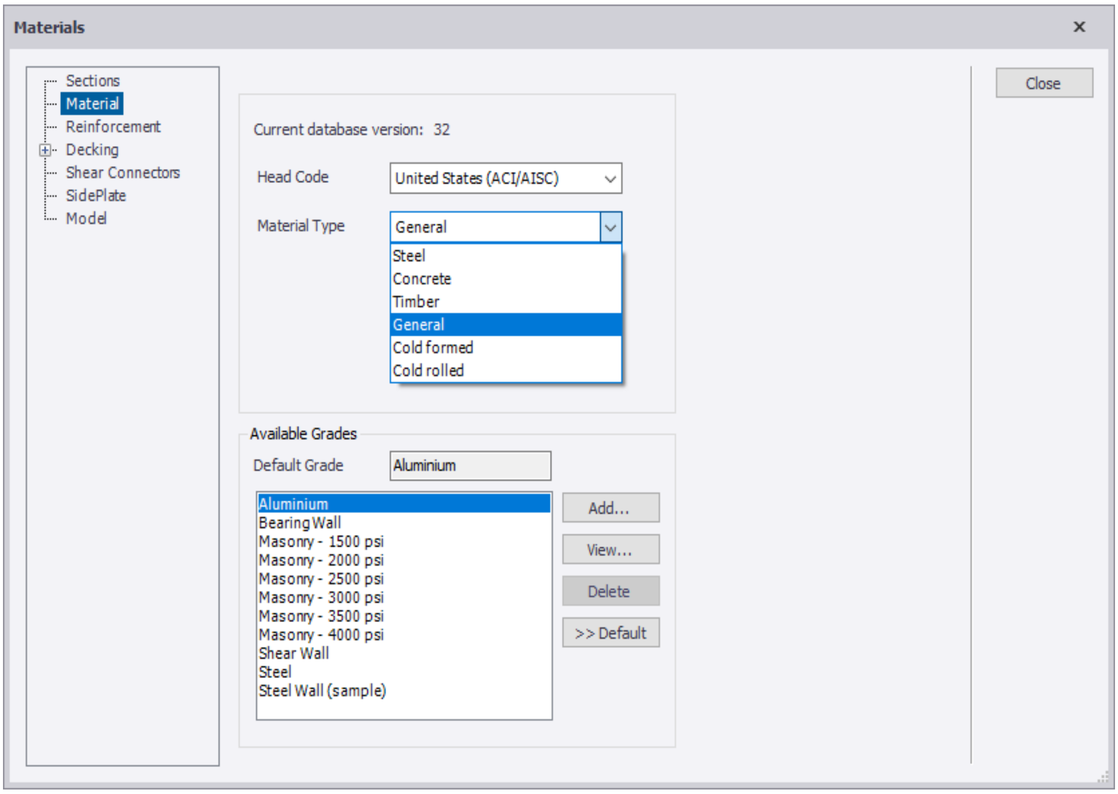
Material type

The **Grade** lists all the available **General** materials in the Materials database under the current head code.



NOTE Care should be taken when choosing the grade, as some general materials in the database may not be appropriate for meshed walls. Meshed walls resist loads in all directions and isotropic stiffness properties are assumed.

If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1016), taking care to first select the General material type as shown below.



Geometric rules

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 432) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.

- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Create general walls in a level view

1. On the **Model** tab, click the arrow below **General Wall**.
2. Go to the **Properties** window.
3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---

7. Click the end point of the wall.

   Tekla Structural Designer creates the wall.

Create general walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

---

1. On the **Model** tab, click the arrow below **General Wall**.
2. Go to the **Properties** window.
3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click the point where the base of the wall should start.
7. Click the point where the base of the wall should end.

8. Click the point where the top of the wall should start.
9. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

General wall properties (page 2106)

How meshed walls are represented in solver models (page 750)

Specify extensions and releases (page 429)

Create and modify wall supports (page 430)

Create door or window openings (page 432)

Create slabs and decks

This section focuses on the operations required to create slabs and decks.

Overview of the slab model

Slabs allow you to decompose loads placed on a floor back to the supporting structure. You can create slabs in either levels or inclined planes. For more information, see the following paragraphs.

Slabs and slab items

Tekla Structural Designer stores slab data in the form of parent slabs, each consisting of one or more individual panels, called slab items. Slab items can be either connected or separated from each other. However, they must be on the same level.

Every parent slab has a unique name. A slab name used at one level can not be re-used at a different level.

Every parent slab has general properties of slab type (and associated deck type) as follows:

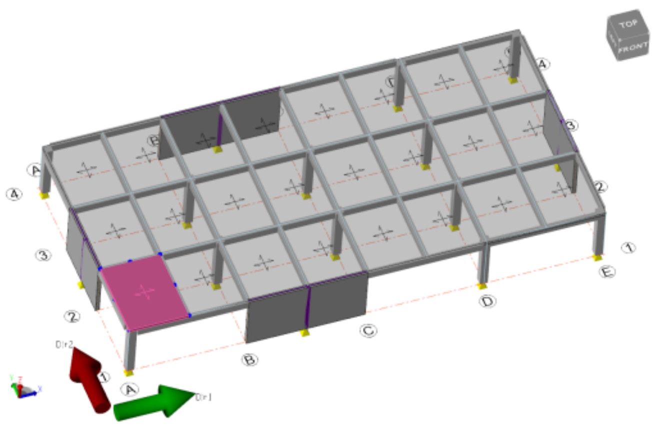
- Slab on beams
  - Reinforced concrete
  - Post tension
- Flat slab
  - Reinforced concrete
  - Post tension
- Precast
  - Precast concrete planks
- Composite slab
  - Profiled metal decking
- General
  - Steel plate
  - Timber
  - General

Every parent slab also has a load decomposition property:

- One-way
- Two-way

Slab items initially inherit their properties from the parent slab. However, once the slab items have been created, you can modify them to amend certain properties: you can change the rotation angle, override the slab depth, or exclude a panel from the diaphragm. Once you have overridden the slab depth, you can also apply a vertical offset in order to model a slab step.

In the view below, there are 24 slab items grouped together in one slab.



Tekla Structural Designer sets some data on the slab level and is common to all slab items, while other data is set at the slab item level. In simple overview terms, the data breakdown is as follows:

Slab data:

- Slab type
- Deck type
- Decomposition
- Thickness
- Vertical offset
- Material properties
- Analysis settings
- General design settings

Slab item data:

- Cover
- Reinforcement information
- Specific design settings

Therefore, you can create slabs over a wide area. While creating slabs, you do not have to consider sub-sections of the slab - you can simply create one big expanse of slab.

When it comes to design, you need to conceptualize the slab as a series of design panels, or slab items. Each slab item will have its own design settings and its own design results. You can select different reinforcement in different panels. You also have to consider pattern loading, where some panels are loaded, and others not.

When results are later presented in calculations and drawings, you can specifically reference the design panels.

Comparison of slab types

The different slab types available in Tekla Structural Designer are compared in the below table.

	Slab on beams	**Flat slab**	**Precast**	**Composite**	**General**
Deck Type	• Reinforced concrete	• Reinforced concrete	• Precast concrete planks	• Profiled metal decking	• steel plate • timber • general
	See: Create slab items (page 453)				
Decomposition	• 1-way • 2-way	• 2-way	• 1-way	• 1-way	• 1-way • 2-way
	See: Overview of one-way and two-way load decomposition (page 557)				

	Slab on beams	Flat slab	Precast	Composite	General
Diaphragm action	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none
	See: Overview of diaphragm action in roof panels and slabs (page 655)				
Load	• all panel load types	• all panel load types	• all panel load types	• all panel load types	• all panel load types
	See: Apply panel loads (page 538)				
Modification factors for 2-way spanning slabs	• yes	• yes	• n/a	• n/a	• yes
	See: Analysis Settings>Modification factors (page 2290)				
Meshed in 3D analysis and grillage chasedown models	• 1-way: no • 2-way: optional	• optional	• no	• no	• 1-way: no • 2-way: optional
	See: Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis (page 637)				
Meshed in FE Chasedown analysis	• 1-way: no • 2-way: yes	• yes	• no	• no	• 1-way: no • 2-way: yes
	See: Manage FE meshed slabs (page 637)				
Designed in Tekla Structural Designer	• 1-way - beyond scope • 2-way- yes	• yes	• beyond scope	• beyond scope	• beyond scope
	See: Design slabs and run punching shear checks (page 789)				

Vertical alignment

The vertical alignment of all slabs in a given level is determined by the construction level type specified in the Construction Levels dialog. The possible types are:

- **T.O.S.** (Top Of Steel): the base of each slab item aligns with the level.
- **S.S.L.** (Structural Slab Level): the top of each slab item aligns with the level.

- **T.O.F.** (Top Of Foundation): the base of each slab item aligns with the level.

Vertical offsets

A vertical offset can be specified after checking 'Override slab depth' in the slab item properties.

These offsets are not structurally significant in slabs as they have no effect on the analysis mesh position relative to the top of the slab in the solver model. However, the mesh properties would reflect any change in the slab thickness.

Slab edges

Slab edges can be either straight or curved.

See: Apply curved edges to existing slab items (page 459)

Slab openings

You can create openings of any shape within slabs. These are only considered when the slab is meshed for load decomposition or analysis.

See: Create slab or mat openings (page 455)

Column drops

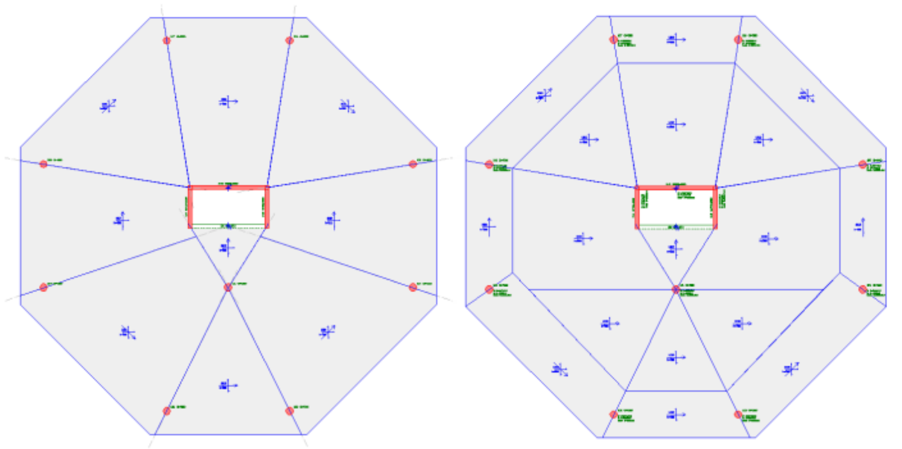
Column drop panels are slab items with an increased thickness. In flat slabs, at points where the slab is supported by columns, you can use column drops to thicken the slab.

See: Create column drops (page 459)

Panel sub-division

Regardless of how the slabs and slab items are initially created, you can further divide or re-form them with the **Slab Split** and **Slab Join** commands. There are several reasons why you may choose to use the previously mentioned commands, related to adding steps, loading patterns, and designing panels.

For flat slabs in particular, the way that slabs are split for the purposes of pattern loading is a matter of engineering judgement. The views below view two options that two different engineers might both justifiably choose for the same slab perimeter.



See: Split and join slabs and mats (page 462)

Create slab items

Slab items are individual slab panels that form parent slabs. You can create slab items either by bay, or by points. For more information, see the following instructions.

Select the slab type and specify slab properties

1. On the **Model** tab, click the arrow under  **Slabs**.
2. In the list that appears, select the desired slab type.
3. In the **Properties** window, define the slab item parameters for the selected slab type.
4. Ensure that the **Slab** and **Select bays** properties are set according to your needs.

Specify the parent slab to which the slab item belongs

1. In the **Properties** window, go to the **Slab** property.
2. Click the arrow on the right side of **Slab**.
3. In the list that appears, select whether you want to:
    - Create a new slab

- Add to existing slab

NOTE When you select this option, at the point of creating the slab item, Tekla Structural Designer checks if there is a existing slab in the plane with same properties.

If Tekla Structural Designer finds an identical slab, the new slab item will be added to it.

If Tekla Structural Designer finds two identical slabs, the closest one is used.

If Tekla Structural Designer does not find an identical slab, a new parent slab is created.

- Manually select the parent slab

NOTE This option is only available in 2D views after one or more slab items have been placed in that view.

4. When the slab item properties are defined, you can place the item either by bays or by picking points.

Create slab items by bay

You can only create slab items by bay in 2D Views.

1. In the **Properties** window, ensure that the **Select bays** option is selected.
2. Do one of the following:

To	Do this
To add an individual slab item	a. Click the outline of a bay bounded by beams or walls.
To add slab items into all bounded bays	• Drag a box that encompasses the bays. **NOTE** Dragging the box from left to right places items in those bays totally enclosed by the box. Dragging the box from right to left places items in all bays that are either enclosed by the box, or cross it. Holding down **Shift** while dragging creates a line instead of a box. Tekla Structural Designer places slab items in all bays that cross the line.

Create slab items by points

You can create slabs in both 2D and 3D Views.

1. In the **Properties** window, ensure that the **Select bays** option is cleared.

2. Click the start point of the slab item.
3. Click other points that define the slab item outline.
4. Double-click the final point to create the slab item.

Create slab or mat openings

When you have created slabs or mats, you can create different kinds of openings to them according to your needs. You can define rectangular, circular, and irregular openings. For more information on how to create and delete slab and mat openings, see the following instructions.

Simple openings

You can quickly define simple openings within existing slabs. Simple openings are rectangular or circular in plan.

The **Slab Opening** command is located on the list in the **Slabs** group. The command is only active in 2D views.

Slab openings can:

- Cross more than one slab item or slab
- Be overlaid or joined to create openings which together have shapes other than rectangular
- Cut across a stepped edge
- Be applied to level an sloping slabs

Openings cannot:

- Be applied to one-way spanning slabs
- Reside within or cut a column drop

Irregular Openings

Alternatively, you can create more complex openings by using construction lines and constructing slab items around an irregular shape.

Create rectangular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select  **Slab Opening**.

Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Rectangular**.
4. If necessary, specify a rotation angle to rotate the opening on plan.
5. Click within the outline of an existing slab item or mat panel to define the first corner of the opening, or press **F2** to define its exact position.
6. Drag the mouse pointer to the opposite corner of the opening.
7. Click the opposite corner of the opening, or press **F2** to define its exact position.

   Tekla Structural Designer creates the opening.

Create circular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Opening**.
Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Circular**.
4. Click within the outline of an existing slab item or mat panel to define the center of the opening, or press **F2** to define its exact position.
5. Drag the mouse pointer to define the radius of the opening, or press **F2** to define the exact radius.

   Tekla Structural Designer creates the opening.

Delete slab or mat openings

1. In the **Structure** tree, open the **Slab Openings** branch.
2. Right-click the name of the slab or mat opening that you want to delete.

3. In the context menu, select **Delete**.

   Tekla Structural Designer deletes the opening.

Add overhangs to existing slab or mat edges

At times, the edge of a slab or mat may extend beyond a grid line, either to the edge of the beam which supports the edge of the slab, or around the perimeter of the building to meet the inside face of the cladding. In order to take these cases in to account, Tekla Structural Designer allows you to define overhangs to the edges of a slab. An overhang may extend across many slab items in one slab and can be curved, or tapered if required.

Any loads that you define over an overhang are included in the total loading on your building.

You can only create overhangs in 2D views.

Add an overhang along the full length of a supporting beam

NOTE This method can only be used if a beam exists along the entire slab/mat edge to which the overhang is to be added.

1. Open a 2D view of the level containing the slab/mat for which you want to create the overhang.
2. Do one of the following:

To	Do this
Create an overhang over a supporting beam to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Overhang**. c. In the **Properties** window, check **LengthOfBeam**. d. If necessary, in the **Properties** window, modify the width and other properties of the overhang. e. Click on the beam over which the overhang is to extend.
Create an overhang over a supporting beam to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**. • In the **Properties** window, check **LengthOfBeam**. • If necessary, in the **Properties** window, modify the width and other properties of the overhang.

	• Click on the beam over which the overhang is to extend.

Tekla Structural Designer creates the overhang along the length of the beam.

Add an overhang to a slab or mat edge between two points

NOTE In order to define a slab overhang between two points, you must have already defined the slab to which it applies.

1. Open a 2D view of the level containing the slab/mat for which you want to create an overhang.
2. Do one of the following:

To	Do this
Create an overhang to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Overhang**.
Create an overhang to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**.

The slab or mat overhang properties are viewed in the **Properties** window.

3. Ensure the **LengthOfBeam** property is unchecked.
4. If necessary, in the **Properties** window, modify the width and other properties of the overhang.
5. Click along the edge of an existing slab item or mat panel to define the start point of the overhang, or press **F2** to define its exact position.
6. Click along the same edge to define the end point of the overhang, or press **F2** to define its exact position.

Tekla Structural Designer creates the overhang between the two points.

Add a curved overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, uncheck the **Edge parallel** property.
2. In the **Properties** window, define the **Curvature** of the overhang.
3. Continue to create the overhang in the normal way.

Add a tapered overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, check the **Tapered** property.
2. Use the **Width1** property to define the length of overhang at end 1.
3. Use the **Width2** property to define the length of overhang at end 2.
4. Continue to create the overhang in the normal way.

Apply curved edges to existing slab items

You may sometimes need to create slab items that have curved edges instead of linear ones. For detailed instructions to do so, see the following paragraphs.

1. Select the slab items that you want to modify.
2. According to you needs, do one of the following:

To	Do this
Apply the same degree of curvature to all selected slab edges	a. In the **Properties** window, go to **All edges**. b. Clear the **Linear** option. c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.
Apply curvature to a specific edge	a. In the **Properties** window, go to the properties of the required edge. The edges are numbered. b. Clear the **Linear** option. c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.

Tekla Structural Designer redraws the slab item with the specified curvature.

Create column drops

In order to increase punching resistance, you can insert drop panels, or column drops, within concrete slabs at points where they are supported by columns. Column drops are slab thickenings that can be positioned above the slab, below the slab, or both. Column drops are rectangular in plan, and aligned to the column axes. Column drops are always positioned centrally over the supporting column.

The extent of the column drop is limited by the extent of the slab boundary. The cut back drop shape may not be rectangular.

NOTE In order to define a slab drop you must have already defined the concrete slab to which it applies.

1. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group.
2. In the list that appears, select **Column Drop**.

   The column drop properties are viewed in the **Properties** window.
3. If necessary, in the **Properties** window, modify the properties of the column drop.
4. Click an existing column connected to a concrete slab panel to create a single column drop, or drag a box around multiple columns to create a series of column drops.

   NOTE To insert the column drop correctly, ensure that the type of the construction level containing the slab is **S.S.L.**, not **T.O.S.**

Specify the material for general slab types

General slab types can have steel plate, timber, or general deck types. The material grade properties for each deck type can either be selected from the material database or they can be user-defined.

Choosing the grade from the material database

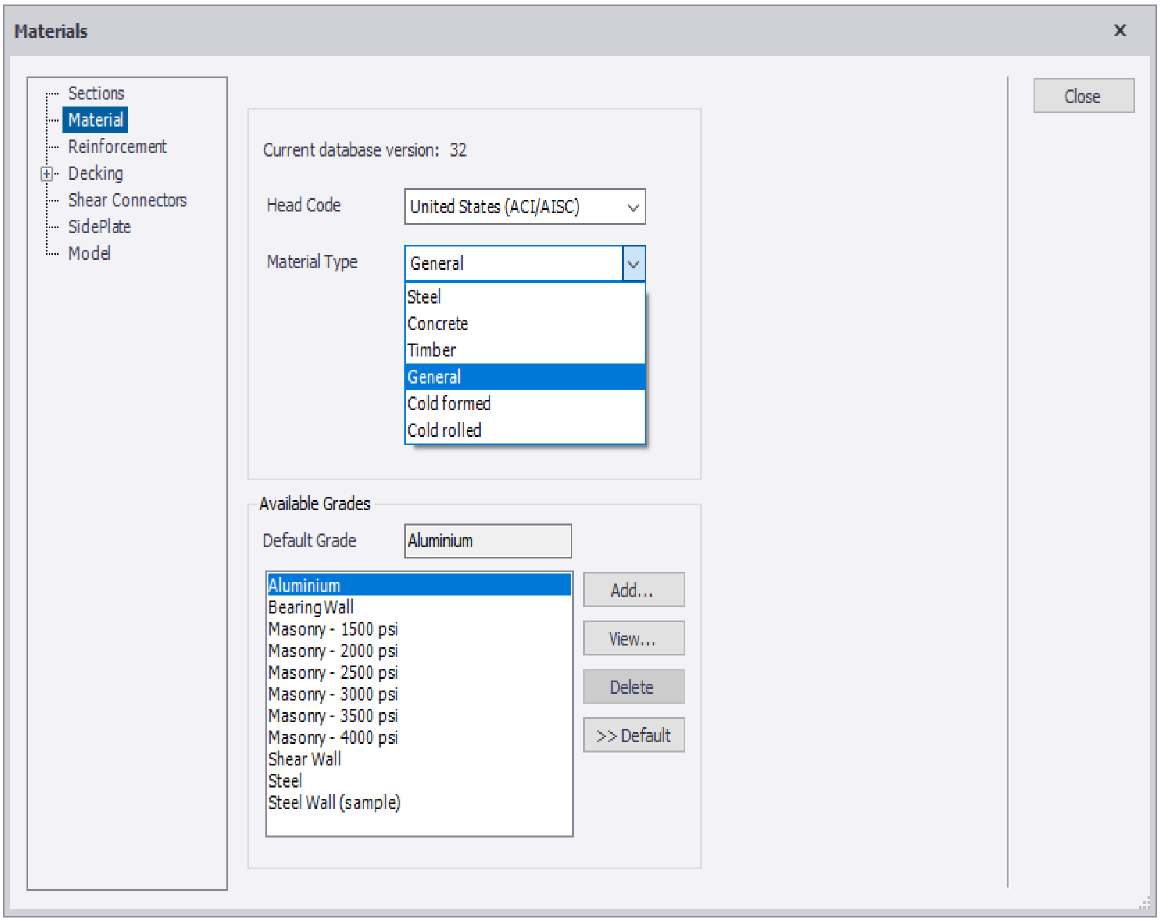
If the **Deck type** is set to:

- **Steel plate** - the **Grade** lists all the available **Steel** materials in the Materials database under the current head code.
- **Timber** - the **Grade** lists all the available **Timber** materials in the Materials database under the current head code.

  NOTE When using a timber deck type, the program will initially default to a material override option rather than displaying a Grade. This is because the Poisson's ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson's ratio (other grade properties such as density and modulus of elasticity will be retained).

- **General** - the **Grade** lists all the available **General** materials in the Materials database under the current head code.

If the general material grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1016), taking care to first select the General material type as shown below.



Override material properties option

Unlike other slab types, general slab types each have an option to override material properties. This option is provided primarily to allow the existing properties of steel or timber deck slabs created in older versions of Tekla Structural Designer (pre 2020 SP5) to be retained, but it can also be used to enter the material properties directly if required.

NOTE When creating new timber decks, the program will default to the override option. This is because the Poisson's ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson's ratio (other grade properties such as density and modulus of elasticity will be retained).

Split and join slabs and mats

Tekla Structural Designer allows you to both sub-divide existing slab items into smaller items, and merge slab items into larger items.

NOTE The commands for splitting and joining slab items are only accessible in 2D views.

Split slab and mat items

1. Do one of the following:

To	Do this
Split slabs	• On the **Model** tab, click **Slab Split**.
Split mats	• On the **Foundations** tab, click **Mat Split**.

2. Hover the mouse pointer over the slab or mat edge, grid point, or other point where you want to start the split.

   NOTE The points used to define the cut line can be outside the boundary of the slabs being split. Thus, they do not need to be on the slab edges.

3. Click the start point of the split.
4. Click the second point of the split on either the same or other item or panel.
5. Do one of the following:
   - To continue the split, click subsequent split points.
   - To end the split, click the second point of the split again.

   Tekla Structural Designer splits any slab items entirely crossed by the split along the cut line.

Join slab and mat items

RESTRICTION You can only join slab items that share a common edge.

1. Do one of the following:

To	Do this

Split slabs	• On the **Model** tab, click **Slab Join**.
Split mats	• On the **Foundations** tab, click **Mat Join**.

2. Select the first of the slab items (the master item) that you wish to join.

NOTE The slab items selected later will adopt the properties of the master item.

3. Select the second slab item.

   Tekla Structural Designer joins the selected slab items to create a new one.

4. Select additional slab items as required, or press **Esc**.

Create trusses and joists

These topics introduce you to the methods of creating trusses and joists.

We recommend you familiarize yourself with how to:

- Create trusses (page 463)
- Create steel joists (page 465)

Create trusses

Trusses are particular arrangements of members that Tekla Structural Designer calculates automatically for you. Once you have created a truss, you can copy the truss throughout your model as necessary. In addition, you can pick a truss in your model, and move it to a desired location.

Create a truss

The **Truss Wizard** helps you to create trusses in your models. For detailed instructions on how to use the **Truss Wizard** to create trusses, see the following instructions.

1. On the **Model** tab, click **Steel Truss**, or **Timber Truss** according to your needs.

   The **Truss Wizard** opens.

2. Select the truss shape.
3. In the model, click the start point of the truss.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

4. In the model, click the end point of the truss.
5. In the **Truss Wizard**, click **Next**.
6. Specify the truss alignment parameters.
7. Click **Next**.
8. Specify the truss details.
9. Click **Finish**.

Create a space truss

Tekla Structural Designer allows you to create linear and planar space trusses. You can create space trusses in a 2D or 3D view. For more information, see the following instructions.

1. On the **Model** tab, click the arrow next to **Steel Truss**, or **Timber Truss**, according to your needs.
2. In the list that appears, select **Space**.

   The **Space Truss Wizard** opens.
3. Define the truss type, the alignment, and the number or bays.
4. Click **Next**.
5. In the model, click the truss location points:
   - For a linear truss, click the start and end points.
   - For a planar truss, click the four corners of the truss.
6. In the **Space Truss Wizard**, specify the truss width and height.

   If you are creating a planar truss, Tekla Structural Designer calculates the width automatically.
7. Click **Next**.
8. Define whether the truss should be straight or curved.
9. Click **Finish**.

Create a free form truss

You can create a free form truss by placing a series of truss members in the required shape. For more information, see the following instructions.

RESTRICTION Free form trusses cannot be created in a 3D view, or a 2D level view.

1. Open the 2D frame view in which you want to create the truss.
2. On the **Model** tab, click the arrow next to **Steel Truss**, or **Timber Truss**, according to your needs.
3. In the list that appears, select **Free Form**.
4. Click the start point of the first truss member.
5. Click the end point of the first truss member.
6. Repeat steps 4 and 5 to place each truss member as required.
7. When the truss geometry is complete, press **Esc** to finish.

Modify the geometry of existing steel and space trusses
After creating a steel, timber, or space truss, you can modify their geometry by using the **Edit** command. For more information, see the following instructions.

1. Hover the mouse pointer over the desired truss so that it becomes highlighted.
2. Right-click the truss.
3. In the context menu, select **Edit [element name]**.

   Depending on the type of the truss, either the **Truss Wizard** or the **Space Truss Wizard** opens.
4. Modify the truss properties according to your needs.
5. To save the changes, click **Finish**.

Modify the properties of existing trusses
You can modify the section sizes, material grades, section orientations, and other truss properties in the **Properties** window. For more information, see the following instructions.

1. Move the mouse pointer over the desired truss, so that it becomes highlighted.
2. Click the truss.
3. In the **Properties** window, modify the truss properties according to your needs.

Create steel joists
Steel joists, or bar joists, are a specific type of members used in the United States. They are simply supported secondary members that, in turn, support slab and roof loads. Steel joists are constrained to standard types specified by the US Steel Joist Institute, and standardized in terms of span, depth and load carrying capacity. To create steel joists in your model, see the following paragraphs.

Specify the section for Standard and Special joist types

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, specify the **Steel joist type**
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select either **Standard** or **Special** as required.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Select Section** dialog box opens.
8. Select the desired joist type and size.
9. Click **Select**.
10. In the **Properties** window, adjust the remaining parameters according to your needs.

Specify the section for the Girder joist type

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, select the **Steel joist type** parameter.
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select **Girder**.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Girder Section** dialog opens.
8. Either:
   - Enter the girder properties and then click **OK**.
   - Click **Add** if you want to add another girder section.
9. In the **Properties** window, adjust the remaining parameters according to your needs.

Create a steel joist

1. Select the joist type and size.
2. Click the start point of the joist.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

3. Click the end point of the joist.

NOTE The ends of steel joists can be on different levels.

Move a steel joist

You can modify the joist position in both 2D and 3D views.

1. Select the joist.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Create portal frames

Tekla Structural Designer allows you to create both single-span and multi-span portal frames. After creating the portal frame, you can modify the properties of either the entire portal frame, or a portal frame member. For more information, see the following paragraphs.

Create a single or multi-span portal frame

1. On the **Model** tab, click **Portal Frame**.
2. In the **Properties** window use **Span Count** to specify the number of spans.
3. Click the start point of the first span of the portal frame.

   The start point sets the local X and Y coordinate origin for the portal frame.

4. Click the end point of the last span of the portal frame.

The start point defines the positive local X axis direction for the portal frame.

NOTE The end point must lie in the same construction level as the start point.

The **Portal Frame** dialog box opens.

5. On the **Spans** page, select the first span and click **Edit...**
6. Define the eaves level, select the frame type, and complete the frame geometry for the span.
7. Click **OK**.
8. According to your needs, do one or all of the following:
   - On the **Rafters** page, review the rafter section sizes.
   - On the **Columns** page, review the column section sizes.
   - On the **Haunches** page, specify eaves and apex haunches. See:Portal frame haunch geometry (page 469)
   - On the **Bases** page, specify the base fixity, and adjust the base levels according to your needs.

     NOTE You can also specify bases as *nominally pinned* or *nominally fixed* if required, as follows:

     a. close the **Portal Frame** dialog box,

     b. open the appropriate **Support** in the Portal Frame **Properties** window,

     c. set the Rotational stiffness x **Type** as *nominally pinned* or *nominally fixed*,

     d. enter the stiffness %

     For more details, see: Partial fixity of column bases (page 492).
   - Use the **Valleys**, **Ties**, **Tie Members**, **Parapets**, and **Parapet Members** pages to specify any additional elements.
9. To create the portal frame, click **OK**.

Modify the properties of an existing portal frame

- According to your needs, do one of the following:

To	Do this
Modify the overall frame properties	1. Hover the mouse pointer over the desired portal frame, so that it becomes highlighted. 2. Right-click the portal frame.

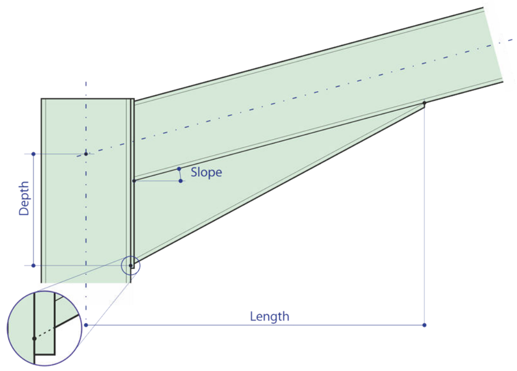
	3. In the context menu, select **Edit [element name]**. The **Portal Frame** dialog box opens. 4. Modify the properties according to your needs. 5. Click **OK**.
Modify the properties of an individual portal frame member	1. Hover the mouse pointer over the desired portal frame member, so that it becomes highlighted. 2. In the **Select Entity** tooltip, scroll until the desired member is highlighted. 3. In the model, right-click the member. 4. In the context menu, select **Edit [element name]**. The **Properties** dialog box opens. 5. Modify the properties according to your needs. 6. Click **OK**.

Add copy or mirror spans in an existing portal frame

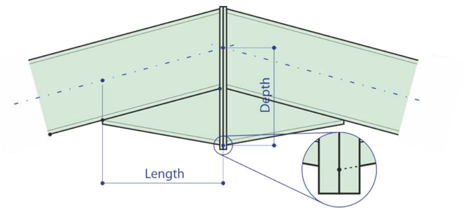
1. Open the **Portal Frame** dialog box (as described in the above topic).
2. Go to the **Spans** page of the **Portal Frame** dialog box.
3. Use the available buttons to introduce additional spans:
   - To introduce a new span at the end of the frame, click **Add**.
   - To insert a new span below the currently highlighted span, click **Insert**.
   - To copy the currently highlighted span to a pre-existing target span, click **Copy...**
   - To mirror the currently highlighted span to a pre-existing target span, click **Mirror...**
4. Click **OK**.

Portal frame haunch geometry

Eaves haunch dimensions



Apex haunch dimensions



Create cold-rolled sections

Tekla Structural Designer allows you to create multiple characteristic types of cold-rolled sections. The types are track, stud, and joist (US), or eaves beam, purlin, and rail (UK), depending on the language that you are using. For more information, see the following paragraphs.

RESTRICTION You can model and analyze cold-rolled in Tekla Structural Designer, but cold-rolled sections are not designed.

Create cold-rolled sections

1. On the **Model** tab, click any steel member type, for example, **Steel Beam**.
2. In the **Properties** window, set the **Characteristic** property to the desired cold-rolled section type.

   The properties in the **Properties** window are updated to match the selected cold-rolled section type.

3. In the **Properties** window, adjust the properties of the cold-rolled section according to your needs.
4. Click the start point of the member.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the member.

   Tekla Structural Designer creates the member.

Modify the position of a cold-rolled section

You can modify a cold-rolled section in both 2D and 3D views.

To move an entire cold-rolled section, see: Move and rotate objects (page 495).

1. Click the section that you want to modify.
2. Click the end node that you want to move.
3. Click the new position of the selected end node.

   The end node moves to the selected position.

Create wall and roof panels

Wall panels, also referred to as wind walls, allow you to apply loads calculated by the **Simple Wind Loading Generator** and **Wind Wizard** to your structure. Wall panels do not introduce any structural strength or stiffness to your structure. If you wish to introduce walls that resist gravity, or lateral loads, you must define them as concrete walls.

Roof panels allow loads placed on a sloping plane to be decomposed back to the supporting structure. Area loads on roofs can act either vertically, or normal to the roof plane. To create and modify roof panels, see the following instructions.

Create wall panels

RESTRICTION Note that:

- In order to define a wall panel, you must have already defined the grid points that define the panel vertices.
- Wall panels must lie in a single plane. Otherwise, Tekla Structural Designer will fail the panel during validation.

- The wall panel must consist of at least 3 points.

1. On the **Model** tab, click **Wall Panel**.
2. Click the start point of the panel.
3. Click the remaining points of the panel.
4. To define the end point of the panel, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates a wall panel between the selected points.

Create wall panels with parapets

NOTE To ensure the wind analysis accounts for the parapet correctly, a wall panel with a parapet should be modeled in two parts:

1. Create an ordinary wall panel up to the roof level
2. Create a second wall panel above the roof level and marking the panel as a parapet

1. Open a frame view in which you want to create the wall.
2. Create the wall panel below the roof level normally.
3. Create the wall panel above the roof level normally.

   NOTE You may need to create a new construction level to define the top level of the parapet.

4. Press the **Esc** key.
5. Hover the mouse pointer over the second wall panel, so that it becomes highlighted.
6. Click the second wall panel.
7. In the **Properties** window, select the **Is a parapet wall** option.

See also: Parapet wall panel load decomposition (page 2139)

Modify the properties of a wall panel

You can modify the properties of a panel wall in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that i becomes highlighted.
2. Click the panel to select it.

TIP If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

Create roof panels

RESTRICTION Note that:

- In order to define a roof panel, you must have already defined the grid points that define its outline.
- Roof panels must lie in a single plane. Otherwise, Tekla Structural Designer will fail the panel during validation.

1. On the **Model** tab, click **Roof Panel**.
2. Click the start point of the panel.
3. Click the remaining points of the panel.
4. To define the end point of the panel, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates a roof panel between the selected points.

Modify the properties of roof panels

You can modify the properties of roof panels in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that it becomes highlighted.
2. Click the panel to select it.

TIP If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

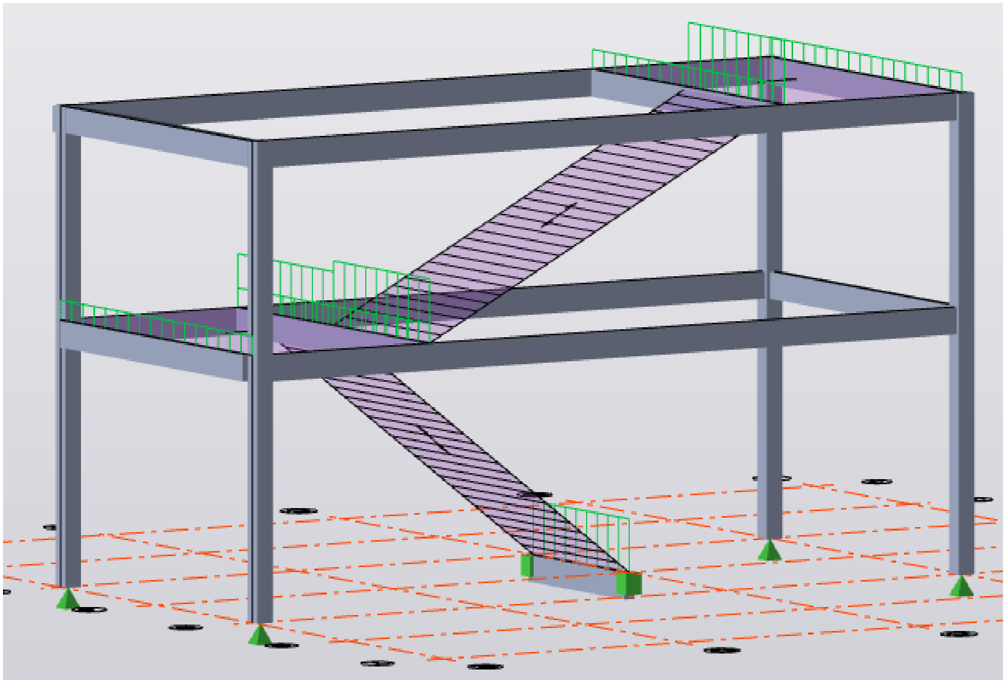
Ancillaries

Ancillaries enable the quick and efficient application of loading from ancillary items such as ladders, stairs and pipework etc - that are not part of the main structural frame.

What are ancillaries used for?

While primarily intended for use in industrial structures, some features associated with ancillary loads are also beneficial to the AEC market - eg:

- Modeling of stairs in steel, concrete, and timber structures,
- The use of stairs in combination with inactive members.



Ancillaries can also be used to model loads from:

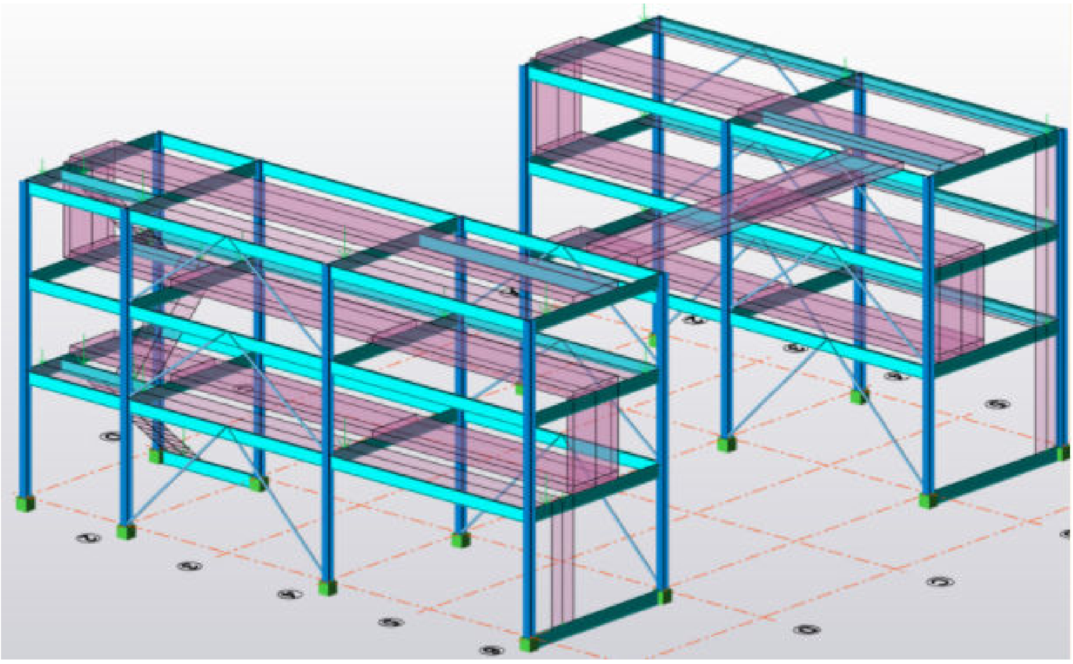
- Walkways/Catwalks
- Ladders (with/without cages)
- Access Platforms
- Operating Platforms (Storage/Standard)

- Lines of Pipework
- Lines of Cable Tray

In Tekla Structural Designer each of the above types can be created in the form of either a *line*, or *area* ancillary.

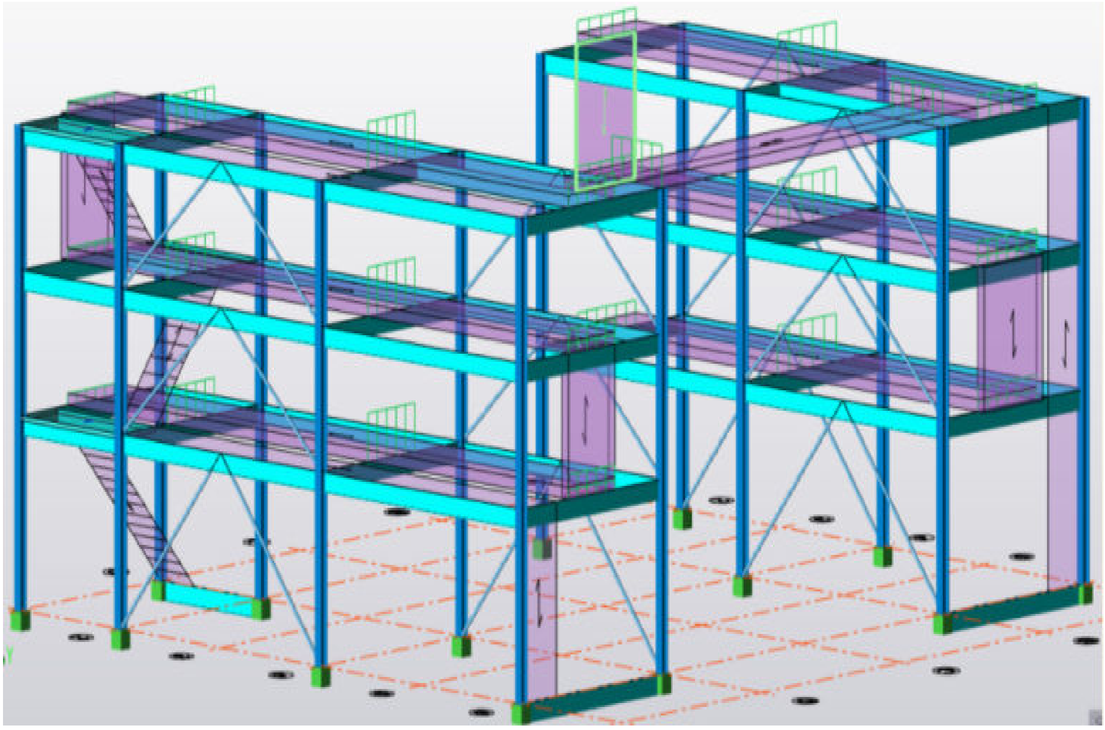
Line ancillaries

Line ancillaries can be defined horizontally, vertically or sloped. They are created by selecting a series of support points in order to create a "run". Each line has a defined width in a user defined plane (perpendicular to length).



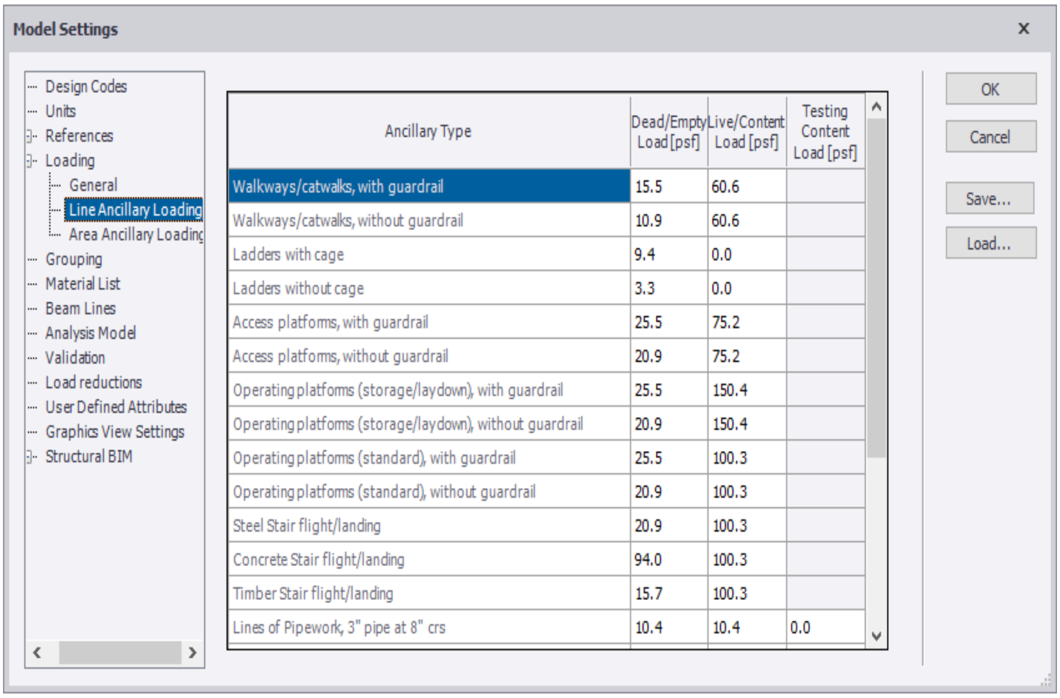
Area ancillaries

Area ancillaries can be defined horizontally, vertically or sloped. They are created in the same way as a panel, by selecting the vertices of the area.



Ancillary load default values

Each Line or Area Ancillary has default Dead Loads and Imposed/Live Loads which can be preset to be project specific from the sub-pages under **Loading** in **Model Settings**.

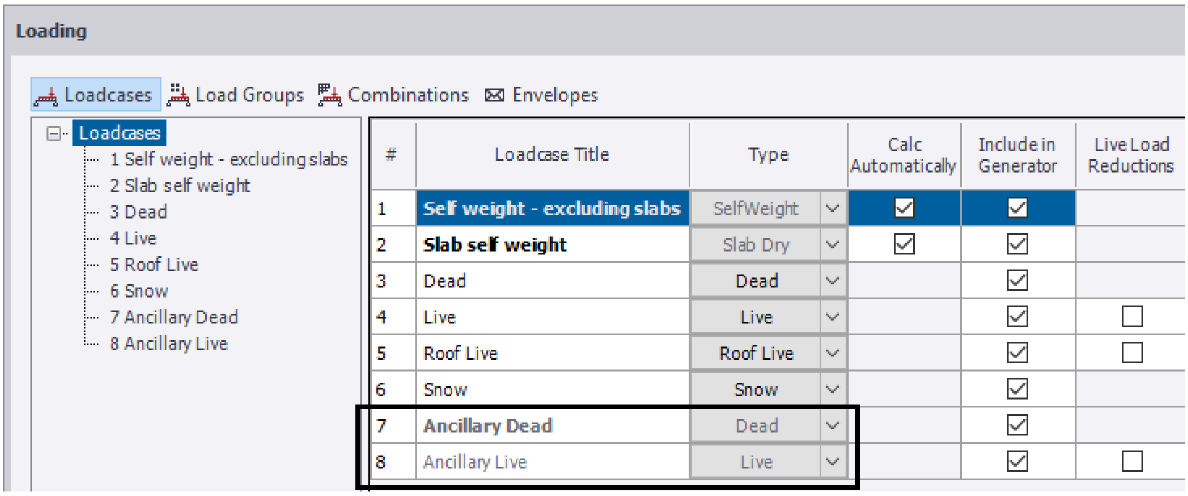


The default values can be overridden when individual ancillaries are created.

NOTE For Pipework Operating & Testing Content Loadcase Types, you can select whether these are considered as Dead or Imposed (Live) loads from **Model Settings > Loading > General**.

Ancillary loadcases

Ancillary loads are always created in dedicated ancillary loadcases which are automatically added and removed as the loads are added/deleted. These dedicated ancillary loadcases specifically aid combination building for Industrial design.



NOTE If working to Eurocodes, the ancillary imposed case Ψ and ϕ factors default to zero and must be manually defined.

Ancillary load decomposition

All decomposed loads from ancillaries are present in the analysis and design.

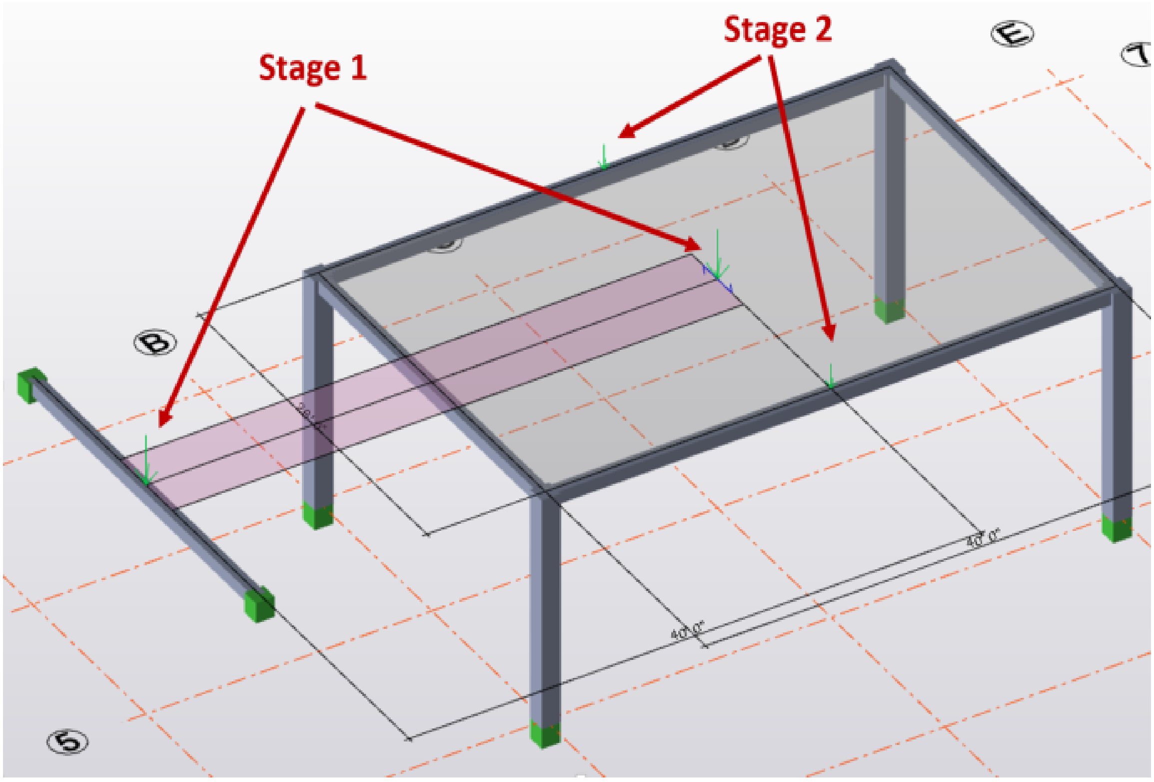
Once loads are decomposed the ancillaries themselves play no further part in analysis and design.

Line ancillary decomposition

Line ancillaries are treated as simply supported beams, spanning onto supporting members/slabs.

Decomposition is in two stages from the line ancillary to its supports then onwards.

In the below example the line ancillary is supported by a beam at one end and a slab spanning perpendicular to the ancillary at the other. At the slab end, stage 2 decomposition occurs to distribute the load on to beams supporting the slab.



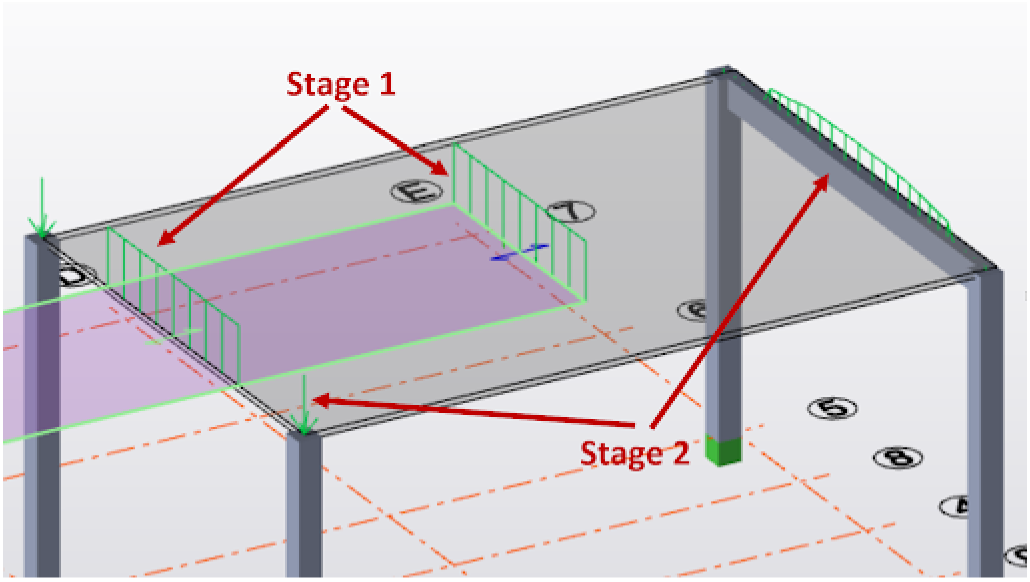
Area ancillary decomposition

In the same way as a roof panel, an area ancillary has a span direction which determines the load decomposition onto surrounding supporting members/ slabs.

Decomposition is in two stages from the area ancillary to its supports then onwards.

In the below example the area ancillary is supported by a slab which spans in the same direction as the ancillary.

The UDLs generated in the stage 1 decomposition are then decomposed on to the members supporting the slab in stage 2.

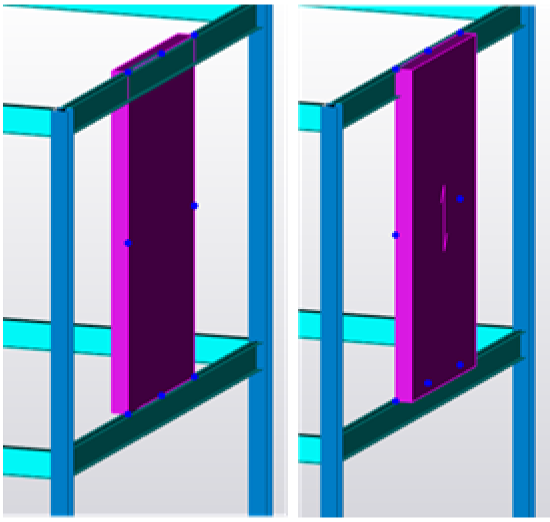


Create line ancillary loads

1. On the **Model** tab, click **Line Ancillary**.
2. In the **Properties** window,
   a. If the load is to only span between two points, uncheck the **Continuous** setting, otherwise leave it checked.
   b. Enter the load width
   c. Select the required **Type**.
   d. Accept the default load values, or define your own.
   e. Specify the remaining properties as appropriate to selected Type.

NOTE 1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.

2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

3. Click the start point of the line.
4. Click the remaining points of the line.
5. Double-click to define the end point of the line.
6. Pick a reference point to define the ancillary plane.

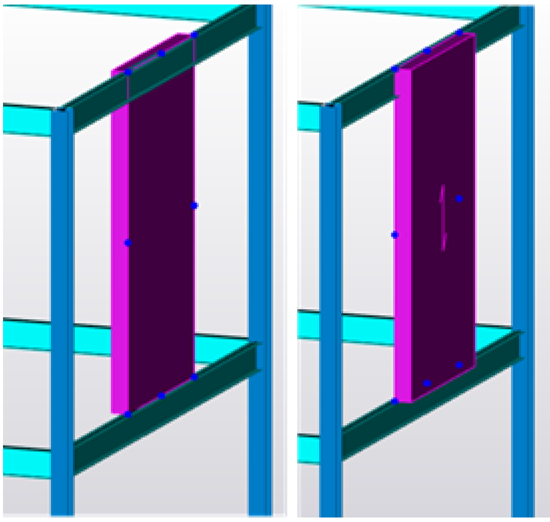
   Tekla Structural Designer creates a line ancillary between the selected points.

Create area ancillary loads

1. On the **Model** tab, click **Area Ancillary**.
2. In the **Properties** window,
   a. The rotation angle defines the span direction measured relative to the first two points clicked to identify the area.
   b. Select the required **Type**.
   c. Accept the default load values, or define your own.
   d. Specify the remaining properties as appropriate to selected Type.

   NOTE 1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.

   2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

3. Click the start point of the area.
4. Click the remaining points of the area.
5. To define the end point of the area, do one of the following:
    - Double-click the end point.
    - Click the end point, and click the start point again.

    Tekla Structural Designer creates an area ancillary load between the selected points.

Inactive members

Single span members can be made inactive for analysis and design while still being kept in the model for load distribution and for determination of effective lengths.

This feature is particularly useful for industrial structures, as frequently not all the secondary beams are included in the analysis and design model, but they are still required to distribute load in the structure and act as restraints to supporting members.

An inactive member is totally ignored in the solver model as is any load carried by it. Therefore, to allow the load to be distributed it is instead decomposed to nodal forces in a pre-analysis decomposition stage - this happens for all applied loads on inactive members on the basis of the members being simply supported.

Which members can be made inactive?

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each have an **Active** setting in the properties which defaults to 'on' but can be unchecked.

NOTE Members cannot be made inactive if:

- they are multi-span, or,
- they support another member (active or inactive)

To make a member inactive

By selecting the Show/Alter State Active (page 875) attribute in a **Review View**, inactive members are color coded allowing the active/inactive setting to be toggled graphically.

Alternatively, you can change the setting manually in the **Properties** window as follows:

1. Select the member in a 2D or 3D view.
2. In the **Properties** window deselect **Active**.

Inactive member load decomposition

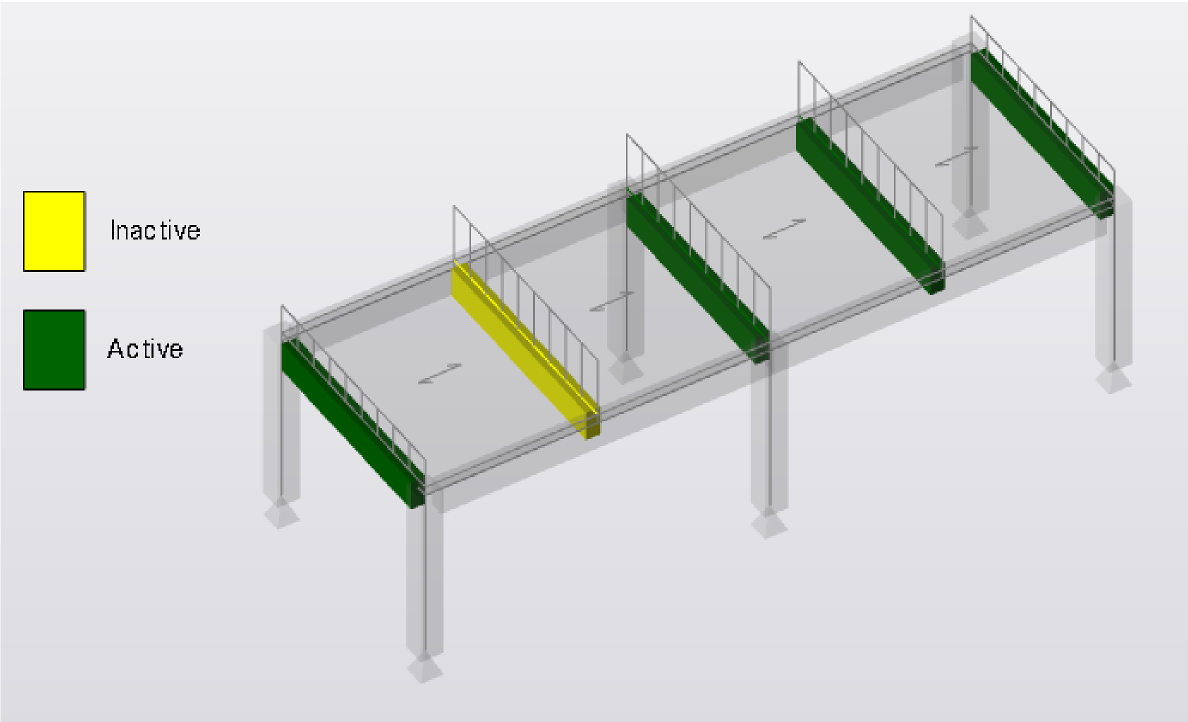
The first stage of load decomposition treats active and inactive members in exactly the same way.

This is then followed by a second stage of load decomposition for the inactive members only, as follows:

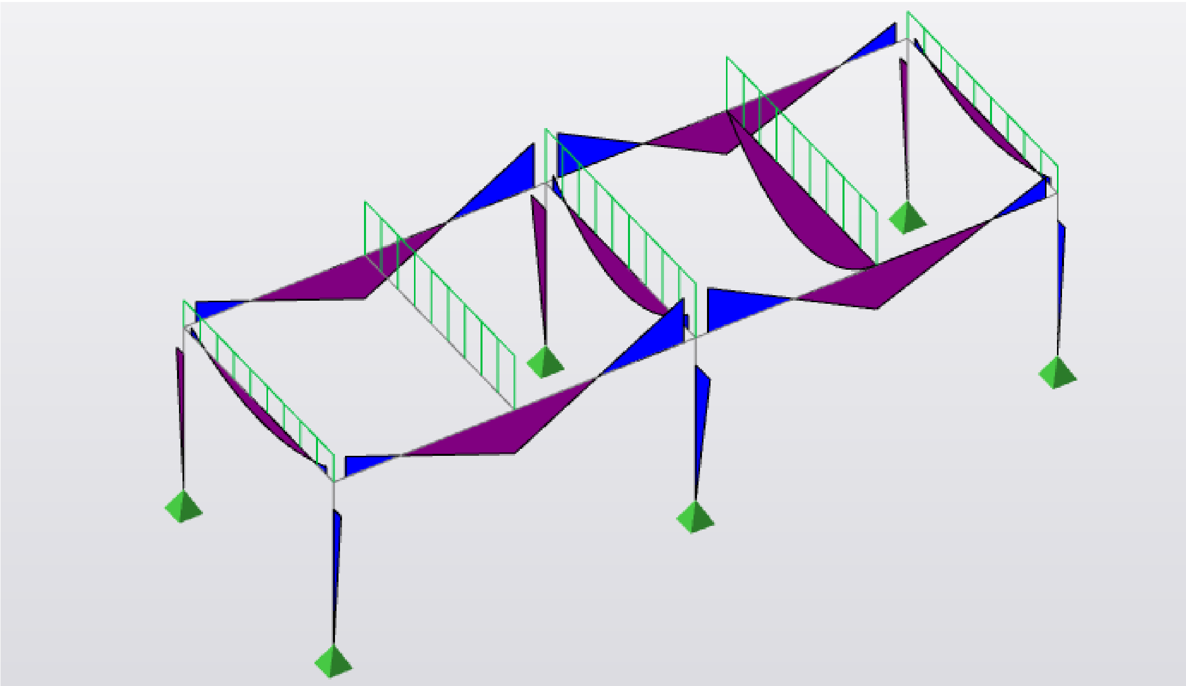
- End reactions are calculated for the inactive members.
- These are then applied as nodal forces to the supporting members.

After decomposition has taken place analysis can proceed with the inactive members removed from the solver model.

In the below example, by using the Show/Alter State Active (page 875) attribute one beam has been made inactive. The view shows decomposed loads - note that the inactive beam still has slab load decomposed to it in the same way as the active beams (as a result of the first decomposition stage).



The effects of the second decomposition stage can be seen by switching to a Results View showing major moments.



In the second stage the end reactions of the inactive beam have been applied as nodal forces to the supporting edge beams in the first bay causing moments to be generated in them, (without this second stage the two supporting beams would have otherwise been completely unloaded.)

No moments are generated in the inactive beam itself because after the decomposition has taken place it does not exist in the solver model.

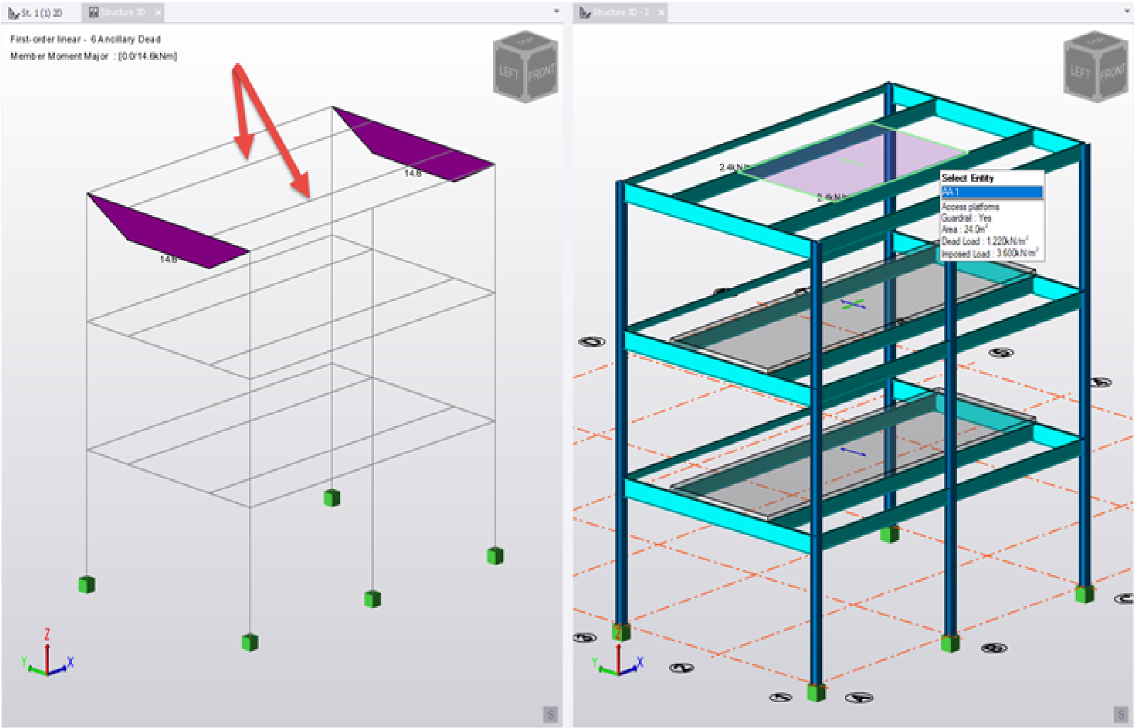
Typical usage cases for inactive members

There are potentially many uses for inactive members, the following being some examples.

Industrial Structures

Inactive members can be used in conjunction with ancillaries in order to apply loads to the structure while not themselves participating in the analysis.

In the below example an area ancillary decomposes load on to two inactive beams, which is then applied as nodal forces on to the supporting members.

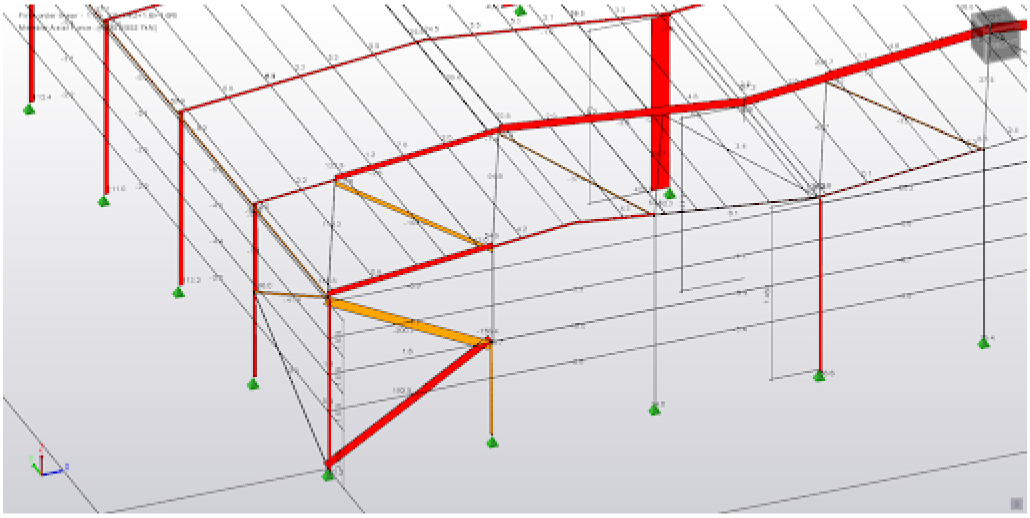


Purlins and Rails

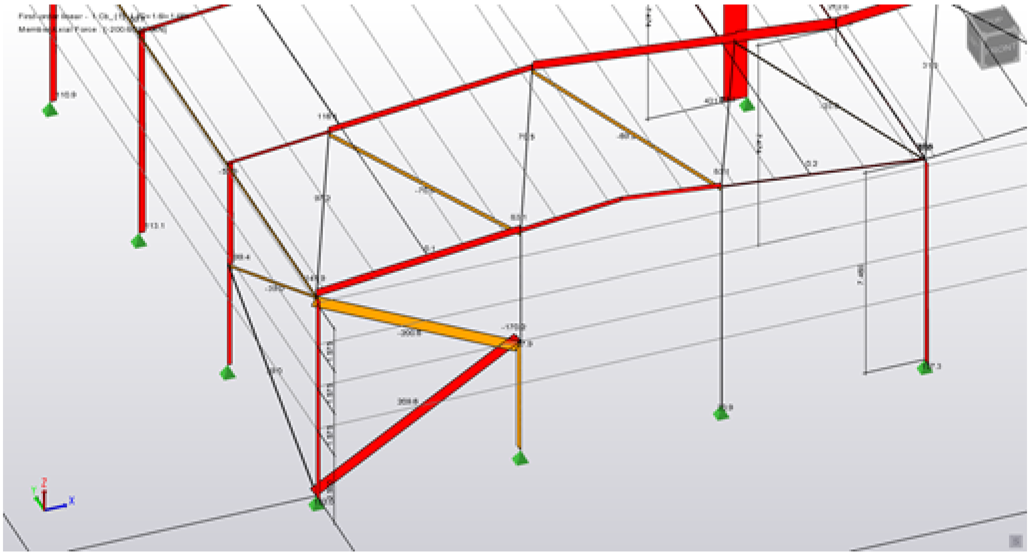
In large portal sheds, purlins and rails may be included in the model to act as a means of load distribution; the engineer would not want to see the forces in them when reviewing forces in the main structural members, or consider the small axial loads that develop in purlins and rails when gravity load cases are analysed.

By setting the purlins and rails to be inactive these goals are achieved.

This is demonstrated in the below example - in the top view the purlins are rails are active, consequently small forces develop in them in gravity combinations and the results views are quite clutterred with values that are of no interest to the engineer.



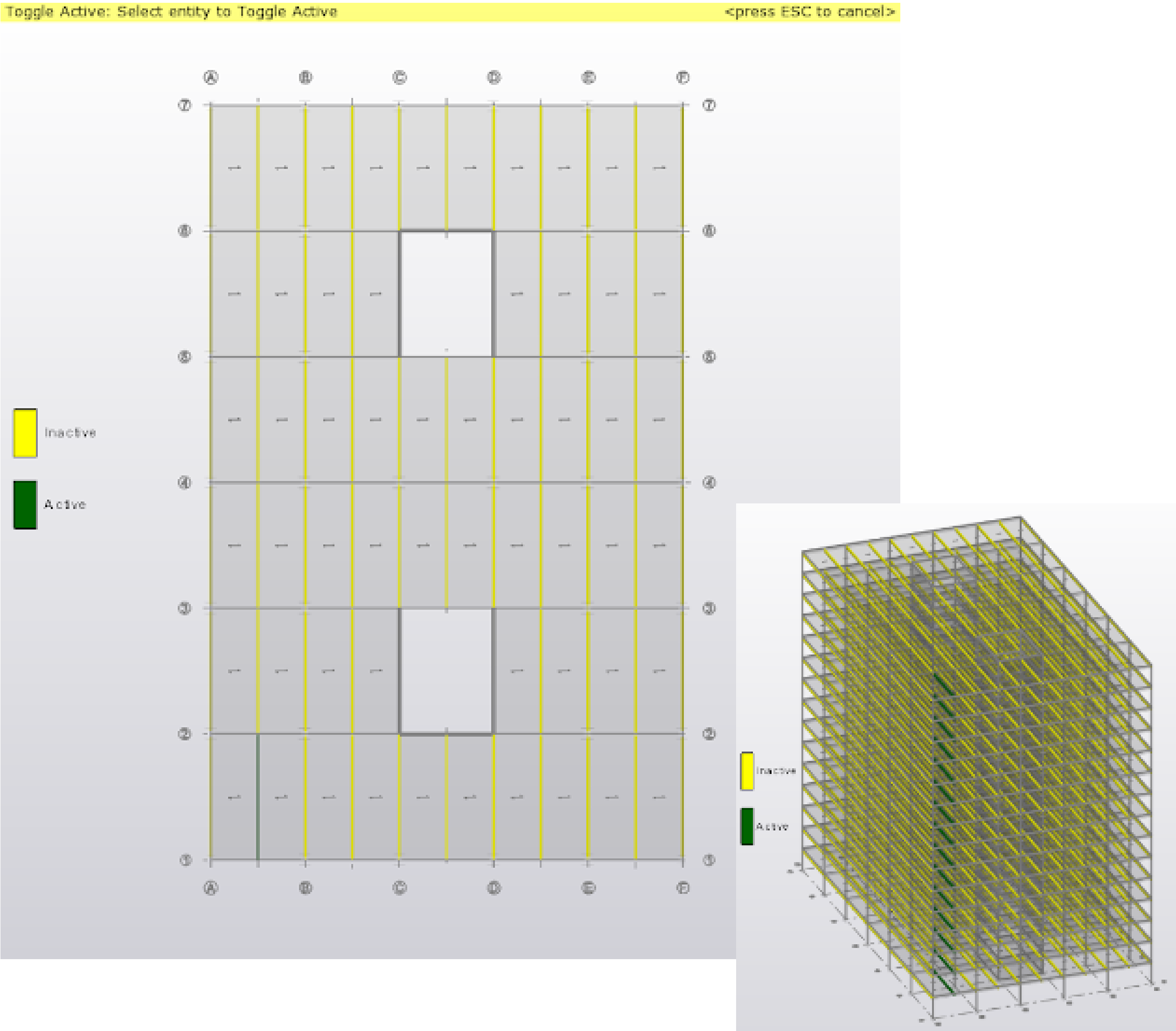
In the second view the purlins and rails have been set inactive, providing a clearer display and showing the values the engineer would expect/assume from such a combination.



Grouped Design

Design can be speeded up by enabling group design and limiting the number of active members in a group.

This is demonstrated in the below example of a regular floor, all beams in the group apart from one have been made inactive which significantly reduces the design time.

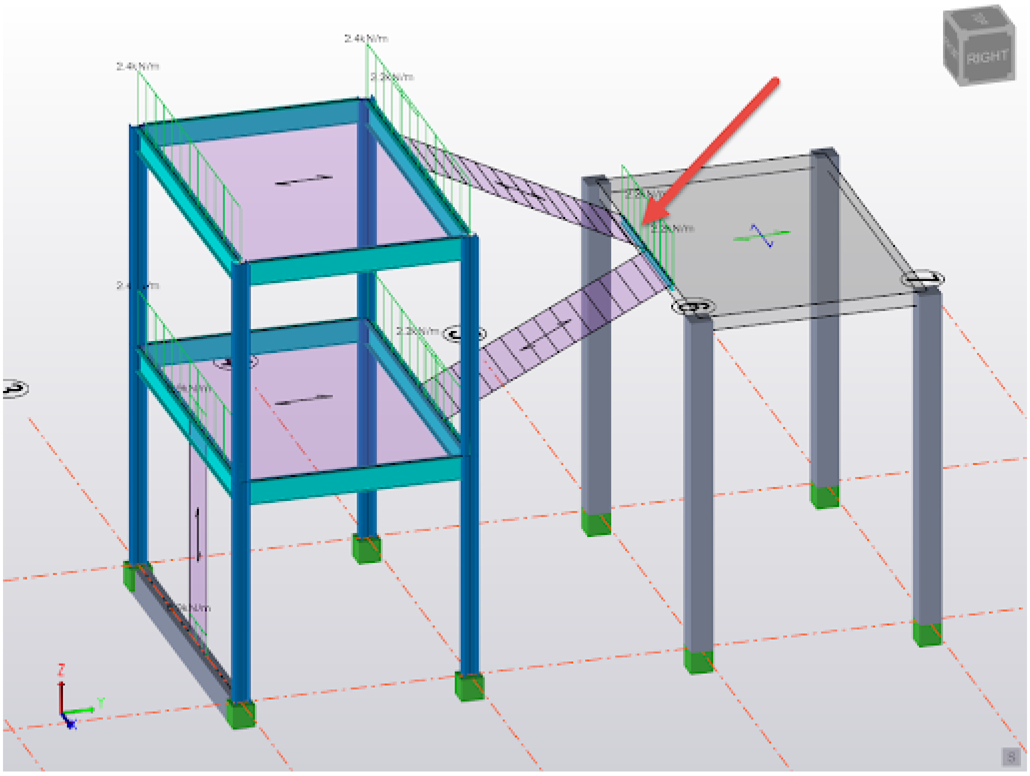


NOTE When making the non-critical beams inactive to speed up design:

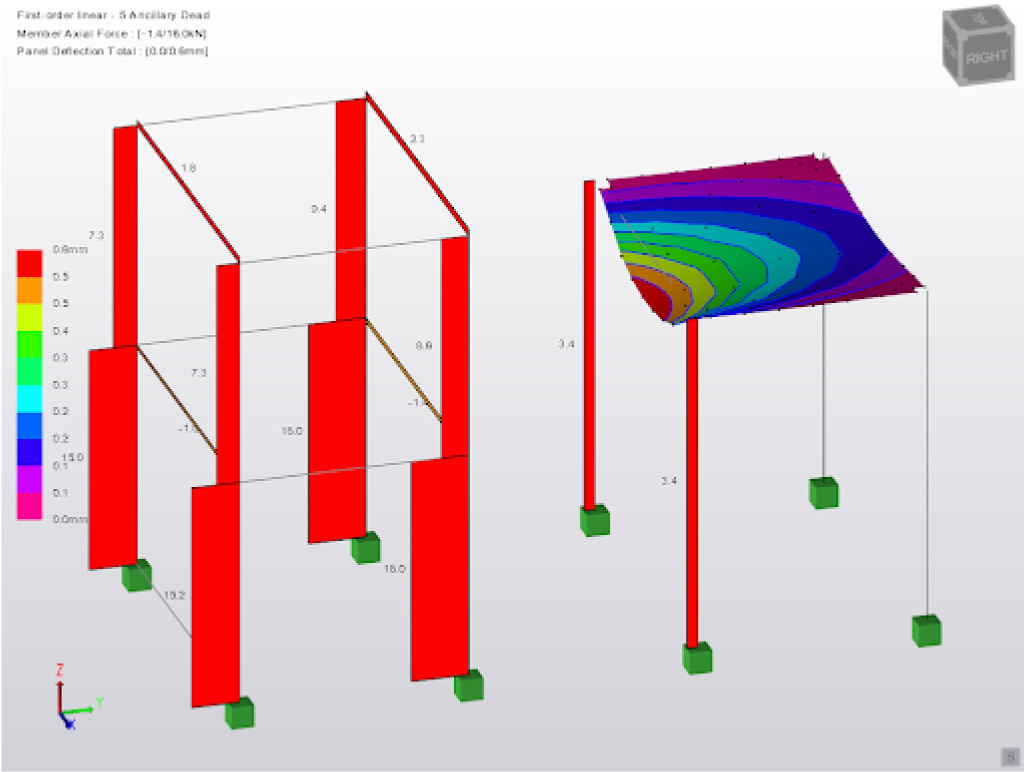
- You are responsible for choosing critical beam(s)
- Primary beams cannot be set to inactive

To support stairs at a slab edge

If an area ancillary stair spans onto a slab edge, in order to decompose the stair load on to the slab you can add an inactive member along just that part of the slab edge to which the stair attaches.



Provided that the slab is meshed for 3D analysis, the inactive member provides a load path - the load becomes point loads on slab at each end inactive member (i.e. at the edges of the stair).



Create supports

Supports allow you to constrain points in your structure vertically and rotationally. You can use supports to model connections to existing structures, so that you do not need to incorporate them in your current model. In addition, you can change the fixity provided at an existing support by modifying the support properties.

Create a single support

RESTRICTION	You can only place supports at existing grid points.

1. On the **Model** tab, click **Support**.

   The support will adopt the properties currently displayed in the **Properties** window.

2. If necessary, in the **Properties** window, adjust the support properties.
3. Click the grid point where you want to create the support.

Create a rotated support using 3 grid points

You can apply a local coordinate system to your supports, allowing the X and Y axes to be rotated about global the Z axis.

NOTE The z direction of the support cannot be changed.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that the **3 Grid Points** option is selected.
4. Click the grid point where you want to create the support.
5. To define the support direction along the x axis, click a second grid point.
6. To define the support direction along the y axis, click a third grid point.

Create spring supports

You can create linear and non linear spring supports in a similar fashion.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that in the direction where you want the spring to act, the degree of freedom is set to **Free**.
4. In the appropriate stiffness properties, do one of the following:
   - Select the **Spring Linear** option, and define a single stiffness value that acts in both the positive and negative directions.
   - Select the **Spring Non-linear** option, and define two stiffness values, one to act in the positive direction and another to act in the negative direction.
5. Click the grid point where you want to create the support.

Create nominally pinned or nominally fixed supports

A support placed under a column is a special case which can be specified as nominally pinned, or nominally fixed if required.

NOTE A validation error is produced if a nominally pinned or nominally fixed support is placed at any other location.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.

3. Ensure that the Mx and/ or My directions about which you require partial fixity are set to **Free**.
4. In the appropriate rotational stiffness properties, do one of the following:
    - Select the **Nominally pinned** option, and define the required stiffness value as a percentage of the column stiffness 4EI/L.
    - Select the **Nominally fixed** option, and leave the stiffness value as 100% of the column stiffness 4EI/L.
5. Click the grid point where you want to create the support.

For more information, see: Partial fixity of column bases (page 492)

Modify support properties

You can modify support properties in both 2D and 3D views.

1. Hover the mouse pointer over the desired support, so that it becomes highlighted.
2. Click the support.

    TIP You can add further supports to the current selection by holding down the **Ctrl** key and clicking the additional supports.

3. Go to the **Properties** window.
4. Modify the support properties according to your needs.

    Tekla Structural Designer automatically applies any changes to the selected supports.

Partial fixity of column bases

Two additional types of rotational linear spring are provided to allow partial fixity to be modelled, these are:

- Nominally pinned
- Nominally fixed

These are specifically provided for supports under columns (of any material), but will result in a validation error if placed under walls, or if they are used for any other supports.

The support stiffness is based on the column properties (E*I/L)

- E = Young's Modulus of the column
- I = relevant bending stiffness (Ixx or Iyy) of the column
- L = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialog, i.e. combined length of all the stacks until a floor is found.

NOTE Where no Floor has been defined in the column above the support, then L is taken as total length of column.

Partial fixity spring stiffness is calculated as follows for each of the two bending releases Mx and My:

- Nominally pinned (spring stiffness) - x% * 4 * E*I/L (default x% = 10%)
- Nominally fixed (spring stiffness) - x% * 4 * E*I/L (default x% = 100%)

Since the spring stiffness is dependent upon stack height and column stiffness (E and I), the spring stiffness will change if any changes are made to column stack height, column E or I values.

In addition, since for steel, Auto Design can change the column size (and hence I value) the spring stiffness will change with any change in column size.

Create analysis elements

Analysis elements allow you to quickly model and analyze both steel, concrete and timber elements that are not in the existing section databases, and elements made of other materials. For more information on how to create and modify analysis elements, see the following paragraphs.

RESTRICTION Tekla Structural Designer analyzes analysis elements, but does not design them.

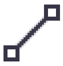
How do I edit the properties of a single model object?

How do I edit the properties of multiple entities?

Create analysis elements

TIP In order to create analysis elements in a new material, first create the :

1. On the **Home** tab, click **Materials**.
2. In the **Materials** dialog box, go to the **Material** page.
3. Set **Material Type** to **General**.
4. Click **Add...**
5. Type the name of the material in the **Grade** field.
6. Specify the material details, and click **OK**.

1. On the **Model** tab, click  **Element**.

   The element will adopt the properties currently displayed in the **Properties** window.

2. In the **Properties** window, adjust the material and properties of the element according to your needs.

3. Click the start point of the element.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

4. Click the end point of the element.

   Tekla Structural Designer creates the element.

Modify the position of analysis elements

You can modify the position of an individual analysis element in both 2D and 3D views.

1. Select the element that you want to move.

2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

4.3 Edit the model

After creating the model and the necessary members within it, you may need to make some modifications.

For example you may want to:

- Edit the properties of (page 360) entities

- Re-position entities (page 361) by moving nodes or edges

Additional model editing commands are located on the **Edit** tab, these allow you to:

- copy objects (page 495) and loads
- move objects (page 495) or move the model
- mirror objects (page 496)
- delete objects (page 503)
- join and split members
- Automatically join all concrete beams (page 505)
- Reverse member axes and panel faces (page 506)
- use cutting planes to hide a part of your model
- remove any unused unused slopes, frames, construction and grid lines
- merge planes (page 511)
- Rationalize the model (page 509)
- create infill members
- edit or add free points

Copy and rotate objects

To copy, or copy and rotate objects, see the following instructions.

1. Select the objects that you want to copy.
2. On the **Edit** toolbar, click **Copy**.
3. If you want to rotate the object when you copy it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to copy the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.
6. Place more copies in your model, or press **Esc** to finish.

Move and rotate objects

To move, or move and rotate objects, see the following instructions.

1. Select the objects that you want to move.
2. On the **Edit** toolbar, click **Move**.

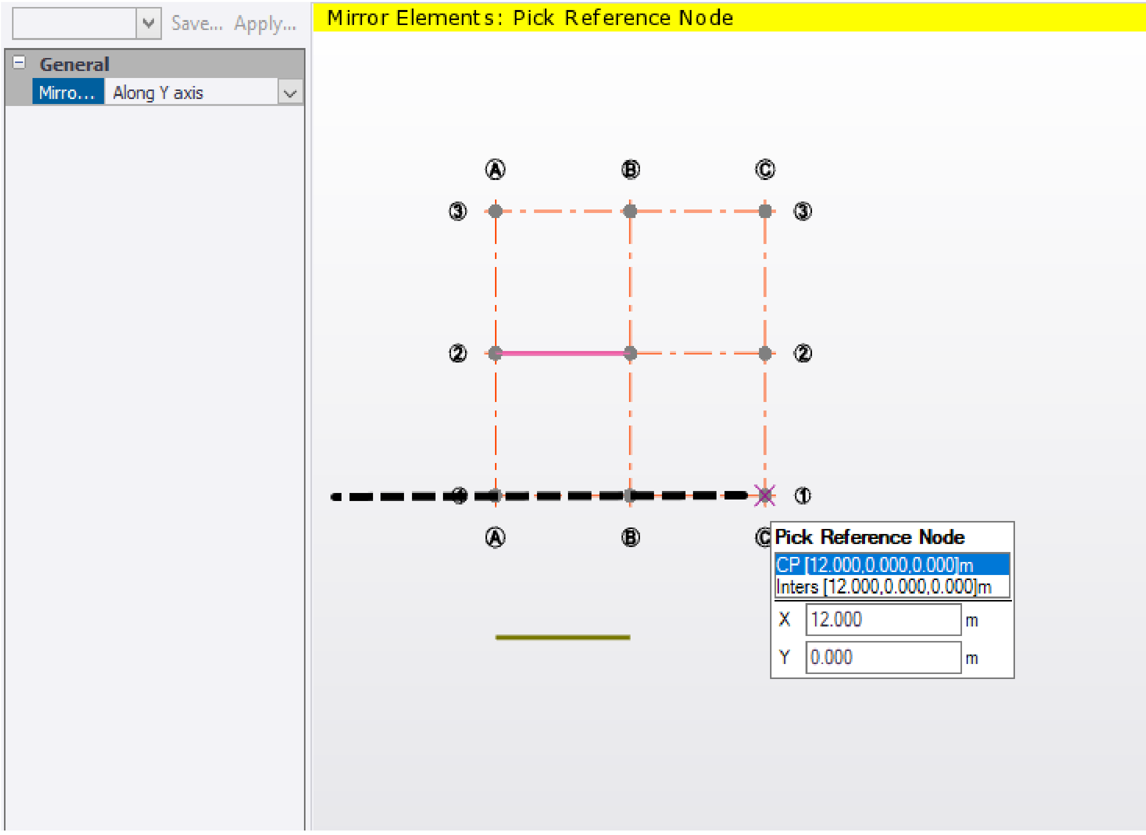
3. If you want to rotate the object while moving it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to move the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.

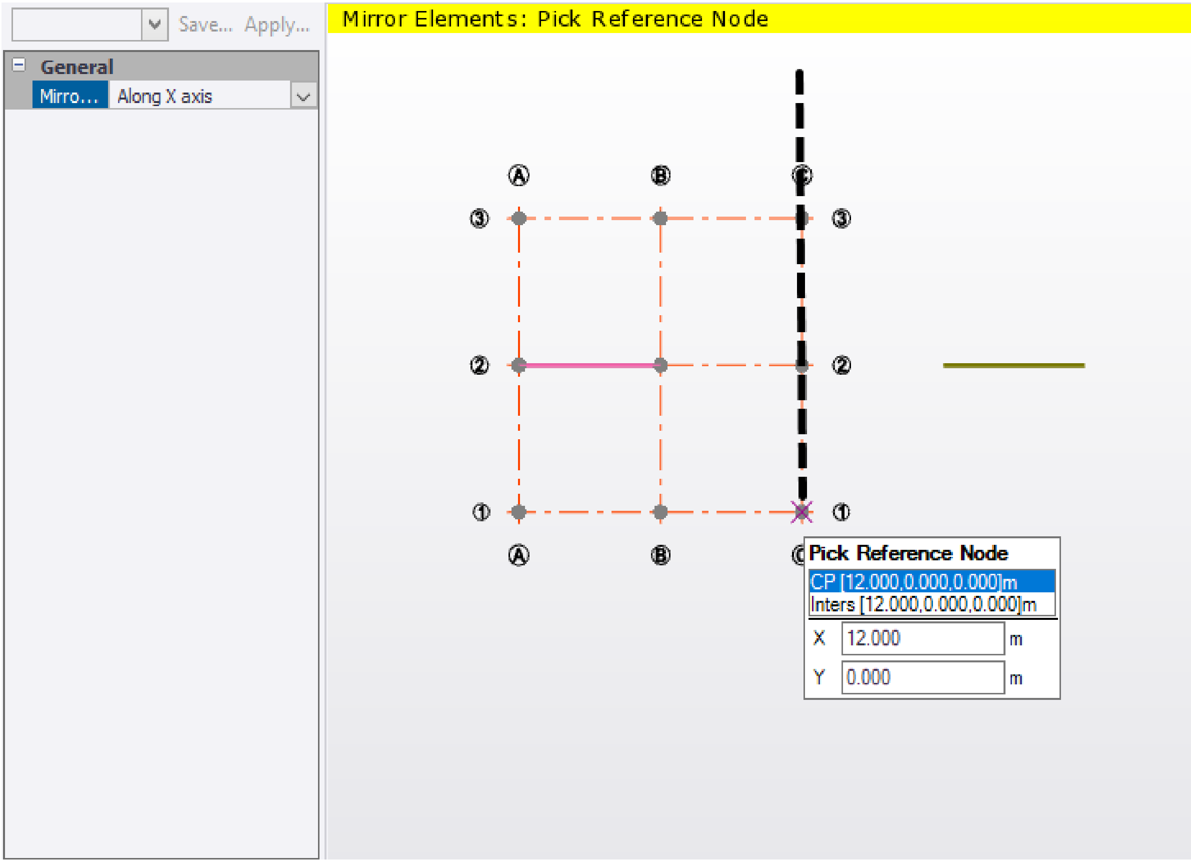
Mirror objects to new locations

To copy existing elements by mirroring them, see the following instructions.

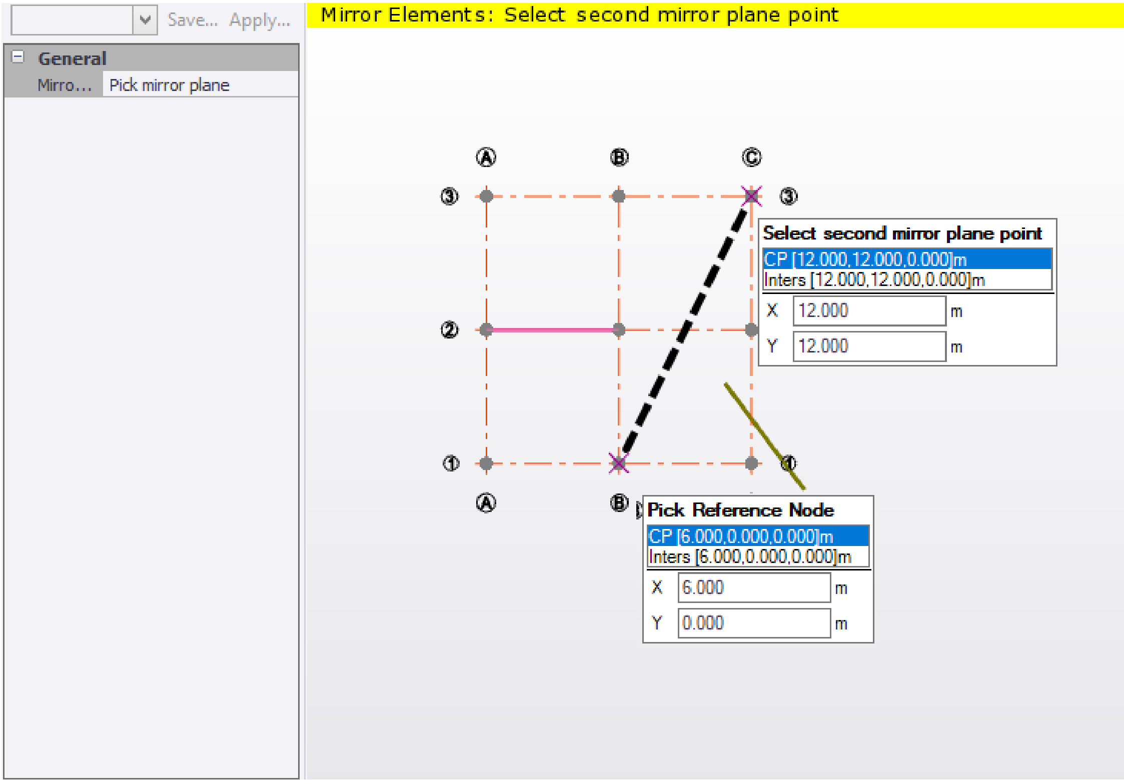
1. Select the objects that you want to mirror.
2. On the **Edit** toolbar, click **Mirror**.
3. In the **Properties** window, select the desired **Mirror mode**:
   - **Along Y axis** mirrors the objects about a global XZ plane defined by a single reference point.



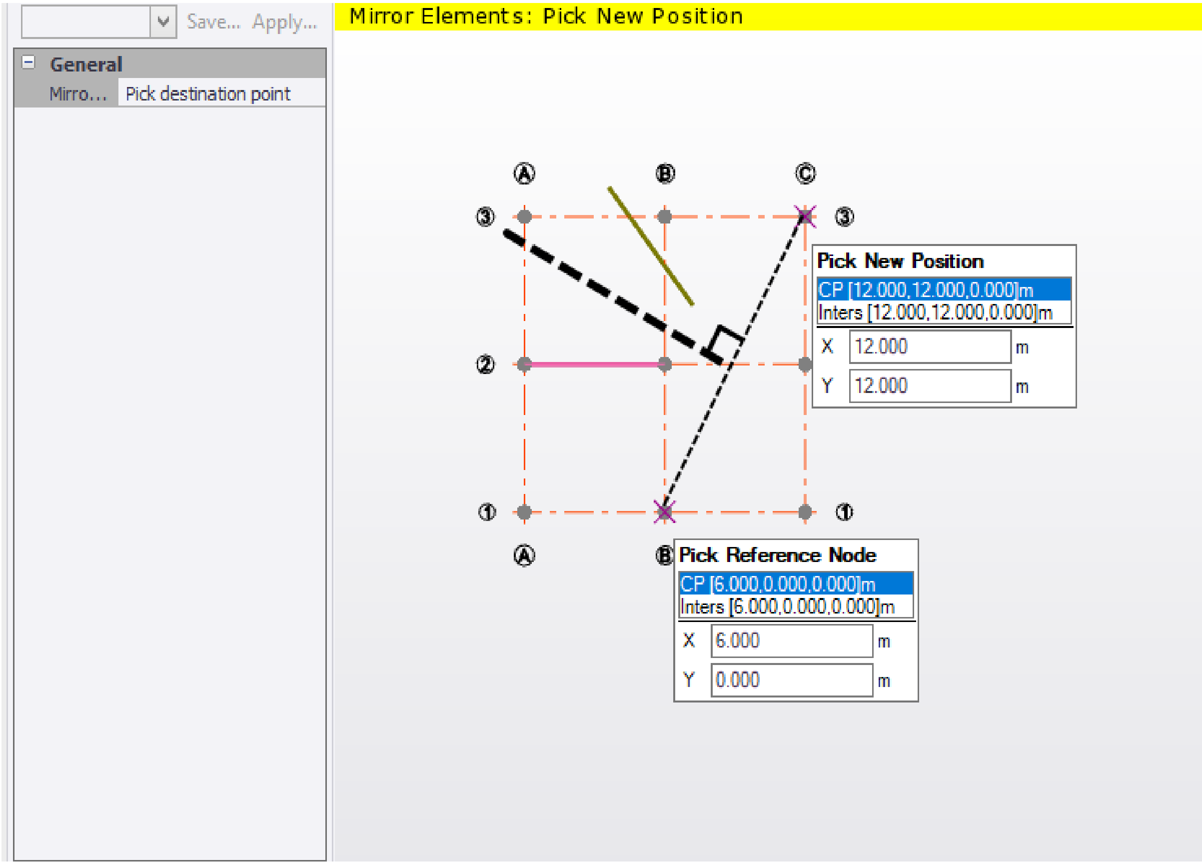
- **Along X axis** mirrors the objects about a global YZ plane defined by a single reference point.



- **Pick mirror plane** mirrors the objects about a plane defined between two points.



- **Pick destination point** mirrors the objects about a plane which perpendicularly bisects a line defined between two points.



4. In the model, click the reference points.
5. If you are using the **Pick mirror plane** or **Pick destination point** mode, click the second point to define the mirror plane.

See also

Copy and rotate objects (page 495)

Copy loads

Tekla Structural Designer allows you to quickly apply the same loads to multiple members by using the **Copy Loads** command.

Copy all member loads from one span to another

1. On the **Edit** toolbar, click **Copy Loads**.

2. Go to the **Properties** window.
3. Set **Mode** to **Copy Span Loading**.
4. Select whether you want to copy loads in the current loadcase or in all loadcases.

   NOTE If you select the current loadcase, remember to select the required loadcase in the (page 247).

5. In the model, click the span that contains the member loads that you want to copy.
6. Click the span that you want to apply the loads to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member loads to the selected span.
7. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different loaded element span to copy loads from.
   - Press **Esc** twice to finish.

Only copy one member load to another span

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
3. In the model, click the member load that you want to copy.
4. Click the span that you want to apply the load to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member load to the selected span.
5. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel area, level, and slab loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 247), select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
4. Click the area, slab, or level load that you want to copy.

5. Click the panel that you want to apply the load to.

   Tekla Structural Designer copies the load to the selected panel.

6. Do one of the following:

   - Click additional panels to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel point, line, and patch loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 247), select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Plane Loads**.
4. Click the panel point, line, or patch load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.

   A red circle indicates the original reference point for the selected loads.

5. To apply the loads, do one of the following:

   - Click a panel node to define a new reference point at that node.

     The loads are applied at the same offset from the new reference point.

   - Click anywhere within a panel boundary at a different level to define a new reference point directly above or below the original reference point.

     The loads are applied at the same offset from the new reference point.

     WARNING If you choose a new reference point that results in the loads being applied outside the panel area, Tekla Structural Designer does not apply the loads to the model. In this situation, a warning appears during validation.

6. Do one of the following:

   - Click to define further reference points to continue applying the loads. .
   - Press **Esc** to select a different loads to copy.
   - Press **Esc** twice to finish.

Copy structure loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
3. In the model, click the structure load to be copied.
4. Click the position you want to apply the load to.

   Tekla Structural Designer copies the load to the new location.
5. Do one of the following:
   - Click additional locations to continue applying the load.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy loads to another loadcase

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 247), select the loadcase from which you want to copy the loads.
3. Go to the **Properties** window.
4. Set **Mode** to **Copy Loads to another Loadcase**.
5. Select the loadcase that you want to copy the loads to.
6. In the model, click the load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.

7. To copy the loads to the loadcase displayed in the **Properties** window, press **Enter**.
8. Do one of the following:
   - Press **Esc** to select additional loads, and then press **Enter** to copy them.
   - Press **Esc** twice to finish.

Delete entities

1. Do one of the following:

To	Do this
Delete using the keyboard	• Select the entity. • Press **Delete**
Delete using the Quick Access toolbar	• Select the entity.

	• From the Quick Access toolbar choose **Delete**
Delete using the context menu	• Right click the entity you want to delete. • From the context menu choose **Delete Element**
Delete from the Edit ribbon	• Select the entity. • From the Edit ribbon choose **Delete**

Join and split members

You can use the **Join** command for joining discontinuous members, thus creating a continuous member. The **Split** command, on the other hand, allows you to split continuous members of any material.

See also

Automatically join all concrete beams (page 505)

Join members

You can use the **Join** command to manually join concrete beams, even when the **Allow automatic join end** options are not selected. This is because the **Allow automatic join end** only applies to the automatic joining that occurs during design process or when you run the **Beam Lines** command.

RESTRICTION You can only join:

- Two similar members if that the angle between the members is less than 45 degrees in both plan and elevation.
- Beams that have an end point in common.

  If Tekla Structural Designer fails to join two beams, their end points have probably been defined using different construction or grid lines.

1. On the **Edit** toolbar, click **Join**.
2. Hover the mouse pointer over the member that you want to join to another member.

   Both the original member and the member to which it will be joined become highlighted. The point where they will be joined is indicated by a red dot.

   NOTE If Tekla Structural Designer tries to join the wrong end, move the mouse pointer towards the other end of the member until Tekla Structural Designer indicates the correct end.

3. Click the highlighted members to join them.
4. Click additional members to join them, or press **Esc** to finish.

Split members

RESTRICTION You can only split members that have previously been joined.

1. On the **Edit** toolbar, click **Split**.
2. Hover the mouse pointer over previously joined members that you want to split.

   The member is highlighted, and Tekla Structural Designer uses a red dot to indicate the point where the member will split.
3. According to your needs, do one of the following:
   - Click the member to split it at the indicated point.
   - Move the mouse pointer further along the member to identify other points where the member could be split.

   NOTE If the member being split is a concrete beam, Tekla Structural Designer splits the member into two separate beams. The first beam has the **Allow automatic join end 2** option cleared, whereas the second one has the **Allow automatic join end 2** option cleared. This prevents Tekla Structural Designer from automatically making the two beams continuous again when the model is designed.

Automatically join all concrete beams

Tekla Structural Designer automatically forms concrete beam lines as part of the combined analysis and design process. However, if you would prefer to have greater control yourself, you can run the **Beam Lines** command manually. This way, you can verify that continuous beam lines are formed as you intend before proceeding with the design.

When run manually, the **Beam Lines** command applies to all concrete beam members in the model, irrespective of whether they are selected or not.

NOTE Discontinuous concrete beam ends can only be joined if the **Allow automatic join end** option is selected for the appropriate beam ends at the join. Then, Tekla Structural Designer only joins the ends if all the following criteria are met:

- The angle in plan at which the two beams meet is less than the **Limiting join angle in plan** specified in **Model Settings** --> **Beam Lines**

- The angle in elevation at which the two beams meet is less than the **Limiting join angle in elevation** specified in **Model Settings** --> **Beam Lines**
- The amount by which the cross sectional areas of the two beams overlap is greater than the **Minimum section overlap** percentage specified in **Model Settings** --> **Beam Lines**

In addition, if either of the two beam ends being joined is pinned, Tekla Structural Designer does not join them unless the **Join pinned beam end** option is selected in **Model Settings** --> **Beam Lines**.

To join the concrete beams in the model automatically, do the following:

- On the **Edit** tab, click **Beam Lines**.

See also

Join and split members (page 504)

Reverse member axes and panel faces

Sometimes when creating a model, you may end up creating beams or wall panels that run or face in the wrong direction. This may lead to problems in the analysis phase. In order to fix these kinds of inconsistencies, you can use the **Reverse** command.

Reverse the local axis of a beam

You can easily end up with beams running forwards and backwards if a consistent approach has not been adopted when selecting start and end nodes. Beams that run in different directions can result in confusing force diagrams. By manually reversing the axes of beams, you can make all beams run left to right and bottom to top in a plan view.

To reverse a beam, do the following:

1. On the **Edit** toolbar, click **Reverse**.
2. Click a beam to reverse its direction.

Reverse the outward face of a wind panel

The front of each wall should be facing outwards in order to correctly determine the wind direction relative to the wall. Ensure that all the outward faces are displayed in the color assigned to the front of the wind wall.

If a wall is facing in the wrong direction, reverse it by doing the following:

1. On the **Edit** toolbar, click **Reverse**.
2. Click a wind panel to reverse its direction.

Manage cutting planes

Initially, the six cutting planes form a cube around the extents of each model. By activating a cutting plane, you can move it inwards so that it slices through the model. Everything on the positive side of the plane is temporarily hidden from the view, making it easier to work on different areas inside the model.

Activate or deactivate a cutting plane

The active cutting plane faces are shown in a different colour. (By default blue indicates the positive side of the plane and red the reverse side).

1. On the **Edit** toolbar, click **Cutting Planes**.

   Tekla Structural Designer displays the six existing cutting planes.
2. Click a cutting plane to activate it.

   Tekla Structural Designer displays the positive side of the plane in blue, and the reverse side in red.

   TIP To deactivate a cutting plane, click an active plane again.

Move a cutting plane to hide a part of the model

When a cutting plane is active, an arrow projects from its center. You can use the arrow to reposition the plane.

1. Click the arrow at the center of the cutting plane.

   Tekla Structural Designer views a line that indicates the direction in which you can move the plane.
2. Click the desired position of the plane, or press **F2** to type the exact distance.

   Tekla Structural Designer redraws the cutting plane at its new position. The cutting plane slices to the model, and everything to the positive side of the plane is hidden.

Re-display a hidden part of the model

- Click the cutting plane to deactivate it.

  Tekla Structural Designer displays any previously hidden parts of the model.

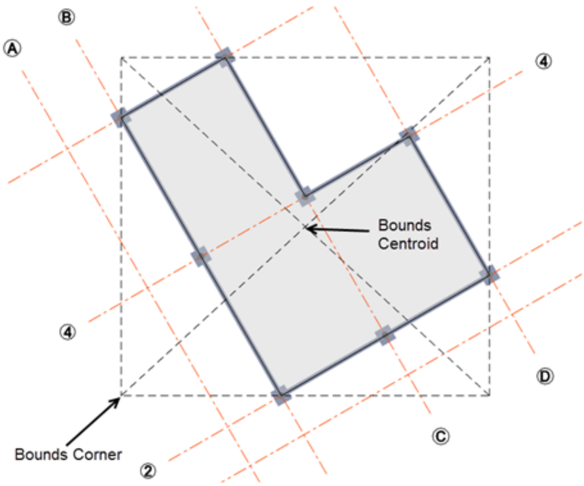
Move the model or the DXF shadow

If necessary, you can relocate the entire model to a new coordinate in the XY plane by using the **Move Model**. You can relocate the DXF shadow of the model similarly with the **Move DXF Shadow** command.

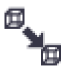
Move the model

In order to move the model, you first have to decide which reference point to use.

You can determine the reference points by drawing an imaginary box aligned to X and Y around the extents of the model, as shown in the following image:



Bounds Corner equals the lower left hand corner of the imaginary box, whereas **Bounds Centroid** equals the centroid of the imaginary box.

1. On the **Edit** toolbar, click  **Move Model**.
2. Set the reference point to either **Bounds Corner** or **Bounds Centroid**.
3. Enter the required target coordinate of the reference point.
4. Click **Move**.

Move the DXF shadow

The **Move DXF Shadow** command is only available if you have previously imported a DXF shadow (page 308).

1. On the **Edit** toolbar, click **Move DXF Shadow**.
2. Select an existing point in the DXF shadow as the reference point.
3. Click the new position of the reference point.

Rationalize the model

In order to remove all unused sloped planes, frames, grids, and construction lines in your model, you can use the **Rationalize** command. If necessary, the **Rationalize** command also allows you to update all grid and construction lines to extend only a short length beyond the point where they are required. After running the command, you can see a summary of the changes in the **Process Window**.

Delete unused sloped planes, frames, grids, and construction lines

1. On the **Edit** toolbar, click **Rationalize**.

   The **Rationalize** dialog box opens.
2. Select the unused items that you want to remove.
3. Click **OK**.

   Tekla Structural Designer deletes the unused items of the selected types, and their associated items.

Update grid and construction line length

1. On the **Edit** toolbar, click **Rationalize**.

   The **Rationalize** dialog box opens.
2. Select **Shrink/Extend Grid & Construction lines**.
3. Click **OK**.

   Tekla Structural Designer updates all grid and construction lines in the model to extend a fixed length (500 mm / 1'8") beyond the point where they are required.

Create infill members

In order to quickly place a pattern of infill members into selected plays in a level, sloped plane, or frame, you can use the **Create Infills** command. Note that only the bays with members attached to all sides can be selected.

RESTRICTION	You cannot create infills in 3D views.

Define the infill properties and pattern

1. On the **Edit** toolbar, click **Create Infills**.
2. Go to the **Properties** window.
3. In **Define Beams**, select one of the following:
    - **By number** allows you to specify the number of equally spaced members in the pattern.
    - **By spacing** allows you to specify the exact positions of the beams, separating them by a comma.
    - **By max spacing** allows you to create as many equally spaced beams in the pattern as possible without exceeding the max spacing.
4. In **Direction**, select one of the following:
    - **Perpendicular** allows you to have the members drawn perpendicular to the highlighted edge member.
    - **Parallel with left** allows you to have the members drawn parallel to the edge member that connects to the end 1 of the highlighted edge member.
    - **Parallel with right** allows you to have the members drawn parallel to the edge member that connect to the end 2 of the highlighted edge member.
5. Define the element parameters according to your needs.

    TIP If you have saved infill properties to a named property set, you can recall them by selecting the set in the drop list at the top of the **Properties** window.

Place the pattern in a single bay

1. Hover the mouse pointer over the required bay, adjacent to the required edge member.
2. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.

3. Click the bay to create the pattern.

Place the pattern in multiple bays

1. Hover the mouse pointer over one of the bays, adjacent to the required edge member.
2. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.
3. Click the bays in which you want to create the pattern.

Merge planes

Where a model contains very close planes, as can often occur in imported models, for example, the Merge Planes command allows the you to merge two planes by selecting a destination plane and then a source plane.

1. On the **Edit** toolbar, click **Merge Planes**.

   Existing Level, Frame, and Slope planes are displayed in the active view.

   NOTE If the view is cluttered by having all the plane types displayed simultaneously, and you only want to merge Levels, consider switching off Frames and Slopes in Scene Content.

2. Pick the destination plane.
3. Pick the source plane.

   If the planes can be merged, everything on the source plane will be moved to the destination plane, and the source plane will be deleted. If the merge is not possible, a message is given in the Process window and the model will be unchanged.

Create and manage free points

Free points can be used to connect beams, columns, braces etc. without having to define an intersection between grid lines or construction lines.

NOTE Once a free point has an item connected to it, if the item is moved a new point is created and the existing free point remains.

Create a free point

A frame is a 2D View of the model, created in a vertical plane defined by an existing grid line. Since only the members that lie within the plane of the frame are displayed, a frame view can be particularly useful for defining bracing.

1. Make sure that Free Points are visible in Scene Content.
2. Right click in the view and pick Add free point... from the context menu.
3. Enter the point co-ordinates and click OK.

Adding, moving or deleting free points from the Edit tab

1. On the **Edit** tab, click **Free Points...**

   A Free Points dialog is displayed.
2. Click **Add** to add a new point, **Delete** to remove an existing point, or click in the table to edit the co-ordinates of existing points.

Related video

Free point defined by XYZ coordinates

4.4 Validate the model

You can validate the model at any time in order to trap errors that will cause the solver to fail before the model is submitted for analysis. Validation is also automatically performed during design.

NOTE You can also measure distances and angles (page 513) to check your modeling.

Run model validation

- On the **Model** or **Load** tab, click **Validate**.

  Tekla Structural Designer performs the validation checks. If your model contains any issues, warning messages appear.

Adjust the conditions considered in model validation

1. On the **Home** tab, click **Model Settings**.

2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

Measure distances and angles

To know the exact distances or angles between different points in the model, you can use the Tekla Structural Designer measuring commands. You can find the measuring commands on the **Model** tab, in the **Miscellaneous** group.

Measure distances

To measure the distance between any two points in the model, see the following instructions.

1. On the **Model** tab, click  **Measure**.
2. Click a node to define the start position.
3. Click a second node to define the end position.

   The distance between the nodes is displayed in the current view. To clear the measurement, press **Esc**.

Measure angles

RESTRICTION	You can only measure angles in 2D Views.

1. On the **Model** tab, click **Measure Angle**.
2. Click a node to define the arc center.
3. Click a second node to define the start position.
4. Click a third node to define the end position.

   The clockwise angle between the start and end position is displayed in the current view. To clear the measurement, press **Esc**.

5 Apply loading

The **Load** tab allows you to:

- Manage load cases, groups, combinations, envelopes and patterns (page 514)
- Apply panel, member, and structure loads (page 538)
- Apply wind, snow, and seismic loads (page 559)

5.1 Manage load cases, groups, combinations, envelopes and patterns

Load cases, groups, combinations and envelopes are all managed in the Loading dialog (page 531). Load patterns are reviewed via **Update Patterns** on the **Load** tab.

- Manage load cases (page 514)
- Manage load groups (page 516)
- Manage load combinations (page 518)
- Manage load patterns (page 527)

Manage load cases

Before applying loads to your model, you must first define the loadcases within which the loads will be contained.

To define loadcases, see:

- Create load cases (page 515)
- Activate reductions in live or imposed load cases (page 515)
- Renumber all load cases (page 516)

Create load cases

When you create a new model, Tekla Structural Designer automatically creates a load case whose type is **Self weight - excluding slabs**. You cannot access the load case because Tekla Structural Designer automatically calculates the loads within it using the objects in your structure. Tekla Structural Designer also creates three other load cases that are initially empty. However, you will almost certainly need to create other load cases that contain the loads that your building must withstand.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 531) opens on the **Loadcases** page. On this page, you can see all currently existing load cases.
2. Click **Add**.
3. In **Loadcase Title**, name the load case.
4. In **Type**, select the desired load case type.
5. Select whether the load case is included when you automatically generate load combinations.
6. Click **OK**.

   Tekla Structural Designer adds the new load case to the list of load cases in the **Loading** list.

Activate reductions in live or imposed load cases

When you create a load case whose type is **Live** (US) or **Imposed** (other head codes), you can allow Tekla Structural Designer to automatically calculate load reductions in accordance with the percentages specified on the Load reductions page in **Model Settings**. To do so, see the following instructions.

Activate reductions in live and roof live load cases (US)

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 531) opens on the **Loadcases** page. On this page, you can see all currently existing load cases.
2. Select the **Live** or **Roof Live**load case to which you want to apply the reductions.
3. Select the **Live Load Reductions** option.
4. Click **OK**.

Activate reductions in imposed load cases (other head codes)

NOTE You cannot activate reductions in load cases whose type is set to **Roof Imposed**.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 531) opens on the **Loadcases** page. On this page, you can see all currently existing load cases.
2. Select the **Imposed** load case to which you want to apply the reductions.
3. Select the **Imposed Load Reductions** option.
4. Click **OK**.

Renumber all load cases

When you delete load cases from the Loading dialog (page 531), the remaining load cases retain their original load case number. If necessary, you can renumber the remaining load cases in sequence.

For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click **Loadcases**.
2. In the context menu, select **Renumber**.

Manage load groups

Load Groups facilitate the generation of combinations (primarily for design of industrial structures where many loading scenarios must be considered).

To manage load groups, see:

- Overview of load groups (page 516)
- Create load groups (page 517)
- Inclusive and exclusive load groups example (page 518)

Overview of load groups

Features of load groups

- Load Groups are an aid to building Combinations - both manually and using the Generator - and their use is entirely optional.
- Once a Combination is built using Load Groups, the link to the Load Group is lost – the Combination is made up only of Loadcases. Only Loadcases and Combinations are analysed – Load Groups are not.

- Each Load Group holds items of a single load type - e.g. “Dead” or “Live”...etc. - and can contain both multiple loadcases and other Load Groups
- The fundamental Load Groups setting is the “Class” which is either “Inclusive” or “Exclusive”
    - When “Inclusive” – all loadcases are added at once into a combination
    - When “Exclusive” – loadcases are used one at a time in combinations. Thus for example, where an Exclusive Load Group contains four loadcases, the Generator will produce four combinations for all required combinations which include the group’s load Type (e.g. “Imposed”), each containing only one of the group’s four cases.

The load groups process

To make use of Load Groups you would proceed as follows:

1. Define Loadcases as normal
2. If required, create Load Groups from Loadcases and/or other Load Groups
3. Create Combinations by combining Loadcases and/or Load Groups

The end result is Loadcases and Combinations (built up of factored loadcases). These are then run through analysis and design.

Create load groups

1. On the **Load** tab, click **Load Groups**.

    The Loading dialog (page 531) opens on the **Load Groups** page. On this page, you can see all currently existing load groups.
2. Click **Add**.
3. Enter the **Load Group Title**.
4. In **Class**, select the desired load group class.
    - Inclusive - all loadcases are added at once into a combination
    - Exclusive – loadcases are used one at a time in combinations
5. Click the load group name in the left hand panel of the dialog to display the available load cases.
6. Select each load case in turn to be included and click the right arrow button to copy it into the load group.

    NOTE A load group can only contain items of one load type.

7. Click **OK**.

   Tekla Structural Designer adds the new load group to the list of load groups.

Inclusive and exclusive load groups example

If you were to manually create a combination to include:

- Loadcase 1
- Load Group 1 ("Inclusive" which contains Loadcases 2 & 3)
- Load Group 2 ("Inclusive" which contains Loadcases 4 & 5)
- Load Group 3 ("Exclusive" which contains Loadcases 6 & 7)
- Load Group 4 ("Exclusive" which contains Loadcases 8 & 9)

When you click OK to close the Loading dialog, the following combinations are created with the relevant factors according to Load Type

Combination	Contains Load cases
Combination 1	1 + 2 + 3 + 4 + 5 + 6 + 8
Combination 2	1 + 2 + 3 + 4 + 5 + 6 + 9
Combination 3	1 + 2 + 3 + 4 + 5 + 7 + 8
Combination 4	1 + 2 + 3 + 4 + 5 + 7 + 9

See also

Create load groups (page 517)

Manage load combinations

Load combinations allow you to assemble sets of load cases, applying the appropriate factors for the strength and service condition. These factors are specific to the design code that you are using.

To manage load combinations, see:

- Load combination classes (page 518)
- Generate load combinations automatically (page 519)
- Create load combinations manually (page 520)
- Create modal mass combinations (page 521)
- Import loadcases and combinations from a spreadsheet (page 522)
- Renumber all load combinations (page 526)

Load combination classes

Combinations fall into five different classes, with a number of options available for each of the classes:

Combination class	Description	Active	Strength	Service
Construction Stage	Only required for design of composite beams	Not applicable	Not applicable	Not applicable
Gravity	Consists of gravity loads only (self weight, dead, slab dry, slab wet, imposed, roof imposed, snow)	On/Off	On/Off	On/Off
Lateral	In addition to gravity loads, contains lateral loads due to notional loads or wind	On/Off	On/Off	On/Off
Seismic	Consists of gravity and/or lateral loads as well as seismic load cases	On/Off	On	Not applicable
Modal Mass	Only required if you perform a modal analysis	On/Off	Not applicable	Not applicable

- **Active**: Selecting and clearing the option switches the combination on and off for analysis and design.
- **Strength**: If the **Strength** option is not selected and the combination is active, the combination is not assessed for design.
- **Service**: If the **Service** options is not selected and the combination is active, the combination is not assessed for deflection.

Generate load combinations automatically

The easiest way to create load combinations is to generate the combinations automatically. In order to do so, see the following instructions.

NOTE **Construction Stage** combinations must be created manually.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 531) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Click **Generate...**

   The **Combination Generator** dialog box opens.

3. On the first page of the dialog, specify the initial parameters according to your needs.

   Depending on the number of combination types that you selected on the first page, Tekla Structural Designer creates one or more pages of combinations.

4. Click **Next**.
5. Review each page of combinations, and adjust them as necessary.
6. Click **Next** to move on to the next page.
7. To save the load combination, click **Finish**.

   Tekla Structural Designer adds the load combination to the list of combinations available in the **Loading** list.

   TIP To review the factors and options that have been applied to a combination, click the combination name in the list on the left side of the **Loading** dialog box.

Create load combinations manually

If necessary, you can create load combinations manually. In this case, Tekla Structural Designer uses a default factor for each load case when you add it to the combination.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 531) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, select the combination class.

5. Depending on the combination class that you selected, do some of the following:
   - Select **Active** in order to include the combination in analysis or design.
   - Select **Strength** in order to assess the combination for design.
   - Select **Service** in order to assess the combination for deflection.
6. In the left side pane of the **Loading** dialog box, click the combination name.
7. Select the load cases that you want to add in the combination, and click **>>**.
8. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

TIP To review the factors and options that have been applied to a combination, click the combination name on the left side pane of the **Loading** dialog box.

Create modal mass combinations

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 531) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, set the combination class to **Modal Mass**.
5. In the left side pane of the **Loading** dialog box, click the combination name.
6. Select the load cases that you want to add in the combination, and click **>>**.
7. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
8. On the **Second order effects** tab, define the amplifier that you want to apply to the combination.
9. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

Import loadcases and combinations from a spreadsheet

Loadcases and combinations defined in an Excel Spreadsheet can be imported via the 'Import...' button on the Loading dialog.

The data must be in the form of a matrix of combination factors in which load cases form the row headers and combinations the column headers or vice-versa. Where a matrix value is blank the associated case is omitted from the combination. The load type can be set by including in the loadcase name "DL" for dead load, "LL" for Imposed and "Wind" for Wind (without quotes).

The selected spreadsheet's Worksheets are automatically listed and one is selected for import.

Flexible controls allow definition of whether Loadcases are listed in a column and combinations in a row or vice-versa and selection of the case and combination column/row sources. Identified cases and combinations can be reviewed prior to import and the case Type can be edited as necessary.

NOTE The following default/ automatically determined load cases are not supported by this feature and would need to be manually added to combinations after import; Self weight - excluding slabs, Slab self weight, Notional Loads (EHF, NHF and NL).

Example

We want to import the following loadcases and combinations:

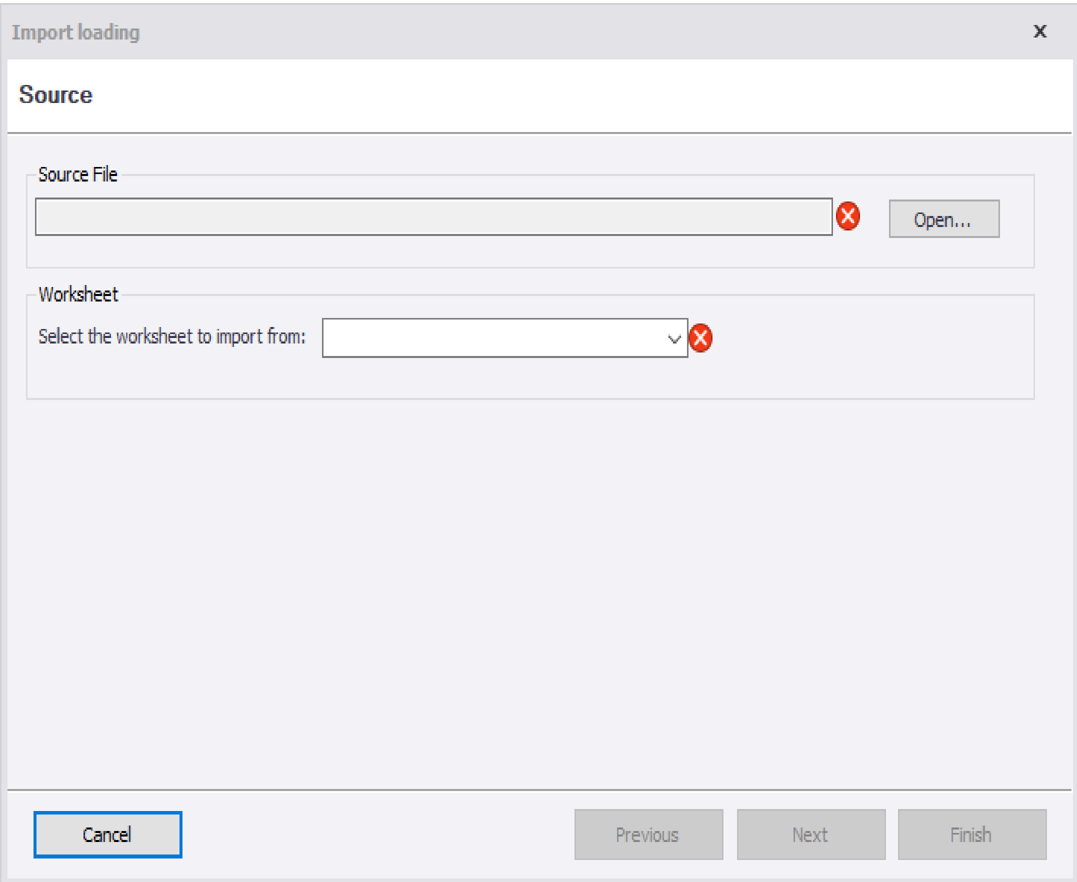
Combination Name	Loadcase	Strength Factor
dead + imposed	DL Dead	1.35
	LL Imposed	1.5
Dead+Imp+Wind Accomp	DL Dead	1.35
	LL Imposed	1.5
	Wind	0.75
Dead+Imp Accomp +Wind	DL Dead	1.35
	LL Imposed	1.05
	Wind	1.5
Dead+Wind	DL Dead	1.35
	Wind	1.5

1. In an Excel spreadsheet a matrix of loadcases and combinations is set out like this:

	A	B	C	D	E
1			Combinations		
2	Cases	Dead+Imposed	Dead+Imp+Wind Accomp	Dead+Imp Accomp+Wind	Dead+Wind
3	DL Dead	1.35	1.35	1.35	1.35
4	LL Imposed	1.50	1.50	1.05	
5	Wind		0.75	1.50	1.50
6					

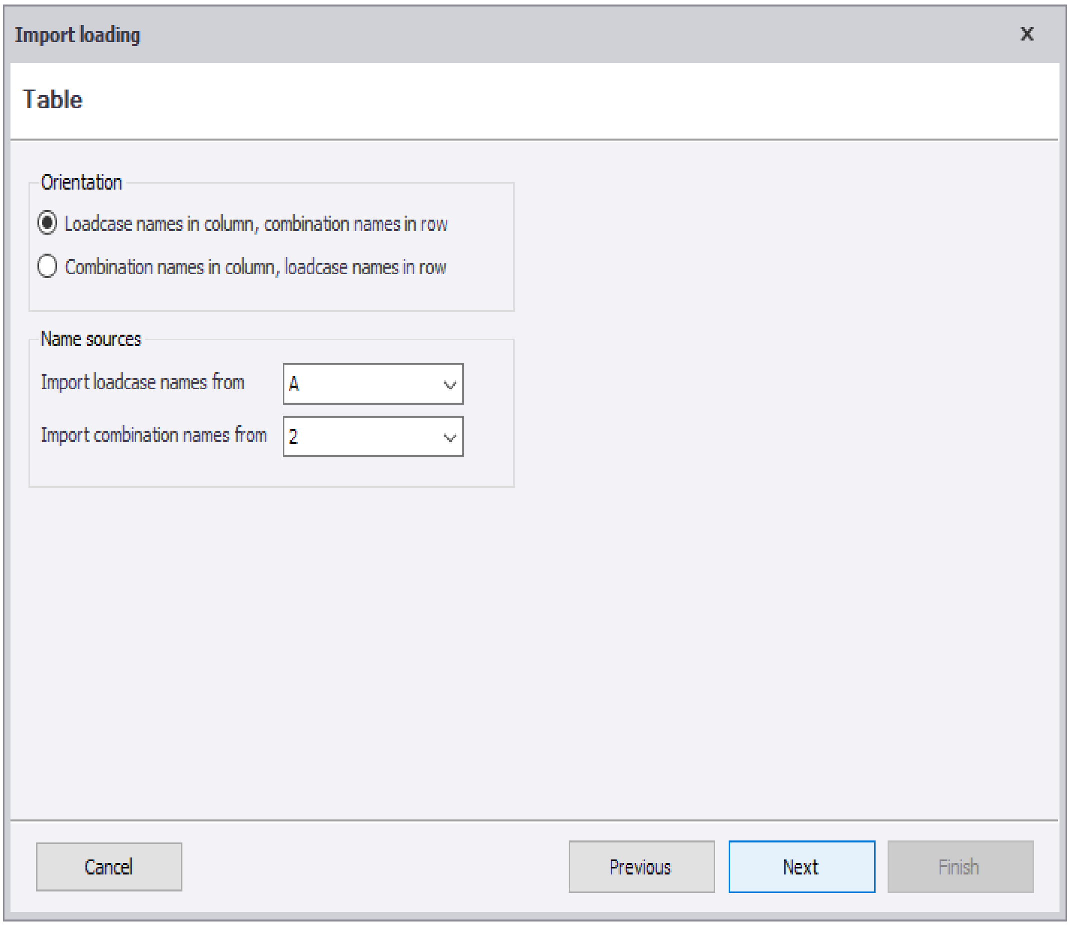
NOTE An empty cell is used to indicate a Loadcase is not to be included in particular Combination.

2. Starting from either the Loadcases or Combinations dialog in Tekla Structural Designer, click Import to start the Import loading wizard, the first page of which is:



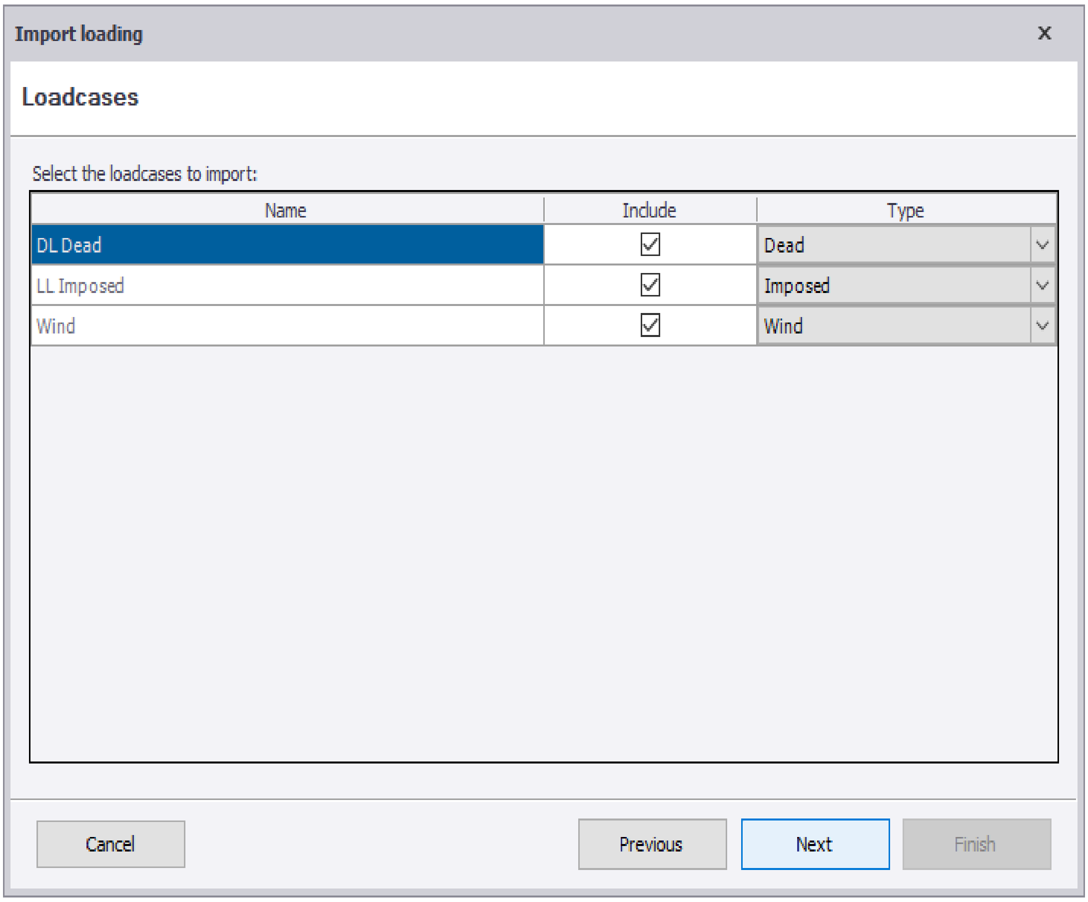
3. Click **Open ...** to navigate to the Excel file and the Sheet within the Excel file which contains the matrix and then click **Next**.

4. In the next page of the dialog you then choose the orientation of your matrix and confirm the Excel column & row which contain the Loadcase and Combination names (i.e. the matrix column and row headers). For this example the correct orientation and name sources are:



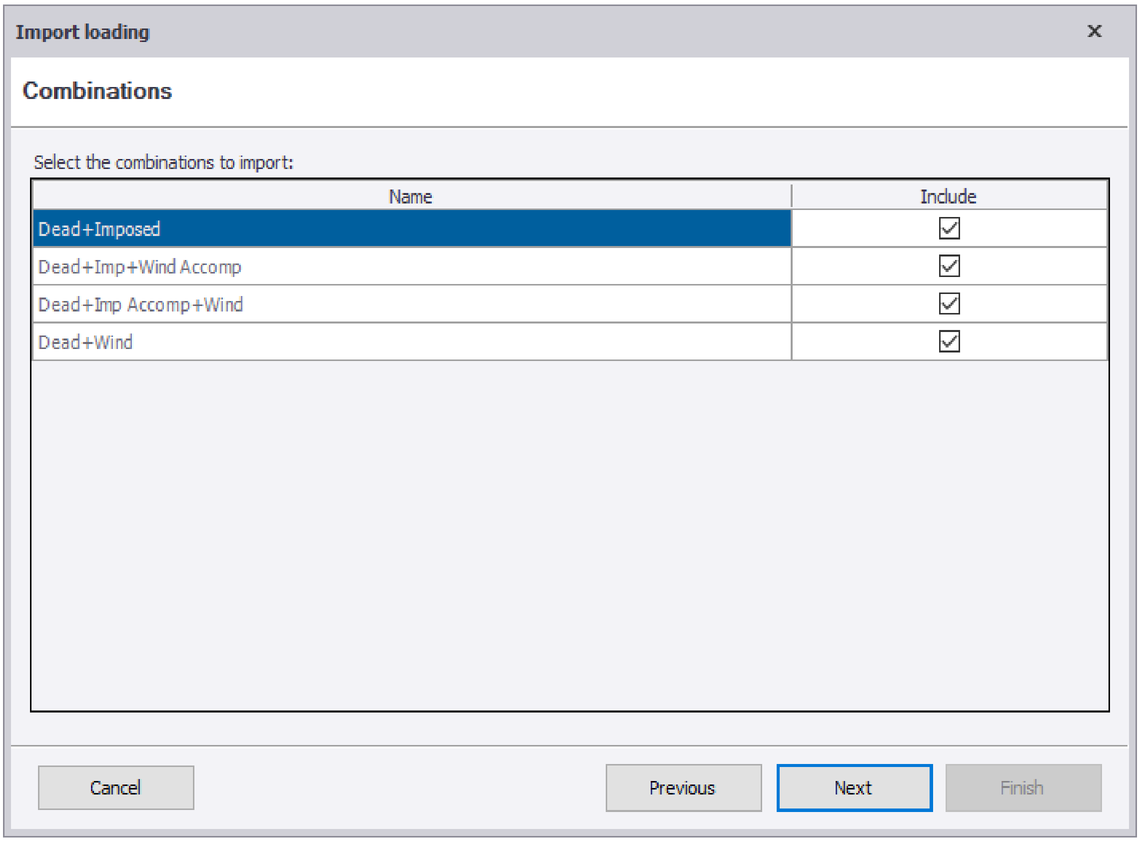
5. The next page of the dialog allows you to confirm/amend Loadcase options (a) **Include** (in the imported loadcases) - default is Include checked on, and (b) **Type** (the Loadcase type from Dead, Slab Dry, Slab Wet, Live, etc) - default is Dead for those names that can't be identified

automatically:



6. The next page of the dialog allows you to confirm/amend the Combination option **Include** (in the imported combinations) - default is

checked on:



7. The final page of the dialog summarizes the number of loadcases and combinations that will be imported, after which click **Finish** to complete the import process.
8. In the Loading dialog review the combinations, amend the combination classes where required, and if necessary include additional loadcases (e.g. automatically determined self weight).

Renumber all load combinations

When you delete load combinations from the **Loading** dialog box, the remaining load combinations retain their original load case number. If necessary, you can renumber the remaining load combinations in sequence. For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click **Combinations**.
2. In the context menu, select **Renumber**.

Manage envelopes

You can use envelopes to view analysis results for multiple combinations simultaneously. When you do so, Tekla Structural Designer displays the maximum positive and negative values along each member from any combination included in the envelope.

NOTE When using envelopes, note the following:

- If you have defined patterned load combinations, you only need to include the base case pattern combination in the envelope. This way, Tekla Structural Designer automatically includes all pattern combinations derived from the base case in the envelope.
- You can include gravity, lateral, and seismic combinations in the same envelope.
- You should not include seismic RSA combinations in envelopes, as Tekla Structural Designer currently cannot display the results.

Create envelopes

1. On the **Load** tab, click **Envelope**.

   The Loading dialog (page 531) opens on the **Envelopes** page. This page displays all currently existing envelopes.

2. Click **Add**.
3. Name the envelope.
4. On the left side pane of the **Loading** dialog box, click the name of the envelope.
5. Select the load combinations that you want to add in the envelope, and click **>>**.
6. Click **OK** to save the envelope.

Manage load patterns

Having live/imposed loads applied only to a portion of the structure can produce a more unfavorable loading than the "fully loaded" condition. More unfavorable loading conditions can be considered in Tekla Structural Designer by the use of load patterns.

To manage load patterns, see:

- Overview of load patterns (page 528)
- Apply patterning to live load cases (page 530)

- Apply patterning to load combinations (page 530)
- Update load patterns (page 530)

Overview of load patterns

NOTE When you apply pattern loading to imposed loads, the factors of the loaded and unloaded spans are specific to the design code that you are using.

The basic steps of applying pattern loads

1. Set individual imposed load cases to be patterned (page 530) according to your needs.

   These load cases are referred to as fully loaded pattern load cases.

2. Set the gravity combinations that contain imposed load cases to be patterned (page 530) according to your needs.

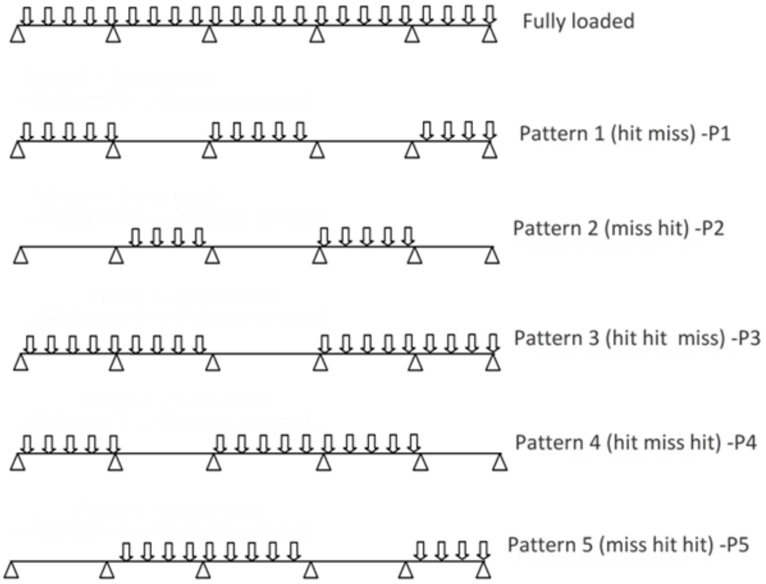
   These gravity combinations are referred to as base case pattern combinations.

After load decomposition, the building analysis automatically sets up the pattern cases for concrete beams in Dir1 and Dir2 directions as follows:

- 10 pattern cases for each pattern gravity combination
- 5 pattern cases for beams along Dir1 and 5 for beams along Dir2

NOTE By default, patterns are only applied to beam loads, and slab loads that have been decomposed on to beams. Loads applied to meshed slabs should be manually patterned using engineering judgement. For more information, see Update load patterns (page 530).

Pattern load cases



A pattern combination containing patterned imposed load cases results in 11 combinations: the base case combination, and 10 pattern combinations derived from the base combination.

If you later modify the building geometry, update load patterns (page 530) to ensure the load patterns reflect the changes.

NOTE Tekla Structural Designer contains a set of 10 pattern combinations associated with each fully loaded pattern combination. The pattern combinations are the same for beams and slabs.

The rules of applying load patterns to slabs and beams

Slab load patterning only applies where two-way slabs have been meshed in the solver model:

- in FE chasedown analysis
- in 3D analysis where two-way slabs are set as meshed

Therefore, the slab load pattern setting has no affect on:

- one-way spanning slabs
- two-way slabs not meshed in 3D analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model
- two-way slab in Grillage chasedown analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model

Therefore, in 3D analysis and Grillage chasedown analysis:

- When a beam is set to **Full Load**, it receives the full decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.
- When a beam is set to **Min Load**, it receives the min decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.

Apply patterning to live load cases

If necessary, you can apply load patterning to live/imposed load cases in a combination. In order to do so, see the following instructions.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 531) opens on the **Loadcases** page. On this page, you can see all currently existing load cases.
2. Click the live/imposed load case to which you want to apply patterning.
3. Select the **Pattern Load** option.
4. Click **OK**.

Apply patterning to load combinations

You can only apply load combinations to gravity combinations, as lateral and seismic combinations do not consider pattern loading. For detailed information on how to apply pattern loading to load combinations, see the following instructions.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 531) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. On the left side pane of the **Loading** dialog box, select the name of the load combination.
3. In **Parameters**, ensure that the **Pattern** option is selected.
4. Click **OK**.

   NOTE If you wish, you can use pattern loading for every gravity combination in your model. However, doing so may create many additional combinations.

Update load patterns

Tekla Structural Designer applies patterning to beam loads automatically. However, if you need to apply patterning to slab loads for slab design or other purposes, you must update the load patterns manually.

1. On the **Load** tab, click **Update Patterns**.
2. In the **Properties** window, select each pattern and adjust the loading status of the existing beams and slabs.
3. To switch the loading status of a slab, click the slab in the model.

Loading dialog

Summary

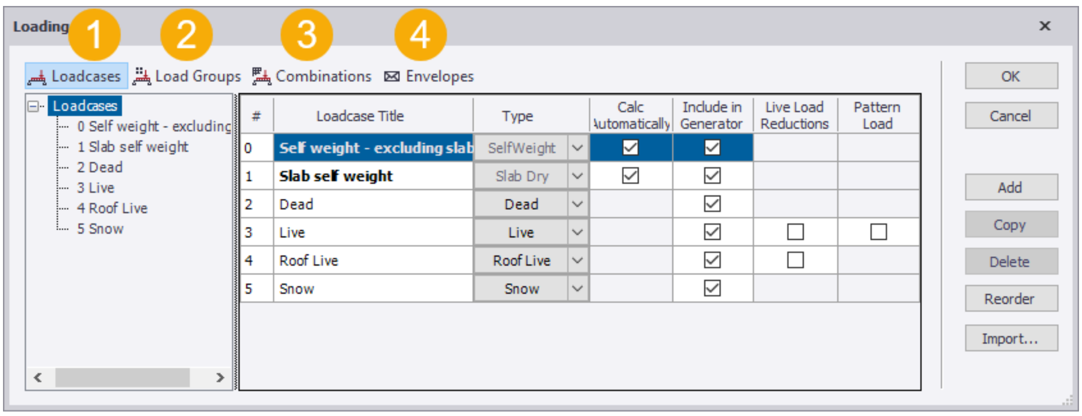
The **Loading** dialog is used to define loadcases, load groups, combinations and envelopes.

Location

On the **Load** tab, click:

- **Loadcases** - to open the dialog on the **Loadcases** page.
- **Load Groups** to open the dialog on the **Load Groups** page.
- **Combination**- to open the dialog on the **Combinations** page.
- **Envelope**- to open the dialog on the **Envelopes** page.

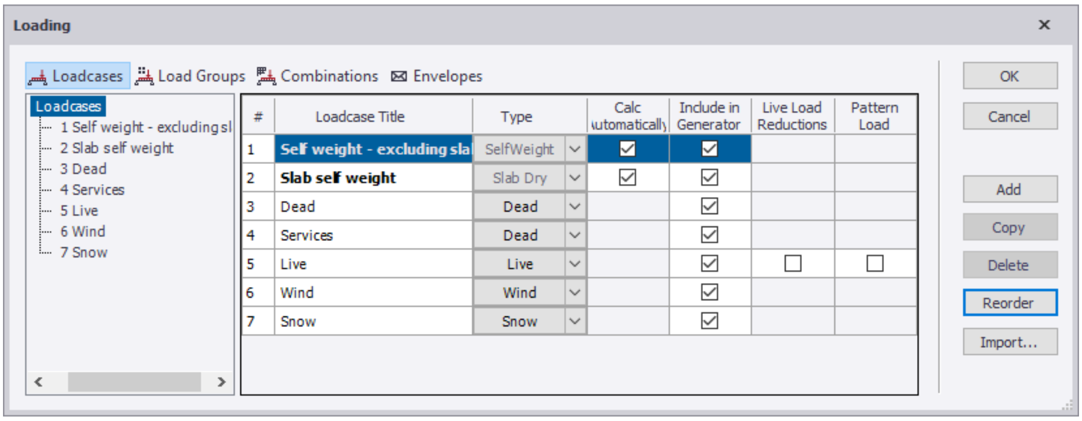
Content



1. Loadcases

Loadcases table

Select **Loadcases** in the left hand pane to show a table of loadcases that have been defined.



You can click within the table to:

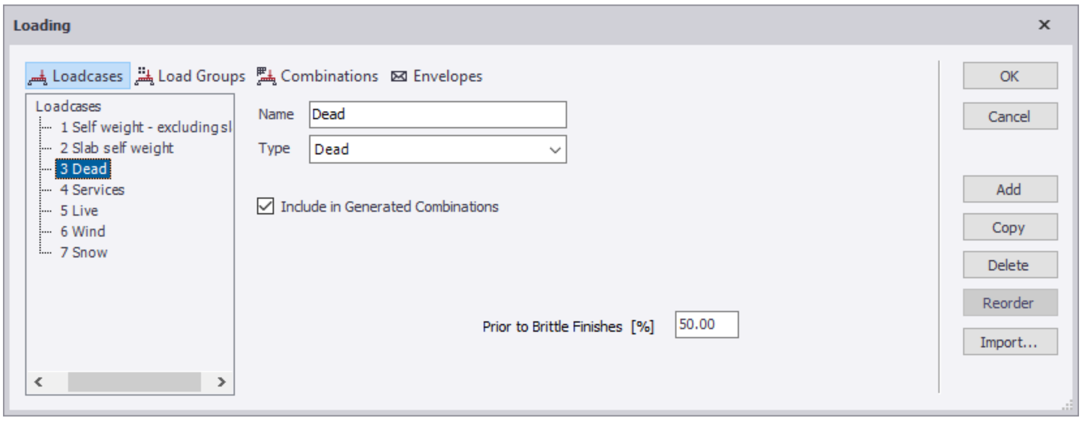
- Rename a Loadcase Title.
- Change the type of a loadcase.
- Choose whether the "Self weight excludiung slabs" and Slab self weight" cases are to be calculated automatically, or defined manually.
- Choose which cases are to be included when generating the load combinations.
- Apply reductions to live/imposed cases.
- Apply pattern loading to live/imposed cases.

Using the buttons you can:

- **Add** a new loadcase to the table.
- **Copy** an existing loadcase.
- **Delete** an existing loadcase.
- **Reorder** to move loadcases up or down the table.
- **Import...** loadcases from a spreadsheet.

Loadcase parameters

Select an individual loadcase in the left hand pane to show parameters specifically applicable to that loadcase.



Certain values can only be specified by selecting individual loadcases, for example:

- For dead loadcases: **Prior to Brittle Finishes %** .
- For live/imposed loadcases: **Long Term %** .

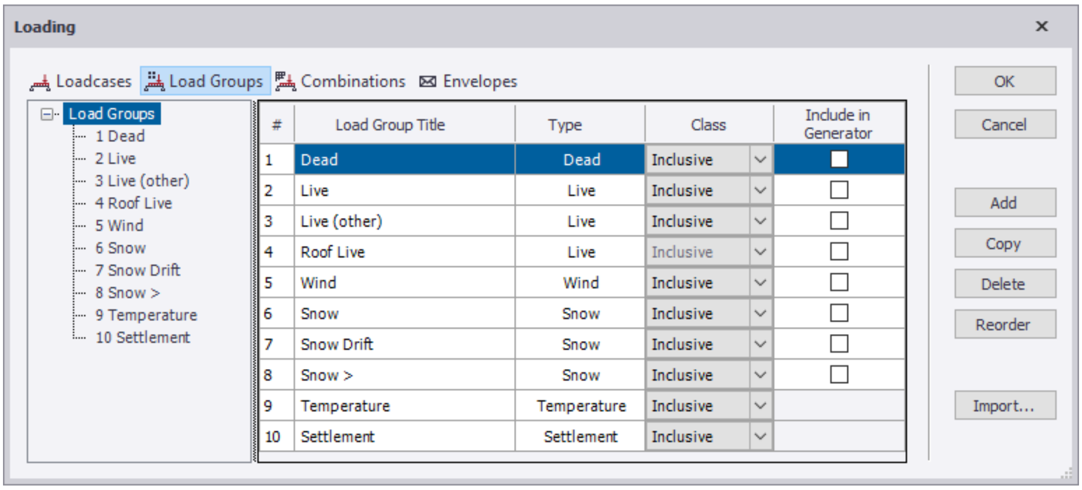
See also

Manage load cases (page 514)

2. Load Groups

Load Groups table

Select **Load Groups** in the left hand pane to show a table of any load groups that have been defined.



Click within the table to:

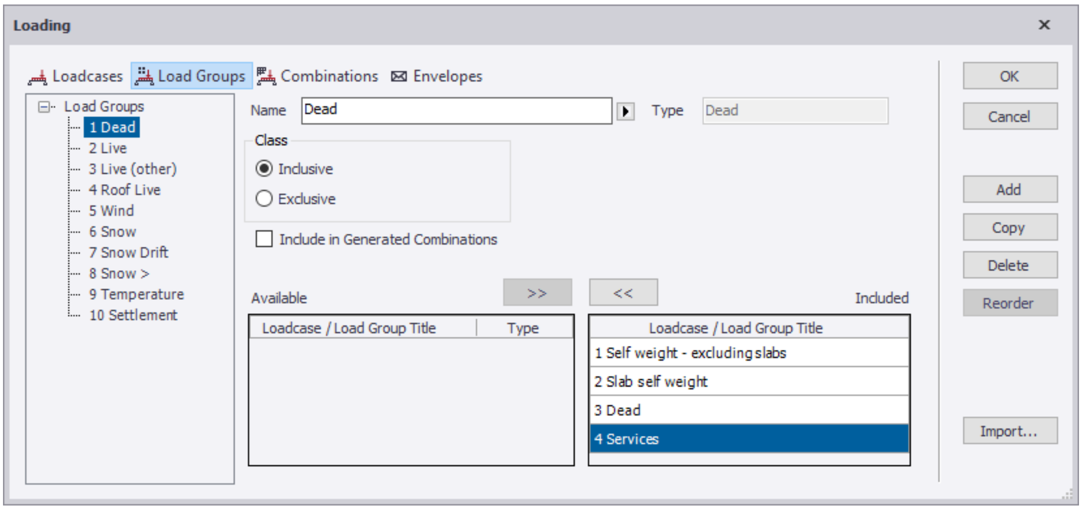
- Rename a load group title.
- Set the load group class as Inclusive, or Exclusive.
- Choose whether the load group is to be included in the generator.

Using the buttons you can:

- **Add** a new load group in the table.
- **Copy** an existing load group.
- **Delete** an existing load group.
- **Reorder** to move load groups up or down the table.
- **Import...** load groups from a spreadsheet.

Load group parameters

Select an individual load group in the left hand pane to set up the load group content.



The load group class is set as Inclusive, or Exclusive.

The loadcases/load groups currently in the load group are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove loadcases/load groups from the Available list to the Included list.

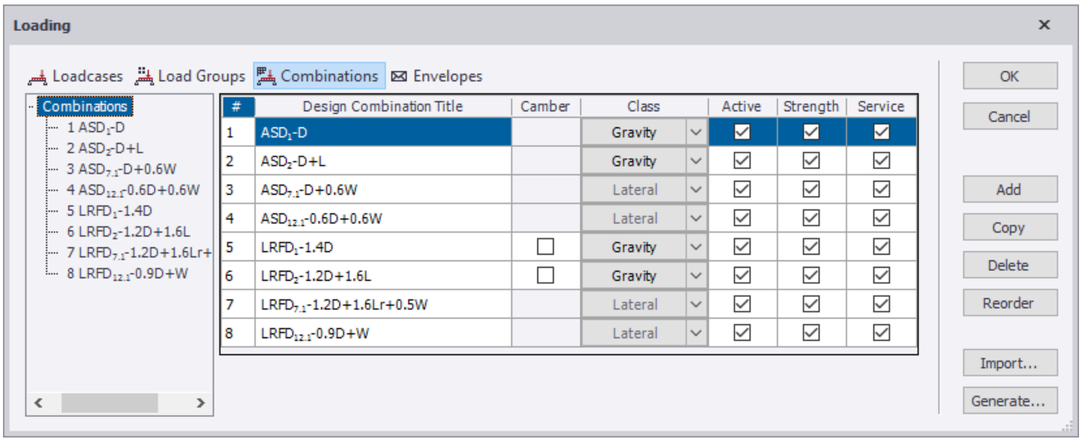
See also

Manage load groups (page 516)

3. Combinations

Combinations table

Select **Combinations** in the left hand pane to show a table of combinations that have already been defined or generated.



You can click within the table to:

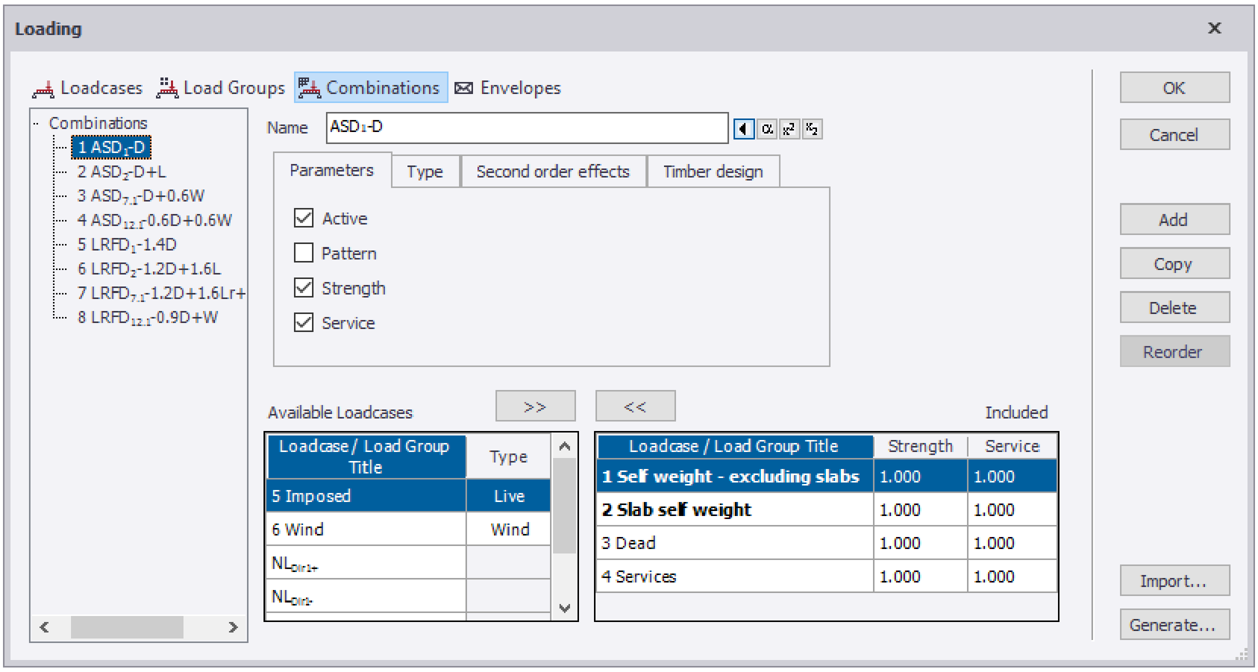
- Rename a combination title.
- Change the class of a combination.
- Choose the gravity combination to be used for any steel beam camber calculations.
- Choose which combinations are active for analysis and design.
- Choose which active combinations are to be assessed for strength.
- Choose which active combinations are to be assessed for service.

Using the buttons you can:

- **Add** a new combination in the table.
- **Copy** an existing combination.
- **Delete** an existing combination.
- **Reorder** to move combinations up or down the table.
- **Import...** combinations from a spreadsheet.

Combination parameters

Select an individual combination in the left hand pane to show the parameters specifically applicable to that combination only.



The tabbed boxes are used to set:

- **Parameters**: (Active, Pattern, Strength and Service).
- **Type**: Head code dependant type settings:
  - US head code - ACI/LRFD or ASD.
  - Eurocode - Formula type (STR, EQU, GEO)
  - India - Limit State or Working Stress
  - BS & Australia - not applicable
- **Second order effects**: the amplification factor, its direction, and whether it is applied to all loads or lateral loads only.
- **Timber design**: (US and Eurocodes only) the load duration/time effect factor.

The loadcases currently in the combination and the strength and serviceability factors that apply are shown in the Included list.

- Use the [>>] and [<<] buttons to add or remove loadcases from the Available Loadcases list to the Included list.
- In the **Strength** column specify the strength factors.
- In the **Service** column specify the service factors.

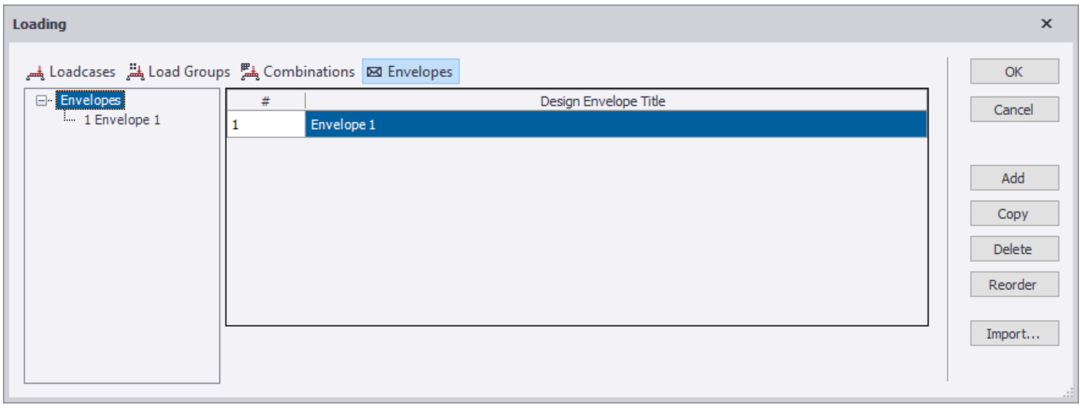
See also

Manage load combinations (page 518)

6. Envelopes

Envelopes table

Select **Envelopes** in the left hand pane to show a table of any envelopes that have been defined.

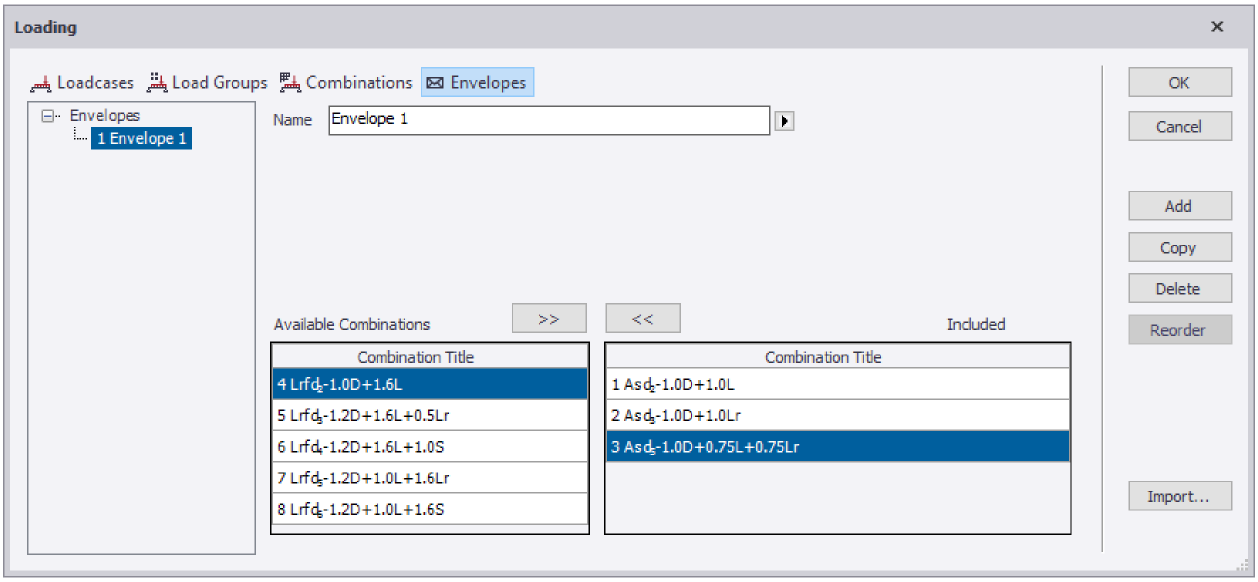


Using the buttons you can:

- **Add** a new envelope in the table.
- **Copy** an existing highlighted envelope.
- **Delete** an existing highlighted envelope.
- **Reorder** to move envelopes up or down the table.
- **Import...** to import envelopes from a spreadsheet.

Envelope parameters

Select an individual envelope in the left hand pane to set up the envelope content.



The combinations currently in the envelope are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove combinations from the Available Combinations list to the Included list.

5.2 Apply panel, member, and structure loads

Apply panel loads

You can use panel loads to apply loads to slab items, roof panels, and wall panels. Tekla Structural Designer contains several types of panel loads, including point loads, line loads, patch loads, polygonal loads, area loads, slab loads, and level loads. In order to apply the different panel loads to your model, see the following instructions.

You can use the different panel loads in the following ways:

Panel load type	Use	Notes
Point load	Can be applied anywhere within an individual or across multiple slab items, roof panels, or wall panels.	Can only be applied in 2D views.
Patch load		
Polygonal load		
Area load	Entirely covers a slab item, a roof panel, or a wall panel.	Not applicable
Slab load	Entirely covers all slap items in a parent slab.	A parent slab can consist of slab items that are physically separate from each other, but are on the same level.
Level load	Entirely covers all parent slabs in a level.	

Create point loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Point**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

6. Click the load position, or press **F2** to type the exact position.

   NOTE If you move the slab item or panel by manually selecting & re-positioning its nodes, the point load does not move with the slab or panel.

   However, if you move any of the grid lines defining the reference node, the load will move as well.

Create line loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.

3. On the **Load** tab, click **Line**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

   ---

6. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.

7. Click the end point of the load, or press **F2** to type the exact position.

   Note that when you type the position using the keyboard, the end point is an offset (X, Y) from the load start position.

Create patch loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Patch**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

   ---

6. Click to specify a corner position of the load, or press **F2** to type the exact corner position.
7. Click to specify the size of the load, or press **F2** to type the size.

   The size is specified as the offset dimensions from the selected corner position.

8. Click to specify the rotation angle of the load, or press **F2** to type the rotation angle.

Create polygonal loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Polygon**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

   ---

6. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.
7. Click a corner point of the load, or press **F2** to type the exact position.
8. Repeat step 7 according to your needs.
9. When the polygon is complete, do one of the following:
   - Press **Esc**.
   - Click the first corner that you defined.

   TIP To modify the shape of an existing polygonal load the following:

   a. In the model, select the load.
   b. Click one vertex of the load.
   c. Click the point where you want to move the vertex.

Create perimeter loads

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Perimeter**.
4. In the **Properties** window, adjust the load details according to your needs.

   TIP If necessary, you can select one or both of the following options:

- **Create as line loads**: creates the load as a series of separate line loads along each external edge.
- **Ignore openings**: only creates the load around external perimeters, and not around internal opening perimeters.

5. Click any slab or mat item.

   Tekla Structural Designer creates the load around the external perimeter of every continuous area of slabs at the selected level.

Create variable patch loads

NOTE Variable patch loads do not have to be quadrilaterals. Instead, they can be any polygonal shape.

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Var. Patch**.
4. In the **Properties** window, specify three load values to describe the variable loads.
5. Adjust the remaining load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

7. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.

8. Click a corner point of the load, or press **F2** to type the exact corner position.
9. Repeat step 8 according to your needs.
10. When the polygon is complete, do one of the following:
    - Press **Esc**.
    - Click the first corner that you defined.
11. Click one of the corners of the polygon to specify the position of **Load 1**.
12. Click another corner to specify the position of **Load 2**.

13. Click a third corner to specify the position of **Load 3**.

TIP To modify the shape of an existing patch load the following:

a. In the model, select the load.

b. Click one vertex of the load.

c. Click the point where you want to move the vertex.

Create area loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Area**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click the panel to which you want to apply the load.

Create variable area loads

Variable area loads can only be applied to non-horizontal slab items and roof panels, as well as wall panels and concrete walls. Therefore, they cannot be applied in a 2D view.

1. Open the 3D view, frame, or sloped plane view in which you want to apply the load.
2. In the **Loading** list, select an appropriate load case.
3. On the **Load** tab, click **Var. Area**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click the slab item or panel to which you want to apply the load.

Create slab loads

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click **Slab**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load to all the slab items within a parent slab, in the model, click any slab item within the parent slab.

Create level loads

Click any slab panel in order to apply the load to all slabs within the level.

1. In the **Loading** list, select an appropriate load case.

2. On the **Load** tab, click **Level**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load to all slabs within the level, in the model, click any slab item.

Apply member loads

By using member loads, you can apply loads to one-dimensional members, such as beams, columns, and braces.

Tekla Structural Designer allows you to create and apply the following types of member loads:

- Full-length uniformly distributed loads (UDLs)
- Partial-length uniformly distributed loads (UDLs)
- Partial-length variable deck loads (VDLs)
- Trapezoidal loads
- Point loads
- Moment loads
- Full-length torsional uniformly distributed loads (UDLs)
- Partial-length torsional uniformly distributed loads (UDLs)
- Partial-length torsional variable deck loads (VDLs)

Create full-length UDLs

In order to create a full-length uniformly distributed load (UDL) and apply it to a member, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click **Full UDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click anywhere along the member.

Create partial-length UDLs or VDLs

In order to create UDLs (uniformly distributed loads) or VDLs (variable deck loads) that only apply to a selected part of the member, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click **UDL** or **VDL**.

3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the start point of the load along the member, or press **F2** to type the exact position.
6. Click the end point of the load along the member, or press **F2** to type the exact position.

Create trapezoidal loads

In order to apply trapezoidal loads to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click **Trapezoidal**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member that you want to apply the load to.
5. Click the load position along the member, or press **F2** to type the exact position.

   The load position defines the point at which the symmetrical trapezoidal load reaches its maximum intensity.

Create point loads and moment loads

In order to create and apply point loads or moment loads to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, in the **Member Loads** group click **Point** or **Moment**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the load position along the member, or press **F2** to type the exact position.

Create full-length torsional UDLs

In order to apply full-length torsional uniformly distributed loads (UDLs) to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click  **Torsion Full UDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click anywhere along the member.

Create partial-length torsional UDLs and VDLS

In order to create and apply torsional uniformly distributed loads (UDLs) or variable deck loads (VDLs) that only apply to a selected part of the member, see the following instructions.

1. In the **Loading** list, select an appropriate load case.
2. On the **Load** tab, click **Torsion UDL** or **Torsion VDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the start point of the load along the member, or press **F2** to type the exact position.
6. Click the end point of the load along the member, or press **F2** to type the exact position.

Apply structure loads

The four types of structure loads that you can apply are:

- Diaphragm loads: applied at any point within a rigid or semi-rigid diaphragm
- Nodal loads (page 554): applied at solver node locations
- Temperature loads (page 554): global rises in temperature applied to elements or panels.
- Settlement loads (page 554): translations or rotations applied to supports in the support UCS system.

Diaphragm loads and diaphragm load tables

Diaphragm loads are Building Direction 1, Building Direction 2, and Mz torsion loads applied at any point within a rigid or semi-rigid diaphragm.

Typically these might be used for the application of externally determined level loads (for example from Wind Tunnel testing of large/ unusual structures).

The **Diaphragm** load command is available when a 2D plan view is active - it can be used to apply a load to any diaphragm within the current view.

If you have multiple loads to apply at different levels, the **Diaphragm Table** of loads provides a quicker means of doing this - either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

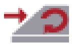
Add a diaphragm load in a 2D view

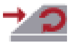
To apply a load to any diaphragm within the current 2D view:

Related video

Diaphragm loading

1. Open a 2D view of the level containing the diaphragm to which you want to apply the load.
2. Click the **Load** tab on the ribbon.
3. In the **Loading** list, select an appropriate load case.

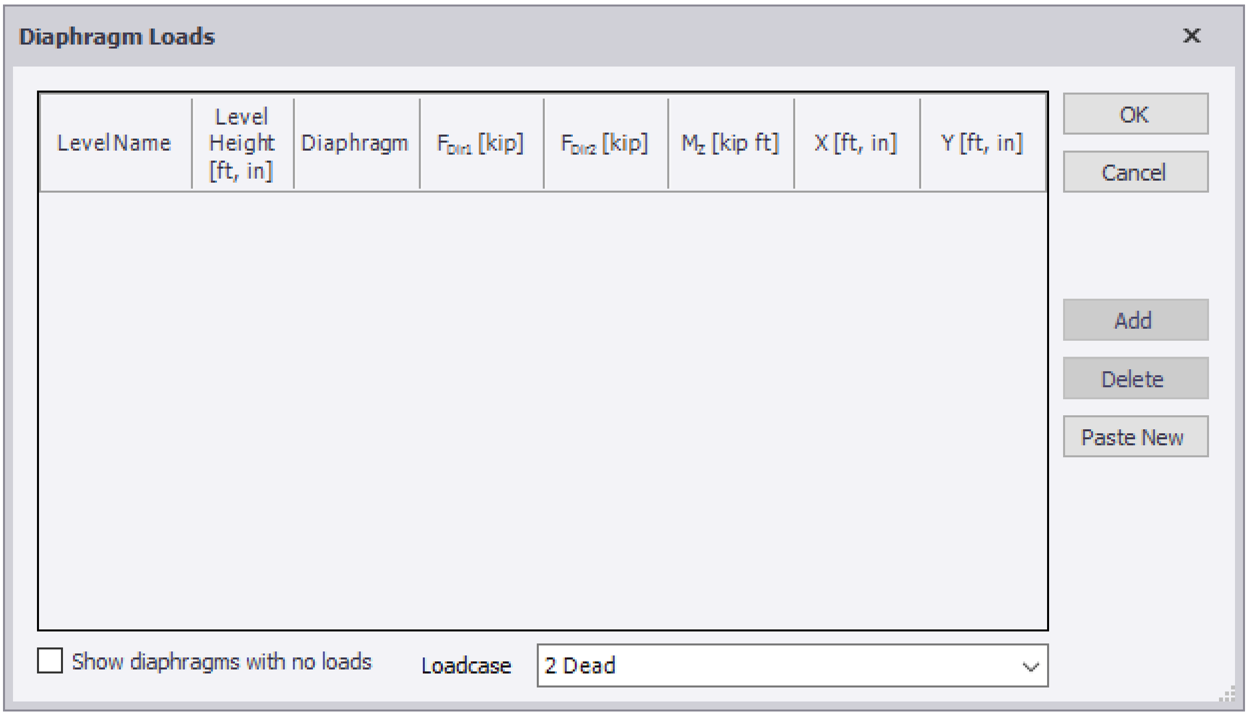
    **Diaphragm** on the **Load** tab should now be active.

4. Click  **Diaphragm**
5. In the **Properties** window, adjust the load details according to your needs.
6. Click the load position, or press **F2** to type the exact position.

Add multiple loads using the diaphragm load table

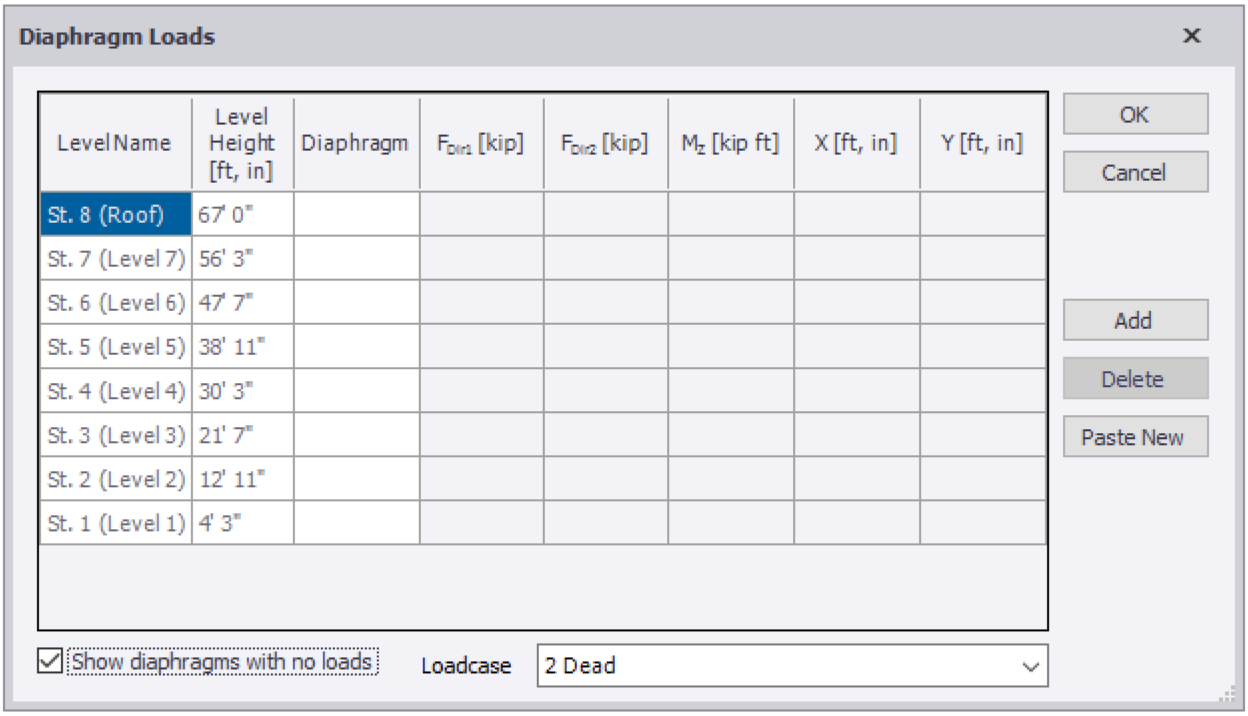
The diaphragm loads table provides a way to quickly apply multiple diaphragm loads at different levels, either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

1. On the **Load** tab, click **Diaphragm Table**.



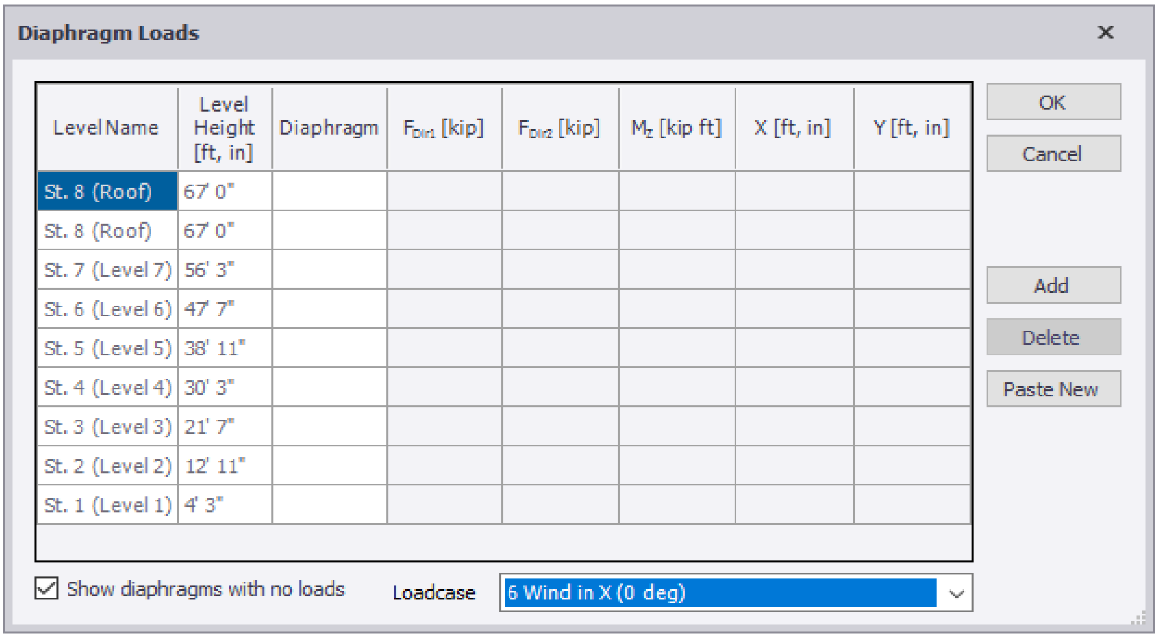
The Diaphragm Loads table opens in a dialog. If any loads had already been added (via the Diaphragm Load command, or the Diaphragm Loads table) these would be listed.

2. In order to see all the separate diaphragms at each level to which loads could be applied, click the **Show diaphragms with no loads** box.

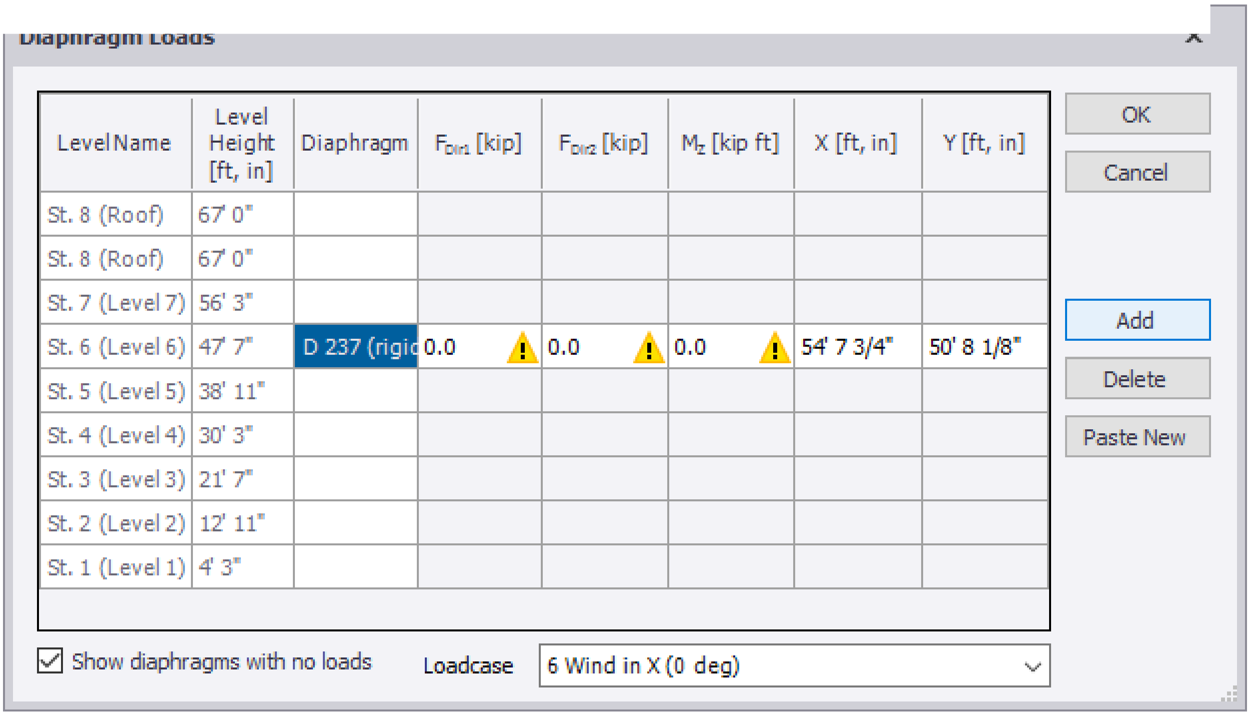


Each level containing a diaphragm is listed, (if more than one diaphragm exists at the same level a separate row is created for each).

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. In the table, click on the diaphragm level to which you want to apply the load, then click **Add**.



A row is displayed for entering the load. The initial default for the load position is simply the middle point in X and Y of the diaphragm extremities.

5. Enter the load values and the actual X and Y coordinates.
6. Add further rows of loading data as required, either at the same level or other levels.

NOTE Multiple loads can be applied to a single diaphragm if required.

7. Click **OK** when done.

The loads are displayed on the model iat their defined locations.

NOTE To edit existing loads in the table, simply open it once more. To delete loads from the table, select the diaphragm in the table and click **Delete**.

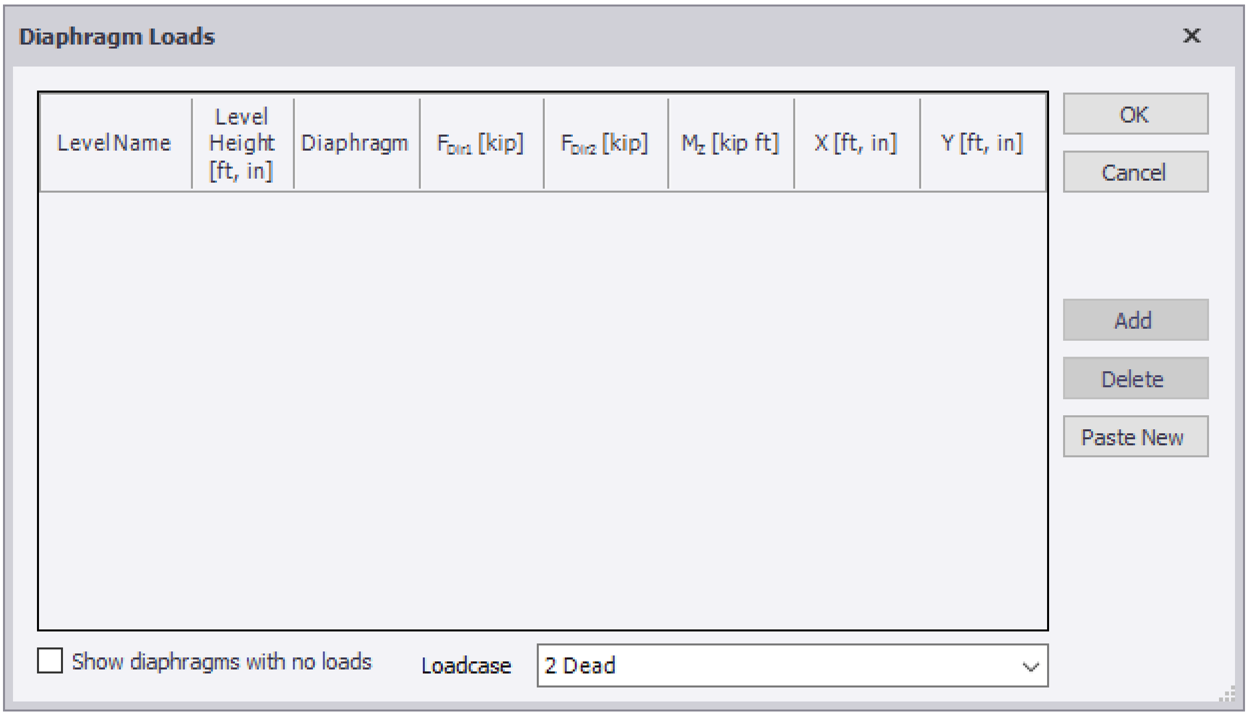
Paste multiple loads into the diaphragm load table

If you have been provided with the loading data in the form of a spreadsheet you can simply paste the data directly into the table.

NOTE The data must includes a Z (Level height) column and you must ensure that it is in the following order (which is not the same order as in the diaphragm loads table):

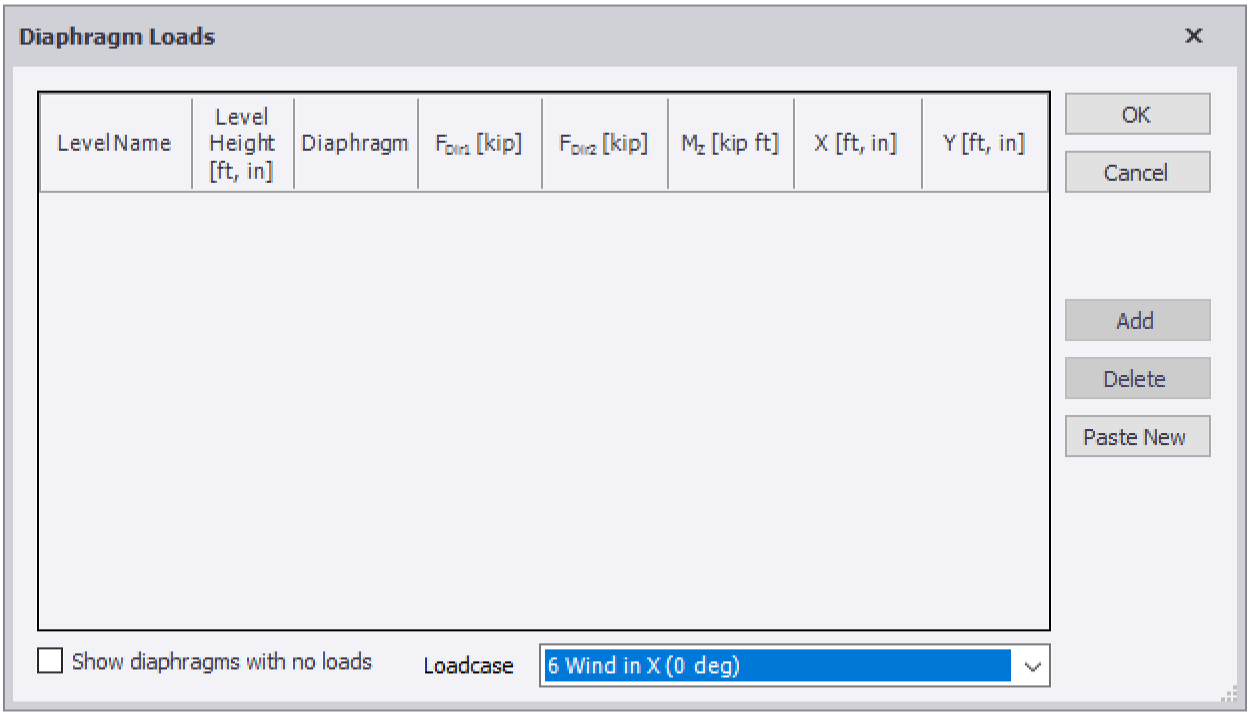
1. Force in direction 1
2. Force in direction 2
3. Torsion moment
4. Position X
5. Position Y
6. Position Z

1. Open the spreadsheet containing the data to be pasted and copy the contents to the clipboard.
2. In Tekla Structural Designer, on the **Load** tab, click **Diaphragm Table**.

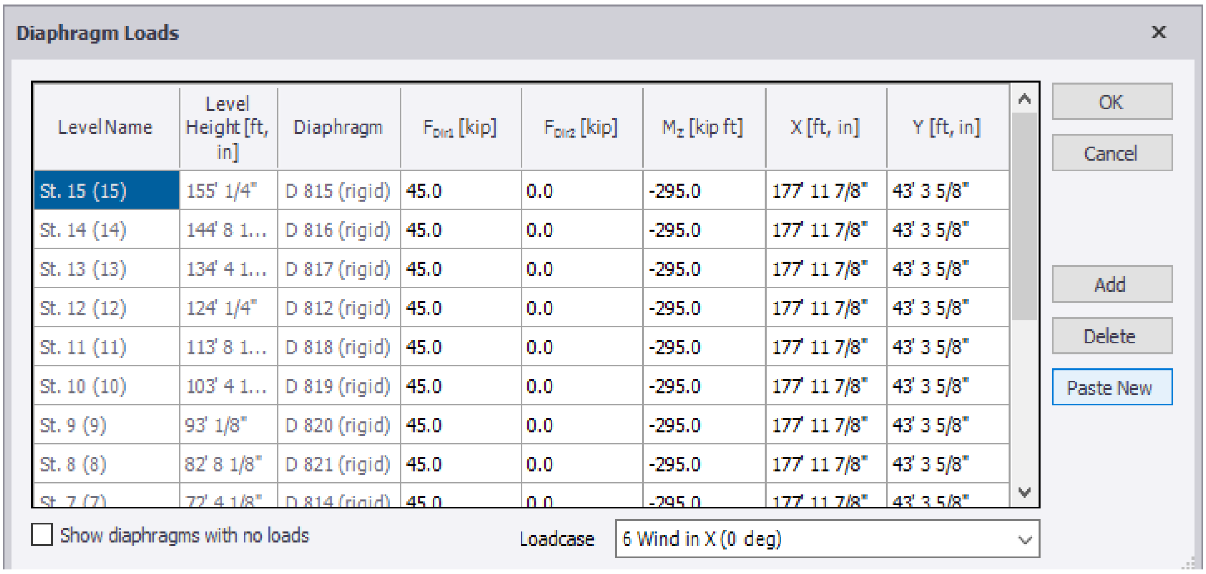


The Diaphragm Loads table opens in a dialog.

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. Click **Paste New**.



Provided the spreadsheet contents were in the expected format, the loads and their positions are pasted into the table.

NOTE Existing diaphragm loads will be deleted / replaced by the pasted data.

5. Click **OK** when done.

Create nodal loads

To create nodal loads at the solver nodes in your model, see the following instructions.

1. Click the **Load** tab on the ribbon.
2. In the **Loading** list, select an appropriate load case.

   Nodal on the **Load** tab should now be active.
3. Click **Nodal**.
4. In the **Properties** window, adjust the load details according to your needs.
5. To define the load position, click the desired node.

Create temperature loads

Temperature loads are global rises in temperature that you can apply to individual elements or panels in the model. For more information on how to create temperature loads, see the following instructions.

1. Click the **Load** tab on the ribbon.
2. In the **Loading** list, select an appropriate load case.

   Temperature on the **Load** tab should now be active.
3. Click **Temperature**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click anywhere along the element to apply the load.

Create settlement loads

Settlement loads are translations or rotations that you can apply to a support. To create settlement loads, see the following instructions.

1. Click the **Load** tab on the ribbon.

2. In the **Loading** list, select an appropriate load case.

   Settlement on the **Load** tab should now be active.

3. Click **Settlement**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a supported node to place the load.

Modify panel, member, and structure loads

To modify the properties of existing loads, see the following instructions.

1. In the model, select the load that you want to modify.
2. In the **Properties** window, adjust the load details according to your needs.

Delete panel, member, and structure loads

If necessary, you can delete existing loads in your model. In order to do so, see the following instructions.

1. Ensure that the load case containing the load is displayed in the **Loading** list.
2. On the **Quick Access** toolbar at the top of the window, click **Delete**.
3. Click the load that you want to delete.

Decompose panel loads

TIP Although Tekla Structural Designer carries out load decomposition automatically when you click to analyze the structure, you have the option to decompose loads manually by using the **Decomposition** command.

In the context of big or complex models, decomposing manually can potentially save time, as it allows you to check that panel loads have been decomposed as you intend before running the analysis.

Decompose panel loads for an individual construction level

1. Open a 2D view of the desired level. (page 276)
2. Click the '2D' toggle button at the bottom right corner of the view to display it in 3D.

3. On the **Load** tab, click **Decomposition**.

   Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

   TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

Decompose panel loads to all required levels

1. Open the **Structure 3D** view.
2. On the **Load** tab, click **Decomposition**.

   Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

   TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

View decomposed loads graphically

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. In **Scene Content**, go to **Loading**.
3. Select the **Decomposed** option.
4. According to your needs, do one of the following:

To	Do this
View the decomposed load values	• In the cell on the right side of **Decomposed**, select both **Geometry** and **Text**.
View the decomposed loads without load values	• In the cell on the right side of **Decomposed**, only select **Geometry**.

5. In the **Loading** list, select the load case.

NOTE Decomposed loads do not exist for 2-way slab items at levels where the **Mesh 2-way Slabs in 3D Analysis** option has been selected.

At levels where this is not the case, you can view the decomposed loads. However, you cannot see any shell results from the FE load decomposition.

View applied and decomposed member loads in a table

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. Right-click on the member, and from the context menu select **Show Member Loading**. Loads that have been applied directly to the member, and loads that have been decomposed to the member are displayed in a table. The 'In Proj.' checkbox is used to indicate applied loads applied in projection as opposed to along the element.
3. To filter the data by Loadcase, Source, Direction, or Type:
    a. Click the appropriate column header to filter by (Loadcase..., Source..., Direction..., or Type...)
    b. From the drop list that appears clear the categories that you don’t want to be displayed.
    c. Click the Close button under the drop list.

Overview of one-way and two-way load decomposition

The way in which Tekla Structural Designer decomposes panel loads depends on how the slabs/panels are modeled and how they are spanning.

Slab/Panel type	Decomposition
One-way spanning slab item	Tekla Structural Designer 1-way decomposes panel loads applied to the slab directly on to those supporting members/two-way spanning slab edges before performing 3D analysis. The rotation angle of the slab item determines the decomposition direction. **NOTE** Any openings in 1 way slabs are ignored and thus have no impact on load decomposition.

<table>
<tr><th>Slab/Panel type</th><th>Decomposition</th></tr>
<tr><td>Roof panel</td><td>Tekla Structural Designer 1-way decomposes panel loads directly on to supporting members/two-way spanning slab edges before performing 3D analysis.

The rotation angle of the roof panel determines the decomposition direction.</td></tr>
<tr><td>Two-way spanning slab item</td><td>• At those levels where the Mesh 2-way Slabs in 3D Analysis option is selected, load decomposition is not required.

• At other levels, any loads applied to two-way slabs are automatically 2-way decomposed back to supporting members during the 3D pre-analysis stage, immediately prior to the 3D analysis stage.

NOTE Where openings have been defined, any portion of a panel load that lies within the opening will not be decomposed.</td></tr>
<tr><td>Roof panel overlapping a one-way spanning slab item</td><td>Tekla Structural Designer 1-way decomposes panel loads applied to the slab directly on to supporting members/two-way spanning slab edges before performing 3D analysis.

The rotation angle of the slab item (and not the roof panel) determines the decomposition direction.

NOTE Any openings in 1 way slabs are ignored and thus have no impact on load decomposition.</td></tr>
<tr><td>Roof panel overlapping a two-way spanning slab item</td><td>• If there are no openings in the slab, decomposition is as per Two-way spanning slab item type described above.

• Where openings exist, any panel load applied inside the opening is</td></tr>
</table>

Slab/Panel type	Decomposition
	first one-way decomposed in the direction defined by the rotation angle of the roof panel on to the two-way slab at the edge of the opening. It is then further decomposed back to supporting members as per **Two-way spanning slab item** type described above. **NOTE** This does not apply when the panel load is an area load that has been applied to the slab item (as opposed to the roof panel), or a level load, or a slab load. In each of these instances the load will automatically be boxed out around the opening.

5.3 Apply wind, snow, and seismic loads

Wind loads can be applied using the **Wind Wizard**, or they can be applied manually - when applied manually they can take the form of panel, member, or structure loads, or simple wind loads.

Similarly, snow loads can either be applied using the **Snow Wizard** or they can be applied manually.

Seismic loads can only be applied using the **Seismic Wizard**.

- Apply wind loads using the wind wizard (page 559)
- Apply wind loads manually (page 565)
- Apply snow loads using the snow wizard (page 566)
- Apply snow loading manually (page 577)
- Apply seismic loads (page 577)

Apply wind loads using the wind wizard

In Tekla Structural Designer, you can apply wind loads to your structure by using the **Wind Wizard** to generate a wind model.

- Create a wind model and wind loads (page 560)
- Modify wind zones and wind zone loads (page 562)
- Create and manage wind load cases (page 564)

See also

Wind modeling handbook (page 1040)

Create a wind model and wind loads

You can use the **Wind Wizard** to automate the wind modeling process. Where appropriate, the **Wind Wizard** uses databases to determine the appropriate wind details for your structure location, and then calculates the appropriate wind loading details according to the selected wind loading code.

RESTRICTION The **Wind Wizard** is not currently available for the Australian **AS:1170.2** wind loading code variant.

Once you have defined the wind directions in which you are interested, Tekla Structural Designer automatically calculates the appropriate wind zones on the roofs and walls of your structure. You can set the type of each roof to achieve the correct zoning, and then tailor the zoning to account for particular features in more detail, if you so require.

The wind modeling process can automatically define standard wind load cases for you based on the usual internal pressure coefficients, or you can define the load case information yourself. In both cases, the appropriate wind pressures are calculated on each zone. You can then combine the wind load cases into design combinations as usual.

NOTE The determination of the wind speeds, the pressures, and the zones is rigorous. However, do remember that the final wind loads that are adopted are your responsibility.

Run the Wind Wizard

NOTE You must define at least one wall or roof panel before running the **Wind Wizard**.

1. On the **Load** tab, click  **Wind Wizard...**

   The **Wind Wizard** opens.

NOTE The **Wind Wizard** varies slightly according to the head code that you are using.

2. Define the necessary information for the wind model. To go to the next page in the wizard, click **Next**.
3. Once you have defined all the necessary information, click **Finish**.

NOTE After running the **Wind Wizard**, you can review the roof and wall zones for each wind direction.

After you have created the wind model with the **Wind Wizard**, you can open a wind view to graphically display the wind zones and loading that apply for a particular wind direction.

Add wind directions

1. On the **Load** tab, click **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click **Add Dir.**
4. Specify the properties of the new wind direction.
5. Click **Finish**.

NOTE Remember to add new wind load cases and design combinations to incorporate the wind loading for the new direction into your calculations.

Delete wind directions

1. On the **Load** tab, click **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click the wind direction that you want to delete.
4. Click **Del Dir.**
5. Click **Finish**.

NOTE Remember to update the existing wind load cases and design combinations to remove the details for the wind direction you have deleted from your calculations.

Delete the entire wind model

To delete the entire wind model and start the wind modeling process from scratch, do the following:

1. On the **Load** tab, click **Delete Wind**.

   Tekla Structural Designer deletes all the previously defined wind directions and wind load cases.

See also

Wind modeling handbook (page 1040)

Modify wind zones and wind zone loads

In order to view and modify the wind zones and loading that apply to a particular wind direction, you have to open the appropriate wind view. Then, you can modify the wind zones and wind zone loads according to your needs.

Open a wind view

NOTE Wind views are only available if you have already created the wind model with the **Wind Wizard**.

---

1. In the **Project Workspace**, go to the  **Wind** tab.
2. Right-click the wind direction that you want to view.
3. In the context menu, click **Open View**.

   Tekla Structural Designer opens the selected wind view, and the **Zone Loads** tab, which allows you to review wind zone data graphically.

View wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. On the **Zone Loads** tab, click **Wind Zones**.

   The wind view displays the zones that are applied to the structure for the selected wind direction.

Modify wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. In the **Project Workspace**, go to the  **Wind** tab.
3. For the wind direction in question, expand the **Roof Zones** or **Wall Zones** branch.

4. In the desired branch, right-click the panel label containing the zone that you want to edit.
5. In the context menu, select **Edit Zones...**

   The **Zone Properties** dialog box opens.
6. Modify the wind zones according to your needs.
7. Click **OK**.

View the wind zone loads

NOTE Remember to update the zone loads when you make changes to your model.

---

1. Open a wind view that displays the details of the desired wind direction.
2. On the **Zone Loads** tab, click **Zone Loads**.
3. In the **Loading** list, select the desired wind load case.

   Tekla Structural Designer displays the loads that are applied to the structure for this load case in this wind direction.

Modify wind zone loads

1. Open a wind view that displays the details of the wind direction whose zone loads you want to modify.
2. On the **Zone Loads** tab, click **Zone Loads**.
3. In the **Loading** list, select the desired wind load case.

   Tekla Structural Designer displays the loads that are applied to the structure for this load case in this wind direction.
4. In the model, click the wind zone that you want to change.
5. In the dialog that opens, clear the **Use Default Values** option.
6. Modify the values that determine the loads.

   TIP If you want to reduce the net pressure for benecifial loads to zero, select the **Beneficial Load** option.

   ---

7. To update the zones, **OK**.

Update wind zone loads

When you change the roofs or walls of your structure, Tekla Structural Designer does not automatically update the changes to existing wind zoning.

Updating the zoning is manual because you may wish to make more alterations before you recalculate the zoning.

Once you have completed your changes, incorporating them and recalculating the zoning details is simple. To reinstate the zoning after making changes, do the following:

1. On the **Load** tab, click **Update Zones**.

   The wind zoning calculations run in the background. Once the calculations are complete, Tekla Structural Designer views the new zoning layout for your structure.

   NOTE If you have defined your own zone layout for any roof or wall, the existing zones will be maintained when you make changes. Make sure to update these zones appropriately.

Create and manage wind load cases

Once you have defined the basic wind data for your model, and calculated wind zoning in the **Wind Wizard**, you can define the required wind load cases manually. For detailed instructions, see the following paragraphs.

Define wind load cases

1. On the **Load** tab, click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Do one of the following:

   - Click **Add** to add the details of each wind load case individually.
   - Click **Auto** to generate standard wind load cases for the wind directions that you have defined in the **Wind Wizard**.

     NOTE You cannot use the **Auto** option once you have created other wind load cases.

Add wind load cases

1. On the **Load** tab, click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Click **Add**.
3. Define the details of the new load case according to your needs.
4. Repeat steps 2 and 3 for each load case that you want to create.

5. Click **OK** to close the **Wind Loadcases** dialog box.

NOTE	Remember to update or add design combinations, so that they take the new wind load cases into account.

Delete wind load cases

1. On the **Load** tab, click **Wind Loadcases**.
2. Click the wind load case that you want to delete.
3. Click **Delete**.
4. Repeat steps 2 and 3 to delete further load cases.
5. Click **OK** to close the **Wind Loadcases** dialog box.

NOTE	Remember to update your design combinations, so that they take the deleted wind load cases into account.

Apply wind loads manually

In case you do not want to construct an entire wind model, you can elect instead to apply the wind loads manually. These can be applied as panel, member, or structure loads in the usual way, or as simple wind loads.

Create load cases for manual wind loads

1. On the **Load** tab, click **Loadcases**.

   The **Loading** dialog box opens.
2. On the **Loadcases** page, click **Add**.
3. Name the new load case.
4. Set the type to **Wind**.
5. Click **OK**.

Create simple wind loads

RESTRICTION	You can only create simple wind loads in the **Structure 3D** view.

NOTE In order access the **Simple Wind Loading** dialog box, you must first create and select the load case where you want to add the simple wind loads.

1. In the **Loading** list, select a manually created wind load case.
2. On the **Load** tab, click **Simple Wind**.
3. In the model, click to define the start point of the load width.
4. Click to define the end point of the load width.

   The **Simple Wind Loading** dialog box opens. The dialog box allows you to specify a single area wind load from the lowest level of the building up to the highest level that contains a rigid diaphragm.

   TIP The wind load does not have to begin at the lowest level of the building.

   If you have one or more levels below ground, you can adjust the value of the lowest level in the **Simple Wind Loading** dialog box.

5. Specify the required area load.
6. To create a stepped profile up the height of the building, use the **Insert Above**, **Insert Below**, **Quick Above...** and **Quick Below...** buttons.
7. Specify the load for each level.
8. When you load is complete, click **OK**.

Modify simple wind load vertical properties

1. Double-click anywhere in an existing simple wind load.

   The **Simple Wind Loading** dialog box opens.
2. Modify the load properties according to your needs.
3. Click **OK**.

Modify the simple wind load width

1. Select the simple wind load that you want to modify.
2. Select one of the two nodes that define the load width.
3. Click where you want to move the selected node.

   Tekla Structural Designer moves the selected node to the new location, and regenerates the load.

Apply snow loads using the snow wizard

In Tekla Structural Designer, you apply snow loads to your structure by running the **Snow Wizard** to input basic snow data and set up the required snow loadcases. Provided that Roof Panels have been modeled, some of these loadcases will be populated with Uniform Snow automatically; the remaining loadcases then have Uniform Snow, Valley Snow and Local Drift Snow applied manually as required. As an alternative, you can also choose to apply the snow loads manually.

NOTE The **Snow Wizard** is not currently available for the Indian, Australian, or British Standard snow code variants.

Apply snow loading manually (page 577)

How do I define snow loading? (Playlist)

Overview of snow loading using the snow wizard

This section provides a brief overview of snow loading in Tekla Structural Designer.

The intensity of snow load is based upon geographic location, building/roof geometry, environmental factors and local roof factors.

All snow loading falls into three categories:

- Uniform Snow load (the first fall of snow)
- Drifted uniform snow (the first fall of snow blown into uneven uniform loading)
- Drift loading (local build-up of snow load behind steps, objects, parapets)

Prior to running the **Snow wizard...** you should ensure that roofed areas of the model are 'clad' with roof panels.

The **Snow wizard...** can then be run in order to define the basic snow load factors and in the case of the Eurocode, the snow load cases and types relevant to the particular National Annex.

From this information all the required snow loadcases are automatically set up and the loads in the undrifted (or balanced) snow load case are created.

Following the wizard, you then manually define the drift cases. To do this, select the relevant snow loadcase, and then define the key attributes for the drift load prior to placing the load in the relevant position on the roof of the building.

The end result is a series of snow loadcases ready to be combined in the Combination Generator with other load cases.

The basic steps are summarized as follow:

1. Apply **Roof Panel**s to the model
2. Run the **Snow wizard...**
3. Apply snow loads to the drift loadcases set up by the wizard
4. Combine snow loadcases into design combinations
5. Perform the static design

How do I define snow loading? (Playlist)

Roof panel types

For the snow loading calculations, Tekla Structural Designer has to distinguish between monopitch (monoslope) and pitched roofs. This is determined from the RoofType property that has been assigned to the roof panel.

The RoofType property is mapped for the determination of snow loading as follows:

RoofType	For snow loading this is considered as:
Default	Monopitch (Monoslope)
Flat	Monopitch (Monoslope)
Monopitch (Monoslope)	Monopitch (Monoslope)
Hip Gable	Pitched
Hip Main	Pitched
Mansard	Pitched

Run the snow load wizard

With the **Snow Wizard**, you can automatically generate snow load cases in both **ASCE7** and **EN 1991-1-3** snow loading codes. Using the wizard, you can set up basic snow data, snow load cases, and snow loads within the balanced, or undrifted, snow load cases.

RESTRICTION

- The **Snow Wizard** is not currently available for the Indian, Australian, or British Standard snow loading code variants.
- Snow loads in drift snow load cases are not set up automatically, and must therefore be manually applied. For more information, see Manually apply snow loads to snow load cases.

---

1. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

   The **Snow Wizard** opens.
2. On the first page of the **Snow Wizard**, define the basic data.
3. Click **Next**.
4. On the second page, specify the load cases that you want to generate, and the number of drift directions to be considered.
5. Click **Finish**.

   If necessary, you can now review the snow load cases that you created.

See also

Snow wizard (ASCE7) (page 2435)

Snow wizard (Eurocode) (page 2426)

How do I define snow loading? (Playlist)

Snow loadcases (ASCE7)

This section will look at the snow loadcases when the headcode is set to AISC.

The snow loadcases that are set up will depend on the head code that is being worked to.

NOTE All loads and lengths of loads are assumed to act on a horizontal projection of the roof surface.

Once set up these loadcases are standard loadcases so you can include them in combinations in the normal manner.

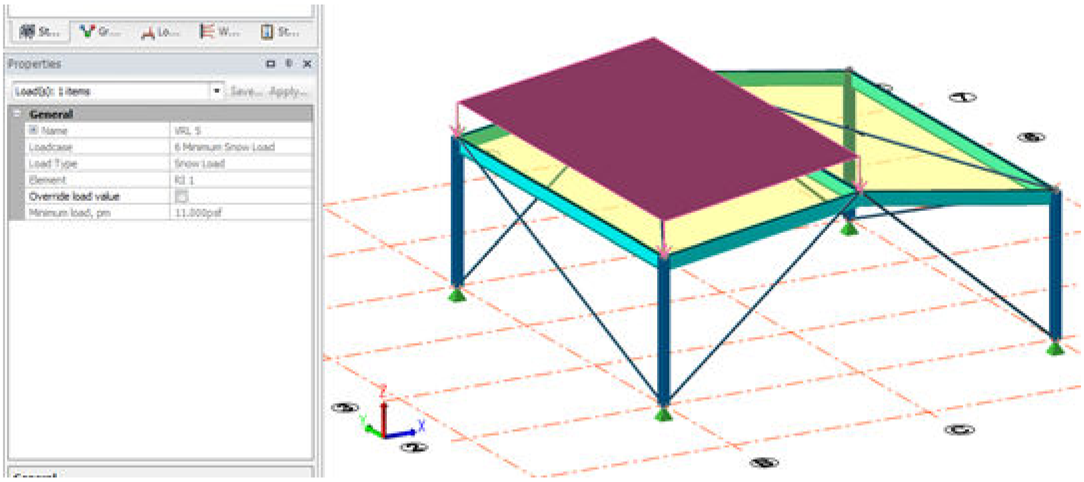
In the **Snow wizard...** you specify the Snow Loadcases to be set up from the following list:

- Minimum Snow Load - for low sloped roofs
- Balanced Snow load - uniform snow load and rain on snow load

- Unbalanced Snow Load - the number of cases required being chosen to reflect the number of wind directions considered
- Drift Snow Load
- Rain on Snow Surcharge

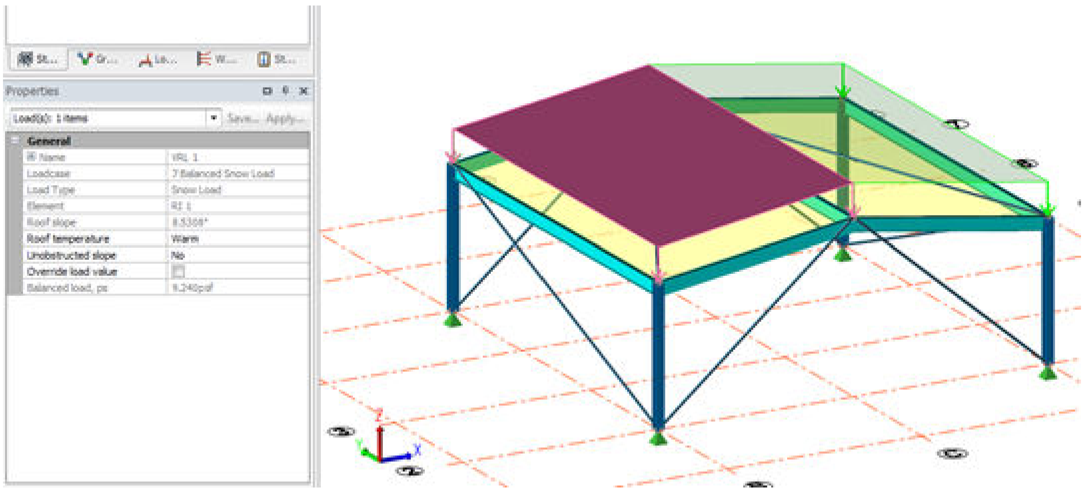
Minimum snow load

Loads would need to be manually applied to this loadcase on completion of the **Snow wizard...** - the only snow load applicable being Uniform Snow.



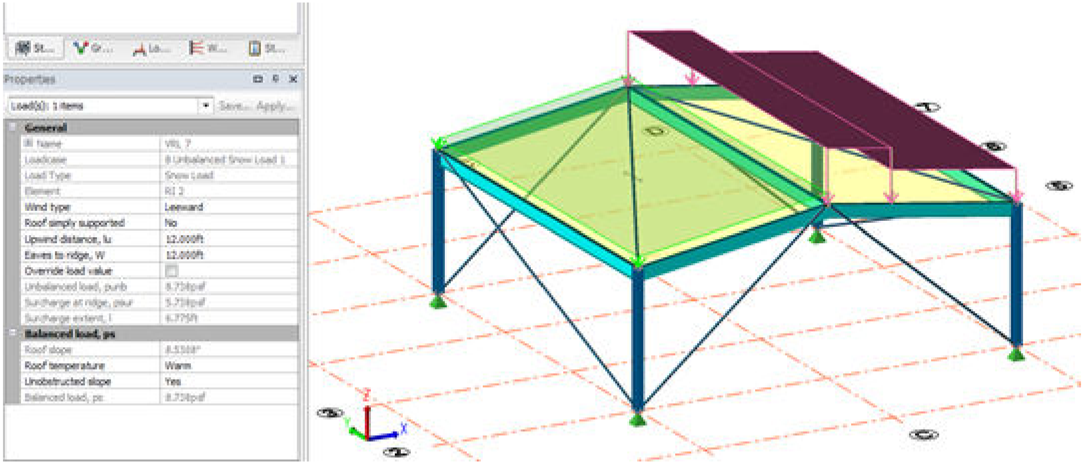
Balanced snow load

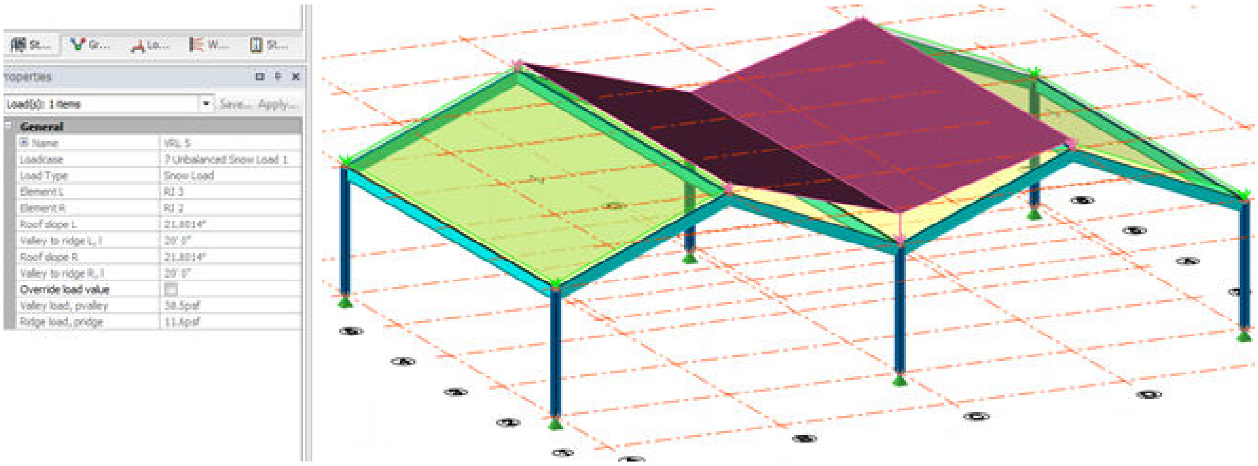
This loadcase is automatically populated on completion of the **Snow wizard...**



Unbalanced snow load

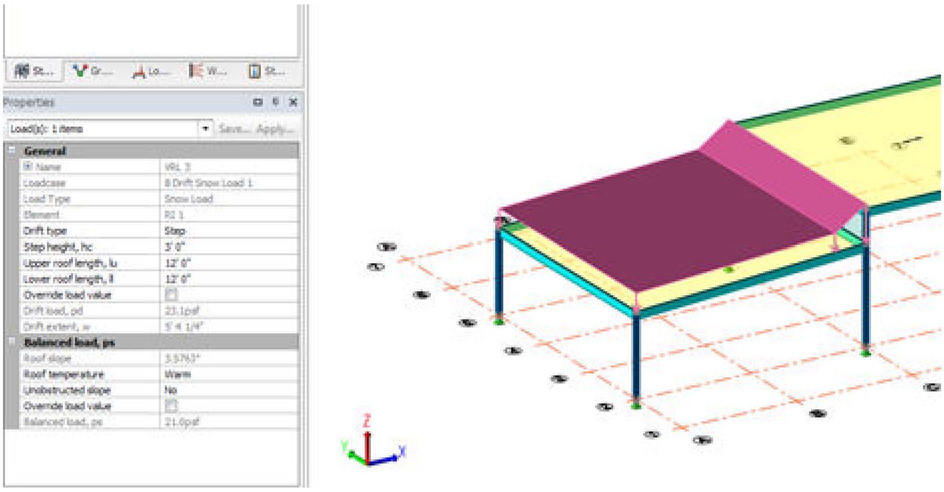
Loads would need to be manually applied to these loadcases on completion of the **Snow wizard...** - both Uniform Snow and Valley Snow loads may be applicable, (depending on the model geometry).





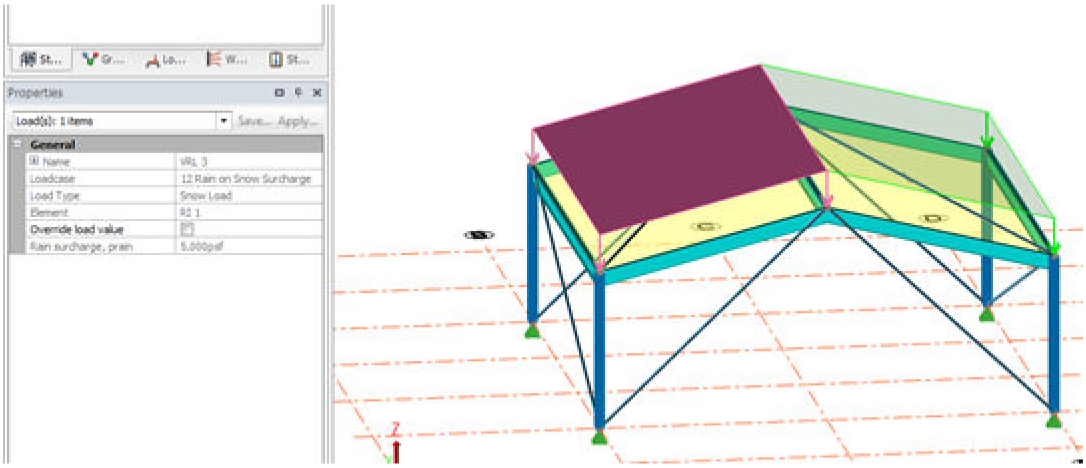
Drift snow loads

Loads would need to be manually applied to these loadcases on completion of the **Snow wizard...** - the only snow load applicable being Local Drift Snow.



Rain on Snow Load Surcharge

This loadcase is automatically populated on completion of the **Snow wizard...**



Snow loadcases (Eurocode)

This section will look at the snow loadcases when the headcode is set to Eurocode.

The snow loadcases that are set up will depend on the head code that is being worked to.

NOTE All loads and lengths of loads are assumed to act on a horizontal projection of the roof surface.

Once set up these loadcases are standard loadcases so you can include them in combinations in the normal manner.

In the **Snow wizard...** you choose the Snow Loadcases to be set up from the following list:

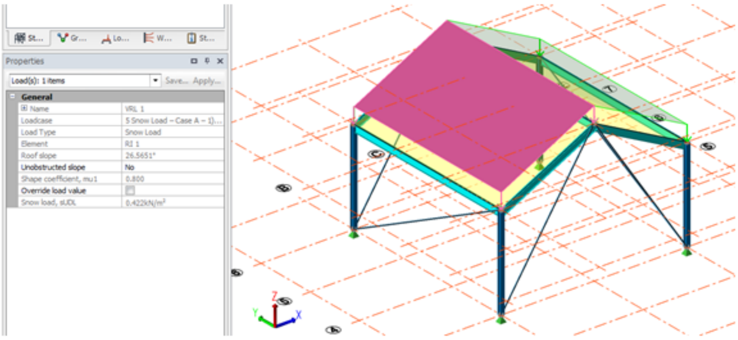
- Snow Load - Case A - 1) Undrifted
- Snow Load - Case A - 2) Drifted *
- Snow Load - Case B1 - 1) Undrifted Snow Load - Case B1 - 2) Drifted *
- Snow Load - Case B1 - 3) Undrifted (Acc)
- Snow Load - Case B1 - 4) Drifted (Acc) *
- Snow Load - Case B2 - 1) Undrifted
- Snow Load - Case B2 - 2) Drifted *
- If any drifts from Annex B are selected
    - Snow Load - Case B2 - 3) Drifted (Annex B) (Acc) *
- Snow Load - Case B3 - 1) Undrifted
- Snow Load - Case B3 - 2) Drifted *
- Snow Load - Case B3 - 3) Undrifted (Acc)
- If any drifts from Annex B are selected
    - • Snow Load - Case B3 - 4) Drifted (Annex B) (Acc) *

NOTE For cases marked * - the number of cases actually set up will depend on the number of wind directions that are asked for.

NOTE The Eurocode / National Annex recommends which loadcases to generate.

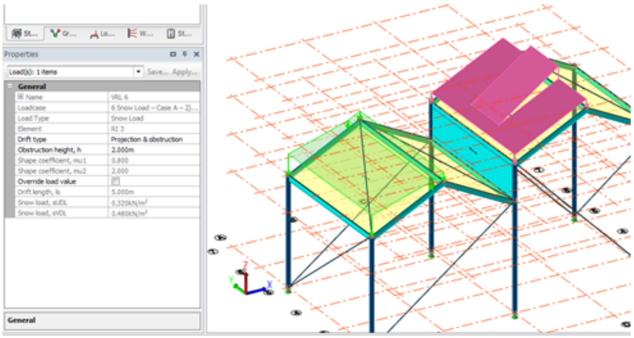
Undrifted loadcases

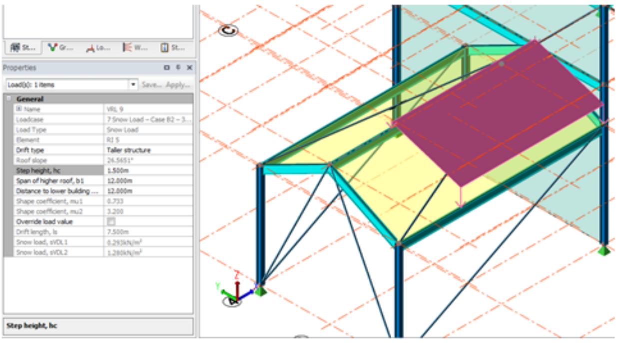
Any undrifted loadcases that have been set up in the **Snow wizard...** are automatically populated with uniform loading on completion of the wizard



Drifted loadcases

All drifted loadcases that have been set up in the **Snow wizard...** would need to be have their snow loads manually applied on completion of the wizard.





Apply drift loads to load cases on completion of the snow wizard

The loads in the drifted snow load cases have to be applied manually as uniform snow, valley snow, or local drift snow loads. Any snow loads that are not appropriate for the selected load case cannot be selected.

Apply uniform snow loads

1. In the **Loading** list, select a snow load case generated by the wizard appropriate for uniform snow.
2. On the **Load** tab, click **Snow Load** --> **Uniform Snow**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click the relevant roof panel.

   NOTE In the unbalanced cases, setting **Roof type** option to **Pitched** allows you to select whether the wind type of the load should be **Windward** or **Leeward**.

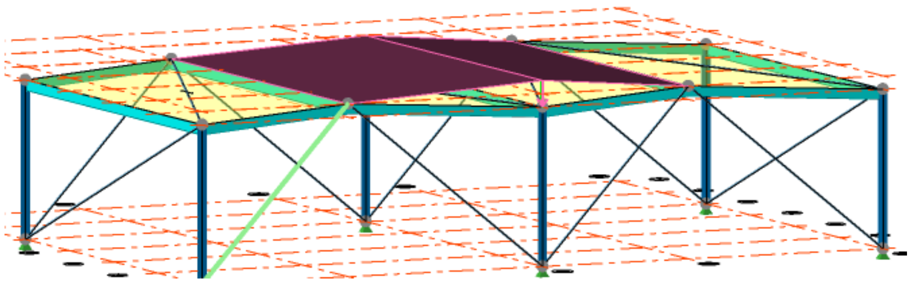
Apply a valley snow load

1. In the **Loading** list,select a snow load case generated by the wizard appropriate for valley snow.
2. On the **Load** tab, click **Snow Load** --> **Valley Snow**.
3. Select the left roof panel of the valley in which the load is applied.

   Tekla Structural Designer automatically determines the right panel, and calculates the valley load.

   TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

   NOTE Graphic displays depict the loading magnitude of a snow load, and not depth of loading. Remember this when reviewing the load that has been calculated on structures like the low pitched valley model shown below:



Although the above image appears to indicate snow load rising above the ridges, the display actually views the loading magnitude.

Apply a local drift snow load

1. In the **Loading** list, select a snow load case generated by the wizard appropriate for local drift snow.
2. On the **Load** tab, click **Snow Load** --> **Local Drift Snow**.
3. Go to the **Properties** window.
4. Specify the drift type, and then enter the height, and any other parameters required to define the drift.
5. In the model, select the roof panel edge where the drift is to form, taking care to click close to the required edge.

TIP The edge closest to the cursor is highlighted, if this isn't where you want the drift to form move the cursor toward the edge required before clicking to select.

6. Click the start point of the drift on the highlighted edge.
7. Click the end point of the drift on the highlighted edge.

TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

Override snow loads

You can apply user overrides for snow load values and drift lengths, if necessary.

1. Select the snow load.
2. In the **Properties** window, select the **Override load value** option.

3. Specify the override value.

   Tekla Structural Designer maintains the user overrides, even if you rerun the **Snow Wizard**.

Update snow loads

If you have created a snow model using the **Snow Wizard**, and later decide to modify the roof geometry, the snow loads need to be recalculated. To do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Update Snow Loads**.

See also

Delete the snow model (page 577)

Delete the snow model

Occasionally, you may need to delete the entire snow model, and start the process from scratch. In order to do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Delete Snow**.

  Tekla Structural Designer resets the basic data that was previously defined in the **Snow Wizard**, and deletes the snow load cases.

Apply snow loading manually

This approach provides an alternative method to apply snow loading to the structure, without having to run the **Snow Wizard**

In order to manually apply snow loads, you must first create a loadcase (page 515) and set its load type to Snow.

You can then manually apply loads (page 538) to this loadcase in the normal way.

NOTE Snow loadcases created manually in this way can have panel, member, and structure loads applied but cannot have snow (i.e. Uniform Snow, Valley Snow and Local Drift Snow) loads applied.

Apply seismic loads

Use the **Seismic Wizard** to define all the parameters required to set up the seismic loadcases and combinations.

RESTRICTION The **Seismic Wizard** is not currently available for the Australian **AS:1170.4** loading code variant.

Create seismic loads in the Seismic Wizard

1. On the **Load** tab, click **Seismic Load** --> **Seismic Wizard...**

   The (page 579) opens.

   NOTE The parameters in the **Seismic Wizard** vary according to the loading code that you have selected in **Model Settings**.

2. In the **Seismic Wizard**, define the necessary parameters for the seismic loading and load cases. To go to the next page, click **Next**.
3. Once you have defined the parameters, click **Finish**.

   The **Combination Generator** dialog box opens.
4. In the **Combination Generator** dialog box, define the seismic load combinations.
5. Click **Finish**.

Display the horizontal design spectrum

After running the **Seismic Wizard**, you can view the horizontal design spectrum.

1. On the **Load** tab, click **Seismic Load** --> **Horizontal Spectrum**.
2. If necessary, in the **Properties** window, you can switch **Direction** between **Dir 1** and **Dir 2**.

Delete seismic loads

- On the **Load** tab, click **Seismic Load** --> **Delete Seismic**.

  Tekla Structural Designer deletes all the previously defined seismic loads in your model.

See also

Seismic wizard in detail (page 579)

Seismic analysis and conventional design (page 1199)

Seismic analysis and seismic design (page 1200)

Seismic analysis and design handbook (page 1186)

Seismic wizard in detail

The **Seismic Wizard** is run to specify the seismic analysis method (RSA or ELF) and to set up the seismic loadcases and combinations.

The information that is entered in the seismic wizard will vary according to the seismic loading code that is being worked to:

- Using the ASCE7 seismic wizard (page 579)
- Using the UBC 1997 seismic wizard (page 587)
- Using the Eurocode EN1998-1:2004 seismic wizard (page 594)
- Using the IS1893 seismic wizard (page 603)
- Code spectra and site specific spectra (page 609)
- Seismic loadcases (page 621)

See also

Seismic analysis and design handbook (page 1186)

Using the ASCE7 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the ASCE Horizontal Design Spectrum (page 611).
Site Specific Spectra (user defined -	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant.

Property	Description
based on S_d and T)	For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 609).
Site Specific Spectra (user defined - generic curve)	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve manually, or by pasting data from a spreadsheet. **NOTE** The spreadsheet data must be in the form of period vs acceleration values and in ascending order of period (the input dialog features an automatic Sort command if the latter is not the case). See: Code spectra and site specific spectra (page 609).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE7-10 (not ASCE7-05) there is an additional value of C_s calculated for a building < 5 stories and where T <0.5s. In this instance S_s is taken as 1.5 and the additional value of C_s should be used to give a final value of C_s. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.

Property	Description
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Site Occupancy	
Site class	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. as defined in ASCE7-05 & ASCE-10 Table 20.3-1.
Occupancy/Risk Category	This field allows you to set the appropriate occupancy category, I-IV, as defined in ASCE 7-05 &-10 Table 1-1.
Importance Factor, I_e	This is automatically derived from the occupancy class.
Override check box	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).
Seismic Design Category (SDC)	
Seismic Design Category (SDC)	User editable (range A-F)
Alternative seismic design category determination	Select this check box in order to use the alternative determination from IBC2009 1613.5.6.1 / IBC2012 1613.5.6.
User Defined SDC	Select this check box in order to specify your own SDC Category
Max earthquake spectral response acceleration	
S_S – short period (0.2s)	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
S_1 – 1.0s period	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
Design spectral response spectral acceleration	
S_{DS} – short period	This is automatically determined from SS and the site class, according to ASCE 7-05&-10

Property	Description	
S_{D1} – 1s period	This is automatically determined from S1 and the site class, according to ASCE 7-05&-10	
Next	Unless the SDC is type A, clicking **Next** takes you to the **Structure Irregularities page** described below; for SDC type A only, clicking **Next** takes you to the **Seismic Loading page**.	

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description	
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.	
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.	
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.	
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.	

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description	
Fundamental Period Definition		

Property	Description
Use approx fundamental period T_a	This is automatically derived from ASCE7-05&-10 clause 12.8.2.1.
User defined fundamental period T	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period	
Long period transition period, T_L	This figure can be determined from figs 22-15, 16, 17, 18, 19 and 20 found in ASCE 7-05 or figs 22-12, 13, 14, 15 and 16 found in ASCE 7--10.
T_S	This value is derived ($T_s = S_{D1}/S_{DS}$)
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems (ref ASCE7-05 / ASCE7-10 Table 12.8-2
Approx fundamental period, T_a	This is automatically derived from ASCE7-05&-10 clause 12.8.2.1.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

For SDC types B to F this page is used to determine the response modification coefficient, R, and other factors in the X and Y directions.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	

Property	Description
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from ASCE7-05 &-10 Table 12.2-1.
Type	This field allows you to set the appropriate type for the chosen system from ASCE7-05 &-10 Table 12.2-1.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.
Deflection amplification factor, C_d	This is automatically derived, but you are given the facility to edit the calculated value.
Redundancy factor, ρ	This is automatically derived from the SDC, (SDC B-C ρ = 1.0, SDC D-F ρ = 1.3) but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those load cases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This “base shear combination” is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

If you select the **Include localization** option, this page is then used to define appropriate localization parameters for using the ASCE7 code outside of the US.

Property	Description
Seismic & RSA	

Property	Description
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (ASCE7 code value = 1/(R/I)).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (ASCE7 code value V = Cs x W and Cs = SDS / (R /Ie)).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (ASCE7 code value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),(ASCE7 factor = 0.2 x S_{DS} x Dead loads).
RSA only (only available if RSA has been selected)	
% Mass participation for RSA	This field allows you to set your own required % mass participation, (ASCE7 code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.

Property	Description
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE7 code value: 85% x V / V_t).
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE7 code value: 85% x V / V_t). **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 621) and open the **ASCE7 Seismic Combination Generator** described below.

ASCE7 Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following ASCE7-5 and ASCE7-10 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run

Property	Description	
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.	
	Clicking **Next** takes you to the **Combinations page** described below.	
Combinations page		
Generate	Select the seismic combinations to be generated with factors as specified.	
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.	
Next	Click **Next** to specify the service factors.	
Finish	Click **Finish** to generate the Seismic combinations.	

Using the UBC 1997 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Uniform Building Code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
   a. Set the Design Code for Seismic Loading to UBC
   b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the UBC Horizontal Design Spectrum.
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 609).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE7-10 (not ASCE7-05) there is an additional value of C_S calculated for a building < 5 stories and where T <0.5s. In this instance S_S is taken as 1.5 and the additional value of C_S should be used to give a final value of C_S. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Soil Profile Occupancy	
S - Soil Profile Type	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. As defined in UBC97 Table 16-J.

Property	Description	
Occupancy Category	This field allows you to set the appropriate occupancy category, I-IV. As defined in UBC97 Table 16-K.	
Importance Factor, I	This is automatically derived from the occupancy category.	
Override check box	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).	
Seismic Zone		
Seismic Zone	This field allows you to set the appropriate zone, I-4. As defined in UBC97 Figure 16-2	
Z - Seismic Zone Factor	This is automatically derived for the seismic zone.	
Near Source (for Seismic Zone 4 only)		
Seismic Source Type	This field allows you to set the appropriate source type, A-C.	
Distance to Seismic Source	The distance to the seismic source.	
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.	

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description	
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.	
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.	
Analysis procedur	Having specified any irregularities, you then choose the analysis procedure.	

Property	Description
e to be used	• Use Static Force Procedure • Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems
Approx fundamental period, T_a	This is automatically derived.

Property	Description	
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.	
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.	

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description	
Seismic force resisting system Dir 1 and Dir 2		
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from UBC97 Table 16-N.	
Type	This field allows you to set the appropriate type for the chosen system from UBC97 Table 16-N.	
Coefficients & Factors Dir 1 and Dir 2		
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.	
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.	
Redundancy factor drift, ρ_{drift}	This factor has to be entered manually.	
Redundancy factor, ρ	This factor has to be entered manually.	
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.	

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those load cases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This “base shear combination” is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

If you select the **Include localization** option, this page is then used to define appropriate localization parameters for using the UBC code outside of the US.

Property	Description
Seismic & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (UBC code value = 1/R).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (UBC code value V = C_V x I x W / (R x T)).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (UBC code value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination), (UBC factor = 0.5 x Ca x I x Dead loads for strength design and 0.0 for allowable stress design).
RSA only (only available if RSA has been selected)	

Property	Description	
% Mass participation for RSA	This field allows you to set your own required % mass participation, (UBC code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.	
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design}	
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design} **NOTE** A zero factor means no scaling will be performed on deflections.	
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 621) and open the **UBC Seismic Combination Generator** described below.	

UBC Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following UBC 1997 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Using the Eurocode EN1998-1:2004 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the EN 1998-1 Horizontal Design Spectrum (page 613).
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 609).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE7-10 (not ASCE7-05) there is an additional value of C_S calculated for a building < 5 stories and where T <0.5s. In this instance S_S is taken as 1.5 and the additional value of C_S should be used to give a final value of C_S. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	

Property	Description
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic inertia combination is determined.
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Ground acceleration	
Region	Only applies for the Malaysia NA: • Peninsular Malaysia, • Sarawak, • Sabah
Reference Peak Ground Acc, a_{gR}	Refer to PD6698:2009
Design Ground Acc, a_g	a_{gR} x γI.
Importance & Ground	
Importance class	For Base Eurocode and UK NA: I- IV as defined in BS EN 1998-1:2004 - Table 4.3. For Singapore NA: Ordinary or Special
Ground type	For Base Eurocode and UK NA: A-E For Malaysia NA: Rock, Stiff Soil, Flexible Soil For Singapore NA: C, D, S_1 (A - Rock, B - Very dense soil/gravel/clay, C - Deposits of dense/medium dense soil, D - Loose to medium soil, E - Surface alluvium. Ref BS EN 1998-1:2004 - Table 3.1.)
Importance Factor, γ_I	This is automatically derived from the occupancy class.

Property	Description	
Spectrum type	For Base Eurocode and UK NA: Spectrum Type - 1 or 2 BS EN 1998-1:2004 - Cl 3.2.2.2.1(P) For Malaysia NA: Spectrum Type - 1 For Singapore and Norway NA: (No choice)	
Lower bound factor, β	This is automatically derived.	
Upper limit of the period of the constant spectral acceleration branch, T_c	This is automatically derived.	
Structural Ductility Class	Low, Medium or High - for whole building (not directional) If ag ≤ 0.78 m/s2 or if ag x S ≤ 0.98 m/s2 - Structure suitable for Low Seismicity (where S is the soil factor) If not - Structure suitable for Medium or High Seismicity	
Site natural period, T_S (Malaysia NA only)	This is only required if the ground type is Flexible Soil.	
Elastic response spectral displacement, S_{DR} (Malaysia NA only)	This is only required if the ground type is Flexible Soil.	
Next	Clicking **Next** takes you to the **Structure regularity page** described below.	

NOTE The Norway NA for EC1998 has the following differences:-

Soil factors specific to Norway

- φ factor for Imposed load types A, B and C = 1.0 (not 0.8)

- γI importance factors specific to Norway- Spectrum specific to Norway (TB, TC and TD factors)

- q values for Ductility Class Medium use for both Medium and High

Structure regularity page

For Medium and High structural ductility this page is used to indicate any irregularities in plan or elevation. (For Low structural ductility only the irregularities in elevation are displayed.)

Property	Description
Structure Plan Regularity - Cl 4.2.3.2	Check the appropriate boxes to define any plan irregularities.
Structure Elevation Regularity - Cl 4.2.3.3	Check the appropriate boxes to define any elevation irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysi **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$

Property	Description
Fundamental Period Definition	
Use approx	If RSA was selected on the previous page this option is dimmed.

Property	Description	
fundamental period T_A		
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.	
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.	
Fundamental Period Dir 1 and Dir 2		
Structure Type	Select from Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures (ref BS EN 1998-1 Cl 4.3.3.2.2)	
Approx fundamental period, T_A	This is automatically derived.	
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_A, or it can be a user defined value, or it will be calculated from the modal analysis.	
Next	Clicking **Next** takes you to the **Behaviour Factor page** described below.	

Behaviour Factor page

For Low Ductility Class, q = 1.5, (but the value can be changed by the user) - all other fields on the page are dimmed and cannot be changed.

For Medium or High Ductility Classes, this page is used to determine the Behaviour Factor in direction 1 and 2.

Property	Description	
Ductility Class	Medium or High.	
Structure Type	Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures	
Frame Type	This options displayed here depend on the structure type selected above.	

Property	Description
α_u / α_l	This is a user defined multiplication factor - for the structure
User defined q	For Medium or High Ductility Classes, select this check box in order to edit the calculated q value.
Behaviour Factor, q	This is automatically derived, but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Seismic Inertia Combination page** described below.

Seismic Inertia Combination page

This page is used to set up the load cases for the Seismic Inertia Combination.

You should include those load cases that you want to contribute to the effective seismic weight of the structure.

NOTE This "Seismic Inertia Combination" is used to develop the seismic design loading and is classed as a modal mass combination for the modal analysis. It is not used in any other analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

If you select the **Include localization** option, this page is then used to define appropriate localization parameters for using the ASCE7 code outside of the US.

Property	Description
Seismic & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (Eurocode value = 1/q).
Design Static Base Shear, V (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (Eurocode value $F_B = S_d (T_l) \times m \times \lambda$).

Property	Description
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (Eurocode value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),((Eurocode - No vertical loads considered - factor = 0.0).
RSA only (only available if RSA has been selected)	
% Mass participation for RSA	This field allows you to set your own required % mass participation, (Eurocode value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (Eurocode - No base shear adjustment).
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (Eurocode - No adjustment). **NOTE** A zero factor means no scaling will be performed on deflections.

Property	Description
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 621) and open the **EC8 Seismic Combination Generator** described below.

EC8 Seismic Combination Generator

The Combination Generator uses the two “core” combinations from EC8 that include seismic loads as below:

Equation 6.12a

$G_{kj} + A_{Ed} + \psi_{2,i} \times Q_{k,i}$

Equation 6.12b

$G_{kj} + A_{Ed}$

Provided the generate option is selected these will then be used, with factors as specified, to generate an appropriate number of combinations to cover for direction +/- Dir1 and Dir2 with +/- eccentricities.

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service combinations.

Property	Description	
Finish	Click **Finish** to generate the Seismic combinations.	

Using the IS1893 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Indian IS1893 code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
    a. Set the Design Code for Seismic Loading to IS1893 (Part 1) and select the year required
    b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description	
Code Spectra	Choose this option to use the IS893 (Part 1) Horizontal Design Spectrum.	
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 609).	
Next	Clicking Next takes you to the **Basic information page** described below.	

NOTE No vertical spectrum is considered in the current release.

Basic information page

The SZone Factor is determined on this page.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Zone & Site	
Seismic Zone	This field allows you to set the appropriate zone, II-V.
Site class	This field allows you to set the appropriate soil conditions, I-III.
Importance Factor, I	This is automatically derived from the occupancy class.
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.

Structure Irregularities page

This page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure

Property	Description
	• Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select the structural type.
Approx fundamental period, T_a	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.

Property	Description	
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.	

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description	
Seismic force resisting system Dir 1 and Dir 2		
Type	This field allows you to set the appropriate type.	
System	This field allows you to set the appropriate system for the chosen type.	
Coefficients & Factors Dir 1 and Dir 2		
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.	
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.	

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those load cases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This “base shear combination” is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

If you select the **Include localization** option, this page is then used to define appropriate localization parameters.

Property	Description	
ELF & RSA		
Scale Factor between elastic and	This field allows you to override the code value for the scale factor.	

Property	Description
design spectra (Dir1 and Dir2)	
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear.
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination).
RSA only (only available if RSA has been selected)	
% Mass participation for RSA	This field allows you to set your own required % mass participation. **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors

Property	Description
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors. **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 621) and open the **Seismic Combination Generator** described below.

Seismic Combination Generator

The Combination Generator sets up seismic combinations from the "core" combinations that include seismic loads.

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.

Property	Description	
Finish	Click **Finish** to generate the Seismic combinations.	

Code spectra and site specific spectra

Code Spectra

Different analysis procedures are provided for determining the elastic design response spectrum, the first of which is simply to adopt the code spectra.

- Default spectra (page 609)

Site Specific Spectra (user defined)

In addition to the code spectra, Tekla Structural Designer also allows you to define your own site specific spectra by specifying significant periods and acceleration. This procedure is applicable to both ELF and RSA analysis.

The curve equations used for the different head codes are as follows:

- ASCE Horizontal Design Spectrum (page 611)
- EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA) (page 613)
- EN 1998-1 Horizontal Design Spectrum (Malaysia NA) (page 615)
- IS893 (Part 1) Horizontal Design Spectrum (page 617)
- ASCE7/UBC Horizontal Design Spectrum - Taiwan (page 619)

This procedure can be used for locations which use another country's loading and design codes where the code spectra are not relevant and so the local site spectra need to be defined.

Site Specific Spectra (user defined - generic curve)

A third route is to set a site specific generic curve, either manually, or by pasting data from a spreadsheet. This procedure requires an RSA to be run (not ELF). This has been added primarily for remote markets using the US codes It is available for the US Headcode with loading codes:

- ASCE7-05
- ASCE7-10
- ASCE7-16

It is not available for UBC, EC or IS codes as these codes do not define the process to be used with a generic spectrum.

User defined RSA curve

Default spectra

The following details the initial values of the parameters for spectra for each of the seismic head codes:

ASCE - Horizontal Design Spectrum

Limits	Equation	Segment	Ref
$0 \leq T \leq T_0$	S_{DS} x (0.4 + 0.6 x T / T_0) x (1/(R/I_e)	Start point/ linear	11.4.5 (1)
$T_0 \leq T \leq T_s$	S_{DS} x (0.4 + 0.6 x T / T0) x (1/(R/Ie)	Constant	11.4.5 (2)
$T_s \leq T \leq T_L$	(S_{D1}/T)/(R/I)	Constant/T	11.4.5 (3)
$T_L \leq T \leq 4s$	(S_{D1} x T_L / T^2)/(R/I)	Constant/T^2	11.4.5 (4)

UBC - Horizontal Design Spectrum

Limits	Equation	Segment
$0 \leq T \leq T_0$	C_a + 1.5 x T / T_0 x (2.5 / R-1)	Start point/linear
$T_0 \leq T \leq T_s$	2.5 x C_a /R	Constant
$T_s \leq T$	C_v / T/R	Constant/T

EC - Horizontal Design Spectrum (UK, Singapore and Europe)

Limits	Equation	Segment	Ref
$0 \leq T \leq T_B$	a_g x S x [2/3 + T/T_B x (2.5 / q - 2/3)]	Start point/ linear	3.13
$T_B \leq T \leq T_C$	a_g x S x 2.5 / q	Constant	3.14
$T_C \leq T \leq T_D$	a_g x S x T_C/T x 2.5 / q	Constant/T	3.15
$T_D \leq T \leq 4s$	a_g x S x 2.5 / q x (T_C x T_D) / T^2	Constant/T^2	3.16

EC - Horizontal Design Spectrum (Malaysia)

For **Rock soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	((2 x π)^2 x γI x S_{DR}(1.25) / (T_C x T_D)) / q	Constant
$T_C \leq T \leq T_D$	((2 x π)^2 x γI x S_{DR}(1.25) / (T x TD)) / q	Constant/T
$T_D \leq T \leq 4s$	((2 x π)^2 / T2 x [λI x S_{DR}(1.25) + γI x m_r x (T - T_D))] / q	Constant/T^2

For **Stiff soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	((2 x π)^2 x γI x S_{DR}(1.25) x 1.5 / (T_C x T_D)) / q	Constant
$T_C \leq T \leq T_D$	((2 x π)^2 x γI x S_{DR}(1.25) x 1.5 / (T x TD)) / q	Constant/T

Limits	Equation	Segment
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda l \times S_{DR}(1.25) \times 1.5 + \gamma l \times m_r \times (T - T_D))] / q$	Constant/T^2

For **Flexible soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	$((2 \times \pi)^2 \times \gamma l \times S_{DR} (1.5T_S) \times 3.6 / (T_C \times T_D)) / q$	Constant
$T_C \leq T \leq T_D$	$((2 \times \pi)^2 \times \gamma l \times S_{DR}(1.5T_S) \times 3.6 / (T \times TD)) / q$	Constant/T
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda l \times S_{DR}(1.5T_S) \times 3.6 + \gamma l \times m_r \times (T - T_D))] / q$	Constant/T^2

IS - Horizontal Design Spectrum

For **rocky, or hard soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.40$	(2.50)	Constant/T
$0.40 \leq T \leq 4.00$	(1.00/T)	Constant/T^2

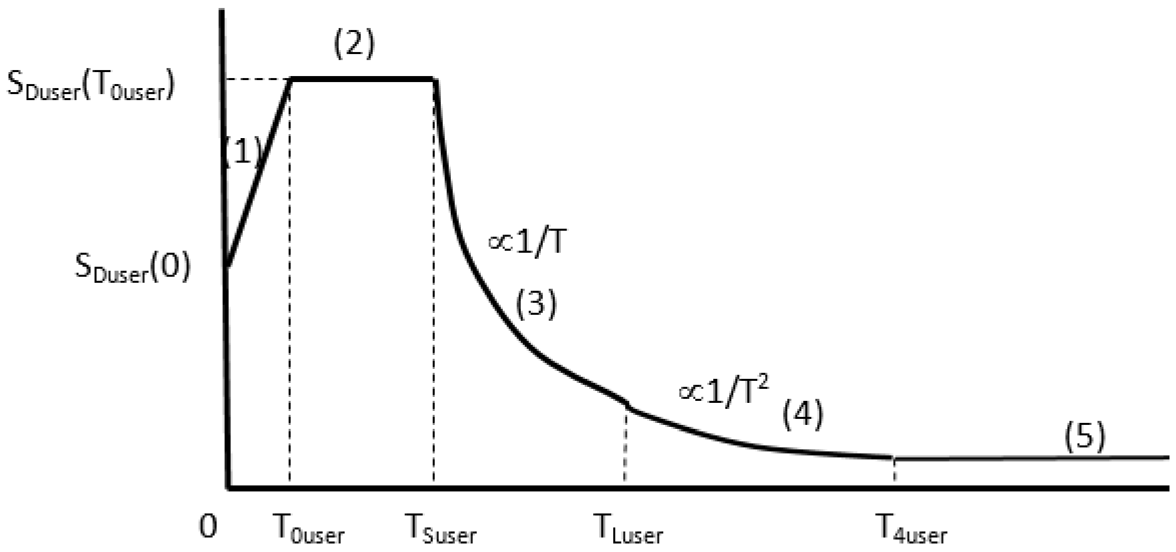
For **medium soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.55$	(2.50)	Constant/T
$0.55 \leq T \leq 4.00$	(1.36/T)	Constant/T^2

For **soft soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.67$	(2.50)	Constant/T
$0.67 \leq T \leq 4.00$	(1.67/T)	Constant/T^2

ASCE Horizontal Design Spectrum



NOTE	ASCE7 has either sloped second segment or horizontal 4th segment - not both.

Parameters

- $S_{Duser}(0)$ – units g
- $S_{Duser}(T_{0user})$ – units g
- T_{0user} – units sec
- T_{Suser} – units sec
- T_{Luser} – units sec
- T_{4user} – units sec, default = 4s

Input limits

- $S_{Duser}(0) > 0$
- $S_{Duser}(T_{0user}) > 0$ if $T_{0user} > 0$
- $0s \leq T_{0user} < T_{Suser} < T_{Luser} \leq T_{4user}$

Curve Equations

Design Response Spectrum curves for S_a(g) are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{0user}$	$S_a(g)(T) = S_{Duser}(0) + ((S_{Duser}(T_{0user}) - S_{Duser}(0)) \times T / T_{0user})$
(2) – straight line	$T_{0user} \le T \le T_{Suser}$	$S_a(g)(T) = S_{Duser}(T_{0user})$
(3) – curve	$T_{Suser} \le T \le T_{Luser}$	$S_a(g)(T) = S_{Duser}(T_{0user}) \times T_{Suser}/ T$
(4) – curve	$T_{Luser} \le T \le T_{4user}$	$S_a(g)(T) = S_{Duser} (T_{0user}) \times T_{Suser} \times T_{Luser}/ T^2$
(5) – straight line continued from (4)		

Adjusted Design Response Spectrum curves for $S_a(g)/(R/I_e)$ are defined by

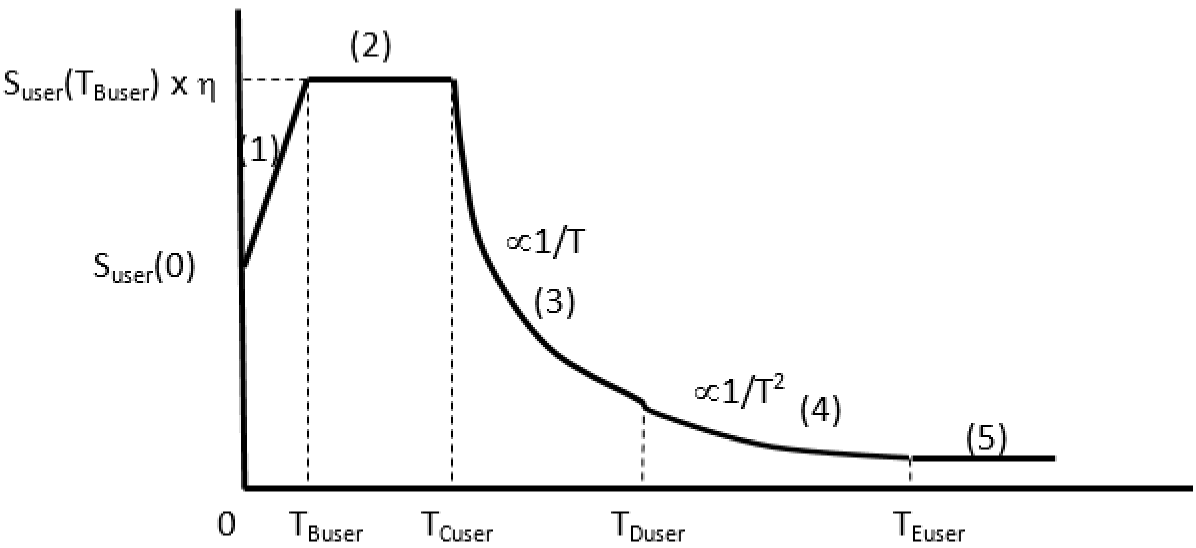
NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{0user}$	$S_D(T) = [S_{Duser}(0) + ((S_{Duser}(T_{0user}) - S_{Duser}(0)) \times T / T_{0user})] /(R/I_e)$
(2) – straight line	$T_{0user} \le T \le T_{Suser}$	$S_D(T) = S_{Duser}(T_{0user}) / (R/I_e)$
(3) – curve	$T_{Suser} \le T \le T_{Luser}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} / T / (R/I_e)$
(4) – curve	$T_{Luser} \le T \le T_{4user}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} \times T_{Luser} / T^2 / (R/I_e)$
(5) – straight line continued from (4)		

NOTE If $T_{0user} = 0$, then no line (1) exists

If $T_{Luser} = T_{4user}$ then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA)



Parameters

- $S_{user}(0)$ – units g
- $S_{user}(T_{Buser})$ – units g
- T_{Buser} – units sec
- T_{Cuser} – units sec
- T_{Duser} – units sec
- T_{Euser} – units sec, default = 4s

Input limits

- $S_{user}(0) > 0$
- $S_{user}(T_{Buser}) > 0$
- $0s \leq T_{Buser} < T_{Cuser} < T_{Duser} \leq T_{Euser}$

Curve Equations

Elastic Response Spectrum curves for $S_e/a_g$are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_e/a_g(T) = S_{user}(0) + ((S_{user}(T_{Buser}) \times \eta - S_{user}(0)) \times T / T_{Buser})$
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_e/a_g(T) = S_{user}(T_{Buser}) \times \eta$

Line	Limits	Equation
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_e/a_g(T) = S_{user}(T_{Buser})$ x η x T_{Cuser}/ T
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_e/a_g(T) = S_{user} (T_{Buser})$ x η x T_{Cuser} x T_{Duser}/ T^2
(5) – straight line continued from (4)		

Design Spectrum for Elastic Analysis curves for S_e/a_g are defined by

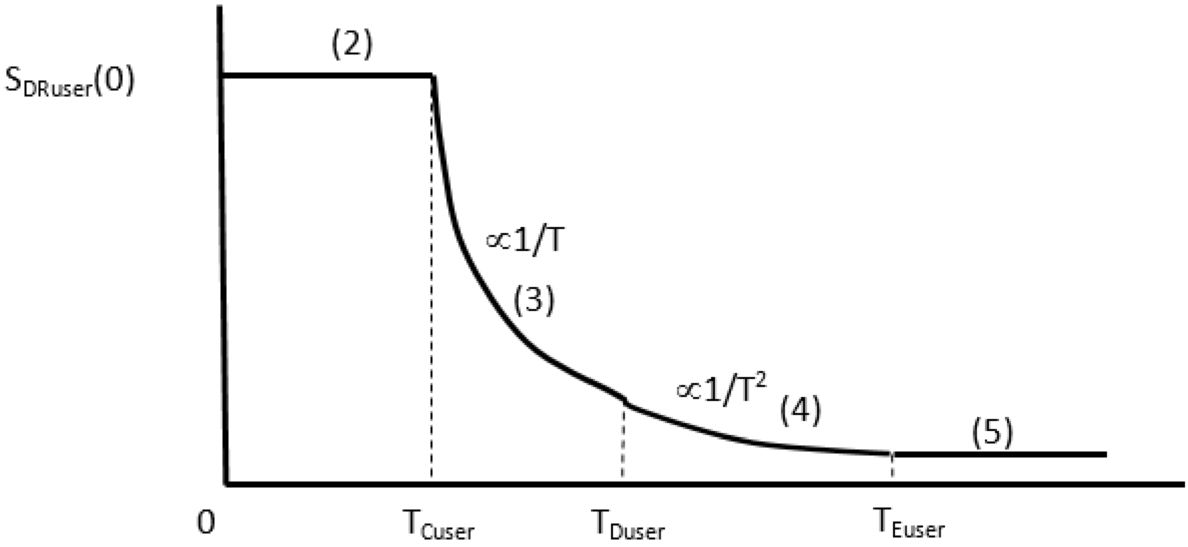
NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_d/a_g(T) = [\ 2/3 \times S_{user}(0) + ((S_{user}(T_{Buser}) / q - 2/3 \times S_{user}(0)) \times T / T_{Buser})]$
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) \times T_{Cuser}/ T / q$
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_d/a_g(T) = S_{user} (T_{Buser}) \times T_{Cuser} \times T_{Duser}/ T^2/ q$
(5) – straight line continued from (4)		

NOTE If $T_{Buser} = 0$, then no line (1) exists

If $T_{Duser} = T_{Euser}$ then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Malaysia NA)



Parameters

- $S_{DRuser}(0)$ – units g
- $S_{DRuser}(T_{Buser})$ – units g
- T_{Buser} – units sec
- T_{Cuser} – units sec
- T_{Duser} – units sec
- T_{Euser} – units sec, default = 4s

Input limits

- $S_{DRuser}(0) > 0$
- $S_{DRuser}(T_{Buser}) > 0$
- $0s \leq T_{Buser} < T_{Cuser} < T_{Duser} \leq T_{Euser}$

Curve Equations

Elastic Response Spectrum curves for $S_e/a_g$are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_e/a_g(T) = S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})$

Line	Limits	Equation
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser})$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2$
(5) – straight line continued from (4)		

Design Spectrum for Elastic Analysis curves for S_d/a_g are defined by

NOTE same curve for both Dir1 and Dir2

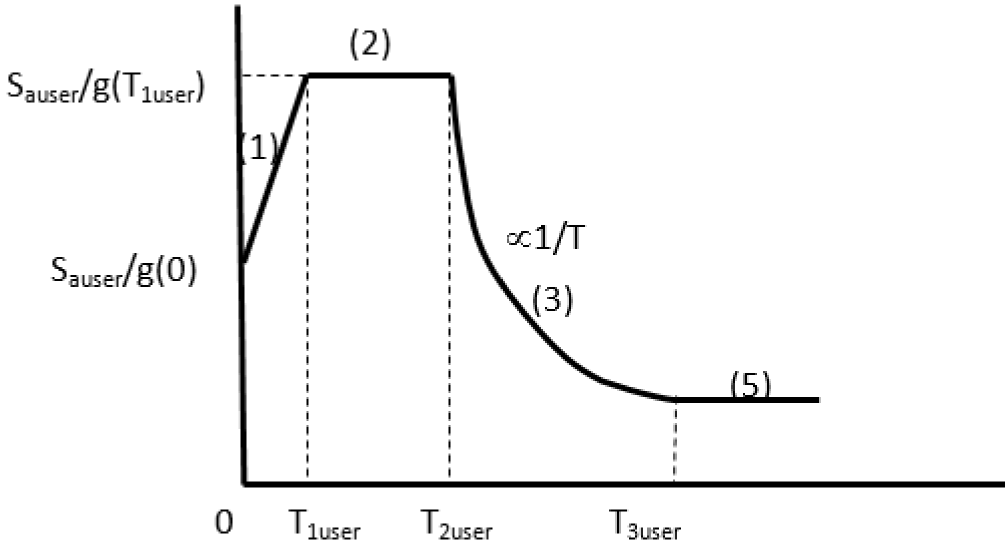
Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{Buser}$	$S_d/a_g(T) = [S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})] / q$
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T / q$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2 / q$
(5) – straight line continued from (4)		

NOTE If $T_{Buser} = 0$, then no line (1) exists(as in the example above)

If $T_{Duser} = T_{Euser}$ then no line (4) exists

NOTE The above by default does not quite equate to Line (4) in the Malaysian NA because there is an extra term $(2 \times \pi)^2 / T^2 \times \gamma l \times m_r \times (T - T_D)$ that tweaks the $1/T^2$ curve slightly. In Sarawak, this term is non-existent, in Peninsular Malaysia it does not exist for Flexible soils. In the Tekla Structural Designer User defined spectrum it is also non-existent.

IS893 (Part 1) Horizontal Design Spectrum



NOTE	In this example T3user = T4user so there is no line (4)

Parameters

- Damping factor_{user}
- $S_{auser}/g(0)$ – units g
- $S_{auser}/g(T_{1user})$ – units g
- T_{1user} – units sec
- T_{2user} – units sec
- T_{3user} – units sec, default = 4s
- T_{4user} – units sec, default = 4s

Input limits

- $S_{auser}/g(0) > 0$
- $S_{auser}/g(T_{1user}) > 0$
- $0s \le T_{1user} < T_{2user} < T_{3user} \le T_{4user}$

Curve Equations

Average Response Spectrum curves for S_a/g are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$S_a(g)(T)$ = Damping $factor_{user}$ x $[S_{auser}/g(0) + ((S_{auser}/g(T_{1user}) - S_{auser}/g(0)) \times T / T_{1user})]$
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$S_a(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user})$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$S_a(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user}) \times T_{2user}/ T$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$S_a(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user}) \times T_{2user} \times T_{3user}/ T^2$
(5) – straight line continued from (4)		

Design Response Spectrum curves for $A_h(g)$ (=$S_a/g/((2xR)/(ZxI))$) are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$A_h(g)(T)$ = Damping $factor_{user}$ x $[S_{auser}/g(0) + ((S_{auser}/g(T_0) - S_{auser}/g(0)) \times T / T_{1user})]$ But not less than Z/2
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$A_h(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user}) / ((2xR)/(ZxI))$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$A_h(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user}) \times T_{2user}/ T / ((2xR)/(ZxI))$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$A_h(g)(T)$ = Damping $factor_{user}$ x $S_{auser}/g (T_{1user}) \times T_{2user} \times T_{3user}/ T^2 / ((2xR)/(ZxI))$
(5) – straight line continued from (4)		

NOTE If $T_{1user} = 0$, then no line (1) exists

If $T_{3user} = T_{4user}$ then no line (4) exists

ASCE7/UBC Horizontal Design Spectrum - Taiwan

Taiwan code Figure 2

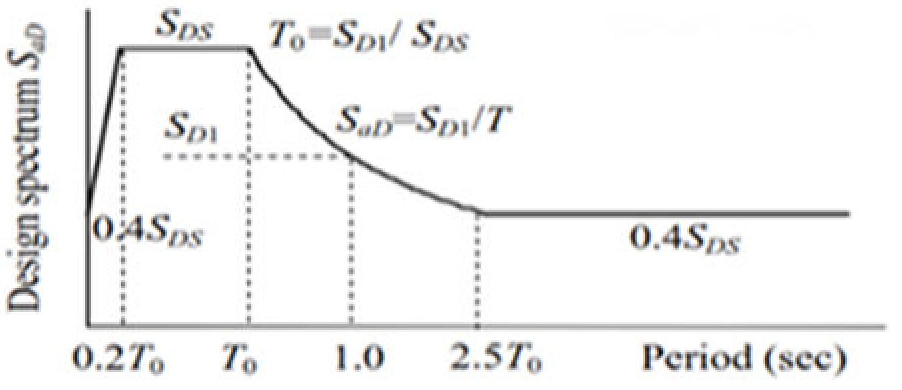


Figure 2. Design response spectrum developed by site-adjusted parameters S_{DS} and S_{D1}

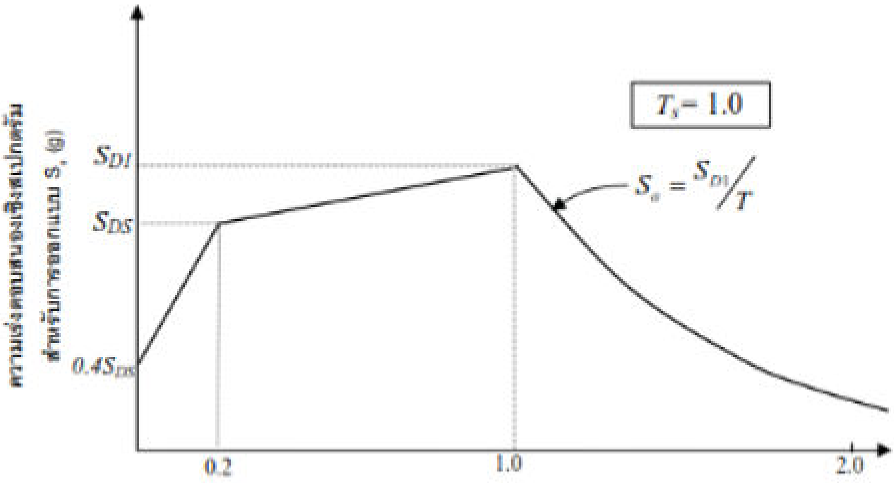
Specifically, the last section of curve (Segment 4) extends horizontally from $2.5T_0$ with a value of $0.4 \times S_{DS}$

NOTE The above curve can be obtained by selecting Horizontal 4th Segment on the Site Specific Spectra page of the Seismic Wizard.

ASCE7/UBC Horizontal Design Spectrum - Thailand

When $S_{d1} > S_{ds}$

Thailand code Figure 1-4-3



Specifically, the previously flat section of curve (Segment 2) rises from S_{DS} at T = 0.2 to S_{D1} At T = 1.0

NOTE The above curve can be obtained by selecting Sloped 2nd Segment on the Site Specific Spectra page of the Seismic Wizard.

Seismic loadcases

Once the Seismic Wizard has been run the loadcases required to be applied to the structure can be generated.

Equivalent Lateral Force (ELF) Loadcases

If an Equivalent Lateral Force (ELF) analysis procedure was selected in the Wizard, the load is applied to the structure at each level as determined for the relevant direction - loadcases being created as follows:

For structures with no torsion

- Seismic Dir1 - loadcase type Seismic
- Seismic Dir2 - loadcase type Seismic

For all other structures:

- Seismic Dir1 - loadcase type Seismic
- Seismic Dir2 - loadcase type Seismic
- Seismic Torsion - loadcase type Seismic

NOTE + and - factors are applied in the combinations to achieve the possible options - +Dir, -Dir, +torsion and -torsion.

Response Spectrum Analysis (RSA) Loadcases

If a response spectrum analysis procedure was selected in the Wizard, the loadcases are created as follows:

For structures with no torsion

- Seismic Dir1 - loadcase type Seismic RSA
- Seismic Dir2 - loadcase type Seismic RSA

For all other structures:

- Seismic Dir1 - loadcase type Seismic RSA
- Seismic Dir2 - loadcase type Seismic RSA
- Seismic Torsion - loadcase type Seismic RSA

NOTE Enveloping is used in the combinations to achieve the possible options - +Dir, -Dir, +torsion and -torsion.

---

6 Analyze models

With Tekla Structural Designer you can perform an extensive range of different analyses.

- Get started with analysis (page 623)
- To run a specific analysis for selected loadcases/combinations, or to run all the analyses necessary for design, see Run analyses (page 664)
- To review the results once the analyses have completed, see Display analysis results (page 671)
- To review the solver model used for a particular analysis type, see View and manage solver models (page 724)

See also

Static analysis and design handbook (page 1159)

Seismic analysis and design handbook (page 1186)

6.1 Get started with analysis

This section explains some basic concepts and procedures you need to know to get started with analysis in Tekla Structural Designer.

Click the links below to find out more:

- Analysis types in
- Adjust and apply analysis settings (page 633)
- What is a solver model (page 633)
- FE meshing, sub models and diaphragms (page 636)

Analysis types in Tekla Structural Designer

The following analysis types can be run in Tekla Structural Designer.

1st order linear

1st order linear static analysis is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: No

Related task: Run 1st order linear analysis (page 664)

1st order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are negligible and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: Yes

Related task: Run a 1st order non-linear analysis (page 665)

1st order modal

This is an unstressed modal analysis which can be used to determine the structure's natural frequencies.

The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly.

Nonlinearity Included:

- Geometric: No
- Material: No

Related task: Run a 1st order modal analysis (page 665)

2nd order linear

This is a 2-stage P-Delta analysis which is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

Related task: Run a 2nd order linear analysis (page 666)

2nd order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: Yes

Related task: Run a 2nd order non-linear analysis (page 666)

2nd order buckling

This is a linear buckling analysis which can be used to determine a structure's susceptibility to buckling.

The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

Related task: Run a 2nd order buckling analysis (page 666)

FE chasedown

This analysis type cannot be run in isolation, it is only performed when it is required as part of another process, such as **Analyze All (Static)**, or **Design All**.

Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.

Grillage chasedown

This analysis type cannot be run in isolation, it is only performed when it is required as part of another process, such as **Analyze All (Static)**, or **Design All**.

Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.

Analyze All (Static)

This is a full series of analyses that would be carried out as part of **Design All (Static)** but with no design. All the analyses required to enable a design to be performed are included:

- 3D analysis
    - 1st order linear/non-linear
    - 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)
- FE chasedown analysis (if required)
- Grillage chasedown analysis (if required)

Related task: Run Analyze All (Static) (page 669)

3D only (Static)

This is a similar to **Analyze All (Static)** but excludes chasedowns to save time during scheme design, (for example while addressing overall stability, sway, drift, wind drift, etc.)

Analyses performed:

- 3D analysis
    - 1st order linear/non-linear

- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)

Related task: Run 3D only (Static) (page 669)

1st order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis

Related task: Run a 1st order RSA seismic analysis (page 667)

2nd order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis

Related task: Run a 2nd order RSA seismic analysis (page 667)

Analysis limitations and assumptions

Certain specific limitations and assumptions relating to the various analysis types are expanded upon below:

Linear analysis of structures containing material nonlinearity

If a structure containing nonlinear springs or nonlinear elements is subjected to a linear (i.e. 1st or 2nd order linear, 1st order modal, or 2nd order buckling) analysis, then the nonlinear springs/elements are constrained to act linearly as described below:

Nonlinear spring supports

In each direction in which a nonlinear spring has been specified, a single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of the spring are ignored.

Tension only, or compression only elements

If either tension only or compression only element types have been specified, they are constrained to act as truss element types instead.

Nonlinear axial spring, or nonlinear torsional spring elements

If either of these element types have been specified, they are constrained to act as linear axial spring, or linear torsional spring element types instead.

A single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of these spring elements are ignored.

Tension only X braces

It is essential that the "X Brace" pattern is used to input cross braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

To determine which brace in each pair is inactivated the program pushes the structure simultaneously in the positive direction 1 and positive direction 2. The brace that goes into tension retains its full stiffness, while the compression brace becomes inactive.

If the above process fails to determine which of the pair goes into tension and which is inactivated then a shear is applied to the structure and the braces are re-assessed.

Analysis of structures containing geometric nonlinearity

It is assumed that where secondary effects are significant (for example the structure is close to buckling), the engineer will elect to undertake a 2nd order analysis. If a 1st order analysis is performed any secondary effects will be ignored.

Analysis of structures containing curved beams

The member analysis for curved members in the plane of the curve is approximated by joining the values at the nodes, which are correct. For detailed analysis of curved members it is your responsibility to ensure sufficient discretization. More refined models can be achieved, if required, by decreasing the maximum facet error.

Analysis of compound (plated) steel beams and columns

Compound (plated) sections are 2 chords or more connected by battens, lattice or welded.

- Static calculations for these section types are the same as for solid sections. The section characteristics are calculated in the basis of the actual section.
- The torsional constant of a compound section is calculated as a total of torsional constants of the chords in a compound section.
- Plane section remain plane.
- The material is homogeneous, isotropic and linearly elastic.
- Saint Venant's principle applies.

Story shears

The story shears that are output are obtained by resolving the loads at column nodes horizontally into Direction 1 and Direction 2. Any loads associated with V & A braces are not included because these occur at mid-beam position and not at column nodes.

Member Deflections

There is a known issue when calculating member deflection profiles in combinations which can affect the following analysis types:

- 2nd Order Linear
- 1st Order Nonlinear
- 2nd Order Nonlinear

This occurs when the structures behavior is significantly nonlinear because the deflection profile is currently based on linear superposition of the load cases within it. Clearly as structural response becomes more nonlinear the assumption that deflections can be superposed becomes less valid. This can cause a deflected profile to be calculated which deviates from the correct profile. The deviation can become significant if load cases fail to solve, but the combination succeeds in solving, as components of the deflected shape are missing entirely. It is suggested that for the three analysis types listed member deflections in combinations be used with caution and engineering judgment.

It should be noted that this limitation only affects member deflection profiles between solver nodes. All other results, including member force profiles and deflection at the solver nodes are correct.

Torsion load analysis - relative angle of twist

Any section that is subject to torsional moment will rotate through an angle, θ. If the cross-section is non-circular this will also be accompanied by warping.

To be able to determine stresses on members subject to torsional moments it is necessary to determine θ (and for "Open" sections its derivatives also).

For single span pinned steel beams only: a torsion load analysis is performed which enables θ (and its derivatives) to be calculated and made available in the Load Analysis View.

For open sections: a more accurate approach is used to determine θ and its derivatives. The 'cases' in DG9, SCI P057 & SCI P385 being used which depend on the end conditions and loading conditions on the beam.

This more accurate analysis is carried out for the following open sections:

i. I Symmetric

ii. I Asymmetric

iii. I Plated (including Westok Plated)

iv. Channel

v. Westok cellular beyond scope

For all other sections not mentioned above (including compound sections): a "Standard" analysis is carried out to determine θ only using the following equation:

$1 / GI_T * \int T(x)$

Where:

I_T = torsion constant

G = shear modulus of steel

T(x) = function of torsion moment

- 2nd Order Linear
- 1st Order Nonlinear
- 2nd Order Nonlinear

Modal analysis - active mass

Modal analysis - active mass

In a 1st order modal analysis mass is assigned to nodes of the analysis model. In simple terms (neglecting rotation terms for the consistent mass matrix) half of each element mass is assigned to each node it is attached to.

Mass that is assigned to a translational support cannot go anywhere - i.e. it is not "active".

Summed Active Mass

Reported in the Dynamic Masses table, this is the actual total active mass for each direction, but expressed in terms of force units rather than mass.

Summed Total Translational Mass

Reported in the **Dynamic Masses** table, this is the total system mass for each direction, again expressed in terms of force units rather than mass.

Translation %

Reported in the **Summed Mass** table, this is the proportion of mass that is active for each direction. For a building this will usually be close to but not quite 100% as some mass always goes to the supports.

Translation % = (Summed Active Mass / Summed Total Translational Mass) x 100

Participation Translation %

Reported in the **Summed Mass** table, this is the sum of Mass Participation (reported in the Modal frequencies table) for all modes for each direction. Design codes stipulate this should be >= 90% for seismic analysis usually for two orthogonal lateral directions.

Solver Model Data

Vibration Frequencies

Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Partic. Trans. Dir 1 [%]	Mass Partic. Trans. Dir 2 [%]	Mass Partic. Trans. Z [%]	Modal Mass Trans. Dir 1 [kip]	Modal Mass Trans. Dir 2 [kip]	Modal Mass Trans. Z [kip]
1	0.2	6.1	0.00	75.26	0.00	0.00	1.6	1.6	1.6
2	0.0	21.8	0.00	0.00	77.08	0.00	1.6	1.6	1.6
3	0.0	31.1	0.00	24.74	0.00	0.00	3.2	3.2	3.2
4	0.0	92.1	0.00	0.00	22.92	0.00	3.3	3.3	3.3
5	0.0	201.7	0.00	0.00	0.00	97.14	2.6	2.6	2.6
6	0.0	487.0	0.00	0.00	0.00	2.86	2.6	2.6	2.6
				Σ=100	Σ=100	Σ=100			

Modal analysis - modal mass

After running a 1st order modal analysis, modal masses for each mode are available in the Modal Frequencies tabular display.

Modal Mass

In Tekla Structural Designer the modal mass, M_i is given by the following matrix equation:

$$M_i = \{\psi\}_i^T [M] \{\psi\}_i$$

Where $\{\Psi\}_i$ is the unity-scaled mode shape (often termed mode vector) of the i^{th} mode (i.e. any single mode) and [M] is the mass matrix. The meaning of the unity-scaled mode shape is that the (numerically) largest modal displacement is set to unity and all other displacements are scaled accordingly. The term $\{\Psi\}_i$ is used to differentiate this mode shape from the mass-normalized shape $\{\Phi\}_i$ which is the mode shape actually reported by Tekla Structural Designer .

This equation comes from modal analysis theory. It may also be termed "generalized mass".

Another way to state this equation, which is found in some design guides, is a summation equation for point masses and their associated modal displacements for a system of discretized mass distribution:

We have from CCIP-016:

$$\hat{m}_j = \sum_{i=1}^{N} \mu^2_{j,i}\, m_i$$

where i is each on N points on the structure, having mass m_i at which the mode shape $\mu_{j,i}$ is the j^{th} mode is known.

According to this reference - "Conceptually, the modal mass can be thought of as the mass of an equivalent single degree of freedom system... which represents the j^{th} mode."

It can be seen how this is equivalent to the matrix equation given above. Actually Tekla Structural Designer makes use of a shortcut calculation since it already has mode shapes which are normalized to mass.

The mass-normalized mode shape $\{\Phi\}_i$ and the unity-normalized mode shape $\{\Psi\}_i$ are related as follows:

$$\{\Phi\}_i = \frac{1}{\sqrt{M_i}} \{\psi\}_i$$

From this we can state the following, where Φ_2 is the largest modal displacement from the mass-normalized mode shape (which we already have from the Tekla Structural Designer modal analysis):

$$\Phi_2 = \frac{1}{\sqrt{M_i}} \times 1$$

Hence:

$$\text{Modal mass} = M_i = \frac{1}{\Phi_2^2}$$

Unstable Structures

Flat Slab Structures

If a concrete structure exists with only flat slabs and columns (i.e. no beams and no shear walls), and the slab is modelled with a diaphragm this is an unstable structure, assuming that the concrete columns are pinned at the foundation level (current default).

To prevent the instability you should mesh the slabs, as the resulting model does then consider the framing action that results from the interaction of the slabs and columns.

Adjust and apply analysis settings

To adjust the analysis settings of the current project or set them as default settings for future projects, see the following instructions.

Adjust analysis settings in the current project

1. On the **Analyze** toolbar, click **Settings**

   The **Model Settings** dialog box opens.
2. Review and modify the analysis settings according to your needs.
3. Do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes to the selected settings set, click **Save...**
   - To revert to the analysis settings specified in the selected settings set, click **Load...**

Adjust analysis settings in future projects

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Analysis Settings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. Review and modify the analysis settings according to your needs.
5. To save the settings as defaults for future projects that use the selected settings set, click **OK**.

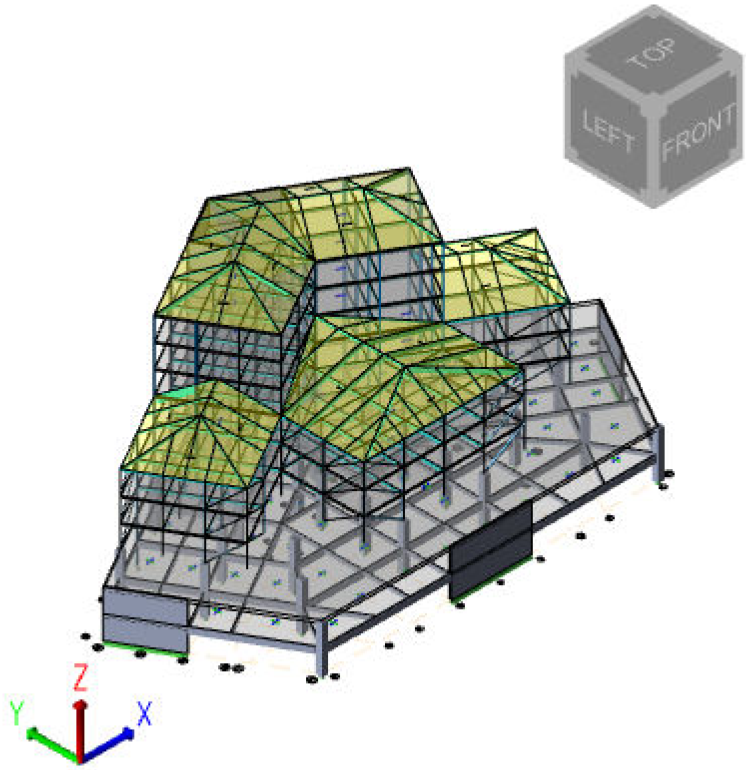
See also

Analysis Settings (page 2278)

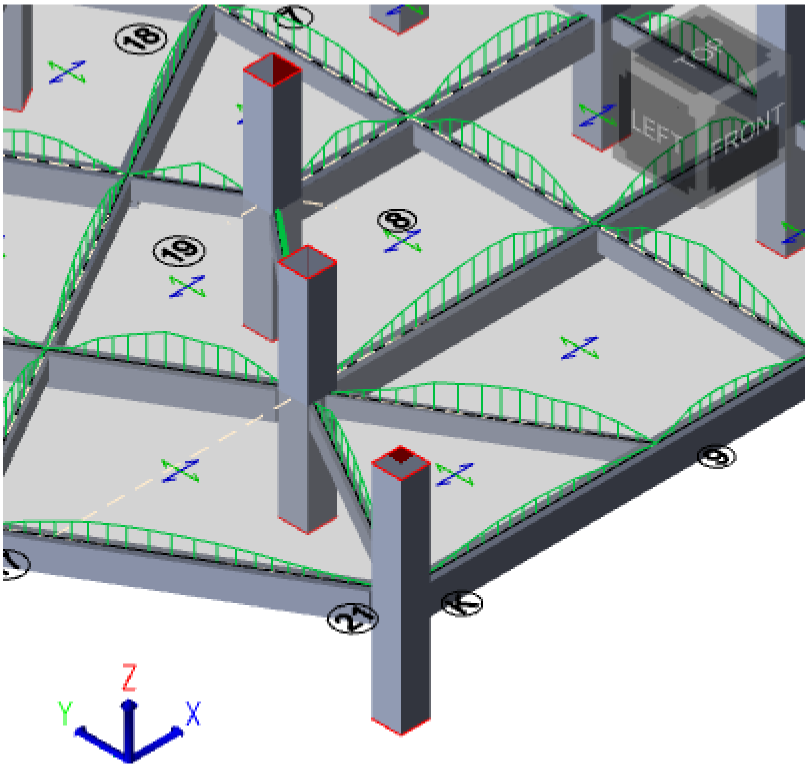
What is a solver model

When you use Tekla Structural Designer to model, analyze, and design structures, you will become familiar with the following concepts:

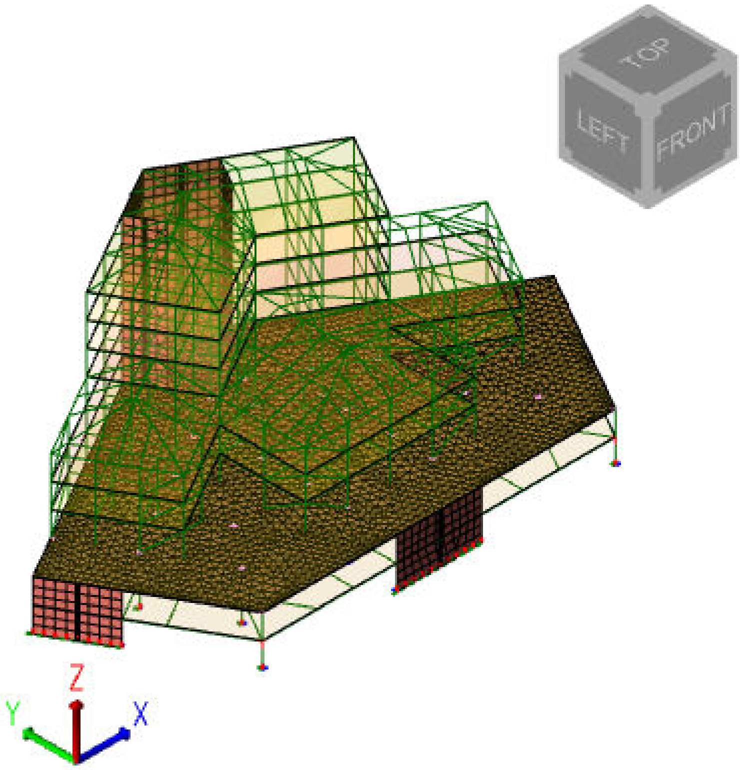
A *structural model* is a 3D model that includes the physical members/objects you create using Tekla Structural Designer, and information related to them. Each member/object in the physical model will exist in the completed structure.



The structural model also contains information about the loadcases and combinations that act on the physical members/objects.



A *solver model* is a 3D model that is created from the structural model. It is used for analyzing structural behavior, and for design.



When you create a solver model, Tekla Structural Designer generates the following analysis objects and includes them in the solver model:

- Solver nodes
- Support conditions for nodes
- 1D solver elements
- 2D solver elements

FE meshing, sub models and diaphragms

Click the links below to find out more about FE meshing, sub models and diaphragms.

To	Click the link below:
Choose whether to mesh slabs for 3D analysis; specify slab mesh	Manage FE meshed slabs (page 637)
Specify wall mesh	Manage FE meshed walls (page 653)
Manage the sub models used in chasedown analyses	Manage sub models (page 661)

To	Click the link below:
Activate and manage rigid and/or semi-rigid diaphragms	Diaphragm action in roof panels and slabs (page 655)

Manage FE meshed slabs

At the levels where two-way spanning slabs exist, Tekla Structural Designer applies FE meshing as follows:

- Two-way spanning slabs are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.
- Two-way slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. This ensures the quickest solution time. A 3D pre-analysis process (in which the slabs are meshed) is used to decompose loads from the slabs on to supporting members.
- Alternatively, you can choose to mesh two-way slabs for 3D building analysis and grillage chasedown analysis, either at all, or selected construction levels. In this case Tekla Structural Designer does not perform the load decomposition during 3D pre-analysis for the two-way slabs at those levels.

In all of the above mentioned situations, the slab mesh density is set globally, but can be overridden at individual levels if required.

Click the links below to find out more:

- Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis (page 637)
- Adjust global slab mesh properties (page 638)
- Apply different mesh properties at different levels (page 639)
- Review the slab mesh before the analysis (page 639)
- How slab properties and features impact on meshing (page 640)
- Slab meshing controls (page 644)

Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis

To ensure the quickest solution time, two-way slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. In this case, the diaphragm option (none, semi-rigid, or rigid) in the slab properties determines their axial rigidity and a 3D pre-analysis process is used to FE decompose loads from the slabs on to supporting members.

If you prefer, you can instead choose to mesh the slabs, either at all, or selected construction levels.

NOTE Two-way spanning slabs are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.

Use meshed two-way slabs in at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select **Levels**.
3. In the **Properties** window, select the **Mesh 2-way Slabs in 3D Analysis** option.

Use FE decomposed slab loads at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select **Levels**.
3. In the **Properties** window, clear the **Mesh 2-way Slabs in 3D Analysis** option.

Use meshed two-way slabs at selected construction levels

1. In the **Structure** tree, expand the **Levels** branch.
2. Click one of the desired construction levels.
3. According to your needs, either select or clear the **Mesh 2-way Slabs in 3D Analysis** option.
4. Repeat step 3 for all the desired construction levels.

Adjust global slab mesh properties

Initially, the same mesh properties are applied globally to all meshed slabs. In order to adjust the global slab mesh properties, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Slab Mesh Size**.

5. Accept or adjust the **Slab Uniformity Factor**. At 100% uniformity all slabs in the mesh will be of approximately equal area.
6. Accept or adjust the **Slab Mesh Type**:
    - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
    - **QuadOnly**: only consists of quadrilateral 2D elements.
    - **Triangular**: only consists of triangular 2D elements.

See also

Apply different mesh properties at different levels (page 639)

Apply different mesh properties at different levels

If you need to apply different mesh parameters at a specific level, you can do so by creating sub models. For more information, see the following instructions.

1. Create a sub model. (page 662)
2. In the **Structure** tree, expand the **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the mesh size and uniformity factor according to your needs.

Review the slab mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

    The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

    If the slab mesh is applicable to the selected solver model, it is displayed.

NOTE	The slab mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

See also

View and manage solver models (page 724)

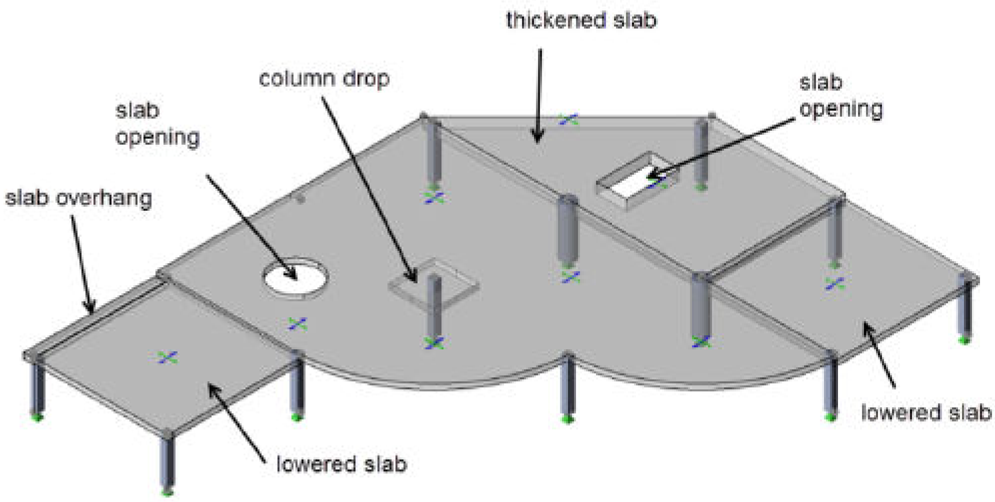
How slab properties and features impact on meshing

The slab mesh is always created in a single plane, but the resulting mesh is can be affected by the slab properties and features as described below.

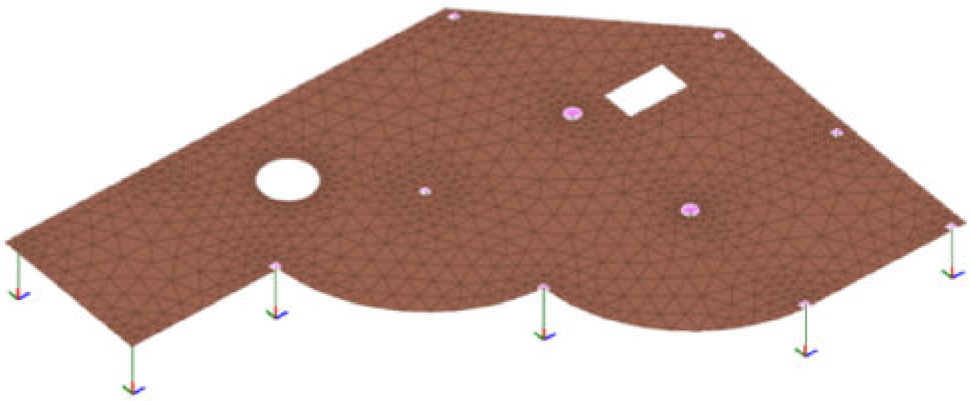
Slab thickness, vertical offsets, column drops and openings

The mesh of shell 2D solver elements is always created in the same plane, irrespective of whether slabs have different thicknesses, slabs items have been raised/lowered via vertical offsets, or column drops have been applied. Shells are not created inside slab openings and any loads placed within openings are not applied to the model.

Consider the example shown below. This features curved slab boundaries, circular and rectangular openings, thickened slab panels, lowered slab panels and a slab overhang. A column drop panel has also been inserted at one of the locations where the slab is supported by a column.



In the resulting FE solver model, since vertical offsets are not structurally significant the analysis mesh is formed at the same level relative to the top of the slab. The mesh properties do however reflect the change to the slab thicknesses in the different slab areas.



NOTE Beam solver elements and slab meshes can only be offset vertically from one another by being defined in different construction levels.

Other slab properties

Rotation Angle

Different slab items in the same slab can have different rotation angles.

This property is used for the following where appropriate:

- Span direction for 1-way load decomposition
- To determine the 2D solver element local axes in the solver model
- Bar direction for Slab on Beam and Flat Slabs.

Include in Diaphragm

This property is only available when the Diaphragm option is Semi-Rigid or Rigid. Individual slab items in the same slab can be included or excluded as required.

The effect of excluding a slab item depends on the Diaphragm option as follows:

- Semi-Rigid - excluded slab items are not meshed with semi-rigid 2D solver elements
- Rigid - internal nodes not considered in the nodal constraints

NOTE Where 2 items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

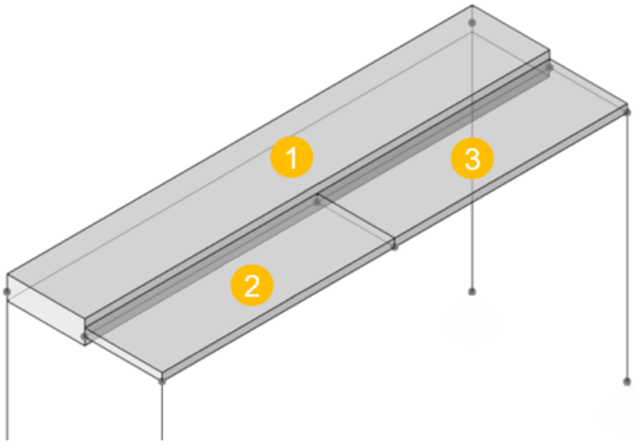
NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Mesh groups

To facilitate meshing, Tekla Structural Designer automatically gathers slab items and features together into mesh groups, and meshes them as a single entity. A mesh group contains one or more slab items with identical analysis attributes. Since slab depth is a key analysis attribute, by definition a slab step, or a column drop, will produce an additional mesh group.

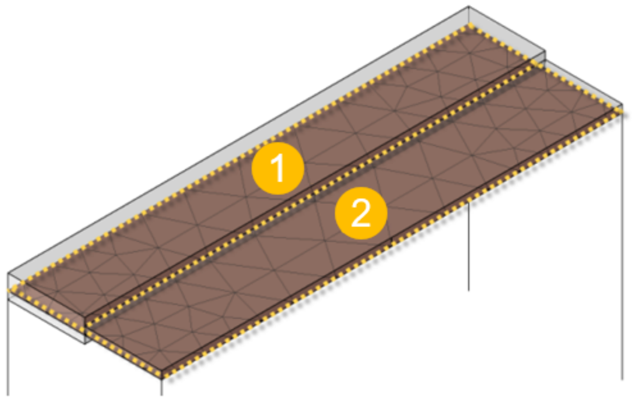
Example: Mesh groups at a slab step

In the following image, you can see three separate slab items:



1. d = 300 mm
2. d = 100 mm
3. d = 100 mm

Although there are three slab items in the previous image, there are only two different slab depths. That is why Tekla Structural Designer only creates two mesh groups, as shown in the following image:



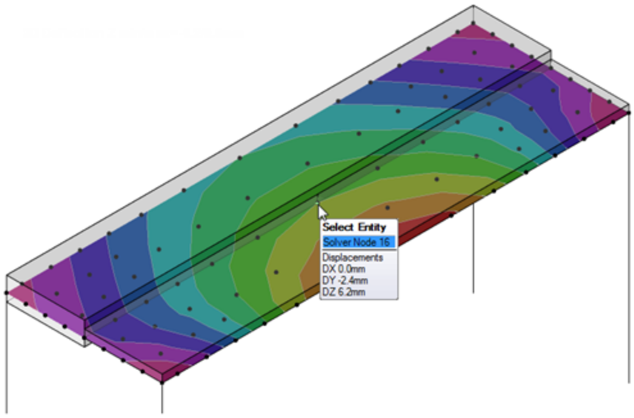
1. mesh group 1
2. mesh group 2

Discontinuity of force contours at slab steps and column drops

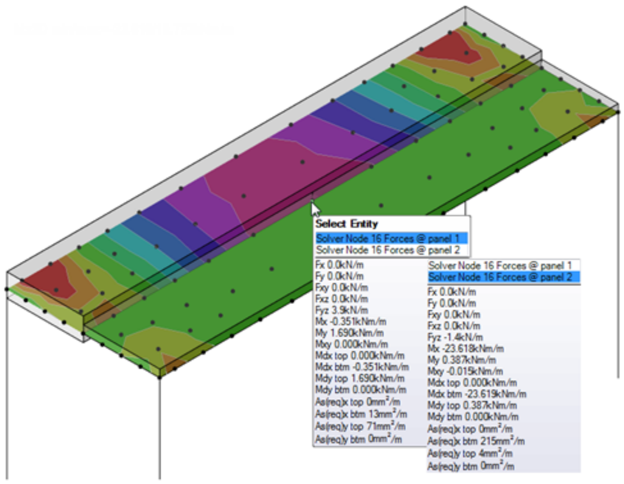
When slab items on either side of a slab step are placed into different mesh groups, the solver nodes along the boundary are shared by both groups. Each node on the boundary reports a single value of deflection, but two values of force, one for each group. That is why there will be a discontinuity of force contours along the boundary.

See the following examples:

Example: Deflection contours (no discontinuity)



Example: Moment contours, discontinuity along boundary



The force discontinuity is a genuine result. The slab items share the same curvature at the step and have the same elastic modulus, so the moment must be directly proportional to the inertia of each slab item.

NOTE Other programs may average the value across the boundary when generating the contours. However, we prefer the approach of Tekla Structural Designer because averaging would result in an unrealistically high design of the thinner slab.

Mesh group boundary warnings

Meshing may fail or produce undesirable results when there is challenging mesh group boundary geometry. In this case, Tekla Structural Designer warns you about the source points source of meshing issues.

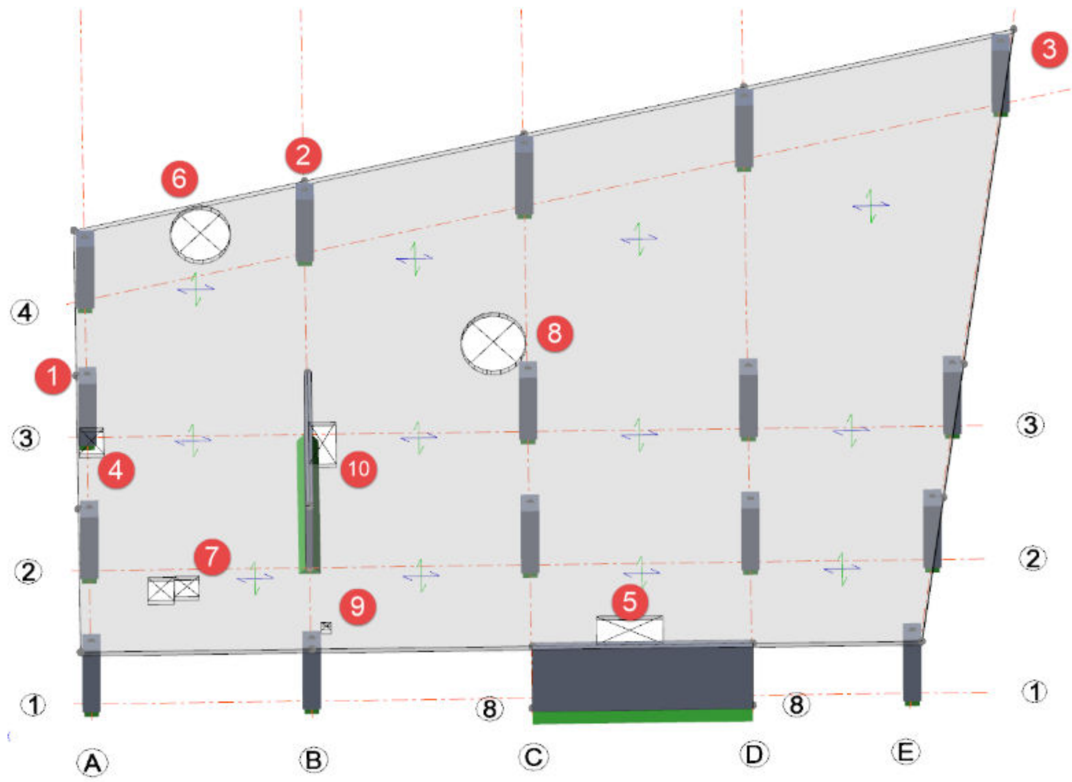
Examples of possible warning triggers are:

- short edges
- distance between a hole and an edge
- small area enclosed by a mesh group

Slab meshing controls

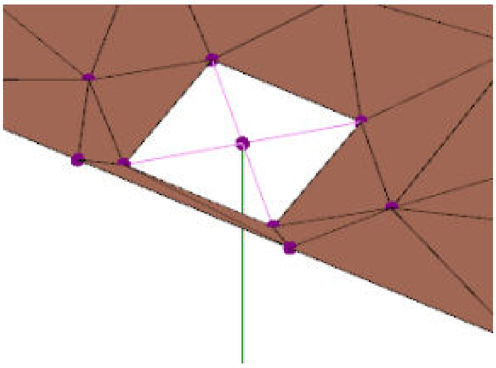
Meshing issues tend to be caused when the area being meshed ends up with really narrow strips of slab, for example when the slab edge ends up just outside the edge of a column.

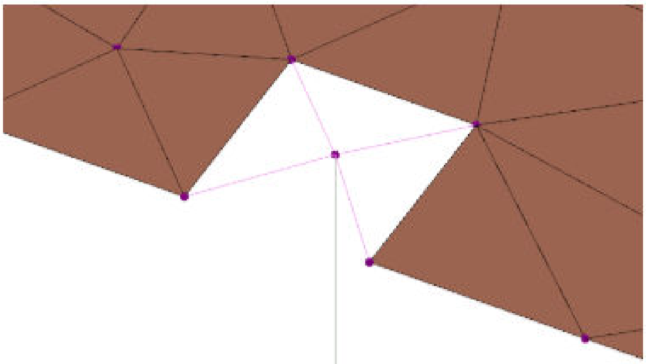
On the **Meshing** page of Analysis options (page 2278) certain meshing tolerance controls can be adjusted when required to solve these type of issue.



The model shown above has various challenging situations which are discussed below along with an indication of whether changing the automatic merging distances in Analysis Options can have an impact:

1. Column face close to parallel slab edge - (yes)

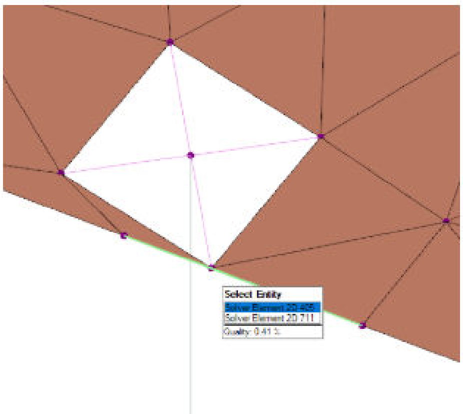


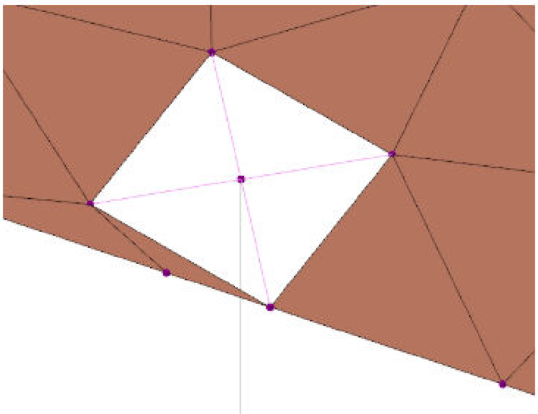


In this example the slab edge is positioned 50mm outside the column face. The view above left shows the analysis model you would get when the Column boundary to slab edge merging distance is < 50mm. The view above right shows the model you get if the merging distance is set to 50mm or above. The column corner nodes merge (move to) the slab edge eliminating the narrow strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

2. Column face close to inclined slab edge - (yes)

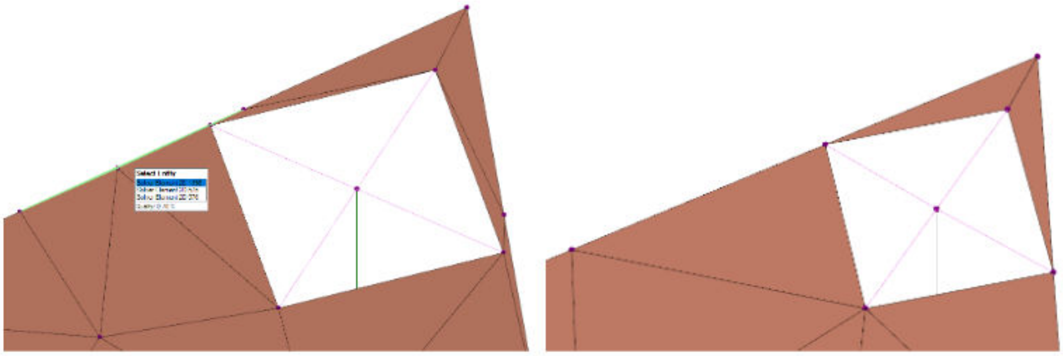




In this example the slab edge sits just a few mm outside the column corner. The view above left shows analysis model where a very poor quality element runs past the corner, (you would only end up with this model if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get if the merging distance is set to the default 5mm or above. One column corner node merges (moves to) the slab edge eliminating the narrowest part of strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

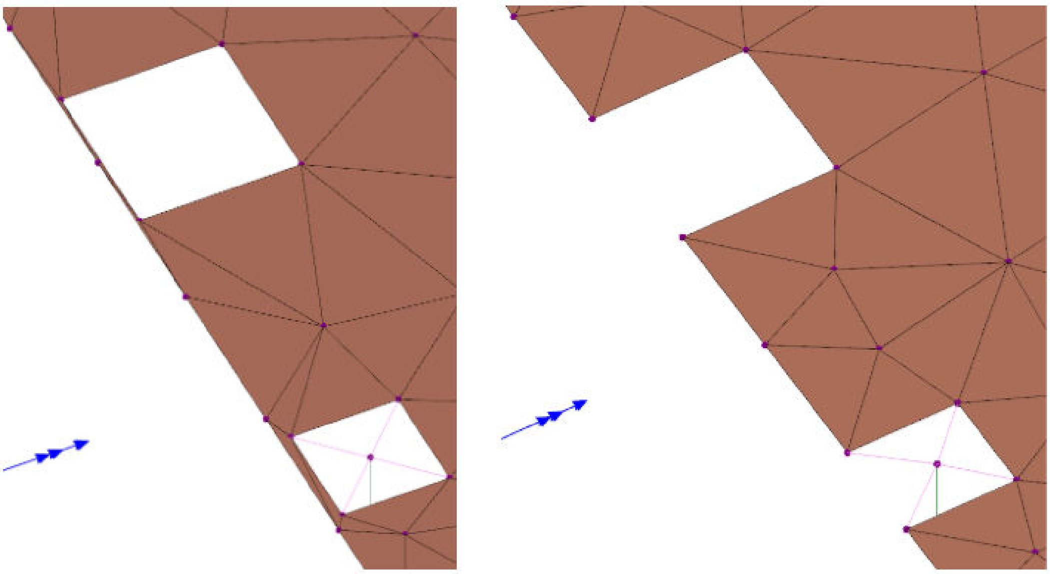
3. Column close to multiple non-parallel slab edges - (yes)



This is just another variation on positions 1 and 2. The view above left shows analysis model you would end up with if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get provided the default merging distance (5mm) is retained - meshing is simplified avoiding mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

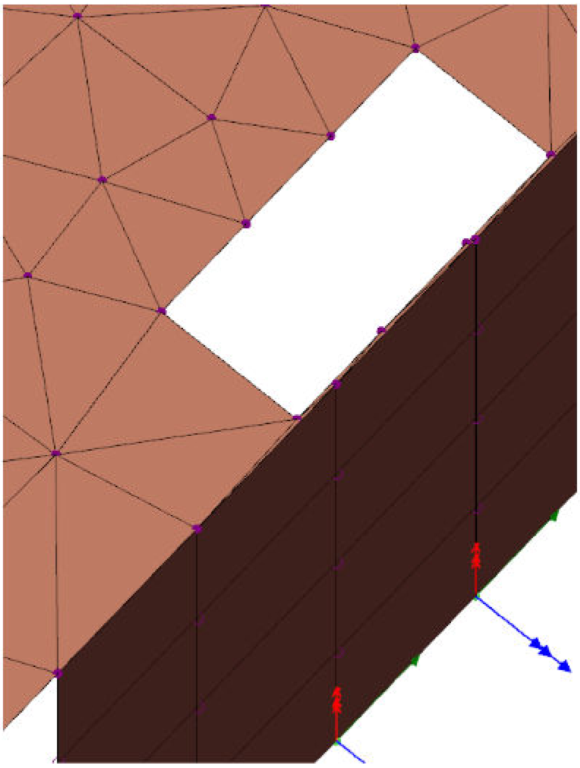
4. Opening near slab edge - (yes)

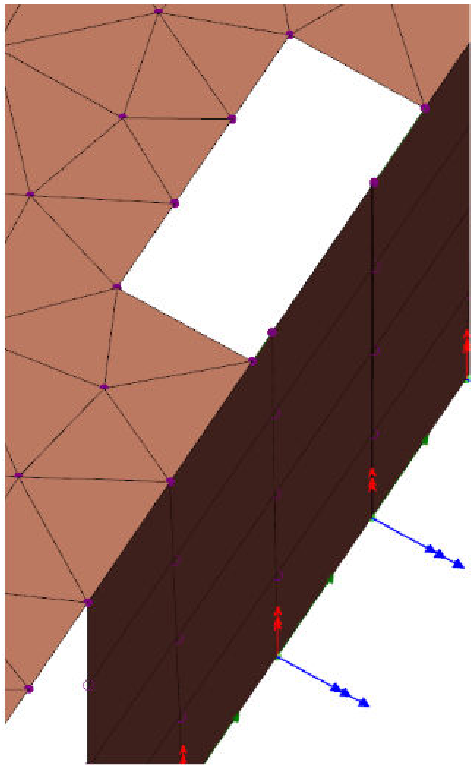
In the following examples merging will make things better, but the reality is that a slab will never be cast with thin isolated strips at the edge - you cannot effectively reinforce a 50 to 100mm wide strip of slab. In all these the ideal solution is that the modelling should reflect the reality of realistic minimum edge distances.



A hole is defined approximately 20mm from the slab edge. The effect is the same as at the column cut-out boundary - above on the right shows how with the Opening to slab edge merge distance set to 50mm the opening corners move to the slab edge. This creates a better analysis model.

5. Opening near wall inserted along slab edge - (yes)

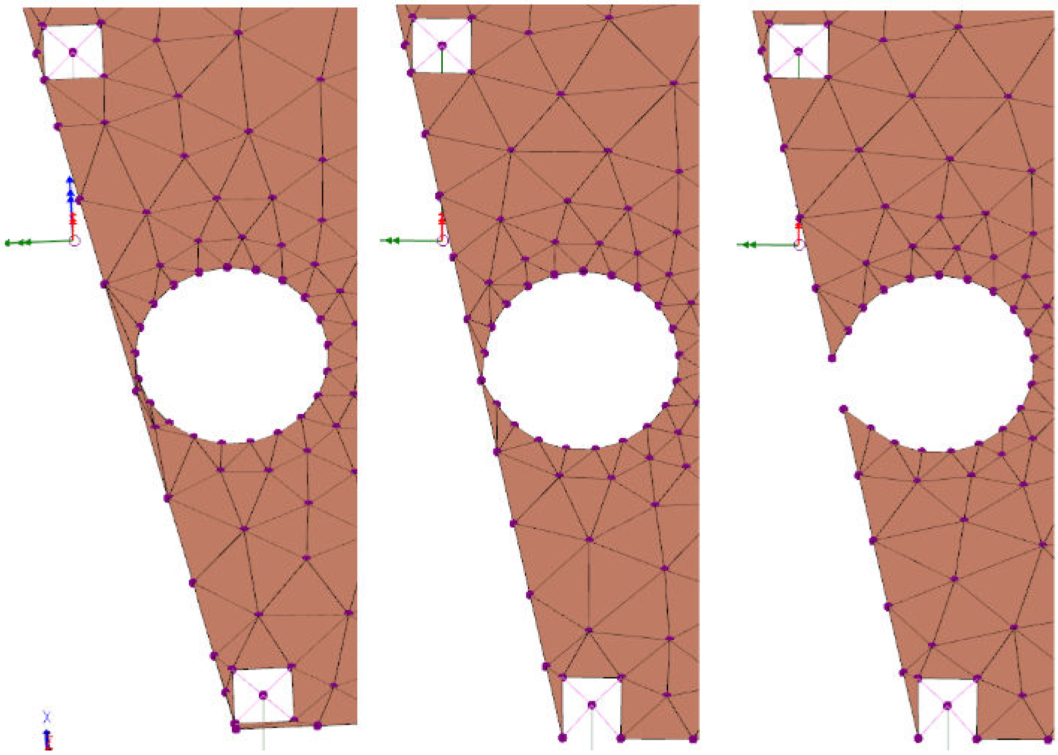




This is basically the same as the previous situation except that a wall happens to be defined along the edge. In such cases the user is often tempted to extend the opening into the physical width of the wall but does not extend it right to the wall insertion line.

There are pros and cons to both models. On the left the narrow meshed strip of slab can give mesh quality warnings and errors. On the right the slab mesh is better, but the model relies a bit more heavily on the wall beam to transfer moments. However, the overall minor axis moment transferred to the wall only drops from 75.7 to 72.5 kNm.

6. Circular opening near slab edge - (yes)

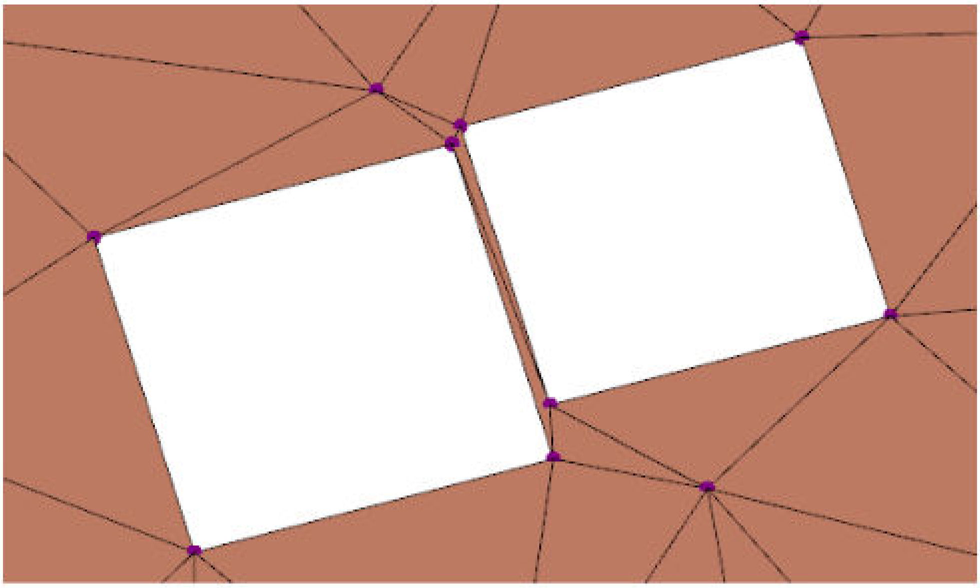


Above left shows the sort of meshing that will occur when a circular opening is created near to the slab edge. Above centre shows an example where closest point on the circle is merged to edge. The meshing looks a bit better but this is still not a great model because you will get strange local results at the connecting node. On the right where theOpening to slab edge merge distance is set to 100mm the edge gets completely broken - this is probably the best model.

7. Closely spaced openings - (no)

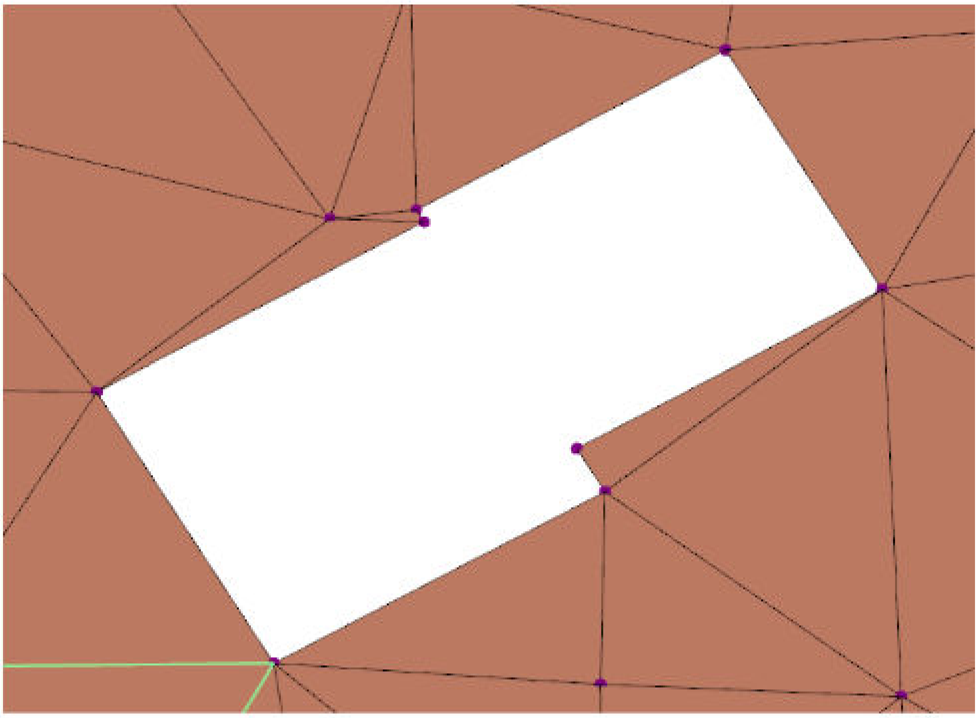
These are all examples where the merging settings have no effect.

In each case the problems can generally be resolved by adjusting the openings so that they overlap slightly.

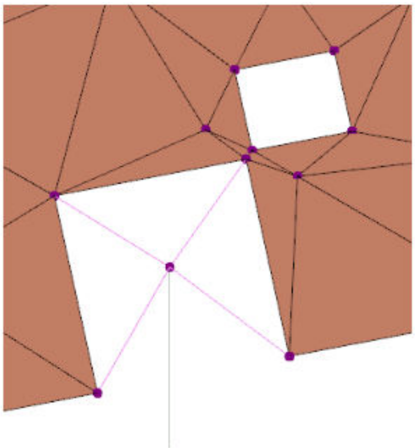


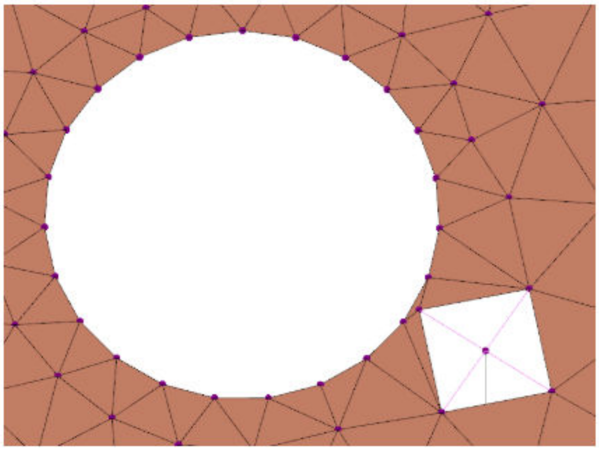
Above - shows meshing when a narrow (unbuildable?) strip of slab is left between openings.

Below shows the model when the openings are manually adjusted to eliminate this strip.

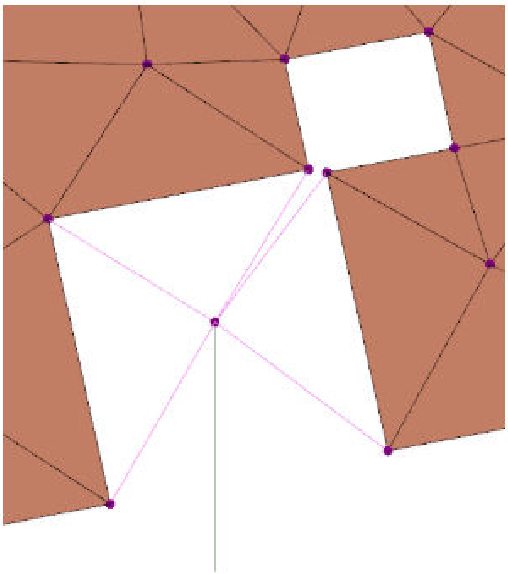


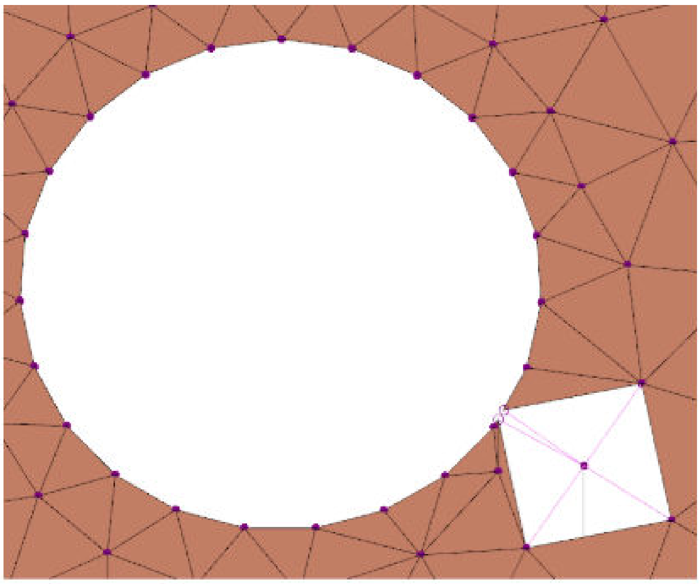
8 and 9. Opening near column - (no)



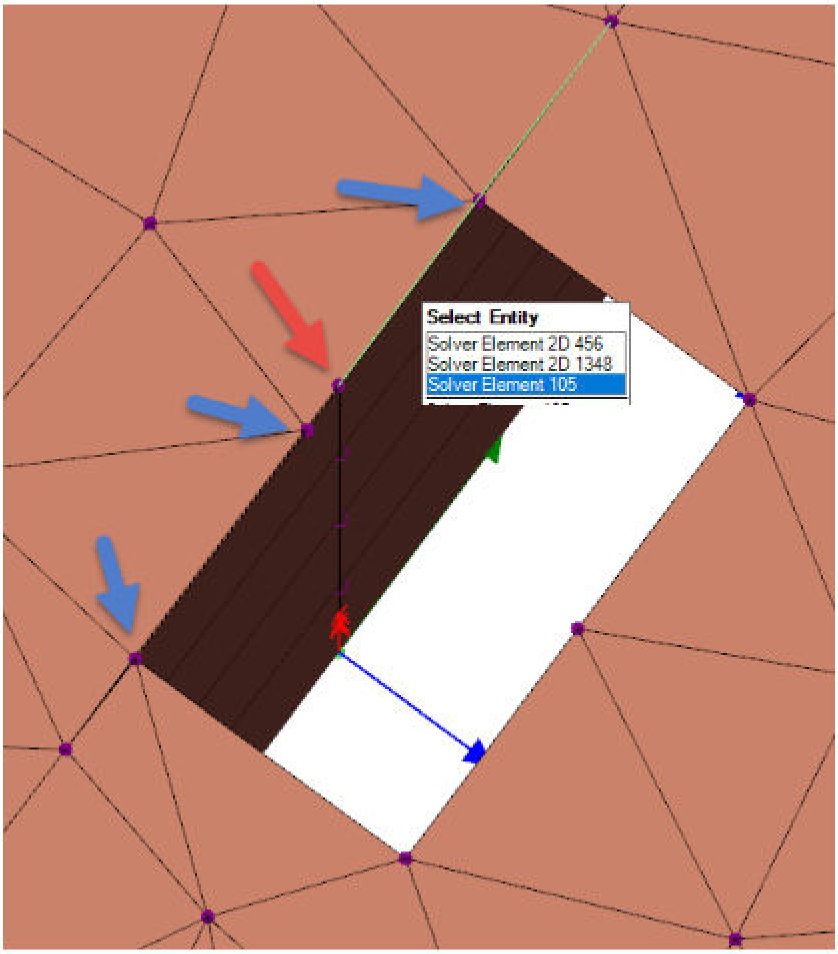


In both these cases it would again be better to adjust the openings so that they overlap the column boundaries as suggested below.



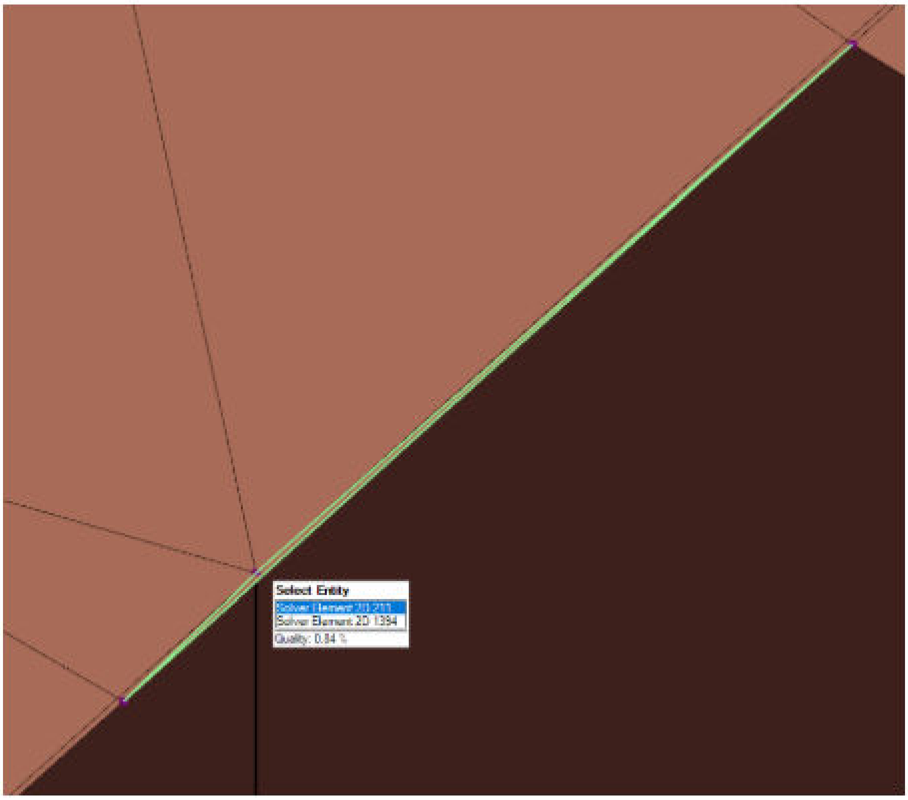


10. Opening near wall with slab on other side - (no, unless certain conditions apply)

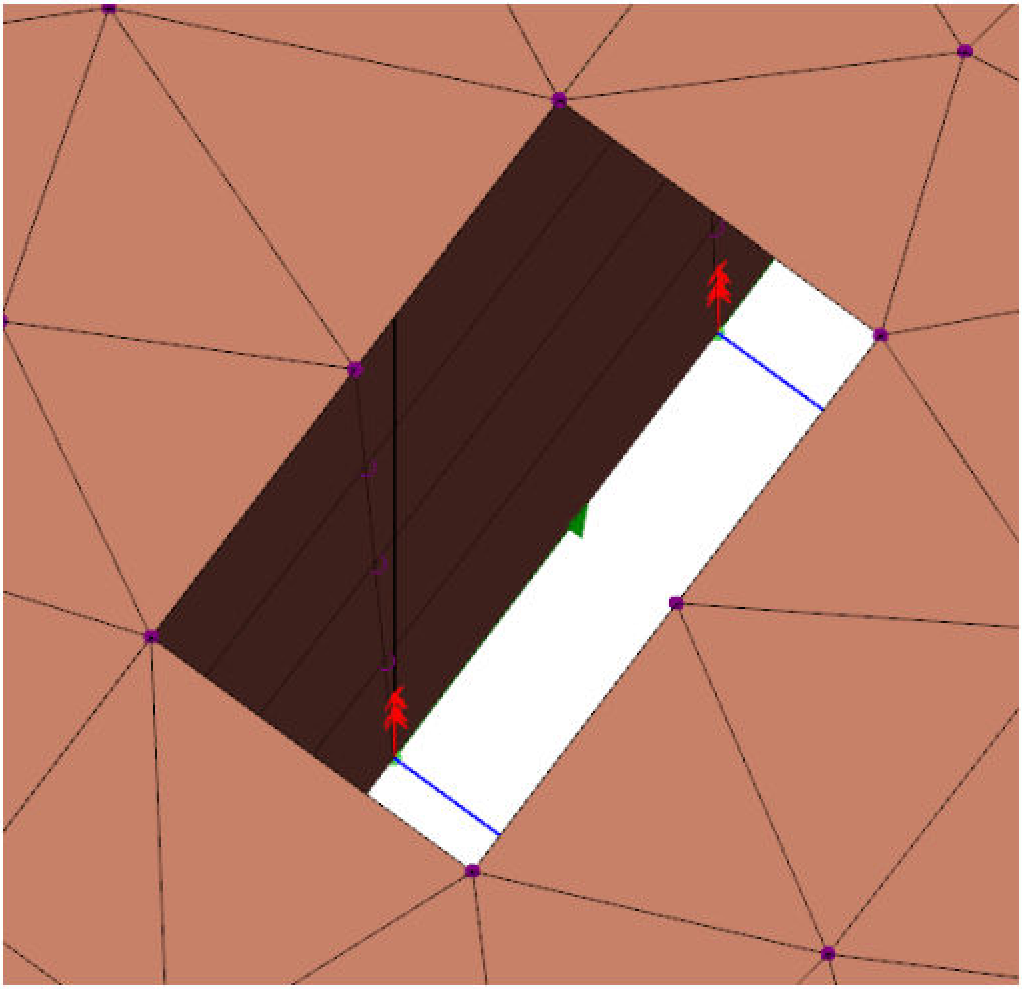


In this case the opening is defined with an edge that is approximately 4mm from the insertion line of the wall. With the Hardpoint to edge distance tolerance at the default of 5mm this creates a model that is poorly connected along the edge of the opening. The node highlighted in red is not included in the slab meshing (because it is just 4mm away from the edge). The 3 slab nodes highlighted in blue are not directly connected to the wall.

If the hard point distance is reduced to 1mm then the model changes as shown below, the node marked in red above is now included in the slab mesh so the adjacent nodes have a better connection to the wall.



However, the best model would be obtained if the opening edge got exactly aligned with the wall, this can actually be quite hard to achieve in the manual positioning of openings. A workaround at present is to force a “slab mesh boundary” along the line of the wall by making the slab on one side slightly different to the slab on the other - then the automatic mergings will create a model as shown below.



Manage FE meshed walls

By default, concrete walls automatically adopt an FE meshed model when the structure is analyzed. Meshed walls automatically use the mesh parameters of the model. However, if necessary, you can override the mesh parameters and apply a user defined mesh to an individual wall.

NOTE If necessary, you can modify concrete walls so that they use a mid-pier model instead of an FE meshed model.

Adjust global wall mesh properties

Initially, the same mesh properties are applied globally to all meshed walls. In order to adjust the global wall mesh properties, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.

4. If necessary, in **Wall Mesh Type**, change the shape of the wall mesh:
    - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
    - **QuadOnly**: only consists of quadrilateral 2D elements.
    - **Triangular**: only consists of triangular 2D elements.
5. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

See also

Apply different mesh properties to an individual wall (page 654)

Apply different mesh properties to individual walls

If you need to apply different wall mesh properties to specific walls, you can override the model mesh properties, and modify the properties of the individual wall. For more information, see the following instructions.

1. In the model, select the walls that you want to modify.
2. Go to the **Properties** window.
3. If necessary, in **Wall Mesh Type**, change the shape of the wall mesh:
    - **QuadDominant**
    - **QuadOnly**
    - **Triangular**
4. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

Review the wall mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

    The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

    If the wall mesh is applicable to the selected solver model, it is displayed.

    NOTE The wall mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

Diaphragm action in roof panels and slabs

Roof panels and slabs both have the potential to act as a diaphragms.

Both rigid and semi-rigid diaphragms can be modelled.

- Overview of diaphragm action in roof panels and slabs (page 655)
- Managing diaphragm action in roof panels and slabs (page 658)

Overview of diaphragm action in roof panels and slabs

Roof panels and slabs will both act as a diaphragms provided they have **Include in diaphragm** checked in their properties.

While **Include in diaphragm** is the only property used to determine diaphragm action in roof panels, for slab items the diaphragm properties are also determined by the **solver model** under consideration and the following properties:

- **Diaphragm option** slab item property
- **Decomposition** slab item property
- **Mesh 2-way Slabs in 3D analysis** level or slope property

How these and other choices affect the diaphragm model are described in the topics below.

Include in Diaphragm

Roof panels:

The **Include in diaphragm** choice works as follows:

- On - a diaphragm is created within the roof panel using a mesh of semi-rigid 2D solver elements
- Off - no diaphragm is created

Slab items:

For slab items the **Include in diaphragm** choice is only available when the **Diaphragm option** is Semi-Rigid or Rigid. Individual slab items can then be included or excluded as follows:

- On - a diaphragm is created within the slab item which may take the form of a shell mesh, a semi-rigid mesh, or nodal constraints
- Off - no diaphragm is created

NOTE Where 2 slab items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Diaphragm option

The choice of **Diaphragm option** is set in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

The available options are:

- Rigid
- Semi-rigid
- None

How your choice affects the 2D solver element types used depends on the method of decomposition selected.

NOTE For roof panels there is no choice of diaphragm option - it is always treated as semi-rigid.

Decomposition

While roof panels are always one-way spanning, for slab panels you can choose the **Decomposition** method in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

Two options are available: Two-Way Spanning or One-Way Spanning - although for certain slab types the value is fixed as follows:

- Composite Slab - One-Way only
- Precast Slab - One-Way only
- Slab on Beams - Two-Way or One-Way
- Flat Slab - Two-Way only
- Steel Deck - One-Way or Two-Way
- Timber Deck - One-Way only

One-way spanning slabs are unmeshed unless the **Diaphragm option** is set to Semi-rigid, in which case they will be meshed with semi-rigid 2D solver elements.

NOTE Semi-rigid 1-way spanning slabs cannot be designed.

Two-way spanning slabs always adopt a mesh of **shell** 2D solver elements for the FE chasedown and FE Load Decomposition solver models. However for the 3D Analysis and Grillage chasedown solver models they are:

- **unmeshed** - if the Diaphragm option is set to None,
- meshed with **semi-rigid** 2D solver elements - if the Diaphragm option is set to Semi-rigid, or,
- meshed with **shell** 2D solver elements - if the Diaphragm option is set to Rigid.

NOTE Semi-rigid 2-way spanning slabs can be designed, but only using the results from the FE Chasedown, not the 3D Analysis or Grillage Chasedown.

Mesh 2-way Slabs in 3D Analysis

Two-way spanning slabs are always meshed with **shell** 2D solver elements in FE Chasedown and Load decomposition solver models. When the **Mesh 2-way Slabs in 3D Analysis** property (specified in the Level, or Slope properties) is checked the same meshing parameters are then extended to the Grillage chasedown and 3D Analysis solver models.

Summary of diaphragm constraint and mesh type configurations

The configurations of mesh and nodal constraints applied to each solver model resulting from the different permutations of the **Decomposition**, **Diaphragm option**, and **Mesh 2-way slabs in 3D analysis** properties are recapped in the table below.

Decomposition	Diaphragm Option	Mesh 2-way slabs in 3D Analysis	FE Load Decomposition & FE Chasedown Models	Grillage Chasedown & 3D Analysis Models
1-way	None	Not Applicable	No mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Not Applicable	Semi-Rigid mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Not Applicable	No mesh; Nodal constraints	No mesh; Nodal constraints
2-way	None	Yes	Shell Mesh; no nodal constraints	Shell Mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Yes	Shell mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints

	Rigid	Yes	Shell Mesh; Nodal constraints	Shell Mesh; Nodal constraints
		No	Shell Mesh; Nodal constraints	No Mesh; Nodal constraints

Managing diaphragm action in roof panels and slabs

Tekla Structural Designer forms diaphragms in every slab item of a parent slab when **Diaphragm option** is set to **Rigid** or **Semi-rigid**.

When a rigid diaphragm is specified, Tekla Structural Designer automatically applies nodal constraints to the associated slab nodes.

In turn, when a semi-rigid diaphragm is specified, Tekla Structural Designer creates a mesh of semi-rigid 2D elements within the slab.

Activate rigid diaphragm option within a slab

By default, Tekla Structural Designer automatically creates rigid diaphragm within slabs. If necessary, you can review and modify the setting for specific slabs as follows:

1. Go to the **Project Workspace**.
2. In the **Structure** tree, expand the **Slabs** branch.
3. Select the slab that contains the diaphragm.
4. In the **Properties** window, set **Diaphragm option** as required.

NOTE You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a slab

You can form semi-rigid diaphragms in both one-way and two-way spanning slabs. In addition, you can review and modify the diaphragm properties by selecting the slab within which they are located.

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the **Slabs** branch.
3. Select the desired slab.
4. In the **Properties** window, set **Diaphragm option** to **Semi-rigid**.
5. If necessary, to adjust the flexibility of the diaphragm, type a value in the **Divide stiffness by** option.

TIP You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a roof panel

You can form semi-rigid diaphragms in roof panels as follows:

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the **Roofs** branch.
3. Select the desired roof panel.
4. In the **Properties** window, select **Include in diaphragm**.

NOTE You cannot form a *rigid* diaphragm within a roof panel.

Adjust semi-rigid diaphragm flexibility

When the Diaphragm option is Semi-Rigid a **Divide stiffness by** value can be set in the slab item properties. The same value is applied to **all** slab items in the same slab.

The stiffness determined from the material properties and slab thickness is divided by this value in order to adjust semi-rigid diaphragm flexibility.

Adjust global semi-rigid mesh properties

When a semi-rigid mesh is created the 2D solver element shape (triangular or quad), the degree of mesh uniformity, and mesh type adopted are obtained from the **Structure Properties**. These parameters can be overridden for individual sub-models by setting different values in the respective **Sub Model Properties**.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Semi-Rigid Mesh Size**.
5. Accept or adjust the **Semi-Rigid Uniformity Factor**.
6. Accept or adjust the **Semi-Rigid Mesh Type**:
   - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.
   - **Triangular**: only consists of triangular 2D elements.

NOTE Beam elements are not split by semi-rigid 2D element nodes.

Apply different semi-rigid mesh properties at different levels

If you need to apply different semi-rigid mesh parameters at a specific level, you can do so by creating sub models.

1. Create a sub model. (page 662)
2. In the **Structure** tree, expand the **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the semi-rigid mesh size, uniformity factor and type according to your needs.

Identify the nodes constrained by rigid diaphragms

In order to see which nodes are constrained by diaphragms, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

   A solver view opens. The rigid and semi-rigid diaphragms are represented as different colored shaded planes.

   Nodes that are constrained by a rigid diaphragm:

   - Must lie within, or be on the edge of, the shaded rigid diaphragm plane.
   - Must be solid rather than hollow, or excluded.

   Therefore, the following nodes are not constrained by a rigid diaphragm:

   - Solid nodes that lie outside the shaded rigid diaphragm plane.
   - Hollow, or excluded, nodes.

Exclude individual nodes from a rigid diaphragm

If necessary, you can exclude specific nodes from a rigid diaphragm. In order to do so, see the following instructions.

1. In the status bar at the bottom of the window, click **Review View**. A review view opens.
2. On the **Review** tab, click **Diaphragm On/Off**.

The nodes are displayed as follows:

- Included nodes are only constrained if they lie within or on the edge of a rigid diaphragm. If they lie outside the rigid diaphragm, they are not constrained by it.
- Excluded nodes are always unconstrained.

3. Click a node to switch whether it is included in the diaphragm or not.

Exclude slab items from a diaphragm
By default, Tekla Structural Designer forms a rigid diaphragm in all the individual slab items within a slab. However, you can decide to exclude specific slab items. In order to do so, see the following instructions.

1. In the structure view, select the slab item that you want to exclude.
2. In the **Properties** window, clear the **Include in diaphragm** option.

TIP You can also exclude individual slab items graphically in the Review View. On the **Review** tab, click **Diaphragm On/Off** and then click the slab item to include/exclude.

Manage sub models
Tekla Structural Designer initially treats each structure as a single sub model. In practice, this means that the Tekla Structural Designer applies the same mesh parameters to all meshed slabs. However, if needed, you can create sub-models in your model by creating horizontal planes between levels. Each individual sub model controls the slab mesh parameters at the levels within it.

Tekla Structural Designer creates additional sub models automatically for every level specified as a **Floor** in the **Construction Levels** dialog box when Tekla Structural Designer performs one of the following:

- Grillage chasedown analysis
- FE chasedown analysis

You can also define sub models manually in the **Sub Models** dialog box.

Tekla Structural Designer uses the he slab mesh parameters specified for a sub model in any analysis that requires the slabs to be meshed, such as load decomposition, building analysis with meshed floors, or FE chasedown analysis.

In both grillage and FE chasedown, Tekla Structural Designer performs the analyses one sub model at a time. Tekla Structural Designer first analyzes the topmost sub model, and then applies its support reactions as loads for the analysis of the sub model below. The sequence continues until Tekla Structural Designer has analyzed all sub models down to the foundation level.

For both grillage and FE chasedown analysis, you can modify the default support conditions applied to the sub-models, if necessary.

Definitions of sub model characteristics

- **Sub model**: a part of the 3D model between two horizontal sub model divide planes.

  Each sub model contains all members entirely between the two horizontal planes. For the columns, wall and braces split by a divide plane, the stacks and brace length above the top plane are included in the sub model, as are the stacks and brace length below the lower plane.
- **Sub model divide planes**: horizontal planes that you can add, delete or move in the 3D structure. Sub model divide planes are notional and infinite. Tekla Structural Designer only allows the planes to cut through the structure where they only split the following members:
  - Column stacks
  - Wall stacks
  - Steel braces
- **Sub model supports**: the artificial supports that Tekla Structural Designer defines for the column and wall stack ends and braces that pass through the divide planes
- **Structure supports**: the supports that the user has defined in the 3D structure
- **Column and wall stacks**: the span length of a column or wall.
- **Volume of the sub model**: the 3D space that exists between any two adjacent sub model divide planes.

Basic rules of sub models

When all sub models are considered together, they form the complete structure. Only column stacks, wall stacks, and braces that are split by sub model divide planes can be in several sub models.

This means that:

- Each member in the 3D model is in at least one sub model.
- A sub model cannot contain a member that is already in another sub model, unless that member is a column, wall, or brace divided by a sub model divide plane.
- A sub model must contain at least one beam member, one truss member, or one slab item.

Create sub models

Sub models allow you to apply different slab mesh parameters within your structure. To create sub models, see the following instructions.

1. In the **Structure** tree, double-click **Sub Models**.

   The **Sub Models** dialog box opens.

2. According to your needs, do one of the following:

To	Do this
Automatically create sub models	• Click **Generate**.
Manually create sub models	• According to your needs, click **Insert Above**, or **Insert Below**.

3. If necessary, modify the height above the base of each level in the **Level** field.
4. Click **OK**.

See also

Sub Models dialog (page 2438)

Sub Model Properties (page 2072)

Open a 3D view of a sub model

To display a sub model in its own 3D view, see the following instructions.

1. In the **Project Workspace**, open the **Structure** tree.
2. Expand the **Sub Models** branch.
3. Double-click the sub model.

   Tekla Structural Designer opens a 3D view of the selected sub model.

   TIP To open a solver view of the sub model, right-click the sub model, and select **Open solver view**.

Delete sub models

If necessary, you can delete existing sub models in the **Sub Models** dialog box. For detailed information on deleting sub models, see the following instructions.

1. In the **Structure** tree, double-click **Sub Models**.

   The **Sub Models** dialog box opens.

2. Select the sub model that you want to delete.
3. Click **Delete**.

   Tekla Structural Designer deletes the sub model, and the floors are transferred to the sub model immediately above the deleted one.

6.2 Run analyses

Various analyses can be run from the **Analyze** toolbar.

Click the links below to find out how to:

- Run a 1st order linear or non-linear analysis (page 664)
- Run a 1st order vibration analysis (page 665)
- Run a 2nd order linear or non-linear analysis (page 666)
- Run a 2nd order buckling analysis (page 666)
- Run a seismic analysis (page 667)
- Run FE chasedown or grillage chasedown analysis (page 668)
- Run Analyze All (Static) (page 669)
- Run 3D only (Static) (page 669)

Once the analysis has completed, the Project Workspace can be used to verify the results in a couple of ways:

- Check sum of reactions against load input (page 670)
- Check stability and overall displacement (page 671)

Run a 1st order linear or non-linear analysis

To run either a linear or non-linear 1st order analysis on your model, see the following instructions.

See also

The Results View (page 671)

View tabular results for support reactions (page 765)

View tabular results for nodal deflections (page 765)

View tabular results for solver element end forces (page 766)

Run 1st order linear analysis

1. On the **Analyze** tab, click **1st Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and load cases that you want to analyze.

3. Click **OK**.

Tekla Structural Designer analyzes the model.

At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 1st order non-linear analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Non-Linear** page.
3. If necessary, adjust the convergence criteria and relaxation factors.
4. Click **OK**.
5. On the **Analyze** tab, click **1st Order Non-linear**.

   The **Select loading** dialog box opens.
6. In the **Select loading** dialog box, select the combinations and load cases that you want to analyze.
7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 1st order modal analysis

In order to run a modal analysis on your model, see the following instructions.

NOTE To run a modal analysis, your model must contain an active modal mass combination.

For more information, see Create modal mass combinations (page 521).

---

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the page (page 2281).
3. If necessary, adjust the analysis options.
4. On the **Analyze** tab, click **1st Order Modal**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** opens to allow you to display the mode shapes graphically. Tabular results can also be viewed in a **Solver Model Data View**.

Run a 2nd order linear or non-linear analysis

In order to run either a linear or a non-linera 2nd order analysis on your model, see the following instructions.

Run a 2nd order linear analysis

1. On the **Analyze** tab, click **2nd Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and load cases that you want to analyze.

3. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order non-linear analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Non-linear** page.

3. If necessary, adjust the convergence criteria and relaxation factors.

4. Click **OK**.

5. On the **Analyze** tab, click **2nd Order Non-linear**.

   The **Select loading** dialog box opens.

6. In the **Select loading** dialog box, select the combinations and load cases that you want to analyze.

7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order buckling analysis

In order to run a buckling analysis on your model, see the following instructions.

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Buckling** page.
3. If necessary, adjust the buckling options.
4. Click **OK**.
5. On the **Analyze** tab, click **2nd Order Buckling**.

   The **Select loading** dialog box opens.
6. In the **Select loading** dialog box, select the combinations and load cases that you want to analyze.
7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

See also

The Results View (page 671)

View buckling factors (page 770)

Run a seismic analysis

In order to run either a 1st or 2nd order seismic analysis on your model, see the following instructions.

Run a 1st order RSA seismic analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Seismic** page.
3. If necessary, adjust the analysis options.
4. Click **OK**.
5. On the **Analyze** tab, click **1st Order RSA Seismic**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order RSA seismic analysis

- On the **Analyze** tab, click **2nd Order RSA Seismic**.

Tekla Structural Designer analyzes the model.

At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

See also

RSA seismic results (page 691)

The Results View (page 671)

View tabular results for support reactions (page 765)

View tabular results for nodal deflections (page 765)

View tabular results for solver element end forces (page 766)

Run FE chasedown or grillage chasedown analysis

You can run FE chasedown and grillage chasedown analyses by using the **Analyze All (Static)** command. The analyses are also run when they are required as a part of the combined analysis and design process.

RESTRICTION FE chasedown and grillage chasedown analyses are run for load cases only, and not for load combinations.

In a combined analysis and design process, Tekla Structural Designer performs the previously mentioned analyses as follows:

- If the model contains two-way slabs or the user has selected the appropriate option in the **Design Settings** dialog box, Tekla Structural Designer performs FE chasedown analysis.

  TIP To control whether Tekla Structural Designer performs FE chasedown analysis, do the following:

  1. On the **Design** tab, click **Settings**.
  2. Under concrete beams, columns, or walls, go to **General Parameters**.
  3. Select whether you want to design the members for FE chasewdown analysis results.
  4. Click **OK**.

- If the model contains one or more concrete members, Tekla Structural Designer performs the grillage chasedown analysis.

See also

Analyze all combinations and load cases (page 669)

Run Analyze All (Static)

Static designs can only be performed provided suitable analysis results exist - while these results are created when you run a combined analysis and member design, in some situations you may prefer to run the analysis separately and then selectively design parts of the model as required.

Analyze All (Static) facilitates this as it performs all the analyses for static loadcases and active static load combinations that are required to enable the selective designs to be carried out.

To run a separate analysis in this way proceed as follows:

1. On the **Analyze** toolbar, click **Analyze All (Static)**.

Tekla Structural Designer performs the following analyses:

- 1st order linear/non-linear
- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**))
- Grillage chasedown (if one or more concrete members exist)
- FE chasedown (if two-way spanning slabs exist)

At the end of the analyses if a 2D or 3D view is active it is switched to a **Review View**.

See also

The Results View (page 671)

View tabular results for support reactions (page 765)

View tabular results for nodal deflections (page 765)

View tabular results for solver element end forces (page 766)

Run 3D only (Static)

3D only (Static) analysis can be used to save time during scheme design, for example while addressing overall stability, sway, drift, wind drift, etc.

To run a 3D only analysis proceed as follows:

1. On the **Analyze** toolbar, click **3D only (Static)**.

Tekla Structural Designer performs the following 3D analyses:

- 1st order linear/non-linear

- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)

At the end of the analysis if a 2D or 3D view is active it is switched to a **Review View**.

See also

The Results View (page 671)

View tabular solver model data and results (page 764)

Review tabular data (page 898)

Check sum of reactions against load input

Once you have performed an analysis in your model, you can use the **Loading** tree to quickly check that the total reaction from each analysis equates to the total load on structure. This way, you can quickly verify that none of the applied loads is missing.

1. In the **Project Workspace**, go to the **Loading** tab.
2. Review the status of each load case and combination.

   The status options are the following:

   - : total reaction equates to the total load on structure
   - : total reaction does not equate to the total load on structure
   - : total reaction is not available

Example cross-checks

The **General** section of the properties for each loadcase provides summations (in global X, Y & Z) of the different load types applied to the structure, from these the total applied load is determined. The total reaction from the 3D Analysis result is also reported in this section. Assuming a 3D analysis, FE chasedown analysis and grillage chasedown analysis have all been performed:

- The **Total User Applied Load** should equate to the **Total Load on Structure**.
- The **Total Load on Structure** should equate to the **Total Reaction** for 3D analysis reported in the **General** section.
- The **Total Load on Structure** should equate to the **Total Reaction** for FE chasedown analysis reported in the **FE ChaseDown** section.
- For each sub-model in the **FE ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**
- The **Total Load on Structure** should equate to the **Total Reaction** for Grillage chasedown analysis reported in the **Grillage ChaseDown** section.

- For each sub-model in the **Grillage ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**

Check stability and overall displacement

You can use the **Status** tree in the **Project Workspace** to review the stability checks and overall displacement of the model.

Review the stability checks and overall displacement in the Status tree

1. In the Project Workspace click the **Status** tab
2. In the**Status** tree , expand the **Design** heading.
3. Expand the required check or overall displacement to review as required.

NOTE Further details of the checks performed are available by reviewing the tabular design data.

See also:

Filter tabular data (page 929)

Stability and imperfections handbook (page 1133)

6.3 Display analysis results

Once analysis has been run, you can switch 2D and 3D scene views to the **Results View** regime to display graphical analysis results. You can also display analysis results for individual members and walls in a **Load Analysis View**

Click the links below to find out more:

- The Results View (page 671)
- The Load Analysis View (page 715)

Tabular results are also available, either by creating reports, or by viewing the tabular solver model data.

See also

View tabular solver model data (page 764)

The Results View

To change to a **Results View** click  in the **Status bar** at the bottom of the main window.

After setting the analysis type and loading (page 673), most of the results can then be displayed simply by selecting from the appropriate toolbar group:

Reactions

- Display reactions (page 673)

1D Results

- Display 1D results (page 675)
- Display 1D deflections (page 675)
- Animate 1D and 2D deflections (page 675)

Sway Drift...

- Display sway drift and story shear (page 676)

Notional Loads

- Display notional forces and seismic equivalent lateral forces (page 677)

2D Results

- Display 2D results (page 677)
- Display 2D deflections (page 683)
- Animate 1D and 2D deflections (page 675)
- Display AsReq contours (page 683)

2D Integrated Results

- Display wall lines (page 684)
- Display core lines (page 684)
- Manage, display and design result lines (page 688)
- Manage and display result strips (page 685)
- **NOTE** Result lines and result strips must be created before they can be displayed.

Mode Shapes

- Display mode shapes (page 691)

RSA Results

- RSA seismic results (page 691)

If required, adjustments can be applied to the diagrams as follows:

- To change contour intervals and colors, see Customize the display of 2D contours (page 696)
- To adjust the amplitude of the diagrams, see Change result diagram scale settings (page 697)
- In 2D views, it is sometimes necessary to switch to an isometric projection, see Display 2D view in isometric projection (page 697)

Set the analysis type and loading for viewing analysis results

When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

1. On the **Results** toolbar, in the **Result Type** group, select the required analysis method.

   NOTE If you select: 1st order modal, 2nd order buckling, 1st order RSA seismic or 2nd order RSA seismic analyses, you should also choose the **Mode** to be displayed.

   ---

2. In the **Result Type** group, if required, click **Reduce Axial Force** if you want to take into account reductions when viewing axial forces in a loadcase or combination in which reductions have been applied.
3. In the **Result Type** group, click **Strength Factors**, or **Service Factors**, as required, to control which factors are used when displaying results for combinations.
4. In the **Loading** list, click either the **Select Loadcase**, **Select Combination**, or **Select Envelope** button.
5. In the **Loading** list, select the desired load case, combination, or envelope.

You can now proceed to select the diagram to be displayed.

See also

The Results View (page 671)

Display reactions

Support reactions

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).

3. In the **Reactions** group, select the reactions to be displayed:

1D	Reactions at supports under 1D elements (e.g. column support reactions),
2D	Reactions at supported nodes of 2D elements (individual meshed wall supports and mat foundation supports),
	Distributed wall reactions (Fz) - available for self-supported meshed, mid-pier, bearing and shear-only wall panels with horizontal bottom levels. When active the nodal support reactions exclusive to the wall are hidden. Summary of assumptions, • Wall panel has enough height to ensure that all loading that contributes to the reaction is far enough from the support such that the principle of St. Venant is considered valid, • Material is perfectly homogeneous and isotropic, • Elastic limit is nowhere exceeded and 'E' is same in tension and compression.
	Integrated reactions at core supports. When active support reactions exclusively from 1D and 2D core members are hidden.

4. For **Distributed wall reactions** the droplist beneath should be set to **Total**, for other support reactions you can select the desired reaction from the droplist.

Fx	support local x axis reaction (corresponds to **Fminor** in the Foundation Reactions Report),
Fy	support local y axis reaction ((corresponds to **Fmajor** in the Foundation Reactions Report),
Fz	support local z axis reaction (corresponds to **Fvert** in the Foundation Reactions Report),
Mx	support local x axis moment (corresponds to **Mmajor** in the Foundation Reactions Report).
My	support local y axis moment (corresponds to **Mminor** in the Foundation Reactions Report)
Mz	support local z axis moment (corresponds to **Mtor** in the Foundation Reactions Report)
Fxyz	support local reactions in **Fx** and **Fy** and **Fz**
Mxyz	support local reactions in **Mx** and **My** and **Mz**
Total	all support local reactions (**Fx**, **Fy**, **Fz**, **Mx**, **My** and **Mz**.

5. From the **Text** group select **Reaction** to display values on the diagram.

Beam end reactions

1. Go to the **Results** toolbar.

2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **Reactions** group, select **Beam Ends**, then from the droplist beneath select the desired reaction.
4. From the **Text** group select **Reaction** to display values on the diagram.

Display 1D results

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **1D Results** group, select **Results**, then from the droplist underneath it, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

See also

Change result diagram scale settings (page 697)

Display 2D view in isometric projection (page 697)

Display 1D deflections

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **1D Results** group, select **Deflections**, then from the droplist underneath it, select the desired deflection type.
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Change result diagram scale settings (page 697)

Animate 1D and 2D deflections (page 675)

Animate 1D and 2D deflections

Once the required deflections are displayed, they can be animated as follows:

1. Right click in the view and from the context menu select **Animate**.

   The animation commences.
2. If required you can change the number of frames per second, the cycle duration and the amplitude by adjusting the sliders and clicking **Apply** in the **Animation** dialog.
3. To end the animation, simply close the **Animation** dialog.

NOTE Animations are available for static displacements and those for modal and buckling analysis.

Display sway drift and story shear

Once you have selected the analysis type, and the load case, combination, or envelope, you can simply view the results by selecting the desired reaction on the **Results** tab. The commands in the **1D Results** group and the **Deflections** group display the results for 1D elements, such as beams, columns, and trusses, and walls modelled using the mid-pier option. Conversely, the commands in the **2D Results** group and the **2D Deflections** group display the results for 2D elements, such as FE slabs and FE walls. For more information, see the following instructions.

Sway

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **Sway Drift and Storey Shear** group, select **Drift**, then from the droplist beneath select **Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Relative sway

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **Sway Drift and Storey Shear** group, select **Drift**, then from the droplist beneath select **Relative Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Wind drift

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **Sway Drift and Storey Shear** group, select **Drift**, then from the droplist beneath select **Wind Drift Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Shear

1. Go to the **Results** toolbar.

2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **Sway Drift and Storey Shear** group, select **Drift**, then from the droplist beneath select **Dir 1 Shear** or **Dir 2 Shear** as required.

See also

Change result diagram scale settings (page 697)

Display notional forces and seismic equivalent lateral forces

To view the notional forces or seismic equivalent lateral forces that apply to a particular load combination or load case, see the following instructions.

View the magnitude of EHF in a combination

RESTRICTION	The command is only applicable to the Eurocode head code.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **EHF**.

View notional loads in a combination

RESTRICTION	The command is only applicable to the ACI/AISC head code.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **NF**.

View the magnitude of NHF in a combination

RESTRICTION	The command is only applicable to the BS and Australian head codes.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **NHF**.

View the magnitude of equivalent lateral forces in a seismic load case or combination

1. In the **Loading** list, select the required seismic load case or combination.
2. On the **Results** tab, click **Seismic**.

Display 2D results

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).

3. In the **2D Results** group, select **Results**, then from the droplist beneath select the result required.

Mdx top	Wood Armer bending along panel x axis, top surface
Mdx bottom	Wood Armer bending along panel x axis, bottom surface
Mdy top	Wood Armer bending along panel y axis, top surface
Mdy bottom	Wood Armer bending along panel y axis, bottom surface
Bearing pressure	Bearing pressure
Fx	Axial force in panel x axis
Fy	Axial force in panel y axis
Fxy	Complimentary in-plane shear
Fxz	Shear in panel z axis in the panel xz plane
Fyz	Shear in panel z axis in the panel yz plane
Mx	Bending along panel x axis
My	Bending along panel y axis
Mxy	Plate torsional moment
σ_x top	In-plane axial stress in the x direction, top surface
σ_y top	In-plane axial stress in the y direction, top surface
τ_{xy} top	In-plane shear stress, top surface
σ_x bottom	In-plane axial stress in the x direction, bottom surface
σ_y bottom	In-plane axial stress in the y direction, bottom surface
τ_{xy} bottom	In-plane shear stress, bottom surface
σ_x max tension	Maximum tension stress in the x direction for both surfaces = max ($\sigma_x$top, $\sigma_x$bottom, 0) ≥ 0
σ_y max tension	Maximum tension stress in the y direction for both surfaces = max ($\sigma_y$top, $\sigma_y$bottom, 0) ≥ 0
σ_x max compression	Maximum compression stress in the x direction for both surfaces = min ($\sigma_x$top, $\sigma_x$bottom, 0) ≤ 0
σ_y max compression	Maximum compression stress in the y direction for both surfaces = min ($\sigma_y$top, $\sigma_y$bottom, 0) ≤ 0
σ_x in-plane	In-plane center stress in the x direction
σ_y in-plane	In-plane center stress in the y direction
σ_x in-plane tension	In-plane center tension stress in the x direction = max ($\sigma_x$in-plane, 0) ≥ 0
σ_y in-plane tension	In-plane center tension stress in the y direction = max ($\sigma_y$in-plane, 0) ≥ 0

σ_x in-plane compression	In-plane center compression stress in the x direction = min ($\sigma_x$in-plane, 0) ≤ 0
σ_y in-plane compression	In-plane center compression stress in the y direction = min ($\sigma_y$in-plane, 0) ≤ 0

NOTE Tooltips are available for the contour plots, by hovering over nodes. These display all the relevant results at the node for the selected loading.

See also

Stresses in 2D elements (page 679)

Stresses in 2D elements

Which stresses can be displayed?

You can view the stresses on the outer faces of 2D elements for both slabs and walls by selecting the required result from droplist beneath **Results** in the **2D Results** group.

The first 6 values are calculated directly from the forces and moments:

1. σ_x top - in-plane axial stress in the x direction, top surface
2. σ_y top - in-plane axial stress in the y direction, top surface
3. τ_{xy} top - in-plane shear stress in xy direction, top surface
4. σ_x bottom - in-plane axial stress in the x direction, bottom surface
5. σ_y bottom - in-plane axial stress in the y direction, bottom surface
6. τ_{xy} bottom - in-plane shear stress in xy direction, bottom surface

NOTE The above 6 values are available for loadcases & combinations, but not envelopes

The next 4 values are determined from the first 6 values:

1. σ_x max tension - maximum tension stress in the x direction for both surfaces
2. σ_y max tension - maximum tension stress in the y direction for both surfaces
3. σ_x max compression - maximum compression stress in the x direction for both surfaces
4. σ_y max compression - maximum compression stress in the y direction for both surfaces

NOTE The above 4 values are available for loadcases, combination & envelopes.

The center stress values are calculated directly from the forces and moments:

1. σ_x in-plane - in-plane axial stress in the x direction, center
2. σ_y in-plane - in-plane axial stress in the y direction, center

The last 4 values are determined from the above 2 values:

1. σ_x in-plane tension - maximum tension stress in the x direction center
2. σ_y in-plane tension - maximum tension stress in the y direction center
3. σ_x in-plane compression - maximum compression stress in the x direction center
4. σ_y in-plane compression - maximum compression stress in the y direction center

How might these results be used?

Users performing the design of structures with concrete core walls are interested to know which panels within the walls are cracked. Which panels are cracked can be determined by comparing the maximum tensile stress in each panel to the concrete tensile strength.

Tekla Structural Designer calculates stress values from the gross section properties (ignoring the reinforcement). To determine cracked panels, you can see the maximum tension (and compression) stress in each direction for each panel, across loadcases, combinations and envelopes.

Calculation of in-plane axial and shear stress

For loadcases and combinations, by using the 2D element thickness, stresses (based on the gross section properties) can be calculated from the forces at the nodes:

$\sigma_x top = F_x / t + 6M_x / t^2$

$\sigma_y top = F_y / t + 6M_y / t^2$

$\tau_{xy} top = F_{xy} / t + 6M_{xy} / t^2$

$\sigma_x bottom = F_x / t - 6M_x / t^2$

$\sigma_y bottom = F_y / t - 6M_y / t^2$

$\tau_{xy} bottom = F_{xy} / t - 6M_{xy} / t^2$

$\sigma_x in\text{-}plane = F_x / t$

$\sigma_y in\text{-}plane = F_y / t$

NOTE	• Tension stresses are positive • Compression stresses are negative

The process for enveloping the above values is the same as that used for other envelopes. For each of the items, a pair of values is found, these are the minimum & maximum values across all loadcases and combinations.

Calculation of maximum tension and compression stress for loadcases and combinations

For loadcases and combinations, the maximum tension and compression values are determined for a specific direction by finding the maximum or minimum of the top and bottom stresses in that direction:

σ_x max tension = Max ($\sigma_x$top, $\sigma_x$bottom, 0.0)

σ_y max tension = Max ($\sigma_y$top, $\sigma_y$bottom, 0.0)

σ_x max compression = Min ($\sigma_x$top, $\sigma_x$bottom, 0.0)

σ_y max compression = Min ($\sigma_y$top, $\sigma_y$bottom, 0.0)

σ_x in-plane tension = Max ($\sigma_x$in-plane, 0.0)

σ_y in-plane tension = Max ($\sigma_y$in-plane, 0.0)

σ_x in-plane compression = Min ($\sigma_x$in-plane, 0.0)

σ_y in-plane compression = Min ($\sigma_y$in-plane, 0.0)

For envelopes, the maximum tension and compression values are determined by applying the above equations to the enveloped values. Envelopes yield two values for each of the 4 entries in the dropdown.

Calculation of maximum tension and compression stress for envelopes

For envelopes, the maximum tension and compression values are determined by applying the above equations for loadcases and combinations to the enveloped values.

Envelopes yield two values for each of the 4 entries in the droplist.

For **tension stresses** (x or y - only x shown for brevity) the values are returned are:

σ_x max tension = m_1 / m_2 , where:

m_1 = Min ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

m_2 = Max ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

For **compression stresses** (x or y - only x shown for brevity) the values returned are:

σ_x max compression = m_1 / m_2 , where:

m_1 = Min ($\sigma_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0)

m_2 = Max ($_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0)

In summary the values visible in the tooltip are:

σ_x max tension	=	Min ($\sigma_x top_{\text{max across all cases \& combs}}$, $\sigma_x bottom_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_x top_{\text{max across all cases \& combs}}$, $\sigma_x bottom_{\text{max across all cases \& combs}}$, 0.0)
σ_y max tension	=	Min ($\sigma_y top_{\text{max across all cases \& combs}}$, $\sigma_y bottom_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_y top_{\text{max across all cases \& combs}}$, $\sigma_y bottom_{\text{max across all cases \& combs}}$, 0.0)
σ_x max compression	=	Min ($\sigma_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0) / Max ($\sigma_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0)
σ_y max compression	=	Min ($\sigma_y top_{\text{min across all cases \& combs}}$, $\sigma_y bottom_{\text{min across all cases \& combs}}$, 0.0) / Max ($\sigma_y top_{\text{min across all cases \& combs}}$, $\sigma_y bottom_{\text{min across all cases \& combs}}$, 0.0)

Key points when using stress values

1. In Walls X direction is horizontal in plane of wall and Y is vertical

2. “Top”/”Bottom” is dependent on shell local axis system, but if you are only concerned about max values you don’t need to worry about this - use the “max” options which consider both faces.

3. For engineers wanting to consider tensile stresses in walls the “σ_y in-plane tension” option will be of greatest interest.

   - This is based purely on the *membrane* tension stress (i.e. ignoring out of plane bending effects).
   - This can be viewed for enveloped results
   - It is very easy to see walls/panels in which no tension stress is developing.
   - In a full 3D view it may be difficult to assess whether a particular cracking stress level is exceeded. Viewing results in 2D views or sub structures may be helpful here.
   - It should be clear that this is based on the concrete section only - reinforcement content is not considered.

See also

Display 2D results (page 677)

Display 2D deflections

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **2D Results** group, select **Deflections**, then from the droplist beneath select the desired deflection type.
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Customize the display of 2D contours (page 696)

Change result diagram scale settings (page 697)

Animate 1D and 2D deflections (page 675)

Display AsReq contours

AsReq contours can either be displayed as values, or as a pass/fail threshold. Displaying as a pass/fail threshold helps to visualize minimum patch sizes when optimizing panel and patch reinforcement as it highlights the specific areas in which the existing reinforcement is not sufficient.

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 673).

NOTE	Typically the analysis method will be FE chase-down

3. In the **2D Results** group , select **As Req**.
4. From the first droplist, select the desired direction and slab face.
5. From the second droplist, select **Value**, or **Pass/Fail** as required.

See also

Customize the display of 2D contours (page 696)

Change result diagram scale settings (page 697)

Display wall lines

Tekla Structural Designer automatically creates a wall line at the centroid of every meshed shear wall to facilitate the display of wall forces.

1. In the **Status bar**, click **Results View**.

   The **Results** toolbar opens.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **2D Integrated Results** group, select **Wall Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the wall line.

TIP	If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 697).

Display core lines

Tekla Structural Designer automatically creates a core line at the centroid of each concrete core to facilitate the display of core forces.

1. In the **Status bar**, click **Results View**.

   The **Results** toolbar opens.
2. Set the analysis type and loading for viewing analysis results (page 673).
3. In the **2D Integrated Results** group, select **Core Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

Tekla Structural Designer displays the selected result on the core line.

NOTE All forces in the results are rotated to be in the axis system of the core line.

TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 697).

See also

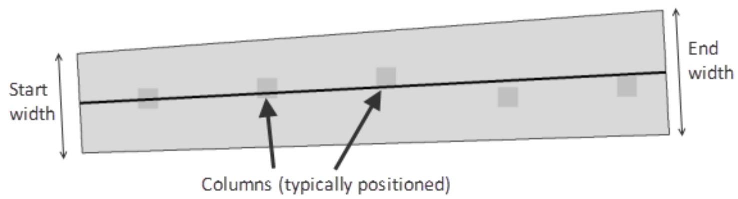
Create concrete cores (page 435)

Manage and display result strips

You can place user-defined result strips across 2D element meshes. For these strips, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used for design purposes, typically for slab design.

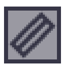
Engineering judgement is required when positioning the strips to ensure suitable design forces are obtained.

By default, strips have parallel edges, but tapering strips can also be defined as shown in the following image:



Create result strips

1. Open a 2D view of the FE mesh where you want to place the strip.
2. In the **Status bar**, click  **Results View**.

   The **Results** toolbar opens.
3. In the **2D Integrated Results** group, click **Result Strip**.
4. In the **Properties** window, adjust the result strip properties according to your needs.
5. Click the start point of the strip.
6. Click the end point of the strip.

TIP Neither the start or the end point have to match nodes in the mesh.

Tekla Structural Designer creates a strip between the selected points.

7. Do one of the following:
   - Continue placing further strips.
   - Press **Esc** to exit the command.

View the results for result strips

Once you have created a result strip in the model, you can obtain results for it without re-running the analysis. In order to view the results, do the following:

1. Open a 3D view containing the strips whose results you want to view.
2. Go to the **Results** toolbar.
3. In the first list of the **Result Type** group, select the desired result type.
4. In the **Loading** list, select the load case or combination that you want to display.
5. In the **2D Integrated Results** group, click the effect that you want to view.

   Tekla Structural Designer displays the selected effect on the strip, and its maximum positive and negative values, calculated according to the result type specified in the strip properties.

NOTE Tekla Structural Designer contains the following three ways to calculate the results:

Method name	Details
Normal	• The results on the center of the result strip are calculated at each station*. • Tekla Structural Designer considers the shell elements local to each station, and calculates a weighted average force based on the distance of the element nodes from the station. The process is repeated for all stations along the center line of the strip to give the results.

Method name	Details
Maximum	• The results on the transverse line across the strip are calculated for each station* along the strip. • Tekla Structural Designer considers the shell elements local to each point, and calculates a weighted average force based on the distance of the element nodes from the point. The maximum result across the strip from all points is taken as the result for the station on the strip center line. The process is repeated for all stations along the center line of the strip to give the results. • The values calculated at points are always weighted averages of results at adjacent nodes. Therefore, they are always less than the peak nodal values. • Maximum values include nodes within the strip.
Average	• Average over strip width. • The results are obtained in the same way as for the maximum option, but in this case, they are averaged to give the results for each station*. The process is repeated for all stations along the center line of the strip to give the results. •

* Along the strip center line, there is a user-defined number of stations. At each station, there is a transverse line with a user defined number of points along it. Final results are always given by station, and obtaining them may or may not use points.

All forces in the results are rotated to be in the axis system of the result strip.

Delete result strips

TIP To delete a strip, ensure that Result Strips are switched on in **Scene Content**.

1. Open a view containing the strip that you want to delete.
2. In the **Quick Access** toolbar, click **Delete**.
3. In the model, click the strip that you want to delete.

   Tekla Structural Designer deletes the selected strip.

See also

Change result diagram scale settings (page 697)

Display 2D view in isometric projection (page 697)

Manage, display and design result lines

You can place user-defined result lines across 2D element meshes. For these lines, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used to assess design solutions, typically for the design of wall panels with openings.

Engineering judgement is required when positioning the lines to ensure suitable design forces are obtained.

Create result lines

1. Open a 2D view containing the wall or slab within which you want to place the strip.
2. In the **Status bar**, click **Results View**. The **Results** toolbar opens.
3. In the **2D Integrated Results** group, click **Result Line**.
4. Click the start point of the line.
5. Click the end point of the line.

   TIP Neither the start or the end point have to match nodes in the mesh.

   Tekla Structural Designer creates a result line between the selected points.
6. Do one of the following:
   - Continue placing further result lines.
   - Press **Esc** to exit the command.

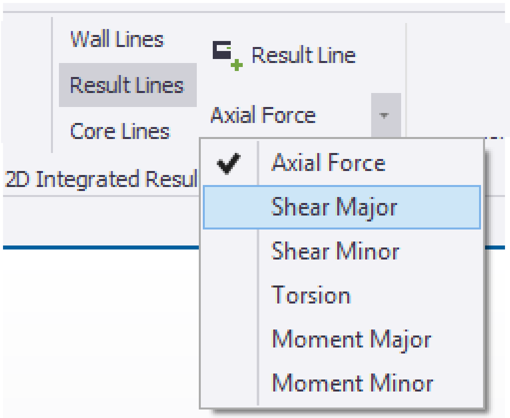
View the analysis results for result lines

NOTE An analysis must be performed after adding or editing result lines in order to obtain up to date results.

To view the results, do the following:

1. Open a view containing the result lines whose results you want to view.
2. In the **Status bar**, click  **Results View**.

   The **Results** toolbar opens.
3. Set the analysis type and loading for viewing analysis results (page 673).
4. In the **2D Integrated Results** group, click **Result Lines**, then from the droplist in the same group, select the desired result.
5. From the droplist in the **2D Integrated Results** group, select the effect that you want to view.

6. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the result line.

   NOTE All forces in the results are rotated to be in the axis system of the result line.

   TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 697).

See also:

View tabular results for result lines (page 767)

Interactively design result lines

1. Open a view containing the result line to be designed.

2. Hover the mouse pointer over the result line that you want to design.

   The **Select Entity** tooltip appears.
3. In the **Select Entity** tooltip, navigate to the result line (RL) name by using the arrow keys.
4. Right-click the result line.
5. In the context menu, select **Interactive Design...** (as a column section, or as a wall section as required).

   The selected (column or wall) interactive design dialog is displayed with the Additional Design Cases page populated with the result line results.
6. Modify the reinforcement parameters as required to achieve a satisfactory design, and click the **Check** button to examine the detailed design results.
7. Click **OK** to save the designed reinforcement to the result line.

   NOTE For help on using the interactive dialogs, see Interactive concrete column design (page 1315) and Interactive concrete wall design (page 1335).

Features of result line design

- The interactive design dialog features full interactive manual selection of both lateral and vertical reinforcement, interaction diagrams and auto-design and check options, just as for a regular wall/ column section interactive design.
- The wall length/ column depth considered is the Result Line length.
- Result lines are not constrained to cross sections in the horizontal plane - vertical sections can be used above/between openings to investigate forces and reinforcement requirements in “coupling beams”.
  - The section design always considers the main bars (running perpendicular to the cross section) as being on the inner layer, from a design perspective this will tend to be conservative but the engineer should give this some consideration when working with non-horizontal sections.
  - The design does not consider the existing reinforcement specified in wall properties - only that which is defined in the Interactive design dialog.
  - Currently this feature is not linked with Reports and so it is envisaged output will be via screenshots of the interactive design and check results dialogs.
- The same result line can be used to interactively design as a column, and as a wall. Both designs are saved to the result line independently.

Delete result lines

TIP To delete a result line, ensure that Result Lines are switched on in **Scene Content**.

1. Open a view containing the result line that you want to delete.
2. In the **Quick Access** toolbar, click **Delete**.
3. In the model, click the result line that you want to delete.

   Tekla Structural Designer deletes the selected result line.

Related video

Interactive design using Result lines

Display mode shapes

When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

1. Open a suitable 2D or 3D View and change the view regime (page 280) to a **Results View**.
2. On the **Results** tab, in the **Result Type** group, select the required analysis type.
3. In the **Loading** list, select the required load case or combination.

   NOTE For RSA seismic analysis, only select load cases, as mode shapes are not applicable for combinations, and therefore cannot be viewed.

4. In the second list of the **Result Type** group, select the desired mode.
5. Select the diagram that you want to display.

See also

The Results View (page 671)

RSA seismic results

For information on how RSA seismic results are displayed for different load cases and combinations, see the following paragraphs.

Result Type

When the result type is 1st or 2nd order RSA seismic, the results that can be displayed depend on the type of the currently selected load case or combination.

Mode Shapes

Mode shapes can be displayed for:

- RSA Seismic load cases:
  - Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**
  - All modes that are relevant for the selected load case
- Effective seismic weight combination:
  - List of all modes returned by the modal analysis

Mode shapes are not displayed for:

- RSA torsion load cases
- Static load cases included in the RSA seismic combination
- RSA seismic combinations

See also

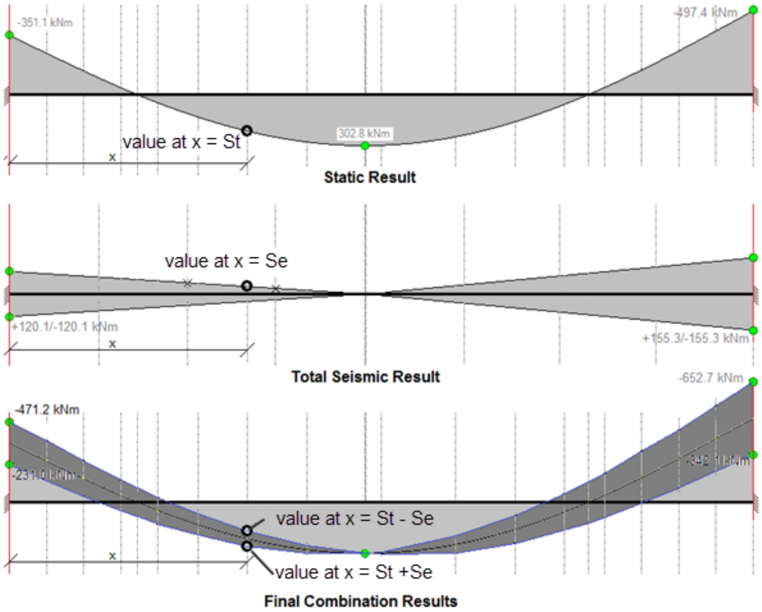
- Display mode shapes (page 691)

1D Element Results

1D element results (and Load Analysis View results) are displayed as follows:

Load case or combination	Display method
RSA seismic load cases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at various points along each member and displayed on both the positive and negative side of the diagram. Therefore, the diagrams are always symmetrical about the base line. All relevant modes: A standard enveloped diagram is displayed.
RSA torsion load cases	Displayed as per 1st order linear analysis.
Static load cases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result:

Load case or combination	Display method
	• Baseline goes through the static values • Top line = static value + seismic value • Bottom line = static value - seismic value



See also

- Display 1D results (page 675)

Story Shear

Story shears are displayed as follows:

Load case or combination	Display method
RSA seismic load cases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each position of interest and the result is displayed as both positive and negative.

Load case or combination	Display method
	All relevant modes: A standard diagram with a single value at each point of interest is displayed.
RSA torsion load cases	Displayed as per 1st order linear analysis.
Static load cases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	The diagram displays two values at each point of interest: • Static value + seismic value • Static value - seismic value

See also

- Display sway drift and story shear (page 676)

Support Reactions

Support reactions are displayed as follows:

Load case or combination	Display method
RSA seismic load cases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each support and the result is displayed as both positive and negative. All relevant modes: A standard diagram is displayed.
RSA torsion load cases	A standard diagram is displayed.
Static load cases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	The diagram displays two values at each support: • Static value + seismic value • Static value - seismic value

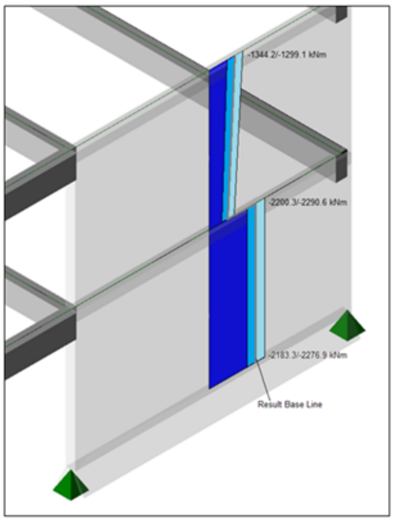
See also

- Display reactions (page 673)

Concrete Wall Results

Concrete wall results are displayed as follows:

Load case or combination	Display method
RSA seismic load cases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at points along the wall line and displayed on both the positive and negative side of the diagram. Therefore, the diagrams are always symmetrical about the wall line. All relevant modes: A standard diagram is displayed.
RSA torsion load cases	A standard diagram is displayed.
Static load cases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result: • Baseline goes through the static values • Top line = static value + seismic value • Bottom line = static value - seismic value



Customize the display of 2D contours

By default, all contour diagrams consist of 10 evenly sized contours, each accounting for 10% of the total range. If necessary, you can increase or decrease the number of contours, and also change the size and the color of individual contours.

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.

   NOTE If you intend to make the changes to the contours in the current model, ensure that you are making changes to the active settings set.

2. Go to **Scene** --> **Contours** .
3. According to your needs, do one or more of the following:

To	Do this
Add new contours	• Click **Split**.
Delete existing contours	• Click **Delete**.
Modify the size of contours	• In the **Size [%]** column, type new values in the appropriate cells.
Change the color of contours	a. In the **Color** column, click the color that you want to change. b. Select a new color. c. Click **OK**.
Revert to the default contour settings	• Click **Reset**.

4. Click **OK**.

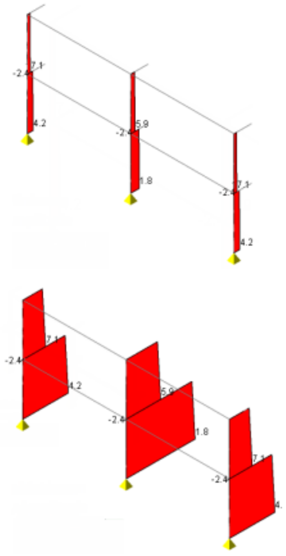
See also

[The Results View (page 671)]

Change result diagram scale settings

- Move the sliders in the **Scale Settings** group according to adjust the diagram amplitude according to your needs.

  The below example shows the effect on an axial force diagram when the 1D Results slider is increased from minimum to maximum.



See also

[The Results View (page 671)]

Display 2D view in isometric projection

Because the diagrams are plotted on each element in the planes in which they act when you are working in a 2D view, you will need switch on an isometric display to see the out of plane forces.

If the 2D view is currently displayed in plan, the **2D**/**3D** toggle button at the bottom right corner of the screen is labeled **3D**.

- To change the current view, click the **2D**/**3D** toggle button.

See also

[The Results View (page 671)]

Sign conventions and coordinate systems

Tekla Structural Designer adopts the standard convention that lower case x, y, and z represent local coordinate systems, whereas upper case X, Y, and Z represent the global coordinate system. For more information on analysis result sign conventions, see the following paragraphs.

Axis systems

The following table presents the axis systems that can be used in Tekla Structural Designer:

Axis system name	Description
Global coordinate system	The global XYZ axis system within which all other systems exist.
Building directions 1 and 2	The principle axes of the building, where dir 1 is rotated at an angle to global X in the horizontal plane.
User coordinate system	A local coordinate system defined by the system or the user.
1D member local coordinate system	The local coordinate system that is applicable to all 1D members, such as beams, columns, and braces.
Mid-pier wall coordinate system	The local coordinate system that is applicable to walls modeled using the mid-pier option.
2D member local coordinate system	The local coordinate system that is applicable to all 2D members, walls, and slabs.
Result line coordinate system	The local coordinate system that is applicable to result lines.
Result strip coordinate system	The local coordinate system that is applicable to result strips.
Foundation reaction coordinate system	The local coordinate system that is applicable to foundations.

General information

All global (XYZ) and local (xyz) axis systems follow the right-hand rule, where:

- x axis is the pointing index finger.
- y axis is the crooked middle finger.
- z axis is the extended thumb.

In the directions of positive rotation:

- About x: the y axis moves toward the z axis.
- About y: the z axis moves toward the x axis.

- About z: the x axis moves toward the y axis.

Object orientation

Tekla Structural Designer considers the orientation of the object when displaying the analysis results. Therefore, to apply the sign convention correctly, you need to know which is the end 1 and which is the end 2 of beams or walls, and which is the face A of columns.

If you select the **Direction** option for an element in **Scene Content**, Tekla Structural Designer displays an arrow on all beams, walls and columns. This arrow points from the start to the end of beams and walls, and from the bottom to the top of columns along the face A. Looking down from the top of a column, Face B, C, and D then follow in the clockwise direction.

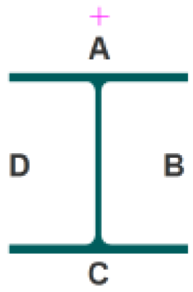
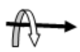


Diagram conventions

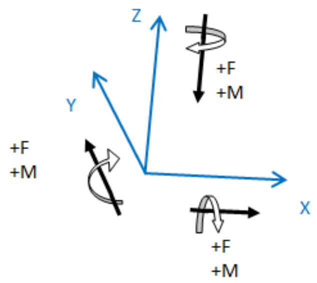
All arrows should point in the direction of the force or moment, as the following image illustrates:



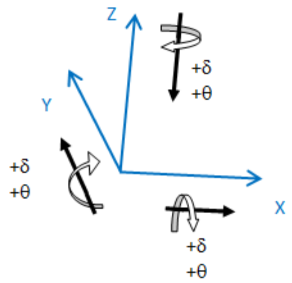
If the arrows are reversed, they become negative forces and moments, as the following image illustrates:

Global coordinate system

The following image illustrates the global axis system and applied load directions.



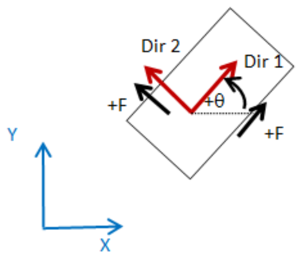
The resulting deflection directions appears as follows.



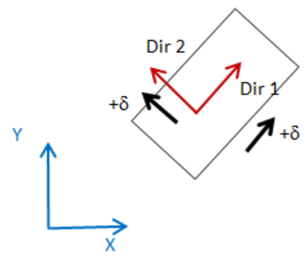
Building directions 1 and 2

Global axes (positive Z vertically up) and angle between X and direction 1 is θ, where θ is positive in right-hand rule about Z.

The following image illustrates the building directions and applied load directions.



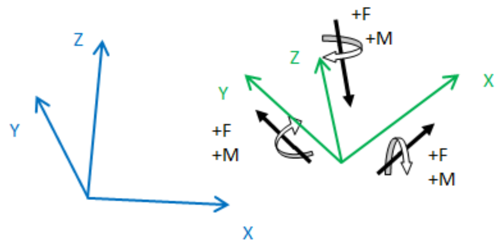
The resulting deflection directions appear as follows.



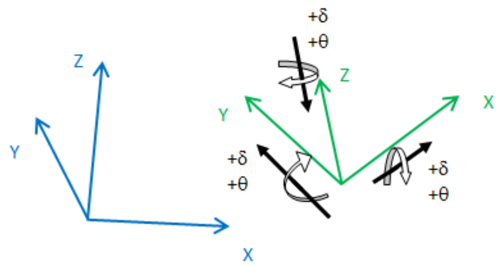
User coordinate system

A user coordinate system can be at any angle to the global coordinate system.

The following image illustrates the axis system of a user coordinate system axis and applied load directions.



The resulting deflection directions appear as follows.



NOTE Every support is given a user coordinate system. Automatically created supports under certain objects default to the following method:

- Support under a single column or wall rotates the foundation forces to align with the y/z-axes of the column or wall
- Support under a mat foundation - uses the global coordinate system.

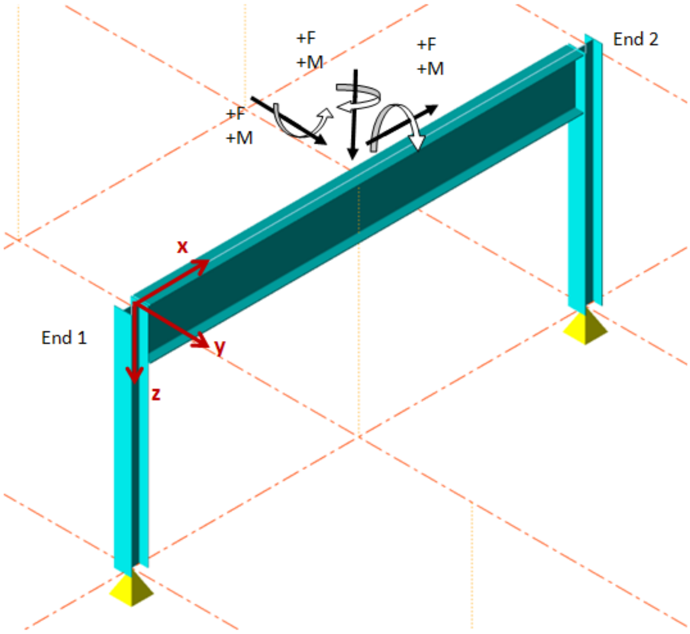
All other supports default to the global coordinate system.

1D member local coordinate system (general case)

Local axis system and applied load directions:

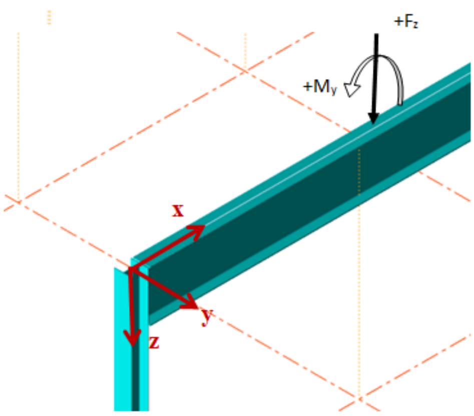
- The local x axis along member starts from end 1 and ends in end 2
- When y = 0:
  - The local z axis lies in the plane created by the local x axis and the global Z axis.
  - The global Z component of the local z axis is always negative.
  - The local y axis follows the right-hand rule.

  y = positive clockwise rotation of y and z axes about the x axis looking towards positive x.

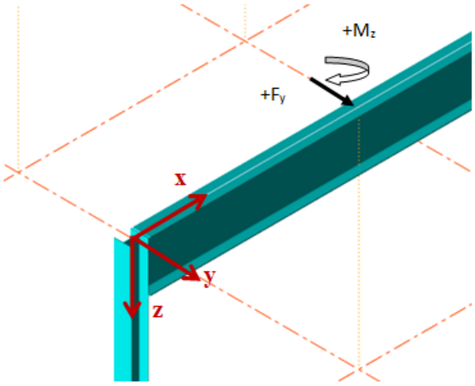


Applied force directions:

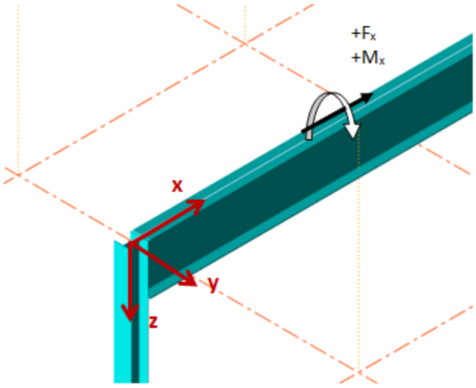
- z = Major (Fz and My):



- y = Minor (Fy and My):



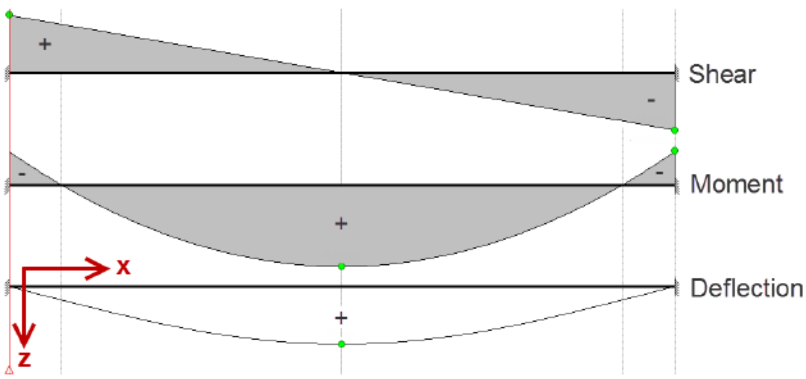
- x = Axial:



Result axis system and directions

In the major axis:

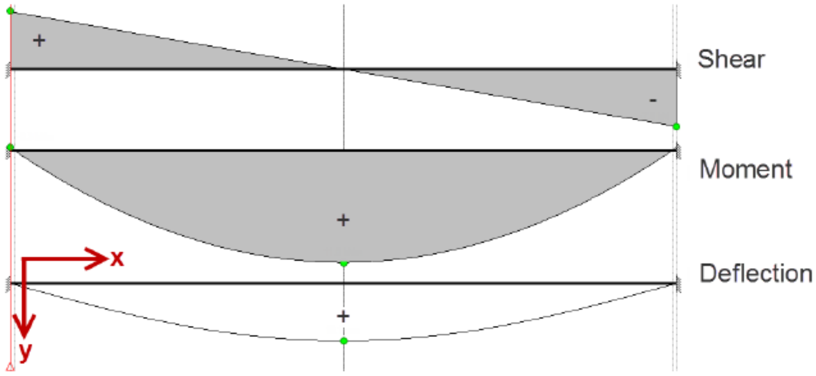
- Moment major = bending about the y axis
- Shear major = shear along the z axis



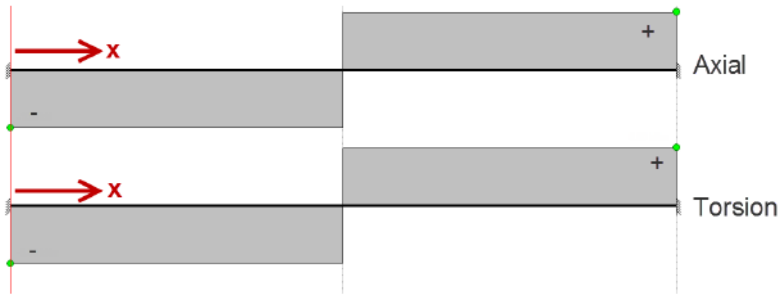
In the minor axis:

- Moment minor = bending about the z axis

- Shear minor = shear along the y axis

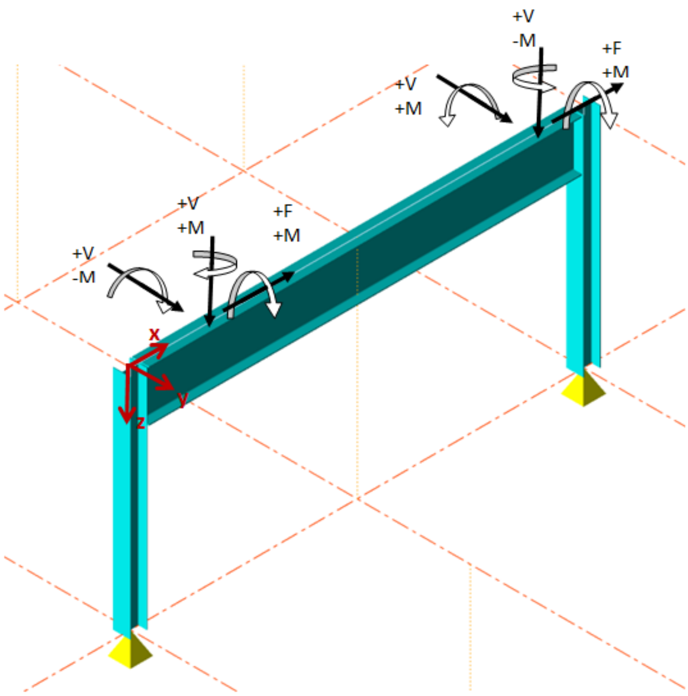


In the axial direction:



Resulting member end forces and directions

Member end forces are the forces applied to the rest of the structure by the member. Based on loading applied above, the forces would be applied as follows:



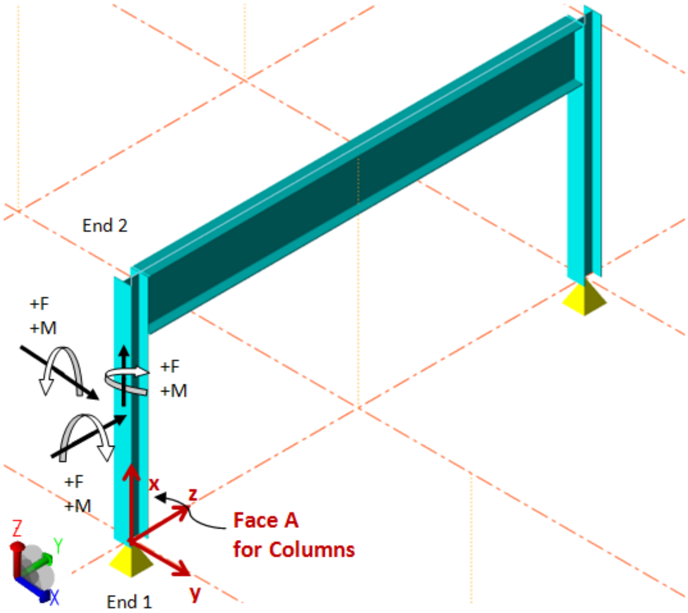
1D member local coordinate system (vertical members)

Local axis system and applied load directions

Local x aligns with global Z (vertical):

- When y = 0:
  - The local y axis aligns with global X.
  - The local z axis according to the right-hand rule.

  y = positive clockwise rotation of y and z about the x axis towards positive x.



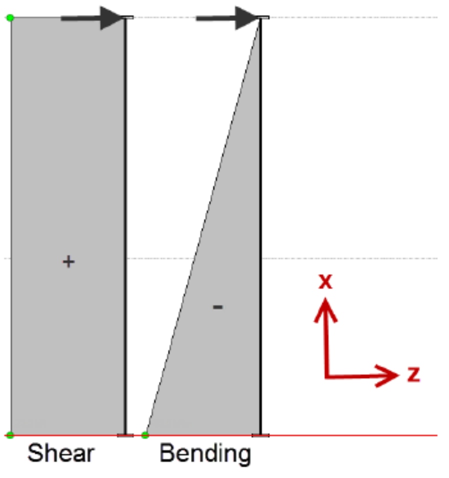
Applied force directions as displayed in the previous image:

- z = Major
- y = Minor
- x = Axial

Result axis system and directions

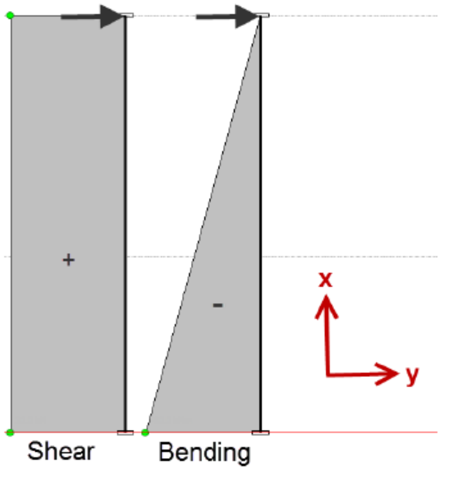
In the major axis:

- Moment major = bending about the y axis
- Shear major = shear along the z axis

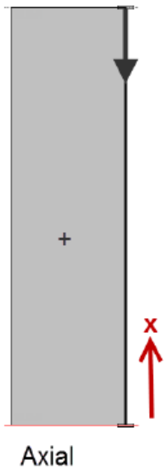


In the minor axis:

- Moment minor = bending about the z axis
- Shear minor = shear along the y axis



In the axial direction:

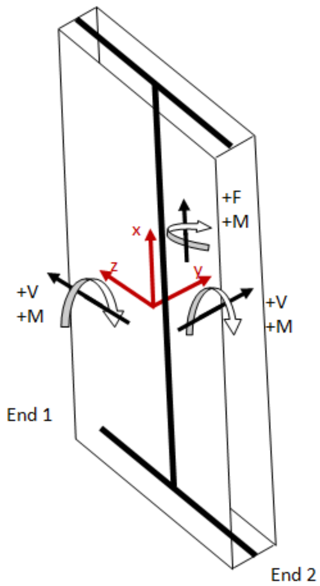


Mid-pier wall coordinate system

Wall axis system and applied load directions

As the following image illustrates, centered on the centroid of the cut section:

- the x axis lies along the stem mid-pier element (positive lowest to highest)
- the z axis lies along the plane of the wall (positive end 2 to end 1)
- the y axis follows the right-hand rule and is normal to the wall.

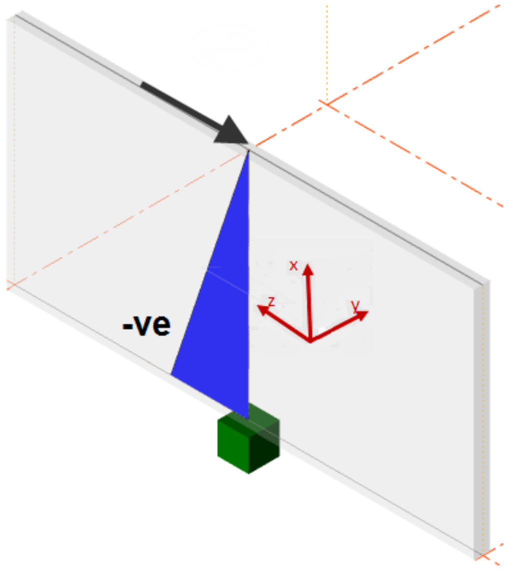


The results from a mid-pier model are in the same axis system as the result line in a meshed wall.

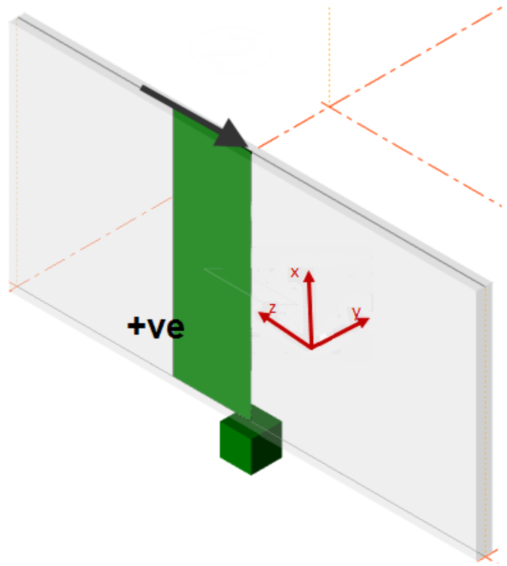
Result axis system and direction

In the major axis:

- Moment major = bending about the y axis:

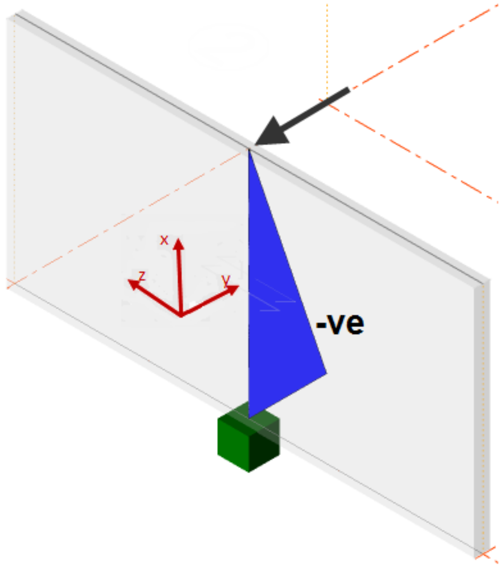


- Shear major = shear along the z axis:

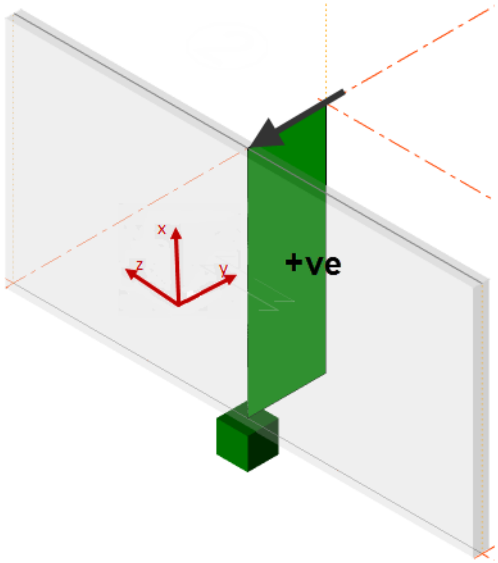


In the minor axis:

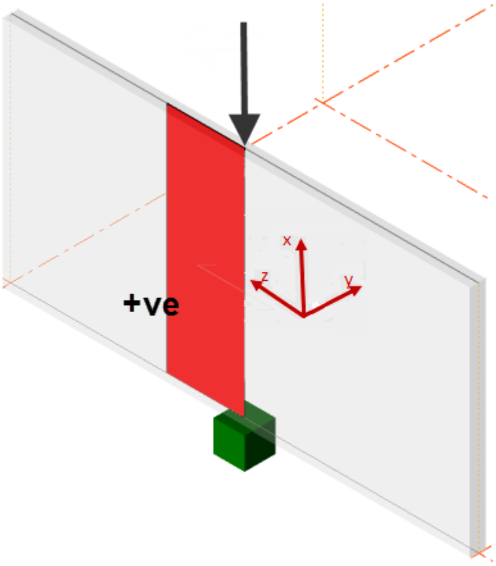
- Moment minor = bending about the z axis:



- Shear minor = shear along the y axis:



In axial and torsion, force is in the x axis and torsion about the x axis:



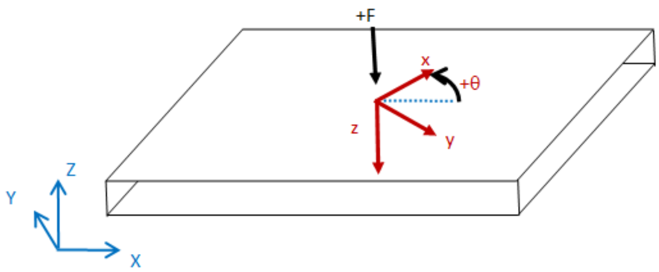
2D member local coordinate system

Horizontal panel local axis system and applied load directions

Horizontal panel local axes are the following:

- The local z axis is normal to the plane of the panel
- When θ = 0:
  - The local x axis plane is in the plane of the panel, aligned with the global X axis and positive in the positive global X direction.
  - The local y axis is in the plane of the panel and follows the right-hand rule.

  θ = positive clockwise rotation of the x and y axis about the z axis looking towards positive z.



Vertical and sloped panel local axis system and applied load directions

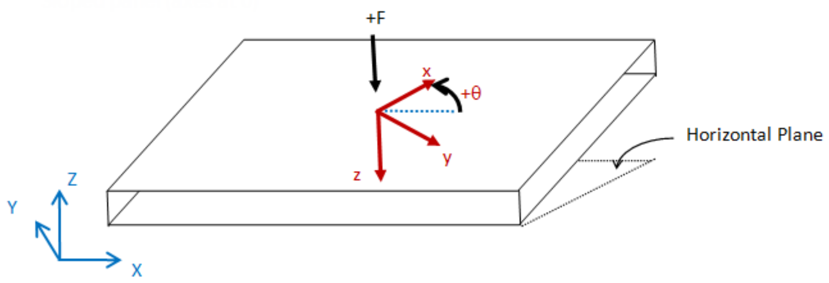
Vertical and sloped panel local axes are the following:

- The local z axis is normal to the plane of the panel.
- When θ = 0:

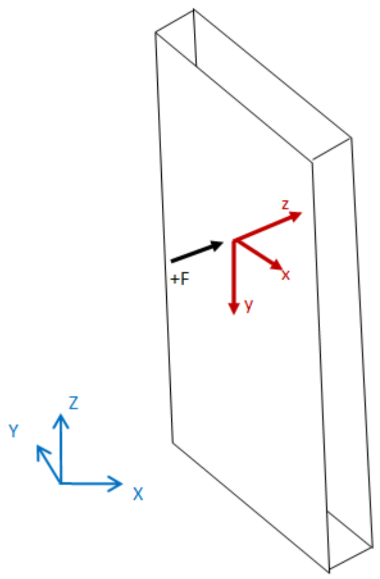
- The local x axis plane is in the plane of the panel and in a horizontal plane.
- The local y axis is in the plane of the panel and follows the line of greatest slope of the plane (positive in the direction of positive global).

θ = positive clockwise rotation of x and y a bout z looking towards positive x.

Sloped panel (axes at θ):

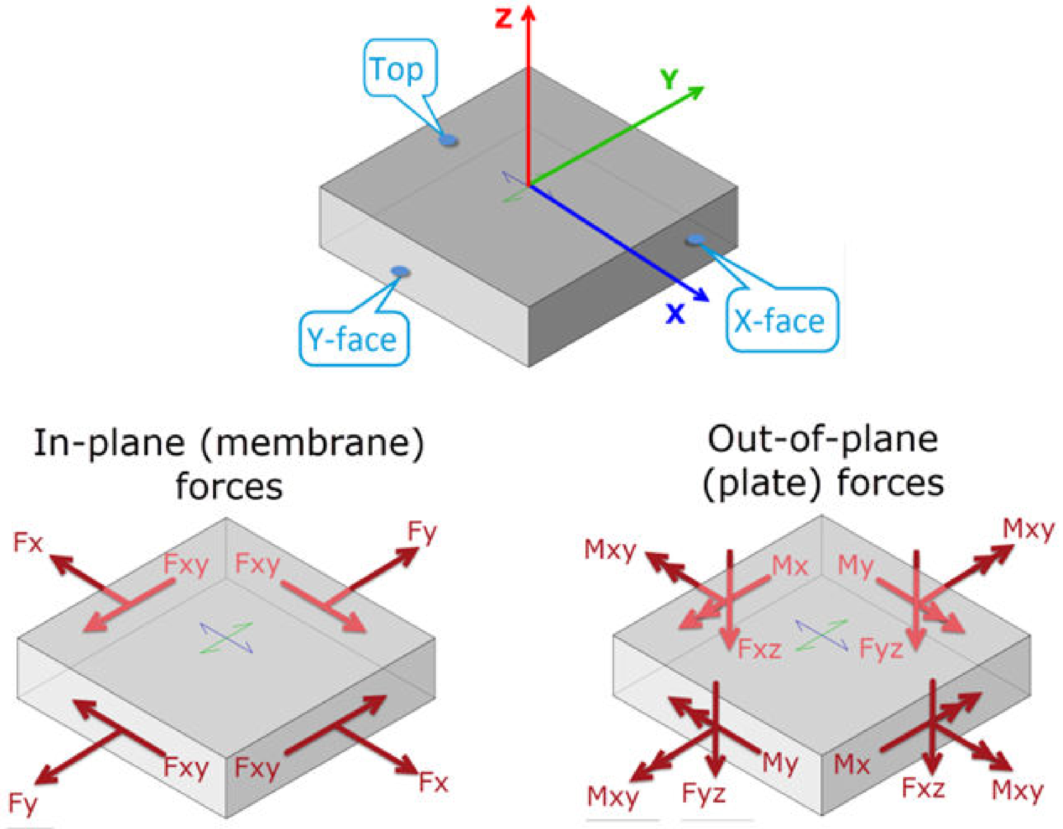


Vertical panel (axes when q = 0):



2D member forces sign convention

The sign convention for 2D member forces is not the same as that of 1D elements. The following diagram illustrates the forces and the panel and 2D element axis system (for results):



- The arrows in the diagram show positive force directions.
    - The double-arrow convention is used for moments: the moment is around the double-arrow, positive being clockwise when looking in the arrow direction.
    - The forces act on a member face cut anywhere in the FE mesh, perpendicular to the force direction.

  Thus, for example, Mx acts on the X face that is perpendicular to the X axis and is moment resulting from spanning in the X direction, Fxz is the out-of-plane shear force acting on the X face, and so on.
- The wood armer design moments (denoted by the *d* suffix) act in the same manner as the unprocessed moments without the *d* suffix. Thus, Mdx acts in the same manner as Mx, and so on.
- The design moments are further classified into top and bottom components for the slab design process.
- The positive Z axis direction (up) follows the right-hand rule and, therefore, is not the same as that for the 2D member local coordinate system. This is

because the 2D member local coordinate system for the applied load directions displays the positive applied load direction convention that, for Z only, is opposite to the convention of the global and 2D element axes.

- A positive moment creates tension in the top surface of the shell. Therefore, the moment over a supporting column is positive, whereas the span moment is negative.
- The conventions for wall results are exactly the same as conventions for columns, so they can be interpreted in the same way.
- The compression of axial loads (Fx and Fy) is negative.
- Out-of-plane shear (Fxz and Fyz) is positive when shear is such that moment is increasing in the positive X or Y direction.

Result line coordinate system

Centered on the centroid of the cut section:

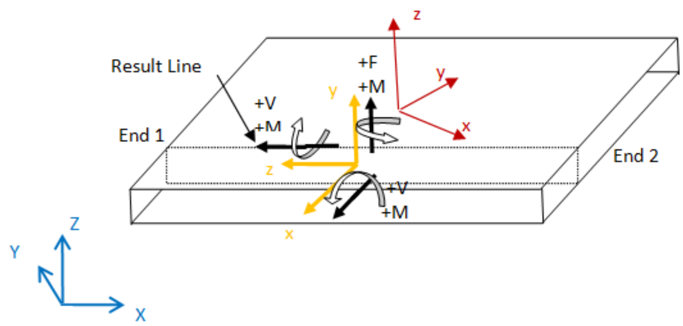
- The z axis lies along the result line (positive end 2 to end 1).
- The y axis normal to plane of mesh (generally positive in the positive Z direction, in special cases positive x towards positive Z).
- The x axis follows the right-hand rule and lies in the mesh, so x is perpendicular to the cut line.

NOTE The results from a result line are exactly like those for a mid-pier model when the cut is horizontal and the cut direction matches the direction required.

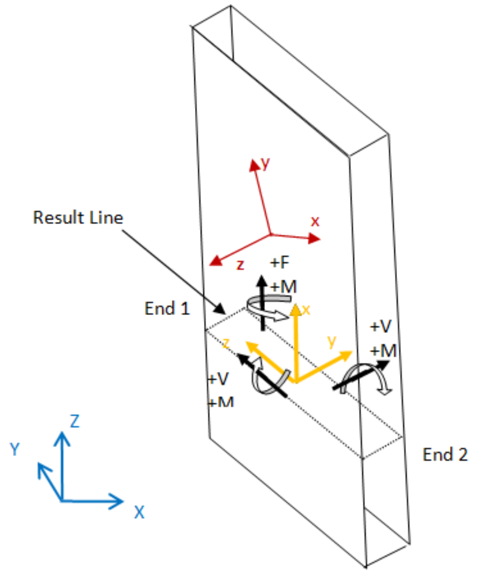
Result Axis System

- In the major axis: bending about the y axis and shear along the z axis
- In the minor axis: bending about the z axis and shear along the y axis
- Axial and torsion: force in x and bending about the x axis

General case:



Special case:



Result strip coordinate system

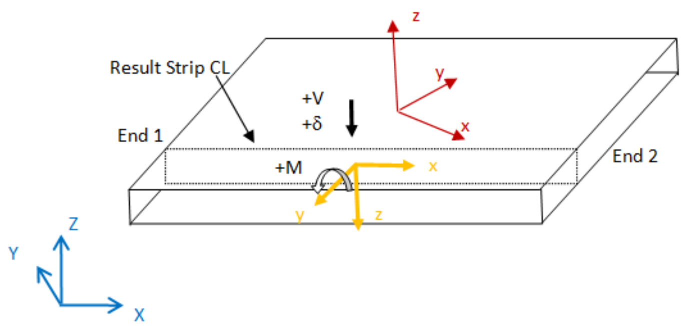
Centered at each station along the strip center line, whether there is a single or several continuous strips:

- The z axis is normal to plane of mesh (generally positive in the negative Z direction, in special cases positive x towards positive Z).
- The x axis lies along the result strip (positive end 1 to end 2)
- The y axis lies along the transverse line to the result strips and follows the right-hand rule, so the y axis is perpendicular to the strip line.

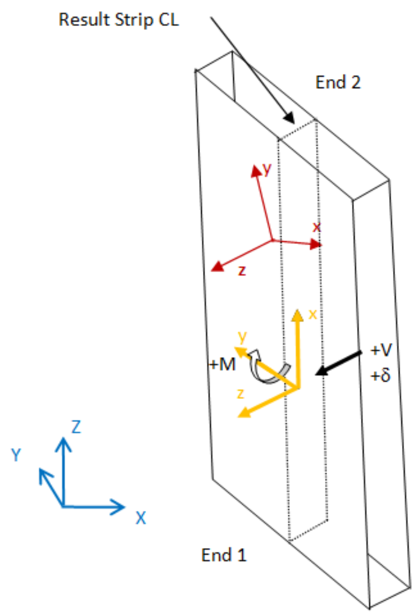
Result axis system

- Deflection in the z direction
- Out of plane moment about the y axis
- Shear in the z direction

General case:

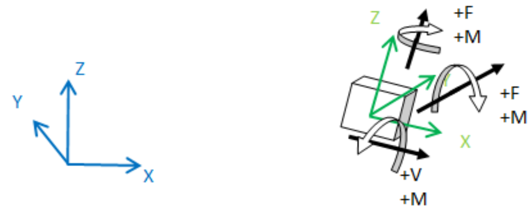


Special case:

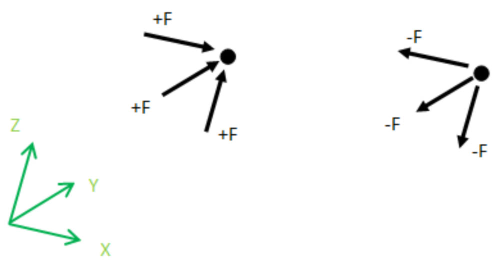


Foundation reaction coordinate system

As the following image illustrates, the foundation reaction coordinate system is aligned with the coordinate system for the support node, whether that is the global coordinate system or a user coordinate system.



Reactions are the forces applied to the structure by the foundation. They appear as follows.



The Load Analysis View

Analysis results for individual members and walls can be displayed in a Load Analysis View.

Open a Load Analysis View

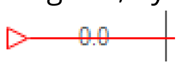
1. Right-click the member whose loading and analysis results you want to view.
2. In the context menu, select **Open Load Analysis View**.

   Tekla Structural Designer opens the load analysis view and the **Loading Analysis** tab.
3. In the **Loading** list, select the load case or combination whose results you want to view.
4. Select the analysis type using the list in the **Result Type** group.
5. If you are viewing the results for a load combination, in the **Result Type** group, select whether you want to view the results based on strength or service factors.
6. In the **Direction** group, select the axis type (**Axial**, **Major**, or **Minor**).

   Tekla Structural Designer displays the results according to the selected options. If necessary, you can control the results in the **Properties** window.

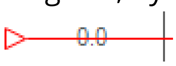
Load Analysis View properties for columns

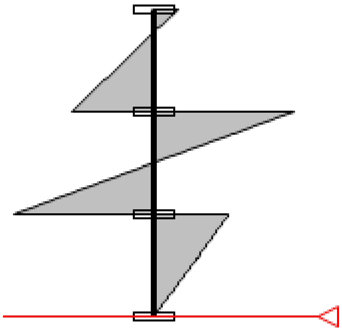
Result Type: Axial

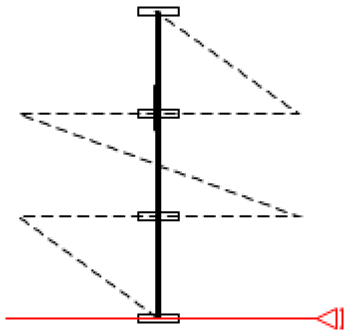
Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Axial force above	The axial force in the column immediately above the cross section at the distance specified.

Property	Description
Axial force below	The axial force in the column immediately below the cross section at the distance specified.
Axial force reduced above	The axial force in the column immediately above the cross section at the distance specified, taking into account imposed load reductions.
Axial force reduced below	The axial force in the column immediately below the cross section at the distance specified, taking into account imposed load reductions.
Torsion moment above	The torsion in the column immediately above the cross section at the distance specified.
Torsion moment below	The torsion in the column immediately below the cross section at the distance specified.
Show axial force	If cleared, the axial force diagram is removed from the view.
Show axial force reduced	If cleared, the axial force diagram is removed from the view.
Show torsion moment	If cleared, the torsion diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor

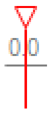
Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is

Property	Description
	displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Shear above	The major or minor shear force immediately above the cross section at the distance specified.
Shear below	The major or minor shear force immediately below the cross section at the distance specified.
Moment above	The major or minor moment immediately above the cross section at the distance specified. The Moments diagram is solid and shaded. 
Moment below	The major or minor moment immediately below the cross section at the distance specified.
Ecc. Moment above	The major or minor moment due to eccentricity immediately above the cross section at the distance specified. The eccentricity moments diagram is dashed and unshaded.

Property	Description
	
Ecc. Moment below	The major or minor moment due to eccentricity immediately below the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load above	The applied distributed load in the major or minor direction immediately above the cross section at the distance specified.
Applied load below	The applied distributed load in the major or minor direction immediately below the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflections diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Load Analysis View properties for beams

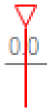
Result Type: Axial

Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Axial force left	The axial force in the beam immediately to the left of the cross section at the distance specified.
Axial force right	The axial force in the beam immediately to the right of the cross section at the distance specified.
Torsion moment left	The torsion in the beam immediately to the left of the cross section at the distance specified.
Torsion moment right	The torsion in the beam immediately to the right of the cross section at the distance specified.
Relative angle of twist	The relative angle of twist (due to torsion) in the beam cross section, at the distance specified.
Angle of twist derivative	A droplist allowing the selection of the derivative of angle of twist: • First • Second

Property	Description
	• Third (This property is displayed for “Open” sections only).
Angle of twist derivative left	The relative angle of twist (due to torsion) in the beam cross section, immediately to the left of the distance specified. (This property is displayed for “Open” sections only).
Angle of twist derivative right	The relative angle of twist (due to torsion) in the beam cross section, immediately to the right of the distance specified. (This property is displayed for “Open” sections only).
Show axial force	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show torsion moment	If cleared, the torsion diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show relative angle of twist	If cleared, the relative angle of twist diagram is removed from the view.
Show angle of twist derivative	If cleared, the angle of twist derivative diagram is removed from the view. (This property is displayed for “Open” sections only).
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor, (plus Major Principal and Minor Principal for angle sections)

Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the

Property	Description
	diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Shear left	The major or minor shear force immediately to the left of the cross section at the distance specified.
Shear right	The major or minor shear force immediately to the right of the cross section at the distance specified.
Moment left	The major or minor moment immediately to the left of the cross section at the distance specified.
Moment right	The major or minor moment immediately to the right of the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load left	The applied distributed load in the major or minor direction immediately to the left of the cross section at the distance specified.
Applied load right	The applied distributed load in the major or minor direction immediately to the right of the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.

Property	Description
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflection diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

RSA Seismic Results in a Load Analysis View

Loading Analysis Views for 1st or 2nd Order RSA Seismic result types use the same rules as those applied to multi-member Results Views for the same result types, i.e. as follows:

RSA Seismic Loadcases

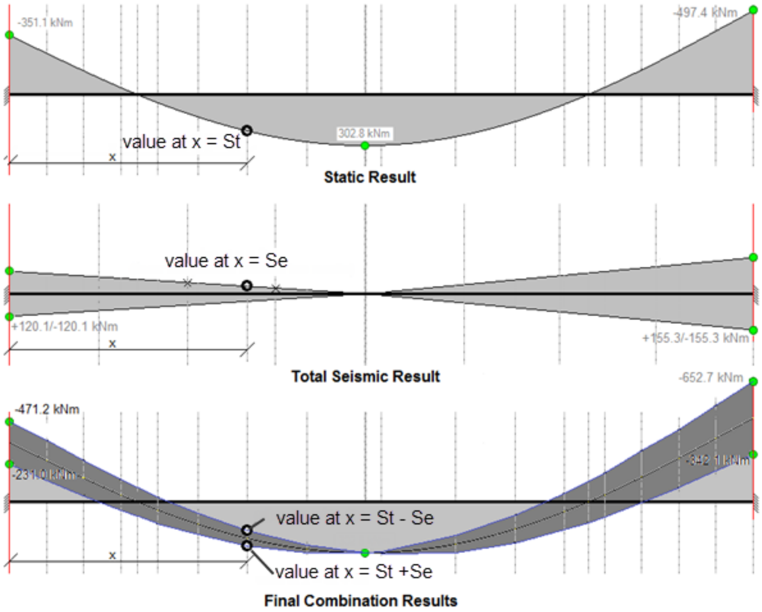
- Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options
    - Absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the base line).
- All relevant modes - a standard enveloped diagram is displayed
- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available

RSA Seismic Combinations

A droplist is provided to allow you to view:

- Design Profile - derived from the Static+Seismic result, the Design Profile is always plotted on the same side of the base line as the Static Only result
- Static Only - displayed as per 1st order linear analysis
- Seismic Only - absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram
- Static+Seismic - an envelope is drawn showing the seismic results above and below the static result.

- Base line is through the static values
- Top line is static value + seismic value
- Bottom line is static value - seismic value



6.4 Solver models

You can display the solver model used for each analysis type in 2D or 3D by opening an appropriate solver view.

If you have performed more than one analysis type, then (providing the geometry and loading have not changed between runs), each solver model is retained.

NOTE Changes to mesh size or uniformity do not constitute a change in the geometry. Hence, if different meshes have been applied for each analysis type, the different solver models are retained.

To	Click the link below:
Learn about the different solver model types and what is displayed within them	Solver model types (page 725)
Open a solver view to display a solver model	Open a solver view (page 732)
Change the solver model type displayed in a solver view	View the solver model used for a particular analysis (page 733)

To	Click the link below:
View properties of solver model objects	View solver model object properties (page 734)
Learn about rigid offsets and rigid zones used in concrete beams and columns	How concrete beams and columns are represented in solver models (page 738)
Learn about the analytical model used for meshed walls	How meshed walls are represented in solver models (page 750)
Learn about the analytical model used for mid-pier concrete walls	How mid-pier walls are represented in solver models (page 755)
Learn about the analytical model used for shear only walls	How shear only walls are represented in solver models (page 757)
Learn about the analytical model used for bearing walls	How bearing walls are represented in solver models (page 761)
View tabular solver model data and solver model results	View tabular solver model data (page 764)

Solver model types

You can review the different solver models by opening a Solver View and then choosing the solver model type required from the right-click menu.

Working Solver Model

The **Working Solver Model** shows the model in its form prior to any analysis.

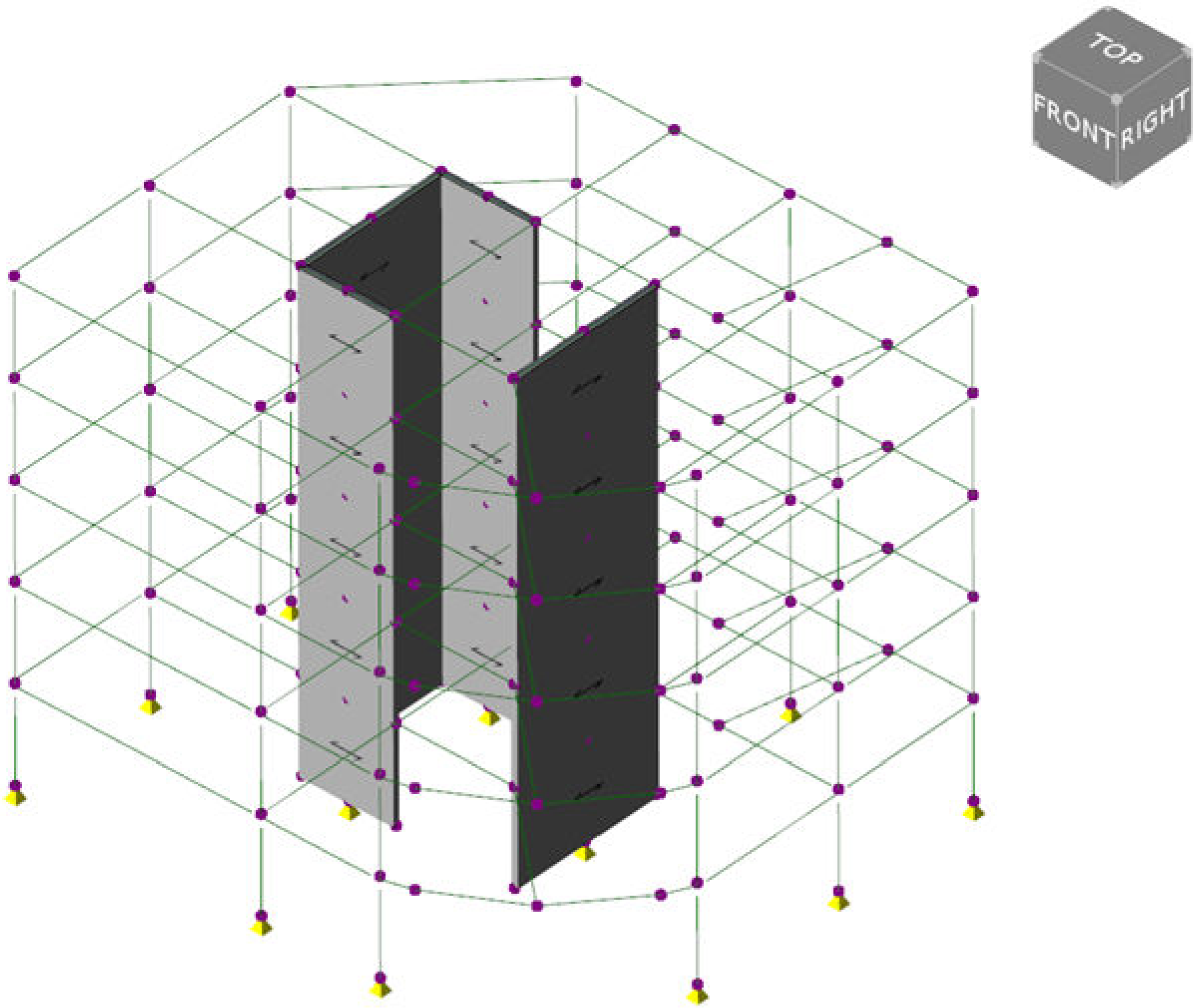
Although 1D elements, solver nodes, and diaphragms are displayed, 2D elements are not (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis.

Solver Model used for 1st Order Linear and 2nd Order Linear

The solver model used for 1st order and 2nd order linear analysis potentially features a mix of 1D analysis elements, FE meshes and diaphragms as follows:

- beams and columns are modelled as 1D analysis elements
- walls are either mid-pier analysis elements, or FE meshes
- slabs (optionally) form rigid, or semi rigid, diaphragms in floors
- 1-way slabs have their loads decomposed on to supporting members at a preliminary stage of the analysis.

- 2-way slabs are (typically) not meshed, in which case they will also have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs (optionally) can be meshed, this is only recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
- supports are user defined



Any FE meshes in the solver model are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order linear or 2nd order linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

2-way slabs meshed

Optionally you can choose to mesh all 2-way slabs - making a fully meshed model (both walls and floors) possible.

This is generally not recommended as it will inevitably increase the model size, (and potentially the time to solve for large models), although it might be considered that a fully meshed model behaves more "correctly" where slabs are considered to be part of the lateral load resisting system of the structure.

It is more likely that you will choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

Solver Model used for 1st Order Non Linear and 2nd Order Non Linear

These solver models are basically the same as those used for 1st order analysis with the exception that they will also feature non linear elements.

Any FE meshes in these solver models are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order non-linear or 2nd order non-linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis types after changes to either geometry or loading will prevent you from displaying results for these models.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 1st Order Modal

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 1st order modal analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Buckling

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 2nd order buckling analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

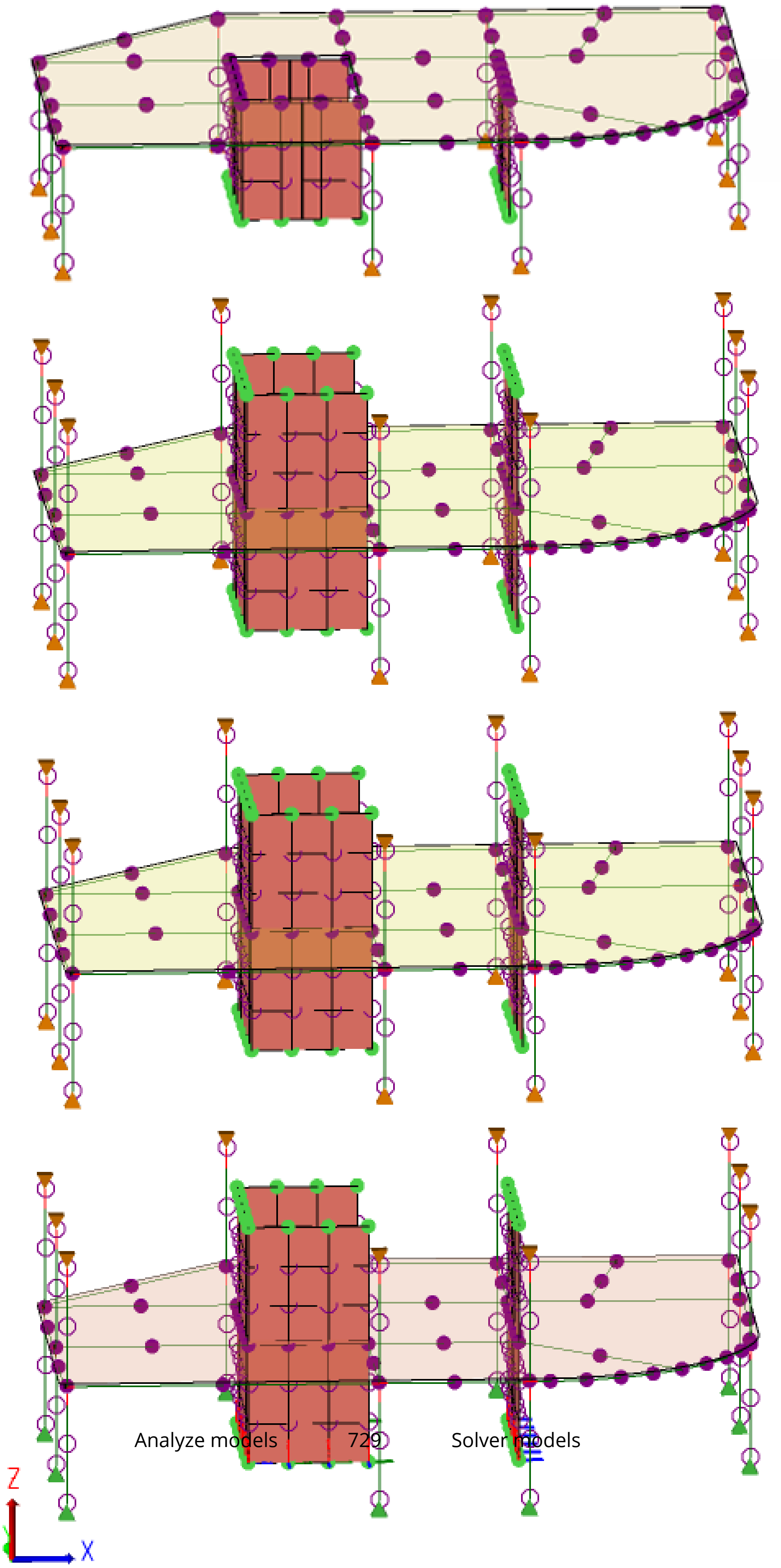
Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

In grillage chasedown a 3D sub model is formed for each floor including those columns and walls that connected to the floor.

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

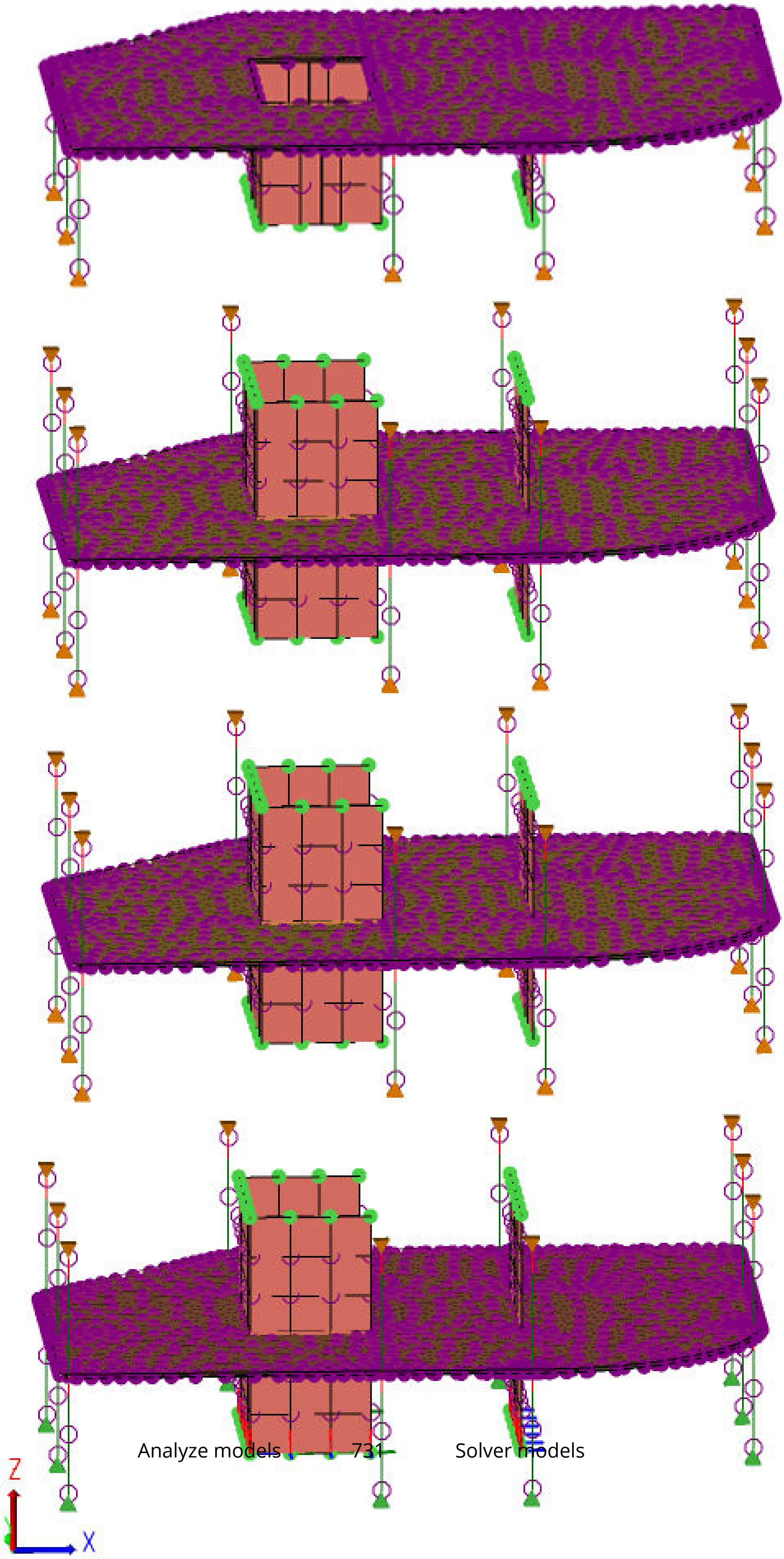

Z
X

they have been set as meshed for 3D Analysis. For all other slabs **load decomposition** is carried out prior to the analysis.

Solver Model used for FE Chasedown

FE chasedown is similar to grillage chasedown, with 3D sub models being formed at each level; the one difference being that in the FE chasedown the two-way slabs are always meshed.

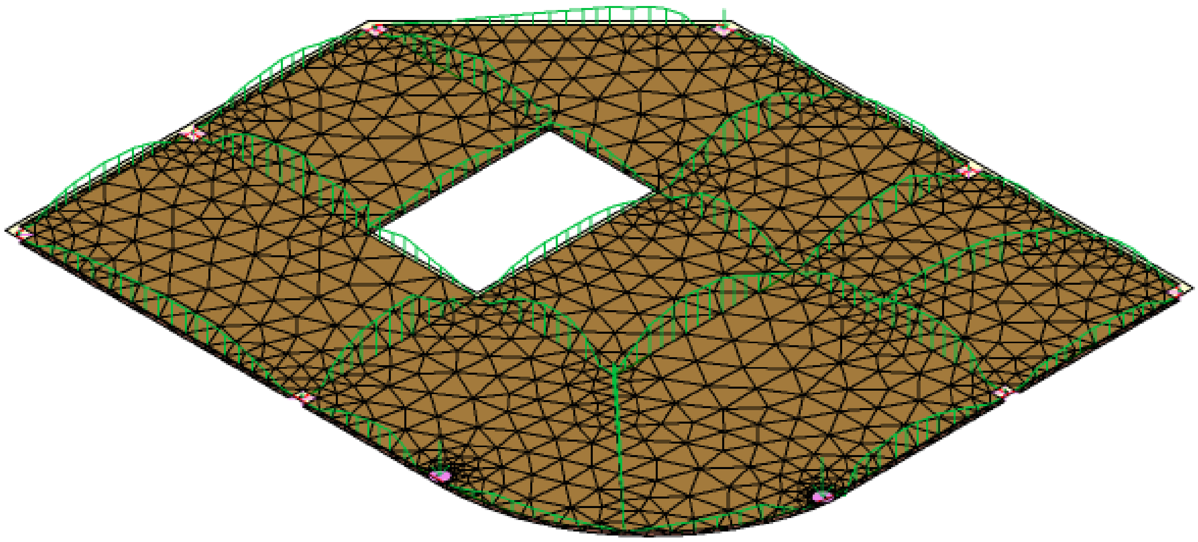
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.


Z
X

Solver Model used for Load Decomposition

At each level, (provided you have not checked the Mesh 2-way Slabs in 3D Analysis option), a solver model is created solely for the purpose of decomposing slab and panel loads back on to the supporting members. As these load decomposition models are only used during the pre-analysis stage, by default they are not retained.

However, if you want to examine the load decomposition model used at a given level this is possible by editing the level properties prior to analysis and selecting **keep solver model**.



Solver Model used for Load Decomposition

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

Refresh Solver Model

The solver model currently displayed is updated to reflect changes that have occurred in the working solver model since the last analysis. The 2D element mesh is also updated to reflect the current mesh parameters. Previous analysis results are also erased for this solver model

Open a solver view

You can either open a new view that displays the solver model, or change another 2D or 3D view to a solver view. In order to open a solver view, see the following instructions.

Open a solver view as a new view

1. To duplicate the existing 2D or 3D view, right-click the tab.
2. In the context menu, select **Duplicate view**.

   A duplicate of the current view opens.
3. To change the view type of the duplicate, do one of the following:

   - Right-click the duplicate tab, and in the context menu, select **Solver view**.
   - In the **Status bar** at the bottom of the window, click **Solver View**.

Change the existing view to a solver view

1. Open an appropriate 2D or 3D view.
2. To change the view type of the view, do one of the following:

   - Right-click the view tab and in the context menu, select **Solver view**.
   - In the **Status bar** at the bottom of the window, click **Solver View**.

See also

View the solver model used for a particular analysis (page 733)

View solver model object properties (page 734)

View the solver model used for a particular analysis

In case you need to view the solver model that Tekla Structural Designer uses for a particular analysis type, see the following instructions.

1. Open a solver view. (page 732)
2. Right-click anywhere in the solver view.
3. In the context menu, select **Solver models**.
4. In the submenu, select the desired solver model.

   Tekla Structural Designer opens the selected solver model.

See also

View solver model object properties (page 734)

View solver model object properties

After opening a solver view (page 732), you can select solver nodes, solver elements, and supports in order to see their properties in the **Properties** window.

- Solver node properties (page 734)
- Solver element properties (page 734)
- Solver element (1D) types (page 735)
- Solver element 2D properties (page 737)
- Support properties (page 2142)

Solver node properties

When in a Solver View, solver node properties are displayed in the Properties Window as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

Property	Description
General	
Fx, Fy, Fz	The translational degrees of freedom at the node.
Fx, Fy, Fz	The rotational degrees of freedom at the node.
Coordinate	The node location.
P-Delta	This property cannot be edited.
Exclude from Diaphragm	Check to remove the node from the diaphragm.
Diaphragm #	Specifies the diaphragm number to which the node is connected.

Solver element properties

When in a Solver View, solver element properties are displayed in the Properties Window as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

Property	Description
General	
Active	When this is set to False the solver element is inactive in the analysis. Only certain member types (braces,

Property	Description
	analysis elements) can be made inactive.
Type	The type of the solver element
Fabrication	The fabrication type of the solver element.
Construction	The construction type of the solver element.
Material	The solver element material.
Gamma angle	Defines the element orientation about its local x axis. When gamma = 0, the local z lies in the plane created by the local x axis and the global z axis.
Length	The solver element length.
Start Releases	
Fx, Fy, Fz	These define the translational end releases at end 1.
Mx, My, Mz	These define the rotational end releases at end 1.
End Releases	
Fx, Fy, Fz	These define the translational end releases at end 2.
Mx, My, Mz	These define the rotational end releases at end 2.

Solver element (1D) types

Eight different 1D solver element Types are available in Tekla Structural Designer as follows:

Beam

An element in any material, with user defined area and inertia properties, and user-definable end releases - used in **all solver models** for:

- Columns (any material)
- Beams (any material)
- Truss top, bottom and side members (any material)
- Mid-pier concrete wall: wall-beam, and wall-column elements
- Bearing wall: wall-beam elements
- Analysis Elements (any material) with element type: Beam

Truss

An element in any material, with user-defined cross sectional area, and pinned ends (releases not being editable) - used in **all solver models** for:

- Braces (any material) that have not been set as tension or compression only
- Truss internal members (any material) that have not been set as tension or compression only
- Bearing wall: wall-column elements
- Analysis Element (any material) with element type: Truss

Truss 1D solver elements are also used in **linear solver models** only for:

- Braces (any material) that have been set as tension or compression only
- Truss internal members (any material) that have been set as tension or compression only

Tension only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer tension. This is a non-linear element which requires non-linear analysis - hence used in **non-linear solver models only** for:.

- Braces (any material) that have been set as tension only
- Truss internal members (any material) that have been set as tension only
- Analysis Element (any material) with element type: Tension only

Compression only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer compression. This is a non-linear element which requires non-linear analysis -hence used in **non-linear solver models only** for:

- Braces (any material) that have been set as compression only
- Truss internal members (any material) that have been set as compression only
- Analysis Element (any material) with element type: Compression only

Linear axial spring

An element that deflects linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear axial spring

Linear torsional spring

An element that rotates linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear torsional spring

Non-linear axial spring

An element that deflects non-linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear axial spring

Non-linear torsional spring

An element that rotates non-linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear torsional spring

Solver element 2D properties

When in a Solver View, solver element 2D properties are displayed in the Properties Window as shown below. None of these properties can be directly edited in the Solver View.

Property	Description
Panel	
Type	The 2D element type will be Shell or Semi-rigid depending on the Slab Type, Decomposition and Diaphragm Options that have been set.
Thickness	The 2D element thickness is derived from a different property depending on the slab type: • Composite Slab; Precast Slab; Slab on Beams; Flat Slab - Overall Depth • Steel Deck; Timber Deck - Thickness
Orientation	The 2D element orientation in the solver model follows the rotation angle defined for the slab item to which it belongs.

Property	Description
DivideStiffnessBy	This property applies to semi-rigid elements only. It adjusts the stiffness determined from the material properties in order to control semi-rigid diaphragm flexibility.
CrackedOption	yes/no
Nodes	
Node 1, Node 2, Node 3	The node numbers associated with this element.

How concrete beams and columns are represented in solver models

Solver elements for most members are created directly between the member insertion points - they do not take into account major and minor snap points, or any offsets that might have been specified in the member properties. The exception to this rule is that solver elements for concrete columns and concrete beams do take into account snap points and offsets - Rigid offsets (page 738) are then automatically introduced where necessary to connect the solver elements.

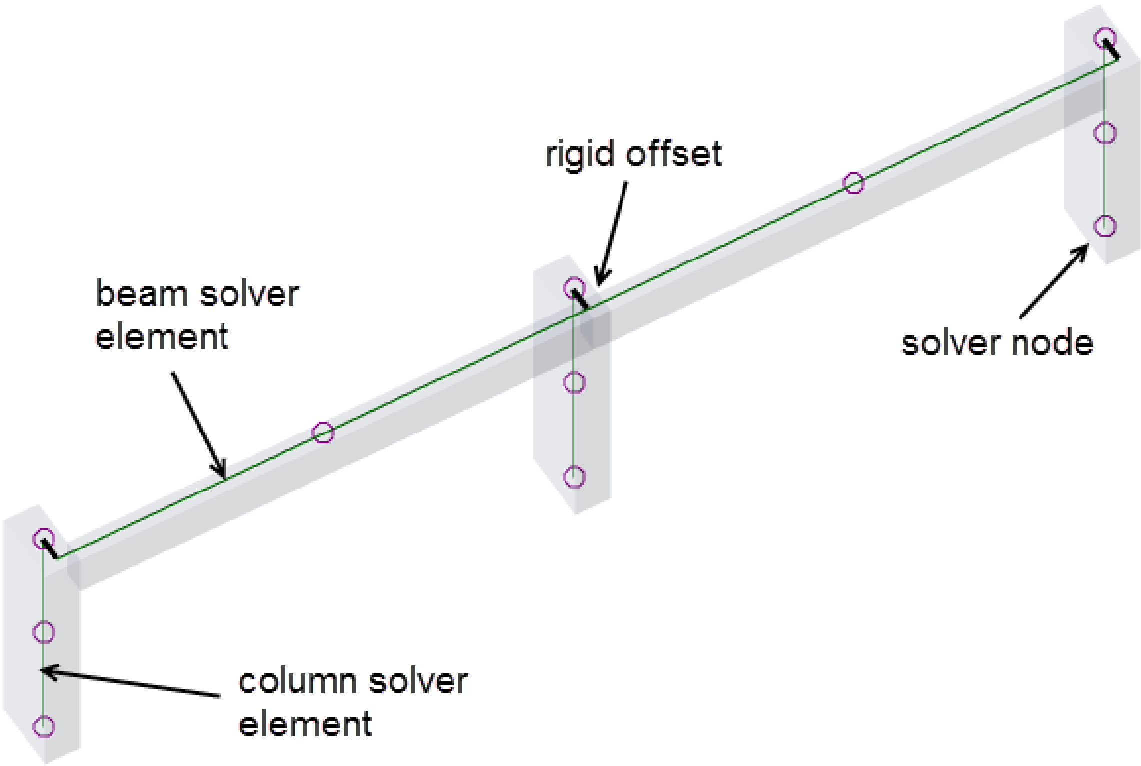
NOTE The rules applied to insertion of solver elements for concrete columns are different to those that are applied to concrete beams. For concrete structures this enables you to simplify the grid layout but then employ offsets to position the members exactly.

Design codes also allow engineers to assume parts of concrete beams and columns are rigid, leading to more efficient designs. Tekla Structural Designer uses Rigid zones (page 739) to cater for this where columns and beams are connected and where beams are connected to other beams. Columns can have rigid zones when they are the supporting or supported member, but beams will only have rigid zones when they are the supported member.

Rigid offsets

For concrete beams and columns rigid offsets are automatically applied to the start and end of solver elements as required to ensure that the solver model is properly connected.

This will be necessary whenever the 1D solver elements are not co-linear. A typical example of this occurs when concrete edge beams are aligned to be flush with the face of the supporting columns, as shown below:



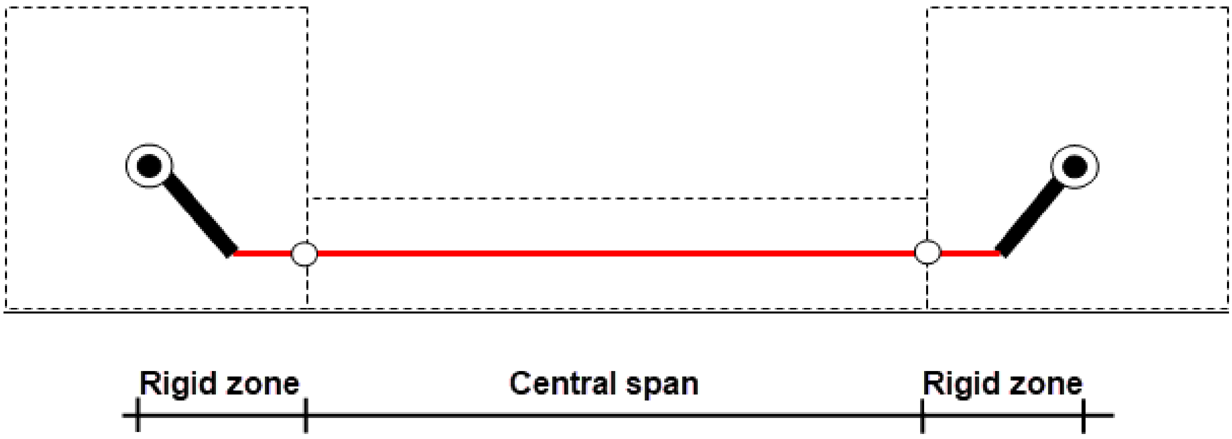
TIP To see rigid offsets: open a Solver View, and then in Scene Content ensure that 1D Elements> RigidOffsets is selected.

Click the below link to view a couple of examples illustrating the use of rigid offsets in concrete columns and beams:

- Rigid offsets examples (page 740)

Rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections. The proportion of the zone which is modelled as rigid (the thick black line shown below) is specified as a percentage, the remaining portion of the rigid zone (the red line inside the rigid zone) remains elastic. The proportion of the rigid zone that is rigid is specified in Model Settings and can vary between 0 - 100%



As shown above, the elastic portion of the rigid zone is aligned with the central span solver element.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the **Rigid zones not applied** option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the **Apply rigid zones** option that is provided in the column properties under the **Design control** heading.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

There is a significant difference between Rigid Zones Not Applied and Rigid Zones Applied with 0% rigidity. The total elastic length of a member is the same in the two models, but the position of releases and start/end of design members will be different.

Rigid zones should not be confused with rigid offsets which are used to ensure that the analysis model is properly connected, i.e. it is possible to have rigid offsets in the model even if rigid zones are turned off.

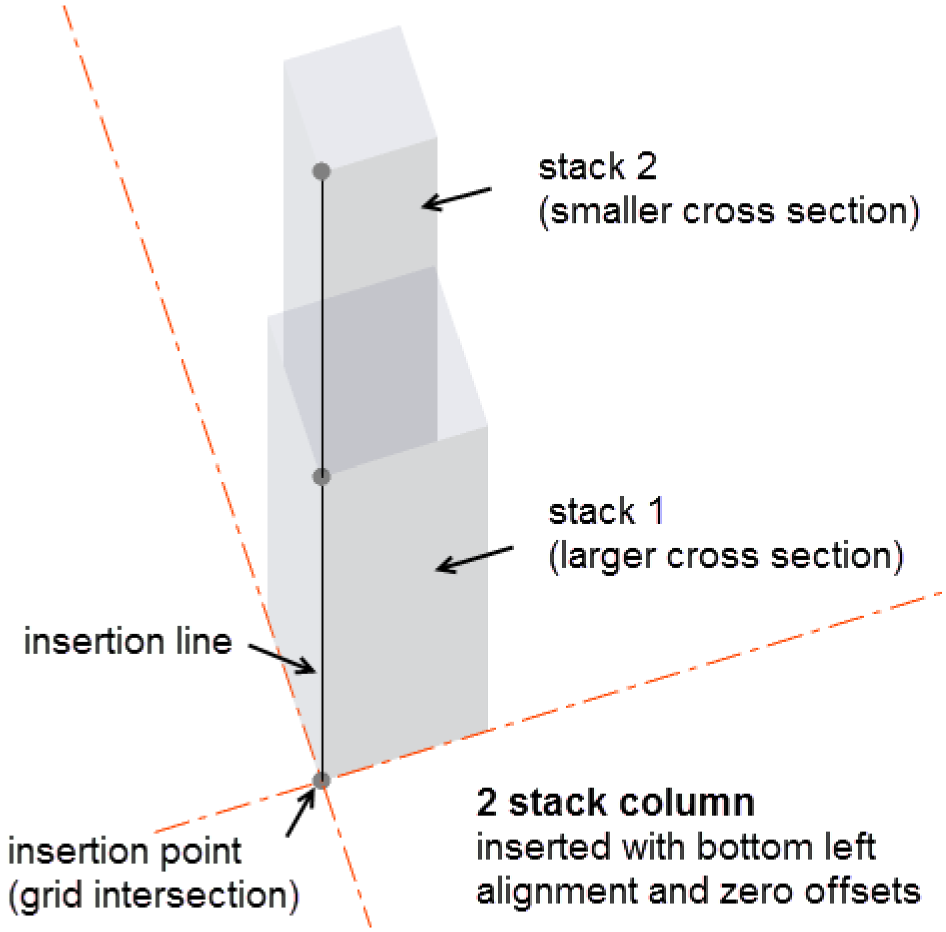
Click the below link to view a couple of examples illustrating the use of rigid zones in fixed and pin ended beams:

- Rigid zones examples (page 744)

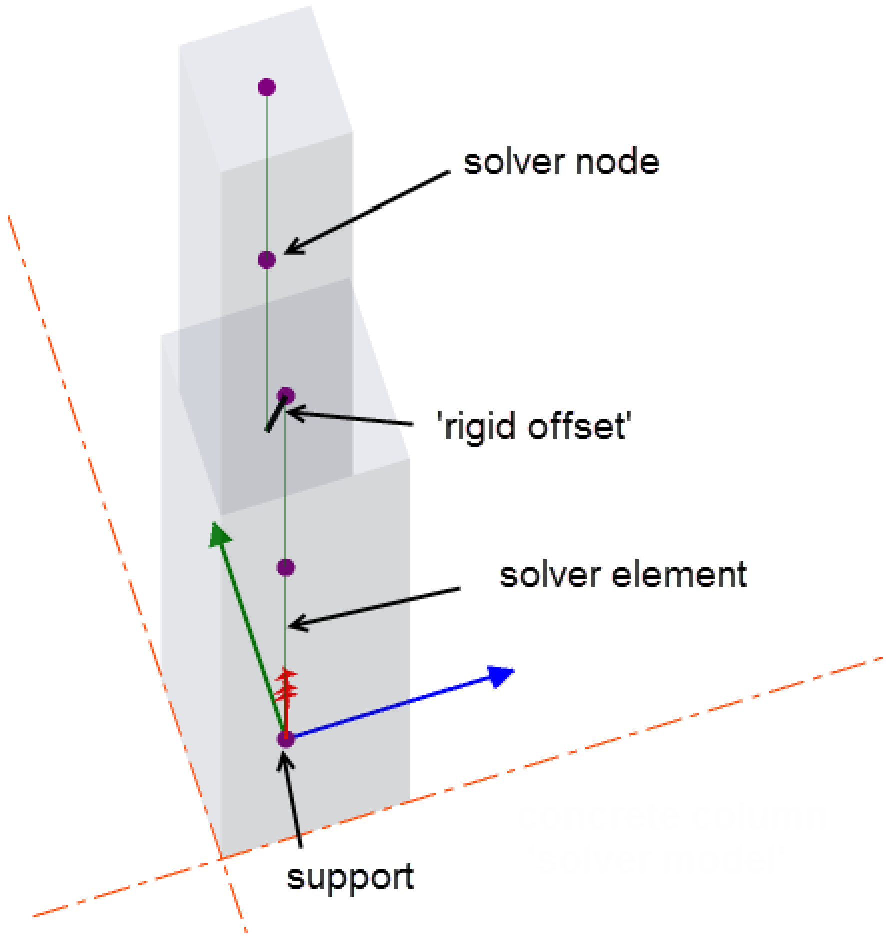
Rigid offsets examples

Rigid offsets example 1 - concrete column

Consider the two stack concrete column shown below - this has been inserted with its alignment properties set to bottom left so that the outer column faces remain flush despite a smaller section being introduced in stack 2.



Since solver elements for concrete columns always take into account any snap points or offsets, they will always be located at the centroid of each stack - thus they do not necessarily coincide with the insertion line used to position the column originally. In this example the centroid position shifts from one stack to the next which causes a “rigid offset” to be created automatically to connect the solver elements. Similar rigid offsets would also be created as required to connect incoming beams into the column centroids.

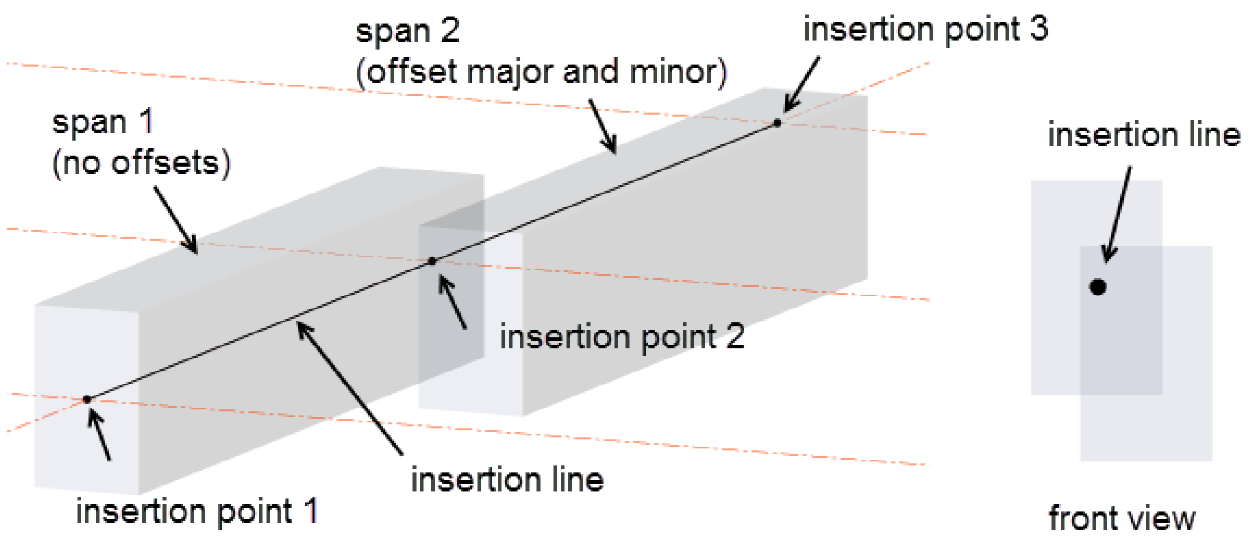


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

As a consequence of this method of modeling, you are freer to simplify the grid layout in order to create the structure more effectively, and then employ column offsets to position each column exactly, knowing that during analysis the program automatically assumes the column is located at its centroid as shown in the plan view.

Rigid offsets example 2 - concrete beam

Consider the two span concrete beam shown below - this has been inserted with both major and minor axis offsets applied to span 2 only.



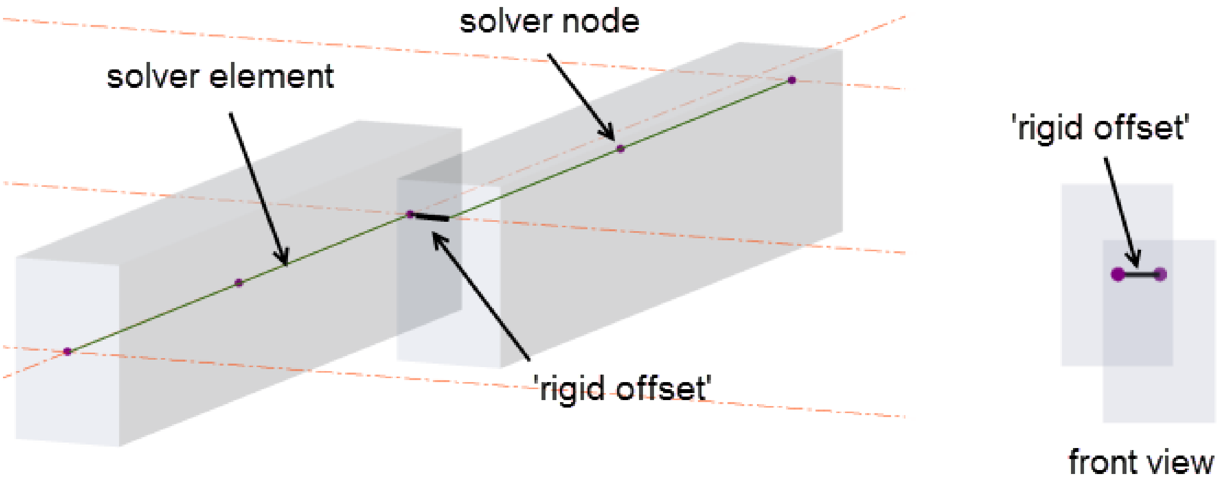
NOTE For concrete beams:

- The **minor** snap points and offsets **are** structurally significant and have an effect on the positioning of the 1D solver elements.
- The **major** snap points and offsets **are not** structurally significant.

In the minor direction beam solver elements are always located at the center of each beam section - as beam widths or minor offsets may vary, this may result in the introduction of lateral rigid offsets to make the connection between spans.

In the major direction beam solver elements are always created at the same level as the insertion line used to position the beam.

Consequently, for this particular example a lateral rigid offset is required to make the connection between the two spans.

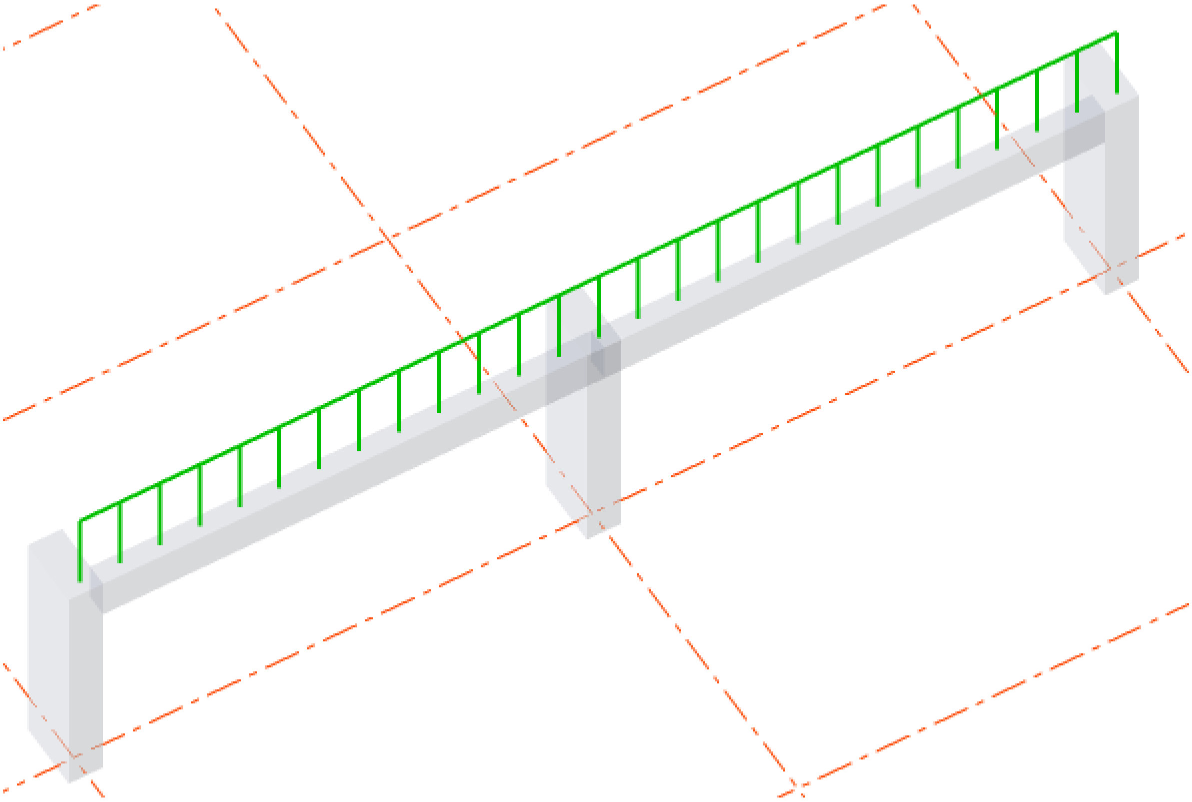


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

Rigid zones examples

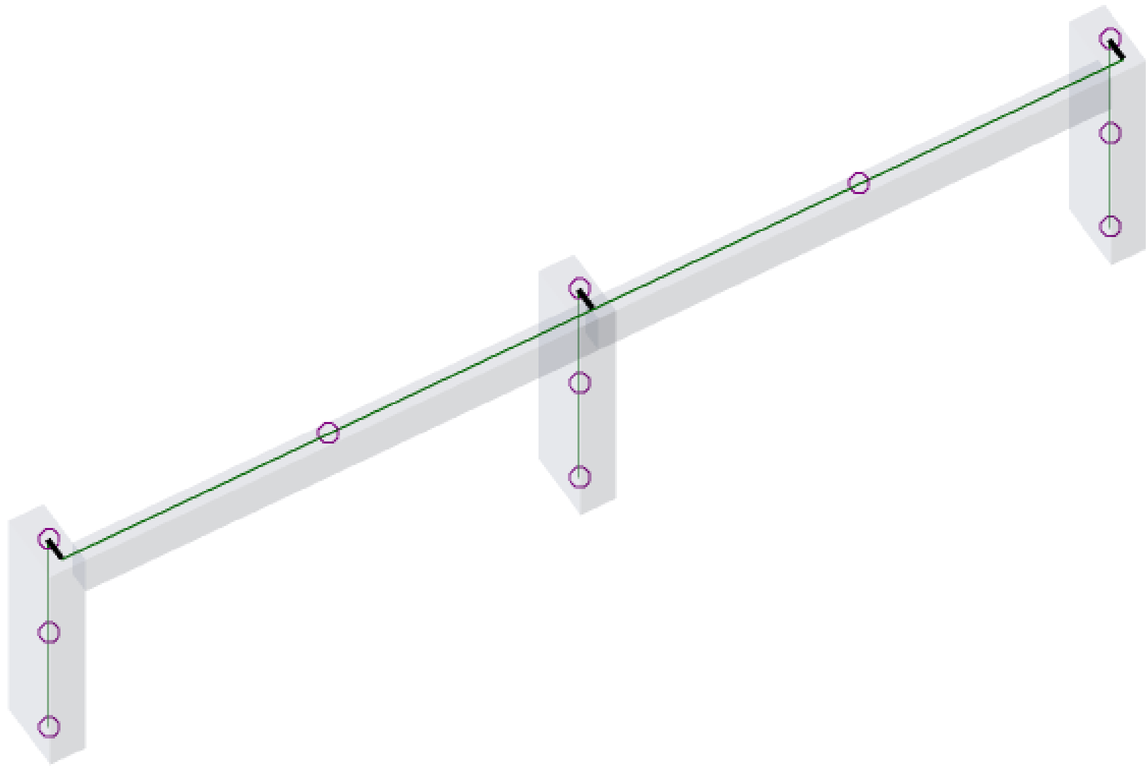
Rigid zones example 1 - fixed ended beam

Consider the following 2 span beam supported on columns and loaded with a udl:

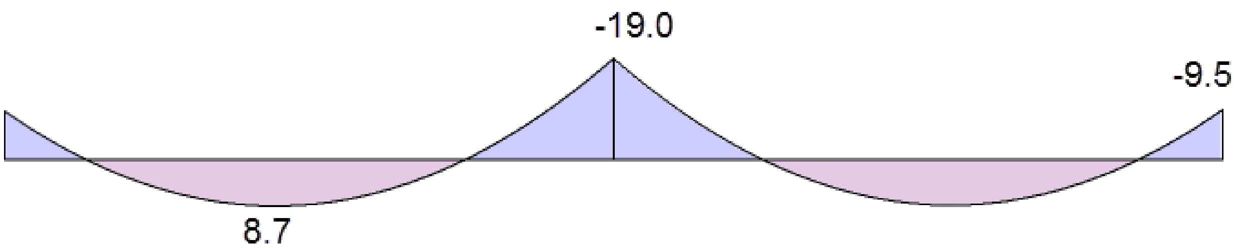


Rigid Zones Not Applied

The analysis model is simply constructed from the solver elements with rigid offsets applied as necessary to connect the beam solver elements to the column solver elements.



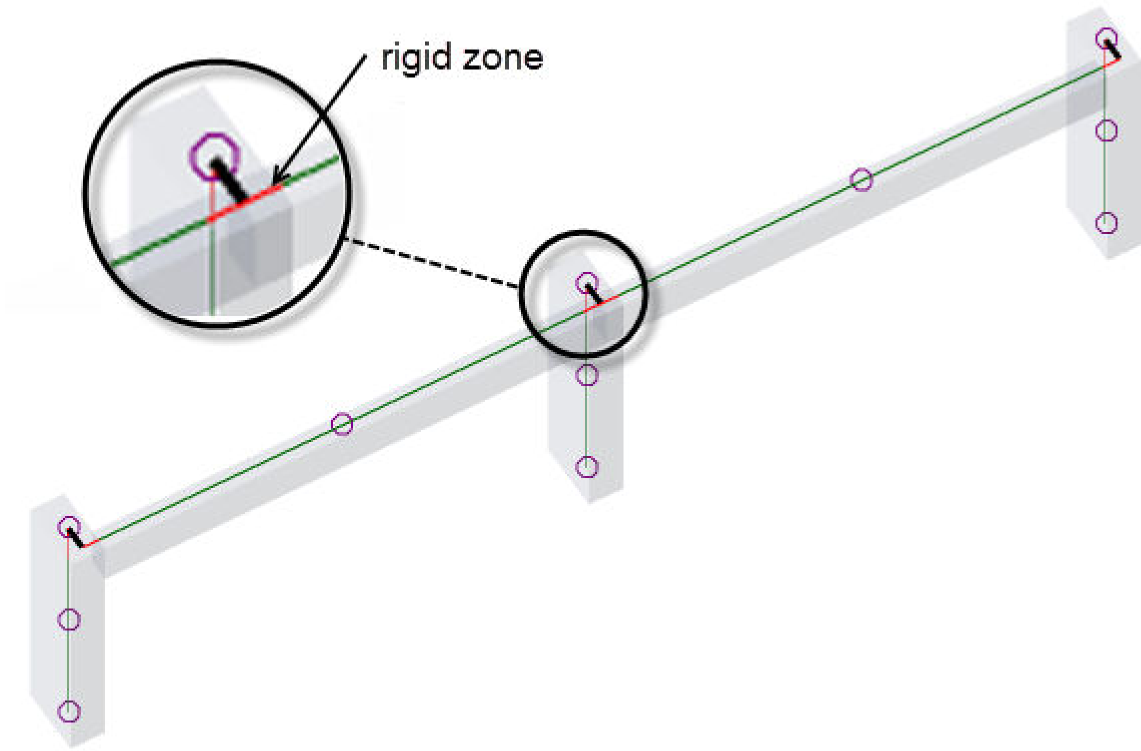
The resulting beam bending moment diagram is as follows:



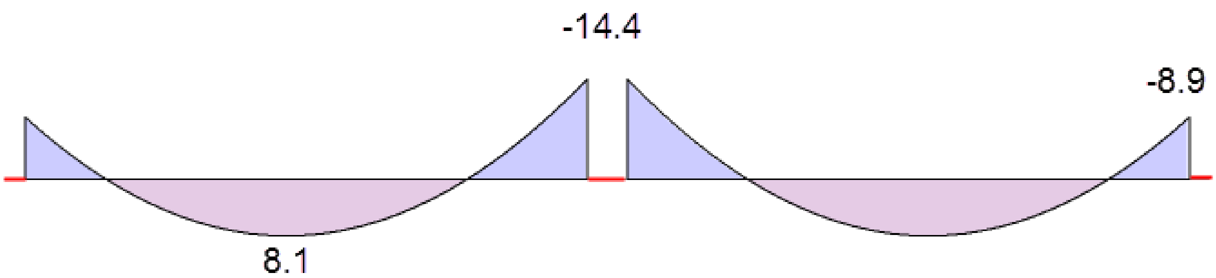
Rigid Zones Applied (default 50%)

Rigid Zones are activated via Model Settings, and this is also where the percentage of rigidity of the zones can be specified. Initially for this example it will be left at the default of 50%.

The revised solver model is as shown below, note the rigid zones that have been formed where the columns and beams connect:



The beam bending moment diagram for the revised model is as shown below.

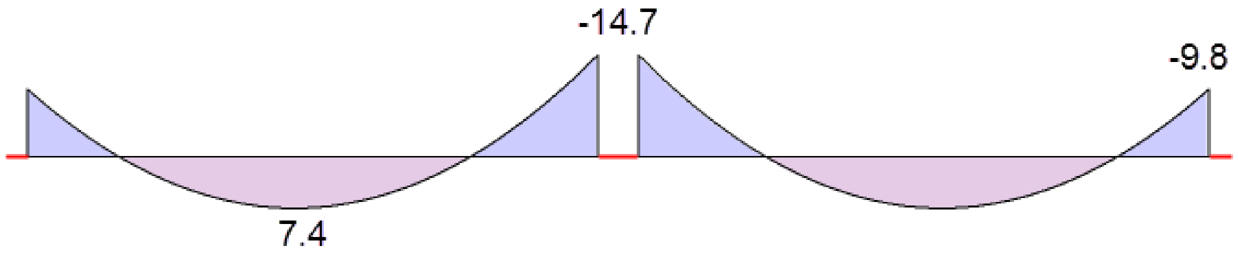


NOTE The above diagram was generated from a Results View to illustrate that a "gap" is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

We might expect the extra stiffness introduced at the supports to increase the hogging moments and reduce the sagging moments, however because the element end forces are now reported at the rigid zone boundaries (rather than the ends of the solver elements) - in this example the main effect is that the hogging moment over the central column support is substantially reduced.

Rigid Zones Applied (100%)

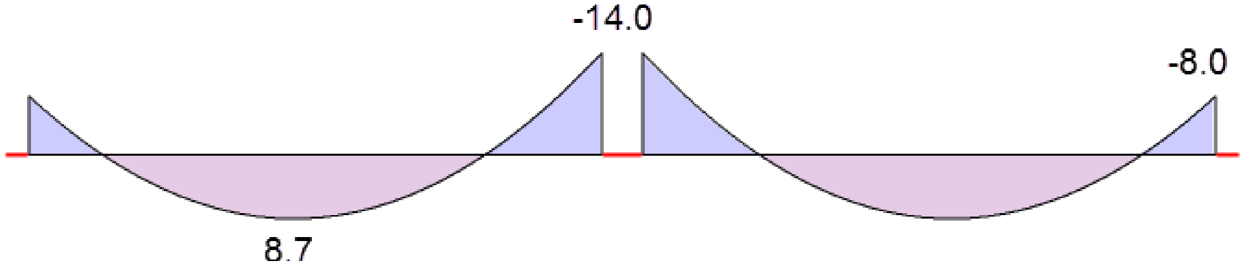
To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:



As expected the extra stiffness introduced at the supports increases the hogging moments and reduce the sagging moments in comparison to the run at 50%.

Rigid Zones Applied (0%)

If the percentage rigidity is reduced to 0% the bending moment is as shown below:



If this result is compared to the run in which rigid zones were not applied, it is clear that although the sagging moments are identical, the hogging moments that are reported are not the same. This is because, although the total elastic length of a member is the same in the two models, the position of the start and end of design members is different (being taken at the rigid zone boundaries when applied).

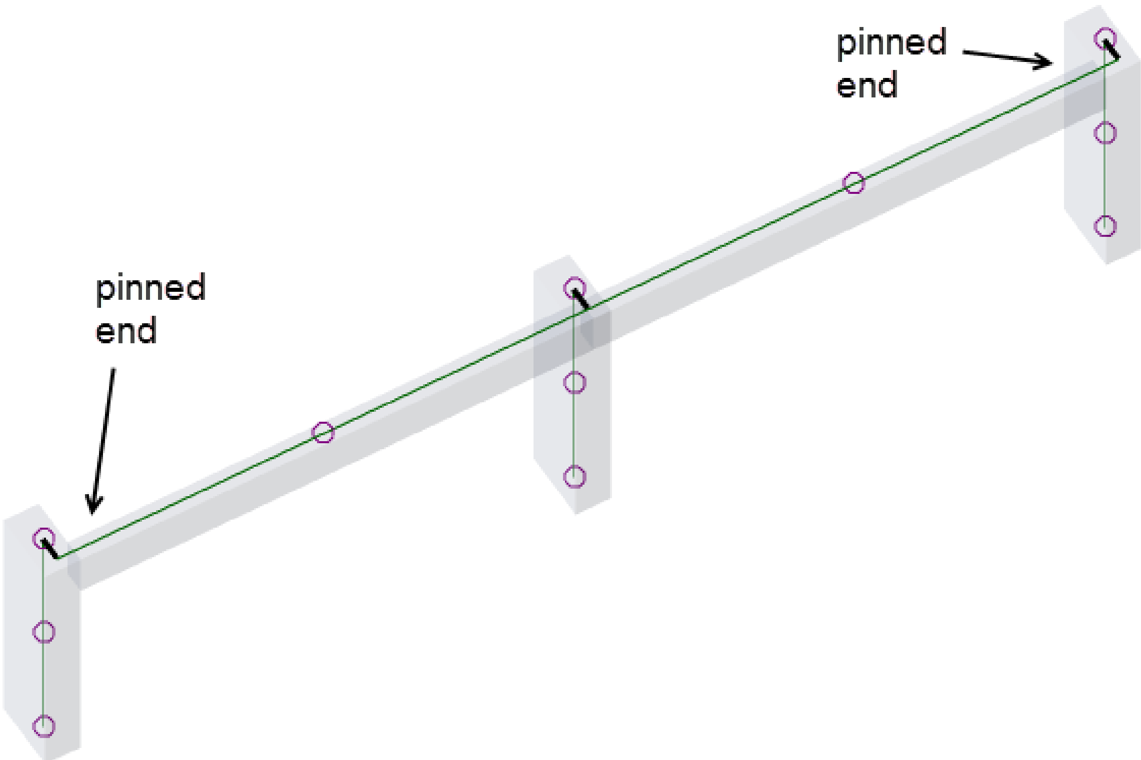
Rigid zones example 2 - pin ended beam

When rigid zones are applied to a pin ended member, the end release is shifted from the end of the solver element to the rigid zone boundary.

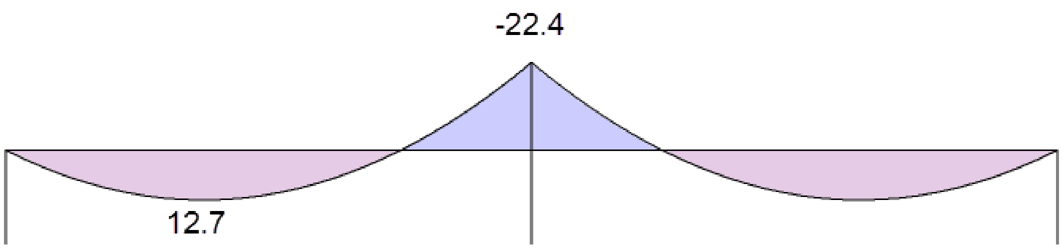
To illustrate this effect the model illustrated in Rigid zones example 1 is modified to have pinned connections introduced at the two remote ends of the beam.

Rigid Zones Not Applied

The analysis model is constructed from the solver elements with rigid offsets applied to connect the beam and column solver elements. Releases are formed at the two remote ends of the beam solver elements.



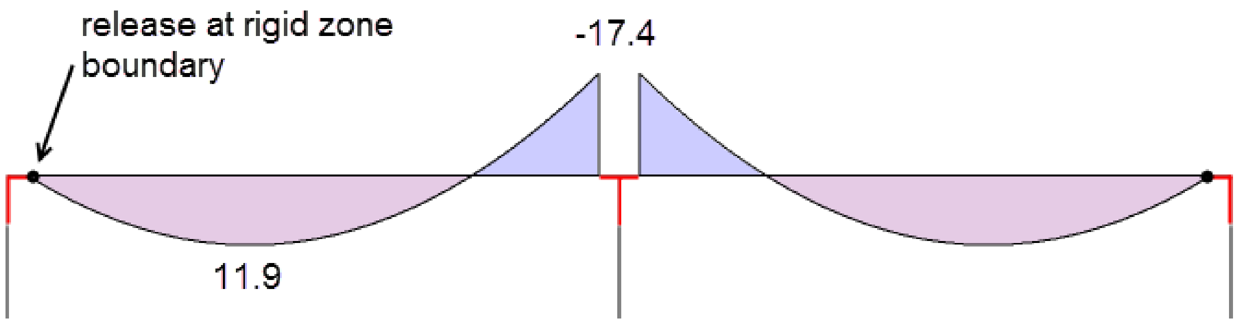
The beam bending moment diagram is as follows:



Rigid Zones Applied (0%)

For comparison, rigid zones are then introduced, (with 0% rigidity in order to keep the total elastic length of the beams the same in both models).

Because the releases are moved to the rigid zone boundaries, this has the effect of reducing the moments in the beams.



How meshed walls are represented in solver models

Meshed walls have isotropic stiffness properties and resist loads in all directions.

Wall beam elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For meshed walls, they are generated along the top of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected. A wall beam element is also generated along the bottom edge of the wall if Generate Support is not selected.

Sloping wall beam elements can be generated by sloping top or bottom edges or connected sloping slabs.

Where horizontal wall beam elements are required, they are generated across the entire width of the wall at that level.

Wall beam elements can also be created where certain properties, (e.g. thickness), differ across panel boundaries.

2D solver elements

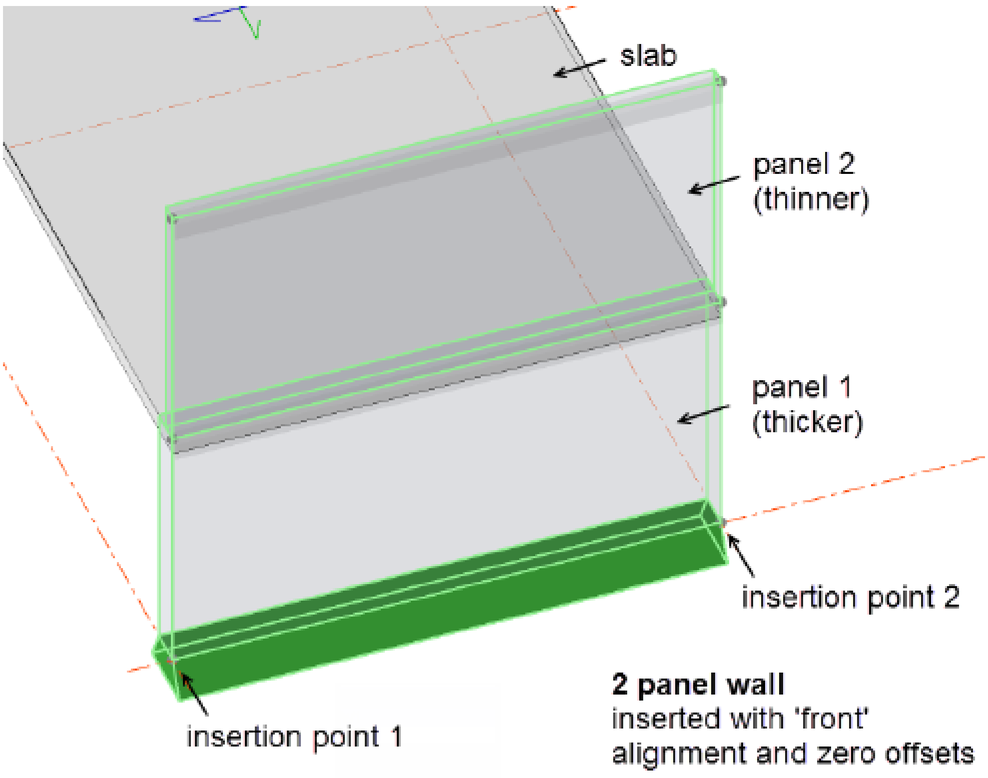
For meshed walls the type of 2D solver element used will depend on whether the wall mesh type is set to Quad only, Tri only, or, Quad dominant.

Modification Factors

Different modification factors applied to meshed walls in the analysis depending on the Material type that has been applied.

Meshed concrete wall example

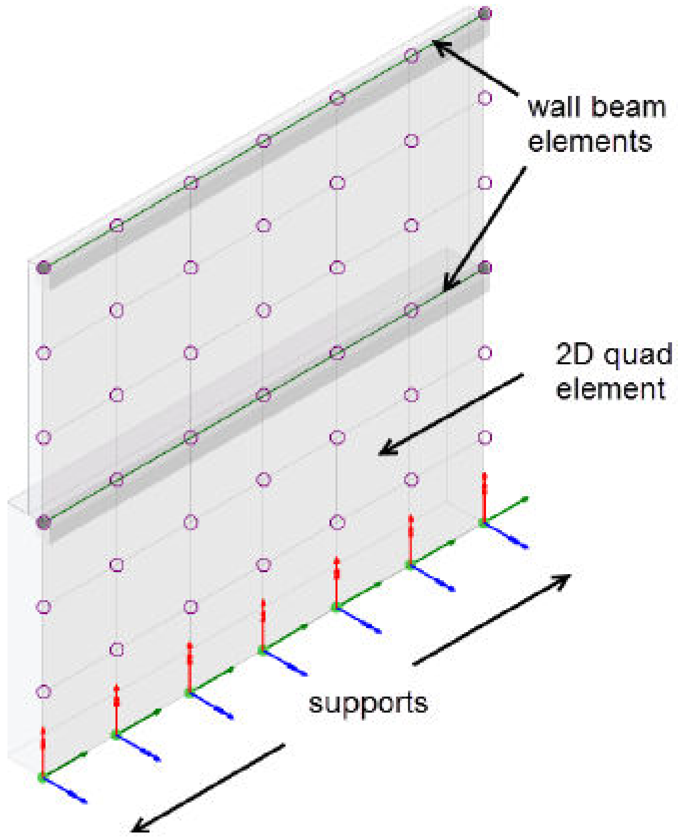
Consider the following two stack wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



1D and 2D solver elements for each wall panel are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified.

Quad only

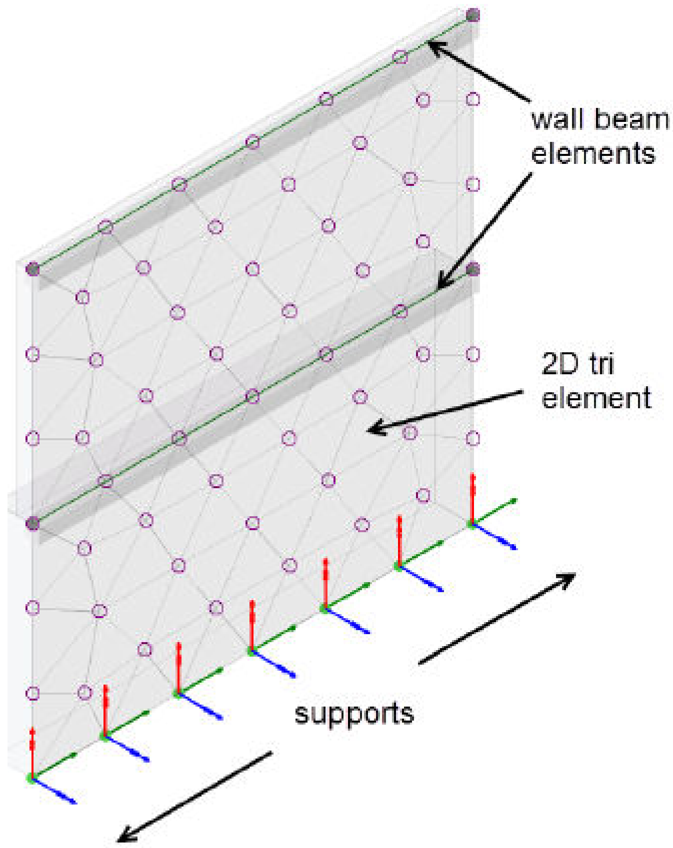
In this two stack example, when the Wall Mesh Type is set to Quad only, solver elements are formed as shown below:



NOTE To see solver elements, solver nodes and 2D elements: open a Solver View, and then in Scene Content check 1D Elements> Geometry, 2D Elements> Geometry and Solver Nodes> Geometry.

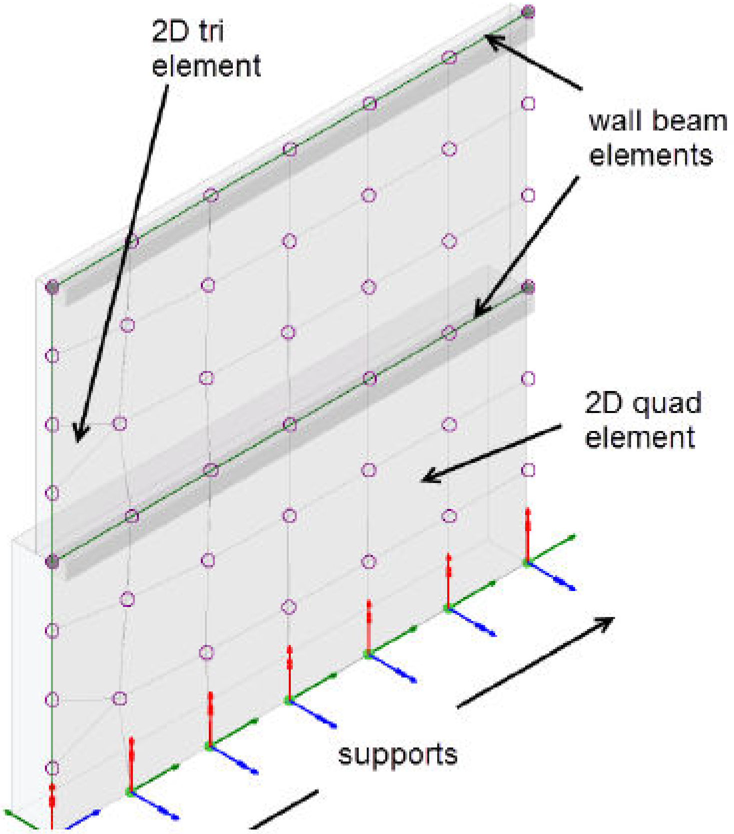
Tri only

In this two stack example, when the Wall Mesh Type is set to Tri only, solver elements are formed as shown below:



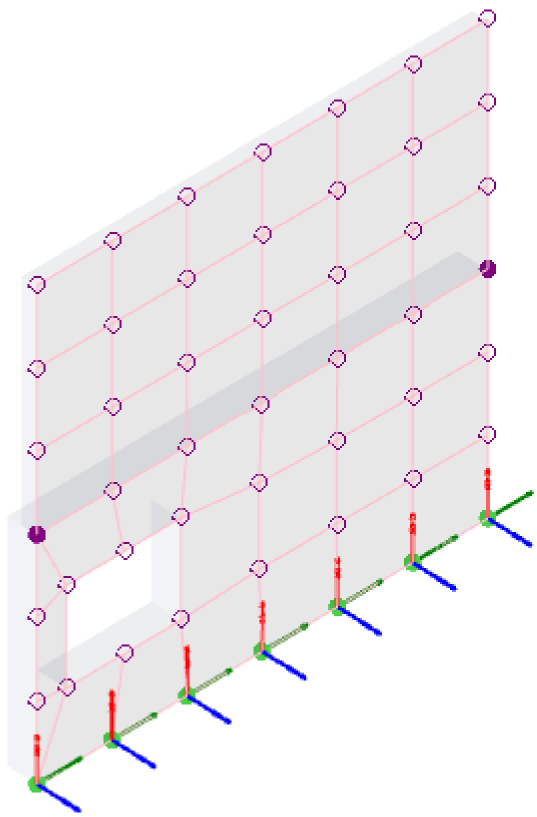
Quad dominant

In this two stack example, when the Wall Mesh Type is set to Quad dominant, solver elements are formed as shown below:



Wall openings

When wall openings are introduced, the mesh will adjust to form around the openings.



See also: Meshed wall openings analysis model (page 432)

How mid-pier walls are represented in solver models

Wall beam and wall column elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For mid-pier walls, they are generated along the top and bottom edges of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected.

NOTE Only horizontal wall beam elements can be generated in mid-pier walls - sloping wall beam elements cannot be generated - this will be indicated by an error in validation.

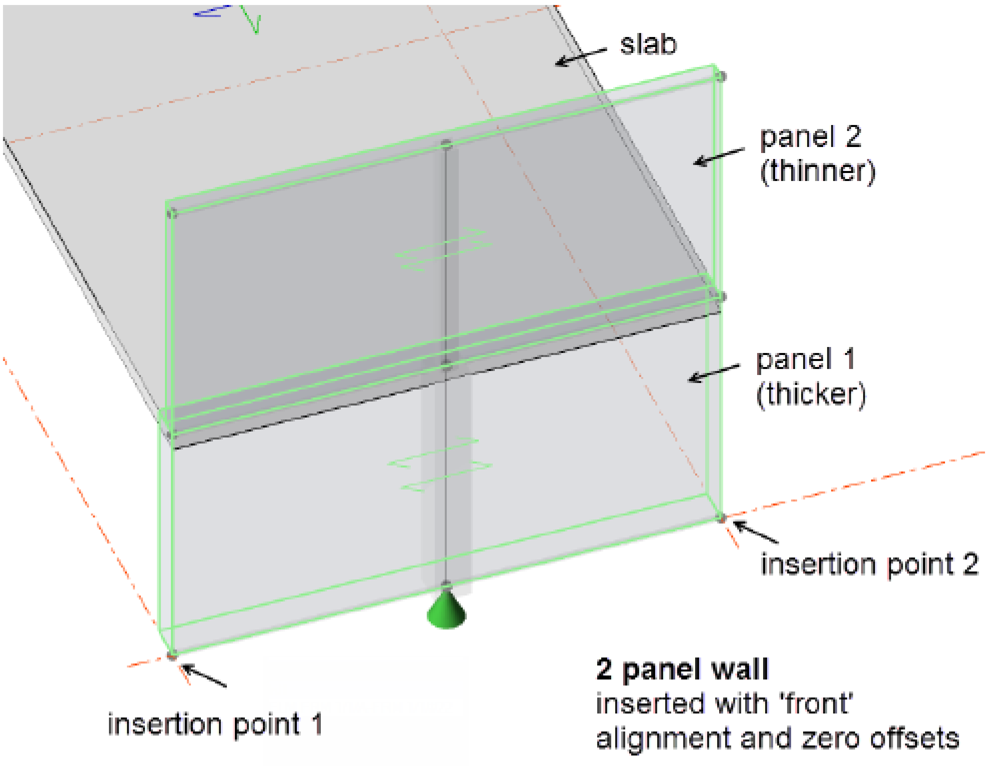
Each mid-pier wall object also has a single vertical wall column element in the middle of the wall, from the top to the bottom level.

Modification Factors

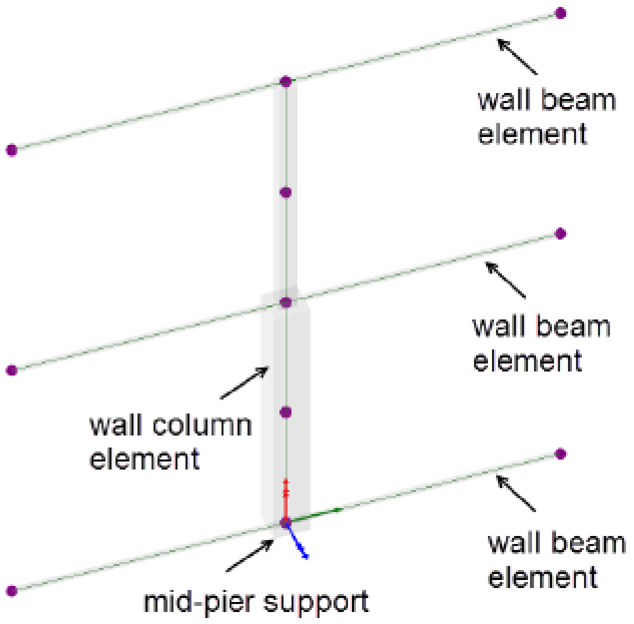
The modification factors applied to concrete walls in the analysis depend on whether they have been specified as meshed or mid-pier.

Mid-pier wall example

Consider the following two stack mid-pier wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



The wall beam and wall column elements are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified, so for this example, the elements are formed as shown below:



NOTE To see wall beam elements, wall column elements and solver nodes: open a Solver View, and then in Scene Content select 1 D Elements> Geometry and Solver Nodes> Geometry.

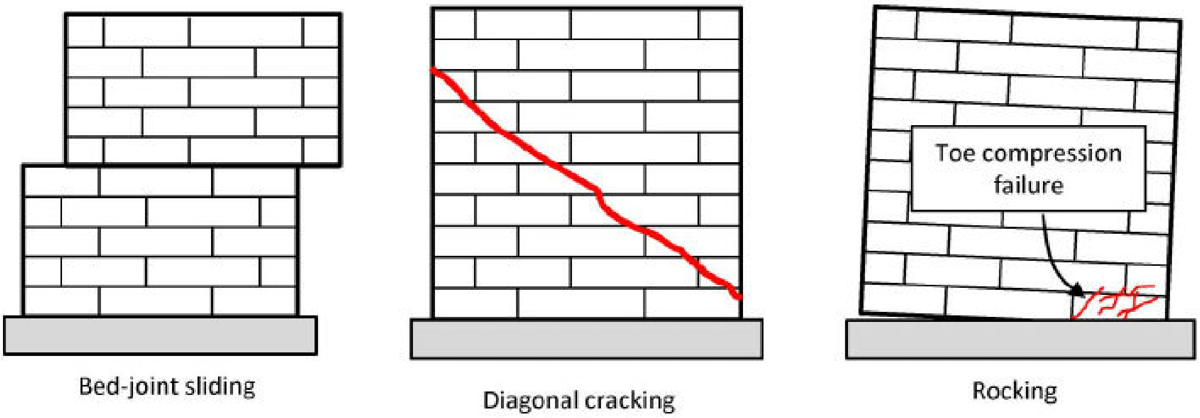
How shear only walls are represented in solver models

Shear only walls resist in-plane shear only and have no out of plane stiffness or load bearing resistance.

Background

The 'shear only wall' is configured to allow, as the name implies, shear forces only to be resisted. These arise from frame action in the lateral load resisting system when subject to lateral loads. The behaviour is typical of unreinforced masonry walls built into a steel or concrete frame. Under lateral loading the top of the column adjacent to the wall panel bears on a relatively short length of wall, creates compression in some width of masonry in the diagonal and exits the wall panel at the toe of the opposite corner.

A masonry wall panel resists this force couple at each corner primarily as a shear panel. The failure mechanism either follows a stepped pattern through the joints or by shear bond along the bed joints. The compression strut manifests as a failure in diagonal tension across the bed joints in a stepped fashion. Failure can also occur by local crushing in the top left or bottom right of the wall (or vice versa). These failure modes are depicted in the figure below.



The third failure mode mentioned, crushing in the corners, is very difficult to model and is believed to have a minor influence on overall behaviour. It is thought to be self compensating to some extent because when the masonry begins to crush more length of wall is brought into play.

The diagonal cracking and sliding in the bed joint are the primary effects of the wall acting as a shear panel. This is the only behaviour that 'shear only walls' are able to model.

The compression strut that is also inferred by the diagonal cracking generates push-pull forces in the frame and complementary axial forces in the beams. These are incorporated in the Tekla Structural Designer implementation by the use of special 'Link Elements'.

The oft used and simplest model of a single brace or pair of braces from corner to corner of the wall panel can adequately represent the lateral stiffness of the infilled frame but introduces unwanted axial forces (from gravity loads), particularly in columns. The Tekla Structural Designer implementation is a significant improvement on this simplest model and requires only the determination of the spring stiffness associated with the shear behaviour.

Crisafulli (2007)**[1]** provides a formula for the stiffness, k_s, of the shear spring as,

k_s	=	γ_s x A_{ms} x E_m/d_m x cos2θ
Where,		
A_{ms}	=	total area of equivalent strut based on a width of strut of the order of ¼ to ⅓ of the diagonal length of the panel

E_m	=	Elastic modulus of the masonry
d_m	=	diagonal length of the wall panel
θ	=	the angle of the 'strut' with the horizontal

The factor γ_s is the proportion of total stiffness that is assigned to the spring whilst the remainder is provided in the Crisafulli model by a pair of 'masonry struts'. In Tekla Structural Designer γ_s is 1.0 i.e. all of the stiffness is provided by the spring whilst the force effects of the 'masonry struts' are replicated by the special 'Link Element'.

Other formulations for the spring stiffness are likely to exist in the literature.

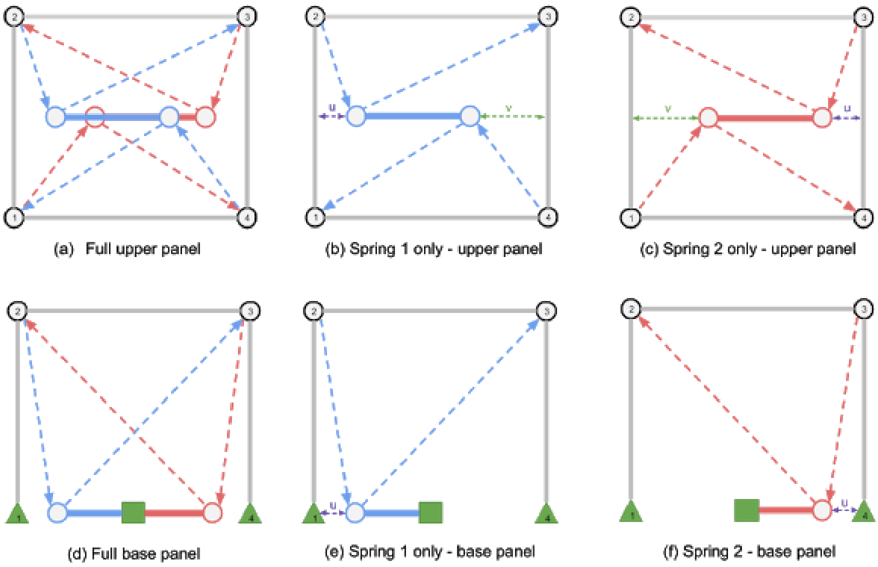
In Tekla Structural Designer, the spring is a either a linear or non-linear uniaxial spring, the former being used when all diaphragms are 'Rigid'.

One of the consequences of the configuration of 'shear only walls' adopted in Tekla Structural Designer is that no loads can be applied out of plane and no members can be connected into the main body of the wall panel. Similarly, the wall panel must be completely surrounded by column and beam members to ensure transfer of lateral loads. For this type of shear wall the head detail is assumed to be such that there is no load transfer from the beam above to the head of the wall.

1 *Crisafulli F. J. and Athol J. C., Proposed macro-model for the analysis of infilled frame structures, Bulletin of the New Zealand Society for Earthquake Engineering, Vol. 40 No. 2 June 2007.*

Solver model in Tekla Structural Designer

Shear only wall panels are modelled using two axial springs between 'panel nodes' connected to 'corner nodes' by link elements. How these are configured differs for interstory panels and the base panel.



Configuration of shear only walls for interstory and base panels. (a) & (d) show the full panel, (b) & (e) show the right to left spring assembly, (c) & (f) show the left to right spring assembly.

NOTE The axial spring and link elements are only shown in the Solver View used for analysis. The above illustrations are not to scale, the actual u and v dimensions being 10mm (25/64 in.) and 20mm (50/64in.)

Interstory panels

For each interstory panel, two springs, each with a pair of nodes are created and connected to the 'corner' nodes where the panel connects to the beam-column node. The connection is made using four special 'Link Elements'. The orientation of the axial spring means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Elements coordinate systems and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces.

Base panel

Where a base panel exists, a single fixed base is created. Two springs are created at the same level, and connected to the 'corner' nodes where the panel connects to the beam-column node using two Link Elements. The orientation of the axial springs means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Element coordinate system and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces. Only horizontal reaction is produced at the support and this must be distributed manually along the wall footing if required.

Walls supported on meshed slabs and foundation mats are treated as base panels with the springs and consequent forces applied to a 'seeded' node in the mesh.

Self weight

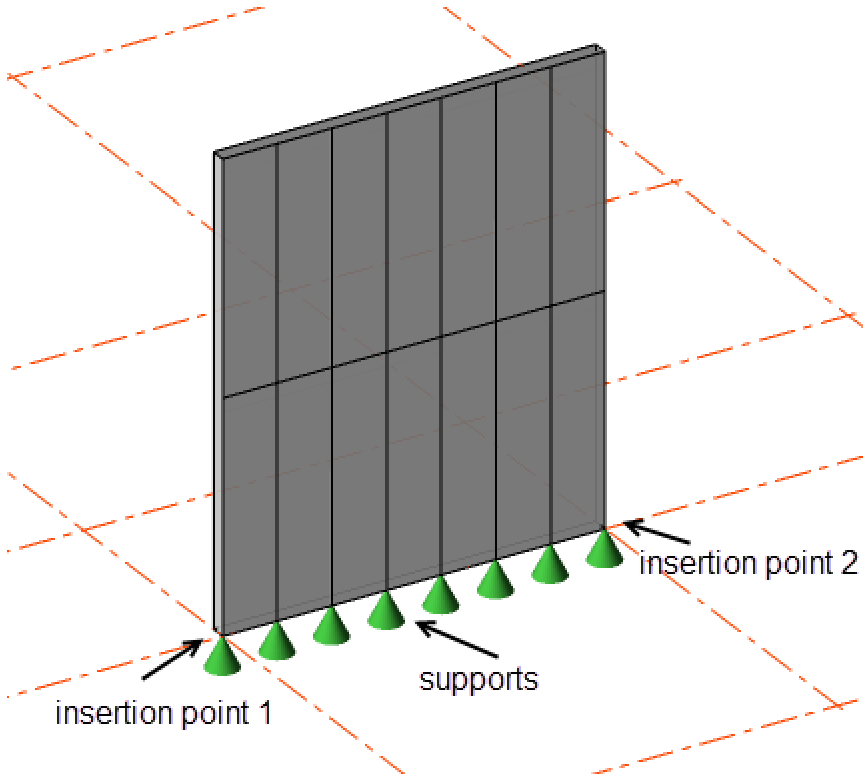
The self weight of each panel is automatically calculated by Tekla Structural Designer and applied to the supporting beam. For a base panel this applied directly to the wall support. For meshed slabs and foundation mats the wall is treated as a base panel and the self weight is applied to a 'seeded' node in the mesh.

How bearing walls are represented in solver models

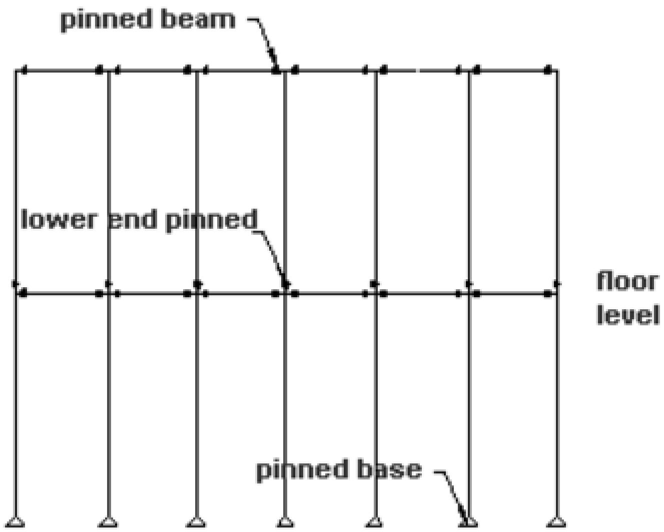
Bearing walls have no in-plane or out-of-plane stiffness and resist vertical load only.

For bearing walls the alignment (Front, Middle, or Back) specified in the wall properties is not structurally significant as it has no effect on the positioning of the solver elements in the solver model.

Consider the two stack bearing wall shown below.

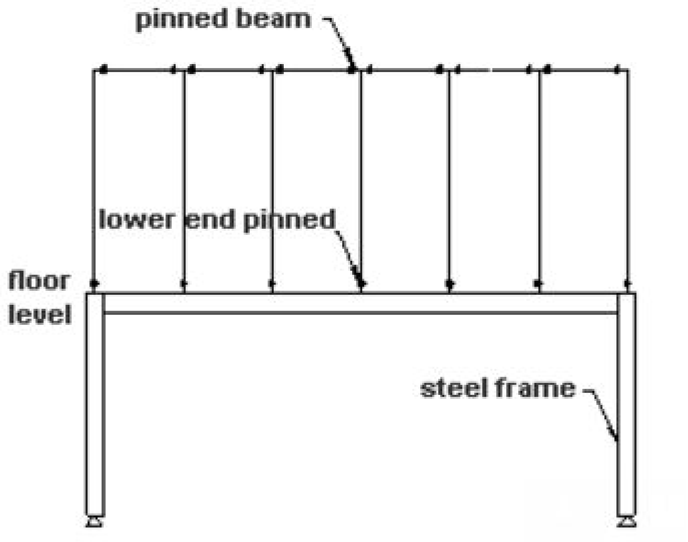


The solver model for this wall is formed using a series of vertical "wall column" and horizontal "wall beam" solver elements. The beams have pinned ends and are placed at the top of the wall spanning between the columns. The next panel above is pinned to the one below and similarly the lower end of a column is pinned to a supporting beam. At the lowest level the column is 'fixed' to a pinned support.

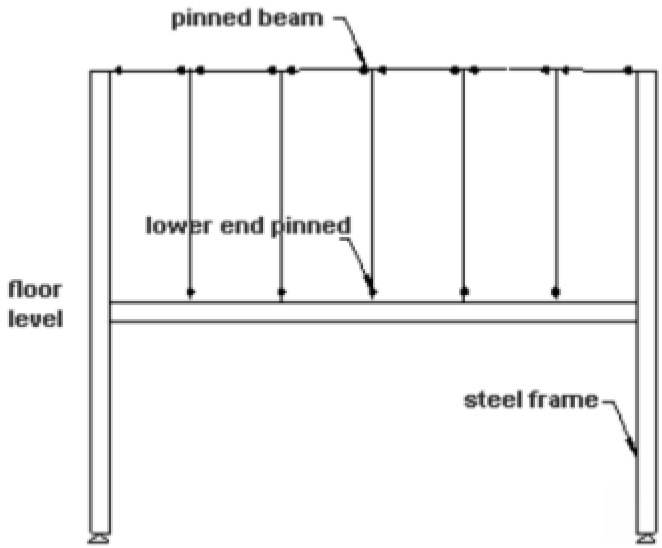


Members supported by the wall either (fortuitously) bear directly on one of the wall columns or on one of the wall beams at the head of the wall. All wall columns and wall beams in an individual bearing wall are given properties automatically by Tekla Structural Designer, based on the width of the bearing wall with which they are associated.

If the bearing wall did not continue to the lowest level, but was instead supported by a transfer beam, then at the lowest level the wall columns would have pinned ends and no supports would be introduced.



For bearing walls that are defined between other vertical column members e.g. steel columns, the wall columns at the edge of the panel are omitted and the associated wall beam is connected to the steel column (for example) and the adjacent wall column - as below.



Wall columns at the edge of the panel are also omitted when it is defined between concrete walls.

Irrespective of whether the wall spans between other vertical column members or not - any load applied to the wall beam at the edge of the panel is shared between the end column and the first internal column. This can result in some load being `lost' directly into the supports.

Load transfer in the bearing wall model is not the same as it would be in for example, a masonry wall. A point load applied at the top of a masonry wall would result in a distributed load on any beam supporting the masonry wall, whereas in a bearing wall the supporting beam would be subjected to a pair of point loads, (or possibly even a single point load if the applied load coincides exactly with a wall column location).

Self weight of the bearing wall is concentrated in the wall beams so seismic weight is concentrated at the top of the wall and not split between the floor above and below.

View tabular solver model data and results

Solver model data views can be used to view tabulated node and element data and specific results for the different solver models.

The full list of tabular data view types is shown below.

NOTE Some of the view types are only available for specific solver model types.

- View solver model object properties (page 734)
- View tabular results for support reactions (page 765)
- View tabular results for nodal deflections (page 765)
- View tabular results for solver element end forces (page 766)
- View tabular results for wall lines (page 767)
- View tabular results for result lines (page 767)
- View tabular results for core lines (page 768)
- View the summed mass for modal mass combinations (page 768)
- View the dynamic masses for modal mass combinations (page 769)
- View active masses by node (page 769)
- View modal frequencies and modal masses (page 770)
- View buckling factors (page 770)

View tabulated solver node and element data

To view node coordinate and degrees of freedom, or solver element properties in tables, see the following instructions.

View tabulated node coordinates and degrees of freedom

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the solver model result type required.
3. In the **View Type** list, select **Nodes**.

   Tekla Structural Designer displays the node coordinates and degrees of freedom in a table.

View tabulated solver element properties

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select the solver model type required.
3. In the **View Type** list, select **Elements**. Tekla Structural Designer displays solver element properties in a table.

View tabular results for support reactions

In order to view the results for support reactions, or nodal forces, in tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view support reactions for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis
3. In the **View Type** list, select **Nodal Forces**.
4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope.

View tabular results for nodal deflections

To view the deflections of the nodes in your model as tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view nodal deflections for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Nodal Deflections**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope.

View tabular results for solver element end forces

To view the raw results for element end forces in tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view solver element end forces for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Element End Forces**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope.

   Tekla Structural Designer displays the raw analysis results of element end forces. This means that no axial load reductions have been applied.

   NOTE The asterisk next to certain element numbers signifies that the results are actually output at the end of a rigid arm that you have modeled, rather than at the node itself.

View tabular results for wall lines

To view the results of the wall lines in your model as tables, see the following instructions. The result tables include information on such aspects as the location, position, axial force, and torsion of the wall lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view wall line forces for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Wall Lines**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for result lines

To view the results of the result lines in your model as tables, see the following instructions. The result tables include information on such aspects as the location, position, axial force, and torsion of the result lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired analysis type.

   NOTE You can view 2D result lines forces for all analysis types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select 2D **Result Lines**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for core lines

To view the results of the core lines in your model as tables, see the following instructions.

The result tables include information on such aspects as the location, position, axial force, and torsion of the wall lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired analysis type.

   NOTE You can view core line forces for all analysis types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

   ---

3. In the **View Type** list, select **Core Lines**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for mode shapes

To view the mode shapes analyzed in a 1st order modal or 2nd order buckling analysis, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the solver model type.

   NOTE You can only view mode shapes for the following solver model types:

   - 1st order modal analysis
   - 2nd order buckling analysis

   ---

3. In the **View Type** list, select **Mode Shape**.

4. In the **Loading** list at the bottom of the window, select the desired load case, combination, or envelope.

5. In the lower list of the **Result Type** group, select the mode.

View the summed mass for modal mass combinations

In order to view the summed total mass of the modal mass combinations in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Summed Mass**.

   Tekla Structural Designer displays the total mass of the modal mass combinations in a table.

View the dynamic masses for modal mass combinations

To view the total dynamic masses for modal mass combinations, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Dynamic Masses**. Tekla Structural Designer displays the total dynamic masses for the modal mass combinations in a table.

View active masses by node

To view the dynamic active masses of the nodes in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Active Masses by Node**.

   Tekla Structural Designer displays the dynamic active masses of the nodes in your model.

View total masses by node

To view a table of the total masses of the nodes in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Total Masses by Node**.

   Tekla Structural Designer displays the dynamic total masses of each node in your model in a table.

View modal frequencies and modal masses

In order to view the modal frequencies and modal masses of a particular load combination, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Modal Frequencies**.
4. In the **Loading** list, select the desired load combination.

   Tekla Structural Designer displays the modal frequencies and modal masses of the selected load combination in a table.

View buckling factors

To view the buckling factors analyzed in the 2nd order buckling analysis in a table, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of **Result Type** group, select **2nd Order Buckling**.
3. In the **View Type** list, select **Buckling Factors**.
4. In the **Loading** list, select the desired load case or combination.

7 Design models

From the **Design** toolbar you can batch design all the members and walls in your model and (separately) batch design all the slabs. From the same toolbar you can also check the response of floors to dynamic excitation.

As an alternative to batch design you might choose to selectively design parts of the model as required.

To get to know about combined analysis and member design and selective member design, see:

- Design steel members and cast-in-place concrete beams, columns and walls (page 772)

To design slabs, see

- Design slabs and run punching shear checks (page 789)

If working to either the Eurocode or US headcode, you may also want to introduce more/less conservatism to the design process by specifying user defined utilation ratios:

- Apply user defined utilization ratios (page 786)

To check the response of floors to dynamic excitation, see:

- Create and run floor vibration checks (page 806)

NOTE Foundation design topics are covered separately, see Create and design foundations (page 836)

Certain member types can be designed using Tekla Tedds, see:

- Design timber and precast members using Tekla Tedds (page 805)

See also

Concrete member and slab design handbook (page 1274)

Steel member design handbook (page 1203)

Vibration of floors to DG11 (page 1843)

Vibration of floors to SCI P354 (page 1989)

7.1 Design steel members and cast-in-place concrete beams, columns and walls

Before commencing the design you should take a moment to ensure design options and autodesign settings are set as required. If working to either the Eurocode , US, or India headcode, you may also want to introduce more/less conservatism to the design process by specifying user defined utilation ratios.

- Apply and modify design options (page 772)
- Autodesign versus check design (page 773)
- Apply user defined utilization ratios (page 786)

Having set the properties and options as required you can then proceed to rapidly assess all steel members and/or concrete members/walls in your model:

- Combined analysis and member design

Alternatively, you may find it more efficient to run an analysis in the first instance and then:

- Check selected members and walls (page 779)
- Design selected members and walls (page 782)

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see Seismic design methods (page 1199) in the Seismic analysis and design handbook.

See also

Concrete member and slab design handbook (page 1274)

Steel member design handbook (page 1203)

Design timber and precast members using Tekla Tedds (page 805)

Apply and modify design settings

The design options in Tekla Structural Designer allow you to adjust the way in which your model is designed.

The different Design Settings (page 2293) sub pages enable you to, for example:

- set which analysis type is run for the design, (on the Analysis sub page)
- set reinforcement and other parameters for concrete design, (on the Concrete sub page)
- set the levels at which forces can be ignored, (on the Design Forces sub page)

- set which members to design using groups, (on the Design Groups sub page)
- control how Autodesign settings will be reset after the design, (on the Auto design sub page)

Modify design settings in the current project

1. On the **Design** tab, click **Settings**.
2. Modify the Design Settings (page 2293) according to your needs.
3. After making the changes, do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes as defaults for future projects, click **Save...**

   TIP To revert the analysis options specified to the default design options of the active settings set, click **Load...**

Modify design settings defaults for future projects

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Go to the Design Settings (page 2293) page.
4. Select the settings set that you want to modify.
5. Modify the settings according to your needs.
6. To save the changes to the settings set, click **OK**.

See also

Design Settings (page 2293)

Autodesign versus check design

Before you begin the design process you should select a suitable design mode for each member in your model. The available design mode options are check design mode and autodesign mode.

Each member in your model is set to one of two design modes:

- Check design mode:

  You assign the desired section size (for steel members), or section size and reinforcement (for concrete members). Then, Tekla Structural Designer determines if the section or reinforcement is sufficient

- Autodesign mode:

  For steel members, you select the desired section type. Tekla Structural Designer then determines a suitable size for the selected section type.

  For concrete members, you assign the desired section size.Tekla Structural Designer then automatically determines a suitable reinforcement configuration.

TIP To quickly review and update the mode applied to all members in the model, open a review view and use the **Auto/Check Design** command to switch between the two modes.

NOTE The Autodesign setting is only considered when you choose to design multiple members, as explained below:

- If you choose to run a **combined analysis and design** (Design Steel, Design Concrete, Design All); or choose a **selective design** of multiple members (Design model, Design plane, Design Selection etc.) - members will either be designed or checked according to their individual 'auto-design' setting.
- If you choose a **check** (Check model, Check plane, Check Selection, Check Member etc.) - each member will be checked irrespective of its **'auto**-design' setting.
- If you **design** an individual member (Design Member, Design Wall etc.) - the member will be designed irrespective of its 'auto-design' setting.

See also

Select whether to design steel, concrete, or all (page 777)

Select between static and gravity design (page 778)

Combined analysis and member design

Run a combined analysis and design in order to rapidly check, or design; every concrete, steel, or, concrete and steel menber in the model for active gravity, static, or RSA load combinations.

Overview

Located on the **Design** toolbar, three groups of commands can be used for combined analysis and design as follows:

- **Design steel** - to design all steel beams and columns in your model.
- **Design concrete** - to design all concrete beams, columns and walls.
- **Design all** - to design all steel beams and columns, and concrete beams, columns and walls.

By selecting a **Gravity**, **Static**, or **RSA** option, you can control which classes of combination are considered in the analysis and design process.

At the end of the chosen process the **Review** tab and the review view open so that you can inspect the results

NOTE As design validation is performed automatically as part combined analysis and design, there is no requirement to run design validation manually beforehand.

NOTE In combined analysis and design each member is either designed or checked according to its individual 'auto-design' setting.

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see Seismic design methods (page 1199) in the Seismic analysis and design handbook.

Run Design Steel (Gravity)

- On the **Design** tab, click  **Design Steel (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel beams and columns in the model.

Run Design Steel (Static)

- On the **Design** tab, click  **Design Steel (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel beams and columns in the model.

Run Design Steel (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 579) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see Seismic design methods (page 1199) in the Seismic analysis and design handbook.

- On the **Design** tab, click  **Design Steel (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel beams and columns in the model.

Run Design Concrete (Gravity)

- On the **Design** tab, click  **Design Concrete (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all concrete beams, columns and walls in the model.

Run Design Concrete (Static)

- On the **Design** tab, Click  **Design Concrete (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all concrete beams, columns and walls in the model.

Run Design Concrete (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 579) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see Seismic design methods (page 1199) in the Seismic analysis and design handbook.

- On the **Design** tab, Click  **Design Concrete (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all concrete beams, columns, and walls in the model.

Run Design All (Gravity)

- On the **Design** tab, click **Design All (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel and concrete beams and columns and all concrete walls in the model.

Run Design All (Static)

- On the **Design** tab, click **Design All (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

Run Design All (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 579) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see Seismic design methods (page 1199) in the Seismic analysis and design handbook.

- On the **Design** tab, Click **Design All (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

Select whether to design steel, concrete, or all

The appropriate design command depends on the materials you have used in the model. For more information on selecting the appropriate command, see the following paragraphs.

In simple terms:

- If your model consists of steel members only, you can run the design by using the **Design Steel** commands.
- If your model consists of concrete members only, you can run the design by using the **Design Concrete** commands.
- If your model consists of a mix of both concrete and steel members, you can run the design by using the **Design All** commands.

TIP For structures that are mostly steel but have a few concrete members: instead of running Design All, you could run Design Steel (in order to focus on the steel design) before switching to Design Concrete for the remaining members.

In this way during the steel design phase you are not running grillage and FE chasedown analyses when they are not required .

For more details on how each command affects the analysis process, see the following:

Design steel

- Performs a 3D analysis.
- Does not perform grillage chasedown or FE chasedown analysis.
- Designs or checks all steel elements and shear walls.
- Does not design or check concrete beams or columns.

Design concrete

- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.
- Designs or checks all concrete beams, columns and shear walls.
- Does not design or check steel elements.

Design all

- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.
- Designs or checks all concrete beams, columns and shear walls.
- Designs or checks all steel elements.

See also

Select between static and gravity design (page 778)

Select between static and gravity design

If you are not sure whether you should use the **Design (Static)** or **Design (Gravity)** commands when designing the model, see the following paragraphs for more details on each option.

Design (Gravity)

Although your final design should consider all combinations, designing for gravity combinations can be a useful way to rapidly pre-size the members in the model that are not subjected to lateral loads.

Designing members for gravity combinations is more common for steel structures than concrete structures.

The gravity design commands involve the use of first order analysis on a limited set of design combinations as follows:

- **Design Steel (Gravity)**: rapid gravity sizing of the majority of steel members
- **Design Concrete (Gravity)**: design of concrete members for gravity combinations only
- **Design All (Gravity)**: design of all steel and concrete members for gravity combinations only

Because you may not have created lateral systems at this stage, the column nodes that are not in a rigid floor diaphragm are, by default, fixed horizontally. To change the default setting, go to **Home** --> **Settings** --> **Design Settings** --> **General**.

Design (Static)

Once initial member sizes have been adequately sized, you should make a sway sensitivity assessment (ACI/AISC), as this can affect the choice of analysis type (ACI/AISC) used in the final static design.

The final static design is invoked by selecting the suitable **Design (Static)** command:

- **Design Steel (Static)**: full design of all steel members
- **Design Concrete (Static)**: full design of all concrete members
- **Design All (Static)**: full design of all steel and concrete members

For each command above, Tekla Structural Designer performs a 3D analysis for all active combinations to establish a set of design forces. The 3D analysis can be either first or second order, depending on the settings specified on the **Analysis** page of the **Design Settings** dialog box.

If the model contains concrete members, Tekla Structural Designer also performs a Grillage chasedown and potentially FE chasedown analysis to establish additional sets of design forces.

All members are checked or designed for the appropriate design requirements. Gravity members are only checked for gravity combinations, while lateral members are checked for all combinations. Only active combinations are checked.

See also

Select whether to design steel, concrete, or all (page 777)

Check selected members and walls

There are various ways to check specific members only, using the results of the analyses that have already been performed.

Using a selective approach can be a more efficient way to work with large models.

Check an individual member, wall, or core

- Do one of the following:

To	Do this
Check a member/wall from a 2D or 3D view	1. Hover the mouse pointer over the member. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Check Member > Static/RSA**, or **Check Wall > Static/RSA** as required.
Check a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Check Member > Static/RSA** as required.
Check a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Check Wall > Static/RSA** as required.
Check all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Check member > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check selected members and walls

1. In the active view, select the members to check.
2. Right-click, then in the context menu, select **Check Selection > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a level, slope, or frame

- In the **Project Workspace**, click **Structure** tab.

- In the **Structure** tree, do one of the following:

To	Do this
Check all members in a construction level	Right click the required **Level** and select **Check plane > Static/RSA** as required.
Check all members and walls in a frame	Right click the required **Frame** and select **Check plane > Static/RSA** as required.
Check all members in a slope	Right click the required **Slope** and select **Check plane > Static/RSA** as required.

NOTE These commands use the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Check walls > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members and walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members of a particular section or type

1. In the **Project Workspace**, click **Structure** tab.

2. In the **Members** tree, right-click the section type or size you want to check.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a group

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to check.
3. In the context menu, select **Check Group > Static/RSA** as required.

NOTE
- **Check Groups** uses the existing analysis results, (even if the analysis status is 'Out of Date').

  It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Check all members and walls in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Check sub structure > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

There are various ways to design specific members only, using the results of the analyses that have already been performed.

Using a selective design approach can be a more efficient way to work with large models.

NOTE In all of the selective design methods, each member in the selection will be individually checked or designed according to its individual autodesign setting, (while also taking account of any member design groups that are active). The exception being that if there is only a single member in the selection it will always be designed irrespective of its autodesign setting.

Design an individual member, wall, or core

Design Member designs an individual member (taking into account any member design groups that are active).

Design Wall designs an individual wall.

Both these commands ignore the auto-design setting of the member/wall. (i.e. a design is always performed even if the autodesign property is off.)

- Do one of the following:

To	Do this
Design a member/wall from a 2D or 3D view	1. Hover the mouse pointer over the member/ wall. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Design Member > Static/RSA**, or **Design Wall > Static/RSA** as required.
Design a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Design Member > Static/RSA** as required.
Design a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Design Wall > Static/RSA** as required.
Design all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Design member > Static/RSA** as required.

NOTE The **Design Member** and **Design Wall**commands both use the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

Design Selection performs a check or design of each entity in the current selection, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the active view, drag a box to make your selection.
2. Right-click, then in the context menu, select **Design Selection > Static/RSA** as required.

   The selected members/walls are designed/checked.

   If any members in the selection are in a design group, other members in the same group are updated and checked as required.

NOTE The **Design Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Interactively design a concrete member

Interactive design is only available for concrete beams columns and walls. A dialog opens showing the existing design of the highlighted member for the currently supplied reinforcement. From here you can interactively modify the reinforcement and instantly see the result.

1. In the active view, hover the mouse pointer over the concrete member you want to interactively design.
2. Right click and select **Interactive Design > Static/RSA** as required.

   The interactive dialog opens from where the member can be designed.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also: Interactive concrete member design (page 1310)

Design all members in a level, slope, or frame

Design plane performs a check or design of each entity in the plane, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Design all members in a construction level	Right click the required **Level** and select **Design plane > Static/RSA** as required.
Design all members and walls in a frame	Right click the required **Frame** and select **Design plane > Static/RSA** as required.
Design all members in a slope	Right click the required **Slope** and select **Design plane > Static/RSA** as required.

NOTE
- The **Design plane** command uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It requires at least one member in the selection to be in auto-design mode.
- If any grouped members are designed and the grouped member design option is on, all members of the group are considered in the design even if they are not in the selected level, slope, or frame.

Design all members

Design members performs performs a check or design of every member according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Design members > Static/RSA** as required.

NOTE	
	• **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available until at least one member is in auto-design mode.

Design all walls

Design walls performs performs a check or design of every wall according to each wall's autodesign setting.

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Design walls > Static/RSA** as required.

NOTE	
	• **Design walls** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available until at least one wall to be in auto-design mode.

Design all members and walls

Design model performs performs a check or design of every member/wall in the model according to the individual member/wall autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right click the **Structure** branch.
3. In the context menu, select **Design model > Static/RSA** as required.

NOTE	
	• **Design model** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available until at least one member is in auto-design mode.

Design all members of a particular section or type

Design members performs performs a check or design the selected members according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Members** tree, right-click the section type or size you want to design.
3. In the context menu, select **Design members > Static/RSA** as required.

NOTE
- **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member is in auto-design mode.

Design all members in a group

Design Group performs performs a check or design of the selected member group according to the autodesign settings in the group.

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to design.
3. In the context menu, select **Design Group > Static/RSA** as required.

NOTE
- **Design Group** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Design all members and walls in a sub structure

Design sub structure performs performs a check or design of all members/ walls in the selected sub structure according to their individual autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to design.
3. In the context menu, select **Design sub structure > Static/RSA** as required.

NOTE
- **Design sub structure** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member in the sub structure is in auto-design mode.

Apply user defined utilization ratios

By default designs and checks are performed against a utilization ratio (U/R) of 1.0.

To facilitate design optimization Tekla Structural Designer allows you to specify user defined U/Rs on an individual member basis (currently for Eurocode, US, and India head codes only).

NOTE Utilization ratio = actual performance value / maximum allowable performance value.

Overview of user defined U/R

When applied to the autodesign process user defined U/Rs introduce a measure of:

- Conservatism - if the value is less than 1.0
- Non conservatism - if the value is greater than 1.0

By not applying user defined U/Rs to the check process, you are still able to work to the original U/R of 1.0 when working in check mode.

User defined U/Rs can be applied to the following objects:

- Members; steel and concrete columns and beams, composite beams, steel joists and braces. This includes separate settings for each span/stack/panel of continuous beams/columns/walls.
- Steel Trusses - includes; Truss member top/bottom, Truss member side/ internal.
- Portal frames (Individual members within Frames are treated as steel columns and steel beams as appropriate).
- Concrete Walls and Slabs including panels, patches, and punching shear checks.
- Foundations; Pad Bases (Spread Footings), Strip Bases, Pile Caps, Mat and Piled Mat Foundation slabs and Piles.

Apply user defined U/R for autodesign only

Applying a value of utilization ratio < 1.0 for autodesign only introduces a measure of conservatism into the autodesign process but doesn't fail the member until the ratio default value of 1.0 is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window
2. Expand **Utilization ratio** and select **Apply (to autodesign)**, but do **not** select **Apply to check**
3. Enter the **Ratio limit** as required.

NOTE For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than 1.0.

Apply user defined U/R for autodesign and check

Applying a value of utilization ratio < 1.0 for autodesign and check introduces a measure of conservatism into the autodesign process and fails the member when the ratio value that was set is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window

2. Expand **Utilization ratio** and select both**Apply (to autodesign)** and **Apply to check**

3. Enter the **Ratio limit** as required.

NOTE For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than the ratio limit.

See also

Review and modify user defined utilization ratios (page 894)

Related video

User defined utilization ratio

Validate the model for design issues

If required, you can manually validate the model for design issues before the model is submitted for design in order to trap errors that will cause the design to fail.

Run design validation

NOTE As design validation is performed automatically as part of all the combined analysis and design processes, there is no requirement to run design validation manually beforehand in those cases. You might however elect to run the validation manually when performing selective member and wall checking or design.

NOTE Design validation cannot be performed if model validation is outdated.

- On the **Design** tab, click **Validate**.

  Tekla Structural Designer performs the design validation checks. If your model contains any design validation issues, warning messages appear in the **Status** tab of the **Project Workspace**.

Adjust the conditions considered in design validation

1. On the **Home** tab, click **Model Settings**.
2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

7.2 Design slabs and run punching shear checks

Slab design in Tekla Structural Designer requires a certain amount of user interaction, (which is why slabs are not considered when any of the combined analysis and design commands are run).

Slab design can therefore be considered in isolation, the process being able to be broken down into the following discreet steps:

1. Create slab patches over columns, beams, walls, or panels
2. Design slabs (the slab areas that lie outside the patches)
3. Design patches (the slab areas inside the patches)
4. Create punching shear checks as required
5. Design punching shear (page 803) for the whole model or individually

See also

Apply user defined utilization ratios (page 786)

Concrete member and slab design handbook (page 1274)

Create and modify patches

You can apply rectangular patches of reinforcement to individual slab items to act in addition to the background reinforcement. After selecting the patch type, you can place patches in the model by clicking or boxing around elements in 2D or 3D views.

Overview of patches and patch types

Patches are used during the design of concrete slabs as a way of managing the physical and design data. Each patch defines a rectangular area of slab within which FE analysis results are collected, enabling Tekla Structural Designer to perform the design process. Design moments are calculated along result strips, embedded within each patch. Depending on the patch type, a patch can contain up to 6 result strips, catering for up to 3 strips of reinforcement in each of two perpendicular directions.

Tekla Structural Designer contains the following patch types:

- Column patch: can be placed at column stack heads
- Beam patch: can be placed along beams
- Wall patch: can be placed along walls
- Panel patch: can be placed at a specified position within the panel boundary
    - not restricted to a centralized position or to existing within one panel
    - can be positioned under loads

Patches can be either on the top or the bottom of the slab and may or may not have reinforcement defined in them. If you have not defined any reinforcement, Tekla Structural Designer uses the background reinforcement. If you have defined reinforcement, then for the top/bottom, x/y direction, you can optionally use the sum of the background + patch reinforcement option, if the patches are reasonably aligned.

NOTE Any patches may overlap on the plan view. However, during design, the eventual overlap is ignored.

Create column patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Column**.
3. In the **Properties** window, adjust the patch properties according to your needs.

a. To specify the patch size, type values in the **Lx** and **Ly** fields.

b. To specify the layer of the patch, select the desired option in **Surface**.

c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

---

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific column	• Click a column node within the slab. Tekla Structural Designer creates a patch to the selected column.
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the columns to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all columns entirely within the box.

Create beam patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Beam**.
3. In the **Properties** window, define the required patch width (perpendicular to the beam span) and the center slab width.

Tekla Structural Designer recalculates the two end strips accordingly. The strips cannot be modified.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific beam span	• Click the required beam span. Tekla Structural Designer creates a patch centered on and orientated to the beam center line.
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the beams to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all beams entirely within the box.

Create wall patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Wall**.
3. In the **Properties** window, adjust the patch properties according to your needs.
    a. To specify the layer of the patch, select the desired option in **Surface**.
    b. To have the reinforcement automatically designed, select the **Autodesign** option.
    c. Define the required patch width perpendicular to the wall.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Create a patch along a specific wall	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall**. b. Click the required wall. Tekla Structural Designer creates a patch, centered on and orientated to the wall center line.
Place an internal patch and two end patches along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal With End Patches**. b. Click the required wall. Tekla Structural Designer creates an internal patch and two end patches, centered and orientated to the wall center line.
Place an end patch at one end of a specific wall:	a. In the **Properties** window, set **Create Mode** to **End Patch at Wall End**. b. Click near the required end of the wall. Tekla Structural Designer creates a patch at the end of the wall that is closest to the point that you clicked.
Place an internal patch part way along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal Patch**. b. Click to define the start point of the patch along the required wall. c. Click to define the end point of the patch along the required wall. Tekla Structural Designer creates a patch between the selected points.
Create patches along multiple walls	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall** or **Internal With End Patches**. b. Move the mouse pointer to one corner of an imaginary box that will encompass the walls to which you want to create patches. c. Hold down the left mouse button. d. Drag the mouse pointer to the opposite corner of the box.

	e. Release the mouse button. Tekla Structural Designer creates patches to all walls entirely within the box.

Create panel patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Panel**.
3. In the **Properties** window, adjust the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place the patch at the centroid of a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected. b. Click anywhere within the required panel. Tekla Structural Designer creates a patch at the panel centroid.
Place the patch elsewhere within a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is cleared. b. Still in the **Properties** window, do one of the following: • To define the patch by its corners, select the **Define corner points** option. • To define the patch by its center, clear the **Define corner points** option.

	c.	In the panel, click the points that define the patch. Tekla Structural Designer creates a patch at the selected position.
Create multiple centroid patches	a.	In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected.
	b.	Move the mouse pointer to one corner of an imaginary box that will encompass the panels to which you want to create patches.
	c.	Hold down the left mouse button.
	d.	Drag the mouse pointer to the opposite corner of the box.
	e.	Release the mouse button. Tekla Structural Designer creates patches to the centroids of all panels entirely within the box.

Modify patch properties

1. Hover the mouse pointer over the patch that you want to modify.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. In the **Properties** window, modify the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.
   d. To specify the directions of the reinforcement that you want to design, select the appropriate option in **Consider Strips**.
   e. To specify the type and direction of the reinforcement that you want to design, select the appropriate option in **Reinforcement**.
   f. To specify the required width for each strip, type a value in the **Width** field.
   g. To specify whether each strip should be designed for the average or maximum design force, select the appropriate option in **Design Force**.

Resize patches

1. Hover the mouse pointer over the patch that you want to resize.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. Hover the mouse pointer over the patch edge or corner that you want to adjust.

   The **Select Entity** tooltip appears.
4. In the **Select Entity** tooltip, use the arrow keys to navigate to the edge or corner node and press **Enter**.
5. Hold down the left mouse button and drag to move the edge or corner.

Design and check slabs

After creating any necessary patches, you can determine the top and bottom reinforcement requirements in all the slab item areas that lie outside of the patch areas. Because patch areas are excluded from slab item design, you should create patches before designing slab items.

NOTE Slabs can only be designed/checked after slab design moments have been established from a suitable analysis. If valid analysis results do not yet exist you would need to create them (by, for example, clicking **Analyze All (Static)** from the **Analyze** toolbar).

Check an individual slab item

- Do one of the following:

<table>
<tr><th>To</th><th>Do this</th></tr>
<tr><td>Check an individual slab from a 2D or 3D view</td><td>1. Hover the mouse pointer over the slab.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right-click, and from the context menu that appears, select Check Panel.</td></tr>
<tr><td>Check an individual slab from the Structure Tree</td><td>1. In the Project Workspace, click Structure tab.
2. In the Slabs branch, locate the slab reference.
3. Right click and select Check Panel.</td></tr>
</table>

NOTE **Check Panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Concrete slab design (page 1354)

Check all slab items

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model slabs** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Concrete slab design (page 1354)

Check all slab items on a single floor

- Do one of the following:

To	Do this
Check all slabs in a floor from a 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Check Slabs**. Tekla Structural Designer checks the reinforcement of all slab items on the floor.
Check all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Levels branch, right click the required **Level** and select **Check plane slabs**.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Concrete slab design (page 1354)

Check all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.

2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to check.
3. In the context menu, select **Check sub structure slabs**.

   Tekla Structural Designer checks the reinforcement of all slab items on the floor.

See also:

- Concrete slab design (page 1354)

Design an individual slab item

- Do one of the following:

To	Do this
Design an individual slab from a 2D or 3D view	1. Hover the mouse pointer over the slab. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Member**.
Design an individual slab from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs branch, locate the slab reference. 3. Right click and select **Design panel**.

NOTE **Design panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Concrete slab design (page 1354)

Design all slab items

- On the **Design** tab, click **Design Slabs**.

  Tekla Structural Designer designs or checks all slab items in the model according to their autodesign settings.

See also:

- Concrete slab design (page 1354)

Design all slab items on a single floor

- Do one of the following:

To	Do this

Design all slabs in a floor from 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Design Slabs**.
Design all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Levels branch, right click the required **Level** and select **Design plane slabs**.

See also:

- Concrete slab design (page 1354)

Design all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to design.
3. In the context menu, select **Design sub structure slabs**.

   Tekla Structural Designer designs all slab items in the sub structure, possibly selecting new reinforcement for them.

See also:

- Concrete slab design (page 1354)

Design and check patches

When that the slab item designs resulting from the slab or mat design process are satisfactory, you can then proceed to design patches. You should design slab items before designing patches, as the additional patch reinforcement requirement takes account of the existing level of reinforcement provided by the slab items.

NOTE For each strip within each patch, Tekla Structural Designer determines the area of the required steel ($A_{s,reqd}$) and the area of the provided steel ($A_{s,prov}$). The $A_{s,prov}$ calculation always considers the patch reinforcement, but if necessary and allowed, you can also include additional background reinforcement in the calculation.

Check an individual patch

1. Hover the mouse pointer over the patch that you want to check.
2. Right-click the patch.

3. In the context menu that appears, select **Check Slab Patch**.

   Tekla Structural Designer displays the results of the check in a new dialog.

Check all patches in the model

- The same command can be accessed in two ways:
  - In a 3D Structural View, right click on the background and choose **Check Patches** from the context menu.
  - In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose Check model patches (command) (page 2239) from the context menu.

  Tekla Structural Designer checks the reinforcement of all patches in the model.

Check all patches on a single floor

1. Open a 2D view of the floor that you plan to design.
2. Right-click anywhere in the view.
3. In the context menu that appears, select **Check Patches**.

   Tekla Structural Designer checks the reinforcement of all patches on the floor.

Design an individual patch

1. Hover the mouse pointer over the patch that you want to design.
2. Right-click the patch.
3. In the context menu that appears, select **Design Slab Patch**. Tekla Structural Designer displays the results of the design in a new dialog.

   The results of the design are displayed in a new dialog.

Design or check all patches in the model

- The same command can be accessed in three ways:
  - On the **Design** tab, click **Design Patches**
  - In a 3D Structural View, right click on the background and choose **Design Patches** from the context menu.

- In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose Design model patches (command) (page 2250) from the context menu.

Tekla Structural Designer designs or checks all patches in the model according to their autodesign settings.

Design all patches on a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the view.
3. In the context menu that appears, select **Design Patches**.

   Tekla Structural Designer designs all patches on the floor, possibly selecting new reinforcement for them.

Create punching shear checks

You can apply punching shear checks to flat slabs and foundation mats by using the **Add Check** command on the **Design** tab.

Punching check locations

You can apply punching shear checks as follows:

In flat slabs:

- Concrete columns supporting a flat slab with and without drops
- Concrete columns supported by a flat slab
- Concrete columns through a flat slab with or without drops
- Concrete walls supporting a flat slab
- Concrete walls supported by a flat slab
- Concrete walls through a flat slab
- Steel columns supported by a flat slab
- Point loads at either face of a flat slab

In foundation mats:

- Concrete columns supported by a foundation mat
- Concrete walls supported by a foundation mat
- Steel columns supported by a foundation mat
- Point loads on a foundation mat
- individual piles supporting a foundation mat

Punching check axis orientation

- When you apply a punching shear check to a column, the check Y and Z axes are automatically orientated to align with the column major and minor axes. The two axis systems are locked together, so if the column is rotated, the punching check axes also rotate.
- When you apply a punching shear check to a point load, you must manually define the punching shear check Z axis orientation. You can do this in the **Properties** window by specifying the point load orientation in relation to the global Y axis.

  NOTE Point load properties (including point load orientation, point load breadth, and point load depth) are not applicable when you apply a check to a column.

Create punching check items

NOTE Punching check items can generally be created in 2D and 3D Structural or Review Views, apart from checks around walls which can only be created in Structural Views, but **not** currently in Review Views.

Punching check items are the objects to which the punching shear information and calculations are attached. To create punching check items, do the following:

1. On the **Design** tab, click **Add Check**.
2. Go to the **Properties** window-
3. Ensure that the location for the slab tension reinforcement is correct.
4. Adjust the remaining properties according to your needs.

   NOTE The point load properties are only relevant if you are adding the check to a point load position.

5. To create the check, do one of the following:
   - Hover the mouse pointer over the desired column node, wall node, or point load. When the **Pick Punching Check Location** tooltip appears, click to add the punching shear at the selected node.
   - Hold down the left mouse button and drag a box to add punching check items to all potential check locations within the box.

Specify stud rail reinforcement

RESTRICTION	Stud rail reinforcement is only available for Eurocode and ACI head codes. You can specify an orthogonal or circular arrangement of stud rails, and then check or design it as required.

1. Create a punching check item as instructed above.
2. Go to the **Properties** window.
3. Select the **Use reinforcement** option.
4. Select whether you want to autodesign the reinforcement.
5. Adjust the remaining reinforcement properties according to your needs.

Modify the properties of existing punching check items

1. Right-click the punching check item.
2. In the context menu, select **Edit Reinforcement**.

Design and check punching shear

Overview of the Design Punching Shear command

The **Design Punching Shear** command calculates an applied load on the slab, accounting for the difference in column/wall axial load and bending. In addition, the command checks the slab shear stresses, accounting for the reinforcement present (background and column/general patches).

Provided that you are using the Eurocode or ACI head code, you can also design the punching reinforcement as an orthogonal or circular arrangement of stud rails at the following locations:

In flat slabs:

- Concrete columns supporting a flat slab without drops
- Concrete columns supported by a flat slab
- Concrete columns through a flat slab without drops
- Point loads at either face of a flat slab

In foundation mats:

- Concrete columns supported by a foundation mat
- Point loads on a foundation mat

Column head drops and the presence of openings within a certain distance of the punching shear boundary are taken into account in the punching shear calculations.

Where punching checks are closely spaced and the perimeters overlap the checks are beyond scope.

NOTE

- Since the checks are dependent on using the correct levels of slab reinforcement (typically provided by patch reinforcement), do not check punching before designing the patches.
- View punching checks applied to walls with caution. Their applicability to long walls is particularly questionable, as the check does not consider the potential for stress concentrations at the ends of the wall.

Check punching shear for an individual punching check item

- Do one of the following:

To	Do this
Check an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Check Punching Shear**.
Check an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Check Punching Shear**.

Check all punching check items

1. In the **Project Workspace**, click **Structure** tab
2. Right click the Structure branch and select **Check Punching Shear**

Check all punching shear check items on a floor

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Check Punching Shear**.

Design all punching check items

- Do one of the following:

To	Do this
Design all punching check items from the ribbon	1. On the **Design** tab, click  **Design Punching Shear**.
Design all punching check items from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. Right click the Structure branch and select **Design Punching Shear**.

Tekla Structural Designer performs punching shear checks for all punching check items in the model based on their individual auto-design setting. If the punching check item does not connect to a flat slab, punching shear is flagged as beyond scope.

Design all punching shear check items on a floor

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Design Punching Shear**.

Design an individual punching check item

- Do one of the following:

To	Do this
Design an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Punching Shear**.
Design an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Design Punching Shear**.

Tekla Structural Designer displays the results of the design in a new dialog box.

7.3 Design timber and precast members using Tekla Tedds

Timber and precast members that have been modelled and analysed in Tekla Structural Designer can then be designed in Tekla Tedds if a licence is available.

For further information, see:

- Timber member design handbook (page 1564)
- Precast member design handbook (page 1527)

7.4 Create and run floor vibration checks

To establish the response of the floor to dynamic excitation you can apply a floor vibration checks. These are applied over a user-defined rectangular or polygon shaped slab area by using the **Add Check** command in the **Floor Vibration** group. In order to create the check, you need to identify a primary beam, secondary beam and critical slab item. In addition, you must specify associated data in order to perform the floor vibration calculation.

See also

Create and modify floor vibration checks (page 806)

Run floor vibration checks (page 809)

Create and modify floor vibration checks

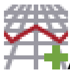
To establish the response of the floor to dynamic excitation you can apply a floor vibration checks. These are applied over a user-defined rectangular or polygon shaped slab area by using the **Add Check** command in the **Floor Vibration** group. In order to create the check, you need to identify a primary beam, secondary beam and critical slab item. In addition, you must specify associated data in order to perform the floor vibration calculation.

Vibration of Floors to SCI P354 Handbook; Vibration of Floors to DG11 Handbook; Vibration of Floors to SCI P354 Example

Create floor vibration check items

Floor vibration check items are the objects to which the floor vibration check information and calculations are attached.

NOTE Floor vibration check items can only be created in 2D Views.

1. On the **Design** tab, click  **Add Check**.
2. Go to the **Properties** window.
3. In **FloorPlateDefinition**, select whether you want to create a rectangular or polygonal slab.
   a. If you selected the **Rectangular** option, adjust the local x angle to define the angle at which the rectangle will be drawn, if necessary.
4. To create a check area, in the model, do one of the following depending on the slab shape:
   - **Rectangular**: Click once to define the first corner of the rectangle, and again to define the opposite corner.
   - **Polygon**: Click to define the corners of the polygon. In order to close the shape, click the first point again.
5. Click to define the primary beam.

   The primary beam properties appear in the **Properties** window.
6. Click to define the secondary beam.

   The secondary beam properties appear in the **Properties** window.
7. Click to define the critical slab item.

   NOTE If the critical slab item is a composite slab, Tekla Structural Designer automatically defines the slab item properties. In other cases, you need to input the properties yourself.

   The critical slab item properties appear in the **Properties** window.
8. In the **Properties** window, review and adjust the check item properties.
9. Click anywhere in the model to create the check item.

   Tekla Structural Designer highlights the primary beam, secondary beam and critical slab item of the current check item.
10. Create new checks, or press **Esc** to finish creating floor vibration check items.

Create floor vibration checks that consider two or three adjoining spans

Creating two-span or three-span floor vibration checks is almost identical to that required for single spans. When creating the check item, do the following:

- In the **Properties** window, set the **Adjoining Spans** property of a primary and/or secondary beam to **Two span** or **Three span**.

NOTE If the **Two span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of two or more spans are available.
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

NOTE If the **Three span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of three or more spans are available to be highlighted,
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

Modify the properties of existing floor vibration check items

1. In **Scene Content**, ensure that Floor Vibration Checks have been turned on.
2. Hover the mouse pointer over the slab area where the floor vibration check item lies.

   The **Select Entity** tooltip appears.
3. Navigate to the floor vibration check item name by using the arrow keys, and click **Enter**.

   The floor vibration check properties appear in the **Properties** window.

4. In the **Properties** window, adjust the check item properties according to your needs.

Run floor vibration checks

Vibration of Floors to SCI P354 Handbook; Vibration of Floors to DG11 Handbook; Vibration of Floors to SCI P354 Example

Check vibration for all floor vibration check items

- On the **Design** tab, click **Check Floor Vibration**.

  Tekla Structural Designer performs floor vibration checks for all floor vibration check items in the model.

Check floor vibration for an individual floor vibration check item

1. Hover the mouse pointer over the floor vibration check item that you want to check.

   The **Select Entity** tooltip appears.
2. If necessary, navigate to the floor vibration check item name by using the arrow keys.
3. Right-click the check item.
4. In the context menu, select **Check Floor Vibration**. Tekla Structural Designer displays the results of the design in a new dialog box.

7.5 Check steel connections

- Check simple connection resistance (page 809)
- Design connections (page 819)
- Export connections to another application for design (page 825)
- SidePlate connections (page 825)

Check simple connection resistance

Tekla Structural Designer provides a quick and simple way of checking the shear capacity of simple beam connections, and the axial capacity of brace connections.

Overview

Tekla Structural Designer has a Connection Resistance database for simple beams and braces. For simple beams the database contains pre-defined types with pre-defined resistances (page 1009) to Eurocode and US head codes.

You can add user-defined connection types (page 1012) and user-defined resistances (page 1012) to the database for any head code, for steel and cold formed materials. Once defined, these can then be used across all projects.

Each resistance value in the database has an 'active' flag which must be 'on' for the resistance to be considered. (Simple beams and braces in the model that have no active resistances are not checked.)

Having configured the active resisistances in the database as required, the checks are then performed in the model as follows:

- For each simple beam the active shear resistances for each available connection type are read from the database.
- These resistances are compared to the maximum shear force at the connection (determined from a 3D analysis)
- The connection configuration is optimized accordingly for each connection type.
- A similar process is then followed for each brace, by comparing the active axial resistances to the maximum applied axial force.

Results can be displayed in a review data table, or output via a connection resistance report.

Specify 'active' connection resistances (Eurocodes)

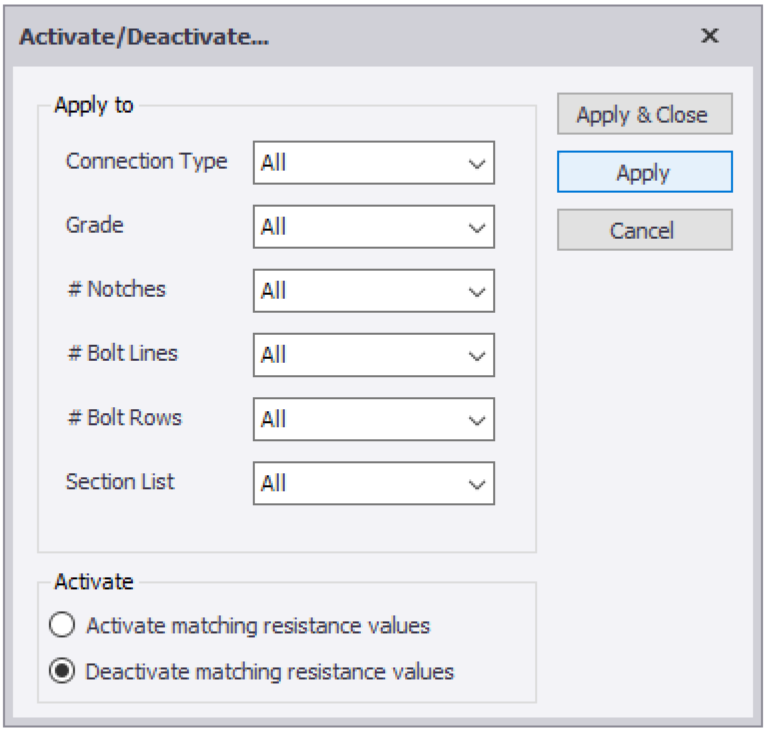
Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

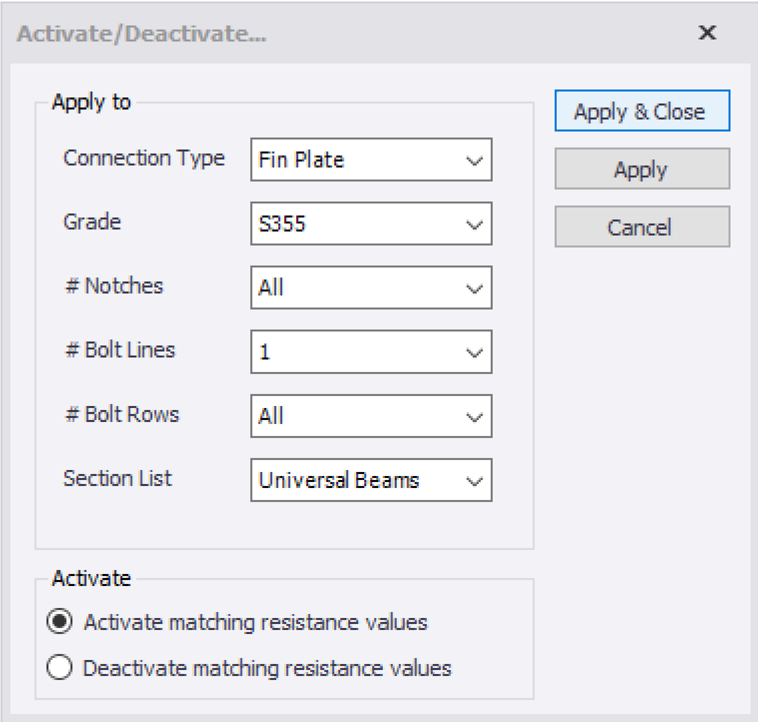
1. In the Connection Resistance dialog (page 2399), click **Activate/ Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

Example: From the pre-defined connection types you want only Fin Plates to be active, only for S355 Universal Beams, and only those Fin Plates with 1 line of bolts. There are a number of ways to achieve this using **Activate/ Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select ‘All’ in each of the Activate/Deactivate filters, select ‘Deactivate matching resistance values’, then **Apply**



3. To activate only Fin Plates for S355 Universal Beams with 1 line of bolts
   a. select Fin Plate in the Connection Type filter,
   b. select S355 in the Grade filter,
   c. select 1 in the # Bolt Lines filter,
   d. select Universal Beams in the Section List filter,
   e. select ‘All’ in the other filters (# Notches, # Bolt Rows),
   f. select ‘Activate matching resistance values’,
   g. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Fin Plates with S355 Universal Beams with 1 line of bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Specify 'active' connection resistances (US)

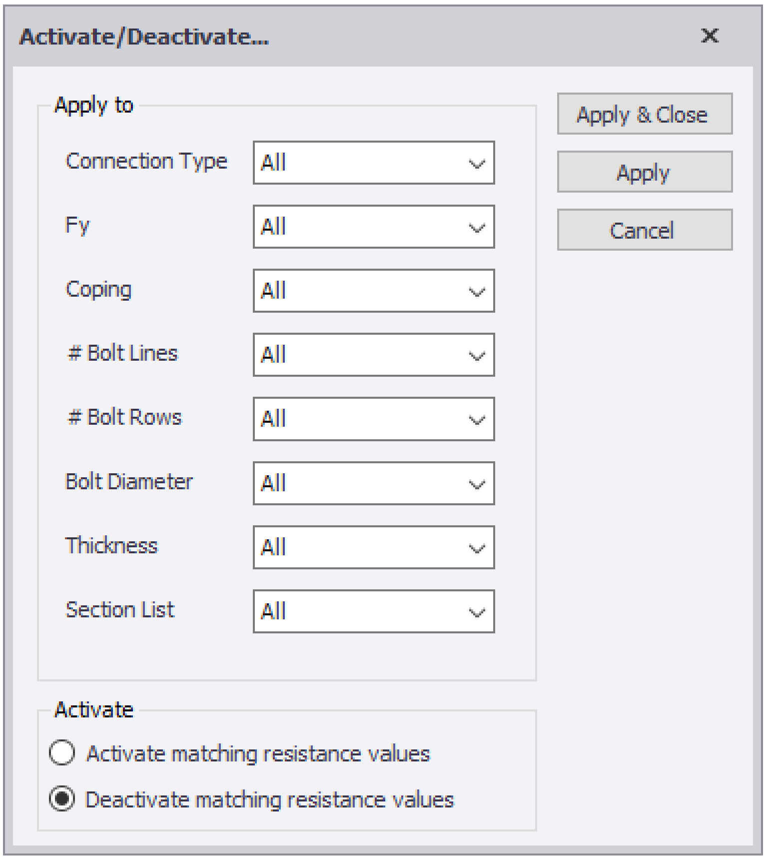
Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

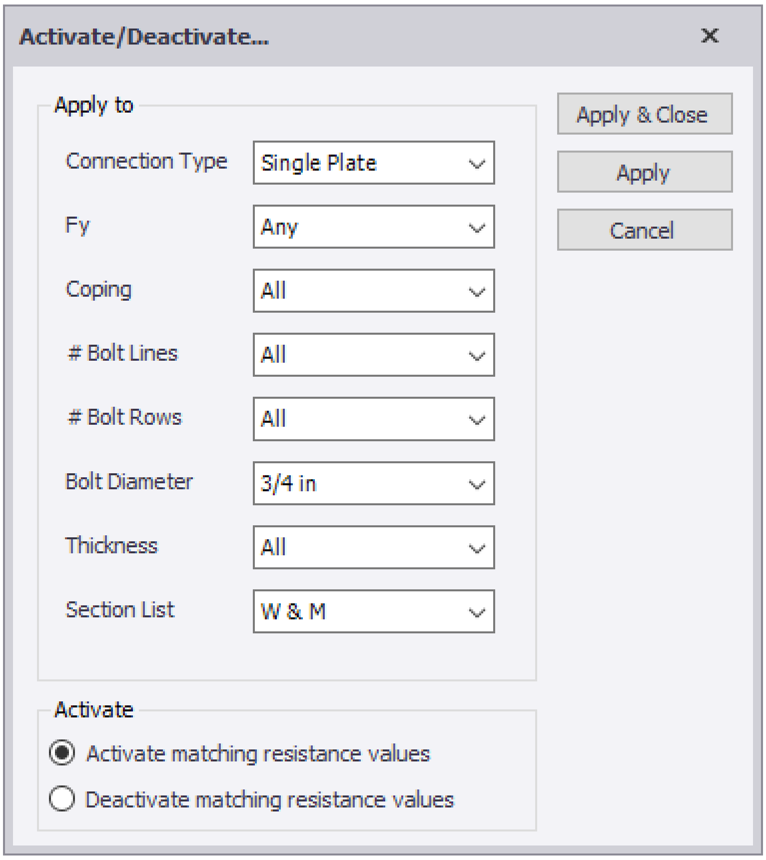
1. In the Connection Resistance dialog (page 2399), click **Activate/ Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

Example: From the pre-defined connection types you want only Single Plates to be active, for W & M beams of 'Any' Fy, only for 3/4 in bolts, but for all defined thicknesses of plate. There are a number of ways to achieve this using **Activate/Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select 'All' in each of the Activate/Deactivate filters, select 'Deactivate matching resistance values', then **Apply**



3. To activate only Single Plates for W & M Beams of 'Any' Fy, with 3/4 in diameter bolts and all plate thickness
   a. select Single Plate in the Connection Type filter,
   b. select Any in the Fy filter, select 3/4 in in the Bolt Diameter filter,
   c. select W & M in the Section List filter,
   d. select 'All' in the other filters (Coping, # Bolt Lines, # Bolt Rows, Thickness),
   e. select 'Activate matching resistance values',
   f. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Single Plates, with Any Fy selected, and W & M Beams with 3/4 in diameter bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Run resistance checks

Provided you have an appropriate set of 'active' resistances, the checks are performed automatically, using the current analysis results - you can proceed directly to a review of the check results.

- If necessary, analyse the model to generate an up-to-date set of analysis results,
- Review the check results, see: ***Display connection resistance checks in a review data table*** below.

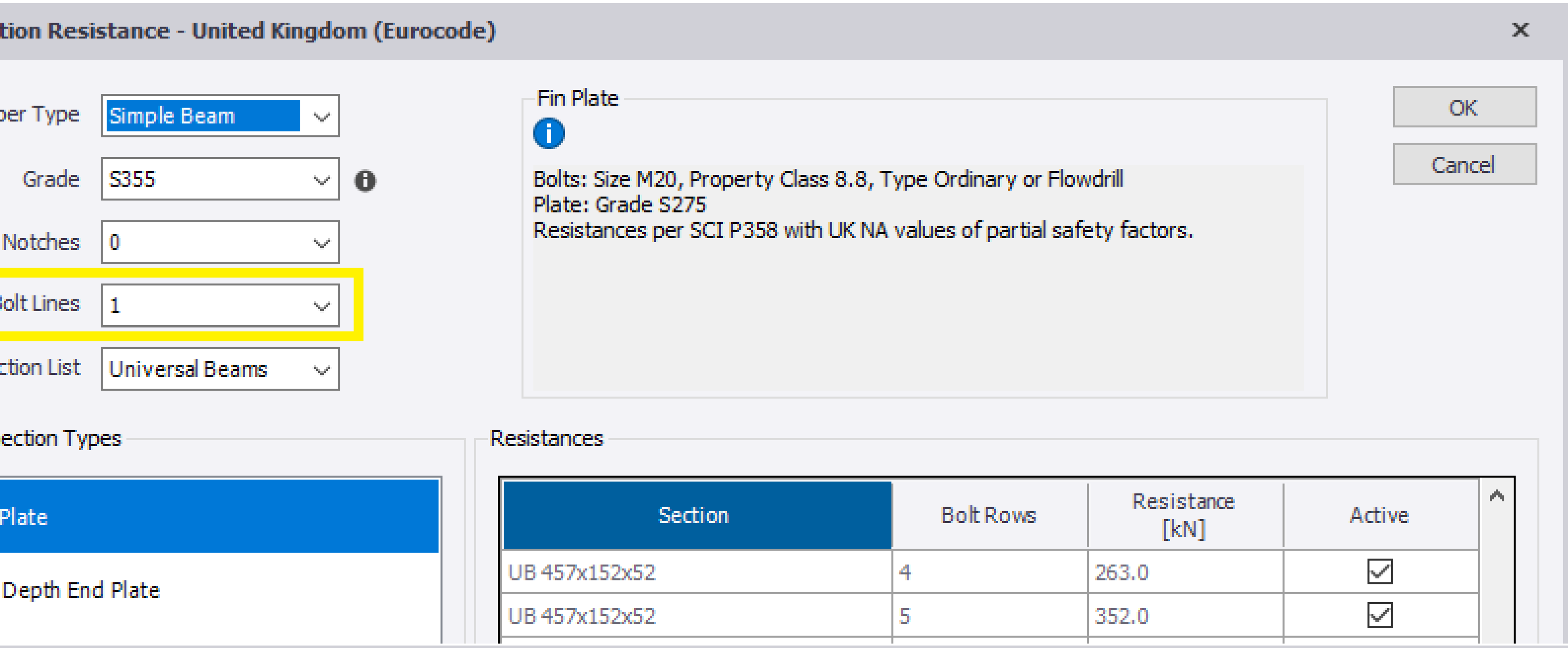
The connection optimization process

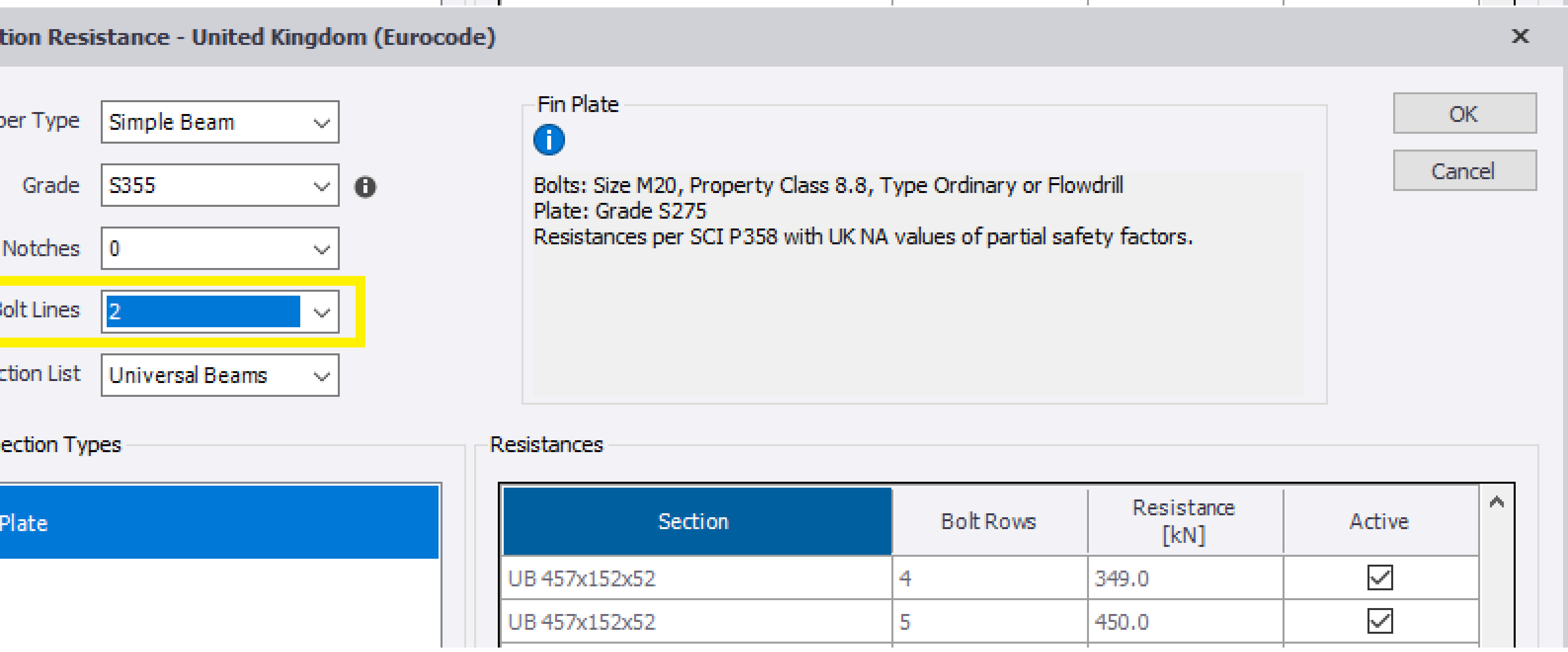
Tekla Structural Designer carries out an optimization process to find the first passing Active resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name.

For Eurocodes, separate check results are reported for each 'active' connection type and each notch variation.

For US codes, separate check results are reported for each 'active' connection type and each coping variation.

Example: to Eurocode, a Fin Plate for an S355 UB 457x152x52 with 1 and then 2 lines of bolts has *pre-defined* resistances as shown below. All resistances are indicated as Active.





If an S355 UB 457x152x52 in a model is found to have an applied major shear force of 250 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 4 bolt rows (resistance 263 kN) is adequate.

If the applied major shear force is 300 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 5 bolt rows (resistance 352 kN) is adequate.

If the applied major shear force is 355 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) is adequate.

For each of the applied major shear forces given above only the *first passing* resistance is reported, by looping through the *Active* bolt rows for 1 bolt line followed, where necessary, by looping through the *Active* bolt rows for 2 bolt lines.

If Tekla Structural Designer gets to the end of these loops and still doesn't find a passing resistance then it reports the last resistance found and a fail status e.g. if the applied major shear force is 455 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) fails.

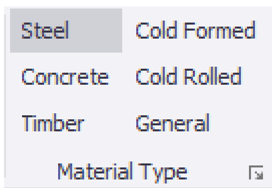
Now suppose that the UB 457x152x52 with 1 bolt line and 5 bolt rows (resistance 352 kN) is *not* Active. The first shear check above (applied force 250 kN) would be unchanged but the second check (applied force 300 kN) would now report that a Fin Plate with 2 bolt lines and 4 bolt rows (resistance 349 kN) is adequate. This illustrates the interplay of the resistance Active status with the optimization process.

Display connection resistance checks in a review data table

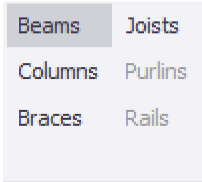
To create a tabular connection resistance summary:

1. If necessary, change the view regime (page 280) to a **Review View**.
2. On the **Review** ribbon tab, click **Tabular Data**.

   A **Review Data** tab opens on the ribbon and a **Review Data View** is displayed.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Connection Resistance**.
4. In the **Material Type** group, select the material (either **Steel**, or **Cold Formed**).

   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.
5. In the **Characteristic** group, select the required characteristic (either **Beams**, or **Braces**).



6. In the **Construction** group, select **Composite**, **Non-composite**, or both.

7. In the **Fabrication** group, select the required fabrication types.

A resistance check is provided for the filtered selection for every section where a resistance has been specified in the database.

NOTE A resistance check is not provided for beams that do not have simple connections.

NOTE If there is uplift in any load combination then the maximum uplift shear force will be displayed, with a warning that connection resistance is not checked for uplift condition.

NOTE Each time you re-analyse the building, if you have the Connection Resistance data table open you must close and then re-open it to update the results.

Create and display a connection resistance report

1. On the **Report** tab, use the droplist to select the existing **Connection Resistance** report.
2. Click **Model Report...**
3. Review the report structure and adjust as required.

4. Click **OK** to save the report.
5. Click **Show Report**

Related video

Predefined connection resistance database for Eurocode and AISC

Design connections

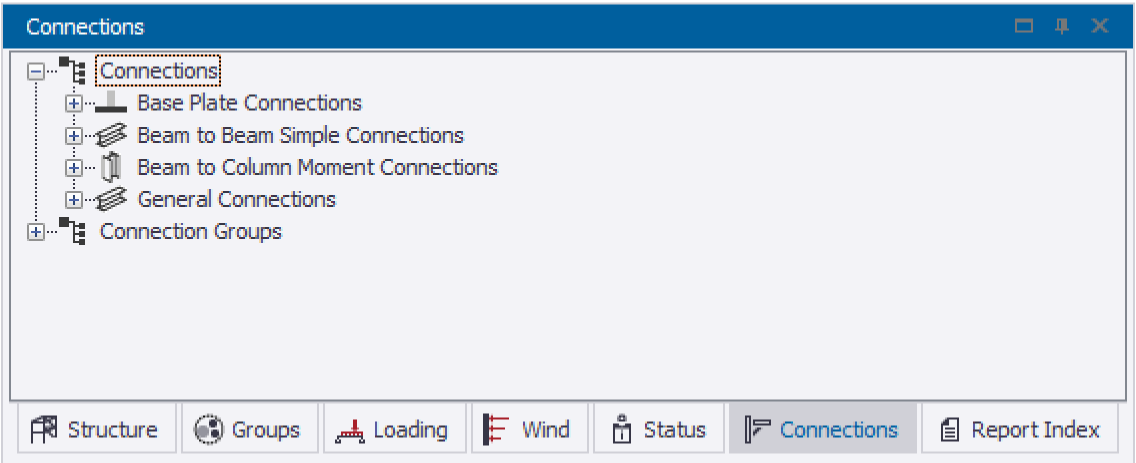
NOTE These topics relate to the *design* of connections using either Tekla Connection Designer, or IDEA StatiCa; they do not cover checking of simple connection resistance (page 809), which is a separate topic.

Overview

Before connections can be designed in Tekla Structural Designer valid connection objects have to be created in the model. This is achieved by running **Update connections** from the Project Workspace. This applies a set of rules (page 822) to create and organize connections into the following types:

- Base Plate Connections
- Beam to Beam Moment Connections
- Beam to Beam Simple Connections
- Beam to Column Moment Connections
- Beam to Column Simple Connections
- Column Splice Connections
- General Connections

Valid connections are then displayed in the scene views and are also listed in the **Connections** tree.



Provided a licence of Tekla Connection Designer is available, some of these can then be designed or exported to Tekla Connection Designer. If you have a licence of IDEA StatiCa you can also export some types to that software for design.

Type	Connection Design Options
Base Plate Connections	• Design Connection • Export to Tekla Connection Designer
Beam to Beam Moment Connections	• Design Connection (Eurocode and BS Head Codes only) • Export to Tekla Connection Designer (Eurocode and BS Head Codes only) • Export to IDEA StatiCa
Beam to Beam Simple Connections	• Export to IDEA StatiCa
Beam to Column Moment Connections	• Design Connection (Eurocode and BS Head Codes only) • Export to Tekla Connection Designer (Eurocode and BS Head Codes only) • Export to IDEA StatiCa
Beam to Column Simple Connections	• Export to IDEA StatiCa
Column Splice	• Design Connection (Eurocode and BS Head Codes only) • Export to Tekla Connection Designer (Eurocode and BS Head Codes only)

Type	Connection Design Options
	• Export to IDEA StatiCa
General Connections	• Export to IDEA StatiCa

See also

- Recommended workflows for specific connection types (page 822)
- Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 824)

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 822) to determine all valid steel connections in the model. The resulting connections are listed in the Connections Tree and are also shown by bounding boxes in the Scene Views.

   NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Design connections

Design using Tekla Connection Designer is initiated in one of two ways:

1. From the Project Workspace:
   - Expand the appropriate branch in the **Connections** tree
   - Right click on the required connection reference and choose **Design connection** from the context menu.
2. From a scene view:
   - Right click on the connection object and choose **Design connection** from the context menu.

See also

Recommended workflows for specific connection types (page 822)

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 824)

[Export connections to another application for design (page 825)]

Steel connection formation rules

The following rules relate to connection formation in Tekla Structural Designer.

1. beam to column connection
    a. beam connections to the column base are ignored
    b. column must be a symmetric section
    c. more then 1 beam attached to the same column face prevents any connection
    d. a continuous beam connected to the column flanges prevents any connection
    e. a beam with a fully fixed or moment end condition connected to the column web prevents any connection
2. beam to beam - for a particular construction point (node)
    a. any column at the node prevents the connection
    b. only two beam ends can be connected, a 3rd beam at the same node prevents any connection.
    c. connections cannot be formed at beam internal nodes
    d. the beams being connected must be symmetric and of the same section size
    e. connected beam ends cannot be free or pinned
    f. top flange haunches prevent the connection
    g. beam ends being connected must have no end plates specified, or both have the same end plate
    h. beam ends being connected must have no bottom haunch specified, or both have the same bottom haunch
3. any non-steel member at the node prevents any connection
4. braces are ignored in the procedure

Recommended workflows for specific connection types

NOTE Connection design requires a Tekla Connection Designer licence.

Portal frame connections

Although it is possible to design portal frame connections in Tekla Structural Designer - i.e. Base Plate, Eaves, and Apex moment connections; it should be noted that the design forces used for connection design within Tekla Structural Designer are those from its own elastic analysis and may not be appropriate.

A more correct approach would be to derive the connection forces from a Tekla Portal Frame Designer plastic analysis. To do this you would need to export the portal frame to Tekla Portal Frame Designer and then design both the frame and its connections within Tekla Portal Frame Designer.

Note that it in addition to the connection forces (usually) being fundamentally different between Tekla Portal Frame Designer and Tekla Structural Designer there are other potential differences also:

- for eaves connections - in Tekla Portal Frame Designer the connection design forces are taken at the column face (rather than the rafter/column intersection),
- for column bases - in Tekla Portal Frame Designer they can be pinned at ULS but % fixity can be taken for SLS, while Tekla Structural Designer will always give an analysis moment for % fixity regardless of ULS/SLS and this moment would be designed for.

NOTE Connection design via Tekla Portal Frame Designer requires both a Tekla Portal Frame Designer and a Tekla Connection Designer licence.

Base plate connections (not in portal frames)

Base plate connections can be edited directly in Tekla Structural Designer.

Provided they are in elastically analyzed frames they can be also be designed using the design forces obtained from Tekla Structural Designer.

NOTE The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.

Base plate connections can be exported to Tekla Connection Designer if you require, in which case if you edit the connection geometry while in Tekla Connection Designer you should use the “Return Connection to Tekla Structural Designer” command (under the “Connection” menu) to return the edited connection data back to Tekla Structural Designer.

Other connections (not in portal frames)

The recommended workflow for connection types other than Base Plate Connections is:

- Select the connection(s) to be designed and export to Tekla Connection Designer always.
- Design in Tekla Connection Designer and print the reports.
- While the “Return Connection to Tekla Structural Designer” command can be used, this does not currently return all connection data to Tekla

Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.

NOTE While the “Return Connection to Tekla Structural Designer” command can be used In Tekla Connection Designer, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/or design processes are re-run; if the connection is exported again for checking, the engineer would need to review the connection data as it may be necessary to re-enter some of the settings previously made.

See also

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 824)

Limitations when using Tekla Connection Designer with Tekla Structural Designer

General

- The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.
- It is always necessary to export all connection types to Tekla Connection Designer for reporting purposes.

Base plate connections

- Base Plate Connections only deal with major axis shear - only one value for shear is available in the Combinations grid in Tekla Connection Designer. (This is a limitation in the Green Book also.)

Moment connections

- While the “Return Connection to TSD” command can be used, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.
- For Beam to Column moment connections, column forces are exported and displayed on the "Column" tab of the Combinations page.

- When designed to Eurocodes, there is a reduction factor for column web in compression to allow for coincident forces in the column itself i.e. moment and axial force. This is called kwc and in most cases is 1.0 i.e. no reduction. But where the stress due to the column forces is high (> 0.7 fy) then kwc is < 1.0. See Clause 6.2.6.2 (2).
- These forces are NOT populated by the TCD integration link and must be entered manually by the engineer.

Simple connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Hollow section connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Column splice connections

- Splice connections with significant 'real' moments (other than eccentricity moments) cannot currently be transferred from Tekla Structural Designer to Tekla Connection Designer.

NOTE Significent 'real' moments being greater than the Ignore forces below (page 2323) limit in Design Settings.

Export connections to another application for design

Click the following links to find out more about exporting specific connections into another application for design:

- Recommended workflow for TCD connection types (page 822)
- Export to Tekla Connection Designer (page 318)
- Export to Tekla Portal Frame Designer (page 319)
- Export to IDEA StatiCa Connection Design (page 344)

SidePlate connections

SidePlates are a US proprietary connection system that can be applied in Tekla Structural Designer but are considered separately to other connection types.

Click the following links to find out more:

- For background to their usage in Tekla Structural Designer, see

  SidePlate connections theory (page 826)

- For modeling instructions, see

  Create SidePlate connections (page 833)

SidePlate connections theory

Overview of SidePlate connections

SidePlate® is a type of moment connection for connecting I or HSS section beams to I, HSS, built up box or built up WF (cruciform) section columns.

SidePlates are welded to the column section in the fabrication shop. Plates or angles are similarly welded to beam ends.

The column and beam units are transferred to site where the columns are erected and the beams are lifted into place and either bolted or welded to the columns. SidePlate connections are used on buildings of 1 to 30 stories tall.

Within the structure, the beam end is held in position by SidePlates. The net result is moment connection with a very stiff section of column and a stiffened beam end.

SidePlate connection types can be:

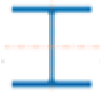
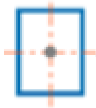
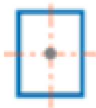
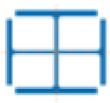
- Non-Seismic moment connections
  - A beam/column moment connection anywhere in the structure
- Seismic moment connections
  - A seismic beam/column moment connection within any of the following seismic force resisting systems Ordinary Moment Frame (OMF), Intermediate Moment Frame (IMF) or Special Moment Frame (SMF).

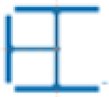
SidePlate connections work for all WF and HSS sections in the AISC steel book, and for all UB and UKB sections. However it is worth noting the SidePlate datafile may NOT have a full set of values for all 'clear spans' or all %M_ps for all these sections; if this is the case then no SidePlate connection can be applied at the joint.

Tekla Structural Designer can perform an initial sizing for the SidePlate connection, but the final detailed design of the SidePlate connection has to be undertaken in SidePlate® software.

Permitted SidePlate connections in Tekla Structural Designer

RESTRICTION	In Tekla Structural Designer, sideplate connections are currently only permitted for US Customary sections and not for metric sections.

Column		Side Plate Permitted Connections					Tekla Structural Designer Status
		Non-Seismic Moment Connections		Seismic Moment Connections			
		Beam		Beam			
		I	HSS	I	HSS		
WF	Flange	yes	yes	yes	yes		Modeled Analyzed Designed
	Web	yes	yes	Warning	Warning		
HSS	Flange	yes	yes	yes	yes		Modeled Analyzed Designed
	Web	yes	yes	yes	yes		
Plated box	Flange	yes	yes	yes	yes		Modeled Analyzed **Not Designed**
	Web	yes	yes	yes	yes		
WF + 2xWT	Flange	yes	yes	yes	yes		**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	yes	yes		

Column		Side Plate Permitted Connections					Tekla Structural Designer Status
		Non-Seismic Moment Connections		Seismic Moment Connections			
		Beam		Beam			
		I	HSS	I	HSS		
WF + 1xWT	Flange	yes	yes	yes	yes		**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	Warning	Warning		

SidePlate workflow in Tekla Structural Designer

A brief description of the SidePlate workflow in Tekla Structural Designer is given here with more detail available in the subsequent topics.

- **Modeling**: You define SidePlate connections by changing properties of the relevant beam or beams.

> **NOTE** The **Update Connections** process used for defining other connection types is **not** applicable for SidePlate.
>
> Connection objects are not created for any point on a steel column where *any* of the beams attached at that point have a SidePlate connection.

- **Visualisation**: Each connection is initially shown in the graphics as a simple box. After sizing has been performed it is shown as a pair of plates.
- **Validation**: Some preliminary checks are carried out in Model validation (page 512), (additional checks are done subsequently in **Initial sizing** and **Analysis model adjustment**). Any problems do not stop the analysis and design process but mean the relevant beam and or column will not benefit from the enhanced properties.
- **Initial sizing**: The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes, using data provided by SidePlate and incorporated into Tekla Structural Designer.

> **NOTE** After the initial sizing process, any changes to loading, (individual loads, load cases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can

reset the status using Review mode (page 892) if you decide that the sizing process needs to be run again.

- **Analysis model adjustment**: Where valid SidePlate dimensions exist, the analysis model passed to the solver is adjusted to reflect the increased stiffness of the SidePlate connections. This is done during stand-alone analysis as well as Design.
- **Steel member design**: If SidePlate connections exist, they are taken into account in the member design for steel beams and columns. The changes affect both static and seismic design.
- **Review**: The SidePlate connection status can be seen easily in a Review view (page 892), and tooltips give more information.
- **Reports**: A “SidePlate Connection Report” can be generated.
- **Final design**: SidePlate developers can use the **Tekla Structural Designer API** in order to fully design a SidePlate connection defined and initially sized in Tekla Structural Designer

NOTE Design of SidePlate connections is beyond scope for both Tekla Connection Designer and export to the IDEA StatiCa Connection Design program.

SidePlate 'joint' and 'connection' terminology

NOTE

- A SidePlate **joint** exists at a column node and consists of 1 to 4 valid connections.
- Every suitable beam framing into the column node can create a SidePlate **connection**.

A SidePlate connection exists at End 1 and / or End 2 of a steel beam where **Apply SidePlate Connection** is true. It will still exist even if other data makes it invalid. The connection position is the reference point at the relevant end of the beam.

All connections sharing the same position on a column are considered as a single SidePlate Joint. No attempt is made to merge separate SidePlate joints even where the relevant levels are close.

Various checks are performed on each connection and each joint, and the analysis model is only be adjusted if they all pass. In particular, if the depths of two SidePlate joints overlap then a warning is generated and the analysis model is not be adjusted.

SidePlate validation

SidePlate validation is performed during model validation.

If warnings are issued, this will typically result in the Connection Status being set to "Not Applied" in the **Properties** window.

NOTE Until the “Not Applied” Connection Status is resolved, the connection will not be considered for the initial sizing process.

Initial sizing of the SidePlates

The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes.

The initial sizing process is only performed where the Sizing Status is “Not Performed” and the Connection Status is **not** “Not Applied”.

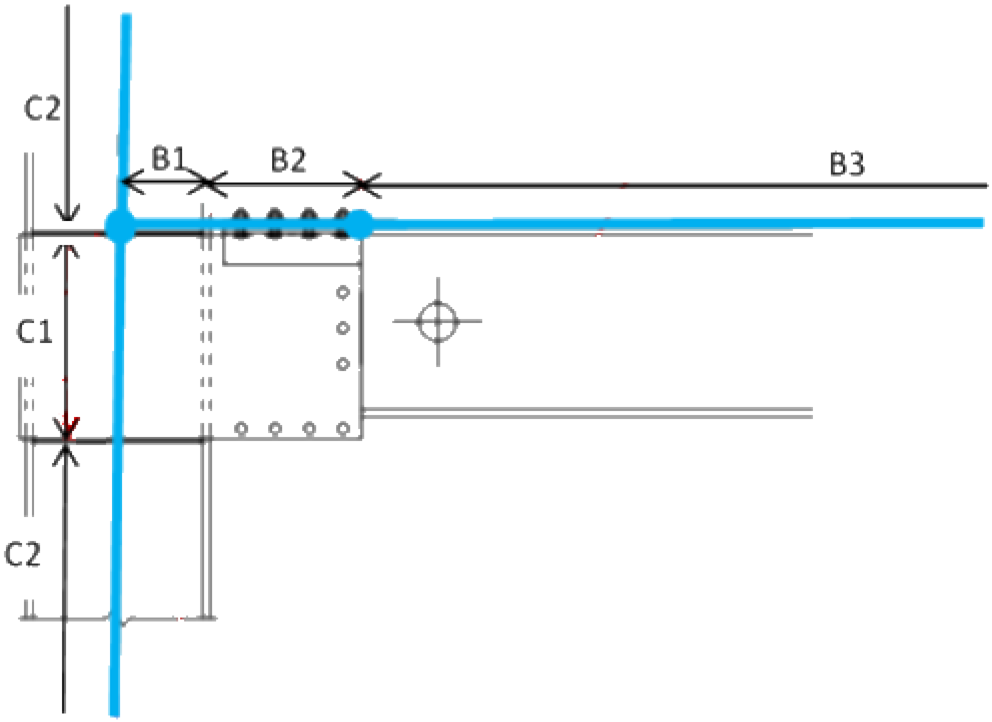
For each connection that is considered, a number of checks are performed in order to verify that SidePlate can be applied. If any of the checks fail, the Connection Status is set to "Not Applied".

NOTE After the initial sizing process, any changes to loading, (individual loads, load cases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can reset the status using Review mode if you decide that the sizing process needs to be run again.

Analysis model adjustment

Analysis model adjustment is only performed for valid SidePlate connections that have successfully completed the initial sizing process, i.e. where the Sizing Status is **not** “Not Performed”.

Both the geometry and the properties of the Solver model are dependent upon the size of the SidePlate connection.



For the beam

- B2 – a new 1d element on the beam, with stiffened beam properties.
- B1 – a rigid offset on the end of B2
- B3 – normal bare beam properties

For the Column

- C1 – a rigid offset at the top of the column 1d element
- C2 – normal column properties

Design/Analysis processes and recommended workflows

Tekla Structural Designer can automatically size members throughout a structure – we call this Autodesign. The process is complex and involves sophisticated initial sizing, multiple analysis runs with repeated design cycles. The interweaving of initial sizing for SidePlate connections into the Autodesign routines is a highly complex process and for the first implementation remains beyond scope. Instead you have to manually reduce section sizes to take advantage of the SidePlate advantages.

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal during this process.

The following examples show the results for 2 different workflows where each step is carried out in the exact order specified and assuming that there are no model issues.

Recommended Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.
- Check “Apply SidePlate Connection” for selected beams - also set Utilization Ratio to 1.25 for those beams.
- Set Utilization Ratio to 1.1 for selected columns
- Run 1st order linear analysis
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because not Static Design
  - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Gravity Design Steel
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because not Static Design
  - Adjust Solver Model will not be performed because no connections will have Sizing Status not “Not Performed”.
- Reset Utilization Ratio to 1.0 for relevant beams and columns
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed for all connections
  - Adjust Solver Model will be performed for subsequent analysis
- Manually adjust section sizes to achieve further size savings with SidePlate connections
  - SidePlate Connection Status will be reset where necessary
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed only where section sizes have been changed
  - Adjust Solver Model will be performed

At the end of this workflow, any subsequent stand-alone analysis will also use the adjusted solver model.

Other Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.
- Check “Apply SidePlate Connection” for selected beams.

- Run Static Design Steel
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because Connection Status will be set to Warning
  - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed for all Connections
  - Adjust Solver Model will be performed for subsequent analysis

Effects of SidePlate on steel member design

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal.

Beams and columns affected by SidePlate connections cannot be AutoDesigned.

Where SidePlate connections exist, changes are required for both static and seismic member design.

For static design

- Beam and column buckling lengths are affected.
- Beam and column design checks are adjusted so that they are applied outside the physical length of any SidePlate connection

For seismic design

- OMF frames - no change
- IMF frames – various changes
- SMF frames – various changes

Create SidePlate connections

To add SidePlate connections:

1. Select the steel beam(s) to which you want to apply SidePlate connections.
2. In the **Properties** window, expand the SidePlate heading and select **Apply SidePlate connection** at one, or both ends as required.
3. In the **Properties** window, select the SidePlate **Connection type** that is appropriate to the SFRS type that has been specified, or, if not in a seismic frame, set the type as **SidePlate MF** (moment frame).

   Each connection is initially shown in the graphics as a simple box. After sizing has been performed it will be shown as a pair of plates.

See also

Beam properties - SidePlate (page 834)

Modify SidePlates (page 892)

Beam properties - SidePlate

For steel beams the **Properties** window has a SidePlate group within which an **Apply SidePlate Connection** check box can be selected for End 1 and End 2 of the beam.

When selected the following properties are displayed:

- Seismic Provision - depending on the head code, one of:
    - Non-Seismic (ACI/AISC only)
    - Seismic OMF
    - Seismic IMF (ACI/AISC only)
    - Seismic SMF
- Field Connection, (user option if Connection Type is Seismic SMF, otherwise fixed to Bolted)
    - Bolted (default)
    - Welded
- Datafile look up criteria (readonly)
    - Use %Mp (if Connection Type is Non-Seismic)
    - Use clear span (otherwise)
- Connection Status - one of
    - Unknown (default)
    - Pass
    - Warning
    - Not Applied
- Failure Reason - one of
    - Unknown (default)
    - Datafile Look Up Failed
    - Geometric Compatibility
    - Beam Element too short for Rigid Offset
    - Column Element too short for Rigid Offset
- Sizing Status - one of
    - Not Performed (default)

  - Initial
  - Final
- SidePlate size and other data set during the Initial Sizing Process, (Invisible if Sizing Status is Not Performed)
  - DimA – the outstand distance required for the SidePlate connection
  - DimB – the depth required for the SidePlate connection
  - Tsp – the thickness of the SidePlates
  - DimH – the length of the SidePlates
  - %M_p- if calculated during initial sizing
  - Clear span - if calculated during initial sizing

NOTE These properties are for the start and end of the whole beam. Where a steel beam is multi-span it is not possible to apply SidePlate connections to the internal points.

8 Create and design foundations

You can create and design both isolated and mat foundations using the **Foundations** toolbar.

8.1 Create isolated foundations

Tekla Structural Designer allows you to create pad base, strip base and pile cap isolated foundations. Before you can create a pile cap the Pile Catalogue must contain at least one pile type.

Click the links below to find out more:

- Create pad bases and strip bases (page 836)
- Create a pile type catalogue (page 838)
- Create pile caps (page 838)

See also

Design isolated foundations (page 840)

Foundation design handbook (page 1580)

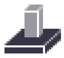
Create pad bases and strip bases

You can create both pad base columns and strip base walls in your model. Pad bases and strip bases are both isolated foundations. The difference is that pad bases support a column, while strip bases support a wall.

Create pad base columns

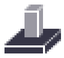
A pad base is an isolated foundation that supports a single column. To place pad bases in your model, see the following instructions.

Create a pad base under a specific column

1. On the **Foundations** tab, click  **Pad Base Column**.
2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Click anywhere in the wall under which you want to place the pad base.

   Tekla Structural Designer places a pad base under the selected column.

Create multiple pad base columns

1. On the **Foundations** tab, click  **Pad Base Column**.
2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pad bases.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pad bases under all columns within the box.

Create strip base walls

A strip base is an isolated foundation that supports a single wall. To place strip bases in your model, see the following instructions.

Create a strip base under a specific wall

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.
3. Click anywhere in the wall under which you want to place the strip base.

   Tekla Structural Designer places a strip base under the selected wall.

Create multiple strip bases

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.

3. Move the mouse pointer to a corner of an imaginary box that will encompass the walls under which you want to place strip bases.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places strip bases under all walls within the box.

Create a pile type catalogue

In order to use pile foundations in your model, you must create a catalogue of pile types. To do so, see the following instructions.

1. On the **Foundations** tab, click  **Catalogue**.

   The **Pile Catalogue** opens.
2. To create a new pile type, click **Add...**

   The **Edit Pile Type** dialog box opens.
3. On the different pages of the **Edit Pile Type** dialog box, define the properties of the new pile type.
4. Click **OK**.

   Tekla Structural Designer creates the pile type.
5. Repeat steps 2–4 as necessary.

TIP In the **Pile Catalogue**, you can also modify or delete the existing pile types:

- To modify a pile type, select the pile type and click **Edit...**
- To delete a pile type, select the pile and click **Delete**.

Create pile caps

A pile cap is an isolated piled foundation that supports a single column.

NOTE Before you can create pile caps, the **Pile Catalogue** must contain at least one pile type.

To create pile types, see Create a pile type catalogue (page 838).

Create pile cap under a specific column

1. On the **Foundations** tab, click **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.
3. Click anywhere in the column under which you want to place the pile cap.

   Tekla Structural Designer places a pile cap under the selected column.

Create multiple pile caps

1. On the **Foundations** tab, click **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pile caps.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pile caps under all columns within the box.

Create a user-defined pile arrangement

If necessary, you can modify the pile arrangement in a pile cap and create user-defined pile arrangements. Creating user-defined pile arrangements can be particularly useful when you want to check a pile cap where the pile positions on site do not exactly match the originally specified arrangement.

Do the following:

1. In the model, select the pile cap that you want to modify.
2. In the **Properties** window, go to **Pile arrangement**.
3. Click the **...** button next to **Pile arrangement**.

   The **Pile Arrangement** dialog box opens.
4. In the **Pile Arrangement** dialog box, select the **User Defined Arrangement** option.

   NOTE If the **Auto-design piles** option is selected in the **Properties** window, Tekla Structural Designer asks you whether you want to turn off the option. Click **Yes**.

5. Specify the pile arrangement properties according to your needs:
   a. Select the pile type.
   b. Define the pile length and width.
   c. If necessary, define the eccentricity of the pile cap centroid from the centroid of the column it supports in X and Y directions.

   TIP To add more piles into the arrangement, click **Add**.

6. If necessary, in the table below the properties, adjust the position of piles by defining the eccentricity of the pile centroid from the pile cap centroid.
7. Click **OK**.

   Tekla Structural Designer creates the pale arrangement that you defined.

8.2 Design isolated foundations

You can design isolated foundations (pad bases, strip bases, and pile caps) either in a batch or individually.

NOTE Before bases or pile caps can be designed, you have to run an analysis to establish the design forces. You can run either the **Analyze All (Static)** command or one of the combined analysis and design commands (such as **Design All**).

Design or check all pad bases and strip bases

1. On the **Foundations** tab, click **Design Pad Bases**.

   Tekla Structural Designer designs or checks all pad bases and strip bases in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Design or check all pile caps

1. On the **Foundations** tab, click **Design Pile Caps**.

   Tekla Structural Designer designs or checks all pile caps in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Check an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to check.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Check Member**. Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to design.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Design Member**. Tekla Structural Designer displays the results of the design in a new dialog box.

See also

Pad base design workflow (page 1581)

Pile cap design workflow (page 1587)

Apply user defined utilization ratios (page 786)

8.3 Create mat foundations

Mat foundations support multiple columns and walls. Mats can be ground bearing, supported on piles, or both. Tekla Structural Designer contains he following commands for creating mat foundations:

- **Minimum Area**: creates an overhanging polygonal mat to minimize the required area
- **Rectangular**: creates an overhanging rectangular mat either at a specified angle to the global axes or at the smallest rectangular area aligned to the global axes
- **Strip**: creates a constant width mat along a series of points that do not have to be on the same line

- **Area**: creates an overhanging polygonal mat when you identify points around its outline
- **Bays**: creates a mat with no overhang when you click within a closed grid area
- **Pile**: creates individual piles underneath existing mats
- **Pile Array**: creates an array of piles underneath existing mats

Click the links below to find out more:

- Create mats (page 842)
- Place piles and pile arrays in mats (page 844)

See also

Create slab or mat openings (page 455)

Add overhangs to existing slab or mat edges (page 457)

Create column drops (page 459)

Split and join slabs and mats (page 462)

Design isolated foundations (page 840)

Foundation design handbook (page 1580)

Create mats

In Tekla Structural Designer, you can create various types of mat foundations, including minimum area mats, rectangular mats, strip mats, area mats, and mats with bays. For detailed information on creating mats, see the following instructions.

Create a minimum area or rectangular mat

1. On the **Foundations** tab, click either  **Minimum Area** or  **Rectangular**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. Do one of the following:

To	Do this
Create a mat in a 3D view	a. Click anywhere on the first column or wall to be supported. b. Click to define the other necessary columns and walls. c. To create the mat, click the last column or wall again or press **Enter**.

Create a mat in a 2D view	a. Do one of the following: 1. To select multiple walls, hold down the left mouse button and drag a box from right to left to encompass the walls under which you want to create the mat. 2. To select multiple columns, hold down the left mouse button and drag a box from left to right to encompass the columns under which you want to create the mat. 3. To add individual columns and walls, click each member individually. b. To create the mat, click one of the previously selected members or press **Enter**.

Create a strip mat

1. Open a 2D view at the base level where you want to place the strip mat.
2. On the **Foundations** tab, click  **Strip**.
3. In the **Properties** window, adjust the strip width and other mat properties according to your needs.
4. In the model, click the start point of the strip.
5. Click to define the other necessary points.
6. To create the strip, click the last point again or press **Enter**.

Create an area mat

Either click once again on the last point, or press <Enter> to create the mat.

1. On the **Foundations** tab, click  **Area**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. In the model, click a construction point at a corner of the area.
4. Click to define the remaining corner points.
5. To create the mat, click the last point again or press **Enter**.

Create a mat within bays

1. On the **Foundations** tab, click **Bays**.
2. In the **Properties** window, adjust the mat properties according to your needs.

3. To select a bay, do one of the following:
   - Click within an enclosed area defined by grid lines.
   - Hold down the left mouse button and drag a box around the required area.

Place piles and pile arrays in mats

A piled mat can either be supported on piles alone, or by a combination of ground springs and piles. Piles can either be placed individually at specific locations within the mat, or an array of equally spaced piles can be created. To place piles in mats and define their inclination, see the following instructions.

Specify if a piled mat is ground bearing

1. Click the desired mat.
2. In the **Properties** window, go to **Soil Parameters**.
3. Do one of the following:

To	Description
Create a mat supported on piles alone	• Clear the **Use Ground Bearing Springs** option.
Create a mat supported by a combination of ground spring and piles	• Ensure that the **Use Ground Bearing Springs** option is selected.

Place an individual pile in a mat

1. On the **Foundations** tab, click **Pile**.
2. Click the mat in which you want to place the pile.
3. Click the pile position in the selected mat.

   Tekla Structural Designer places the pile in the selected position.

Place a pile array in a mat

1. On the **Foundations** tab, click **Pile Array**.
2. In the **Properties** window, adjust the pile type, spacing, and pile direction according to your needs.

   TIP To preview the array, hover the mouse pointer over a mat.

3. Click an individual mat, or hold down the left mouse button and drag a box around multiple mats.

4. If necessary, add further mats to the selection by clicking or boxing them.
5. To create the mat, click the last mat you selected again.

   Tekla Structural Designer places the piles in the mat.
6. Repeat steps 2–4 to place more piles or press **Esc** to finish placing piles.

Specify the pile direction of an inclined pile

1. On the **Foundations** tab, click **Pile**.
2. In the **Properties** window, do one of the following:

To	Do this
Specify the direction as a vector	a. Adjust the X and Y components of the vector. **TIP** If you do not want the pile to slope in the global X or Y direction, leave the appropriate value as 0. b. Type a negative value in the Z component field. For example: To create an inclined pile of 45 degrees in positive X, define the components as follows: `X = 1.0, Y = 0.0, Z = -1.0`. To create an inclined pile of 45 degrees in negative Y, define the components as follows: `X = 0.0, Y = 0.5, Z = -1.0`
Specify the direction as an angular measurement	**NOTE** You can only specify the direction as an angular measurement before the pile is first positioned. a. Select the **Direction by Angles** option. b. Define the inclination of the pile from the vertical. c. Define the azimuth (the horizontal angle measured clocwise about the positive global Z direction. d. If necessary, define the rotation about the longitudinal axis of the pile. **NOTE** Since the pile is only checked for axial capacity, the rotation angle has no effect on the pile checks. However, you can define the rotation to vary how Tekla Structural Designer displays the moment and shear force values of the pile. For example:

	To create an inclined pile of 45 degrees in positive X, define the angles as follows: `inclination = 45 degrees, azimuth = 90 degreeS` To create an inclined pile of 30 degrees in negative Y, define the angles as follows: `inclination = 30 degrees, azimuth = 0.0 degrees`

8.4 Design mat foundations

Design or check all mats in the model

1. On the **Foundations** tab, click **Design Mats**.

   Tekla Structural Designer designs or checks all mats in the model according to their auto design settings.

Check all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.
3. In the context menu, select **Check Slabs**.

   Tekla Structural Designer checks all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Design all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.
3. In the context menu, select **Design Slabs**. Tekla Structural Designer designs all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Check an individual mat

1. Hover the mouse pointer over the mat that you want to check.
2. Right-click the mat.

3. In the context menu that appears, select **Check Panel**.

   Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual mat

1. Hover the mouse pointer over the mat that you want to design.
2. Right-click the mat.
3. In the context menu that appears, select **Design Member**.

   Tekla Structural Designer displays the results of the design in a new dialog box.

See also

Mat foundation design workflow (US customary units) (page 1608)

Mat foundation design workflow (metric units) (page 1594)

Piled mat foundation design workflow (US customary units) (page 1622)

Piled mat foundation design workflow (metric units) (page 1633)

Apply user defined utilization ratios (page 786)

9 Review models

By selecting the **Review** view regime, you are able to graphically display the design status of members, slabs, foundations and connections. In the same view you can interrogate or modify a variety of model parameters and properties.

In addition, by clicking **Tabular Data** from the toolbar, you can open a **Review Data** view, from where you can display tables of Sway/Drift Results, Design Summaries and Material Lists.

TIP The **Review** view opens automatically at the end of the structure design process, but you can access it at any time by clicking the icon in the **Status bar**

9.1 Review designs

By graphically reviewing the design status and ratios you are able to quickly assess the design efficiency.

Click the links below to find out more:

- Set the design type to review (page 848)
- Review member design (page 849)
- Review foundation and pile design (page 850)
- Review slab and mat design (page 851)
- Design review filters (page 853)

Set the design type to review

You can set a specific design type to focus on when reviewing the member, foundation, or slab designs.

The available design type options are Static, RSA, and Combined (i.e. Static and RSA together).

To set the design type, do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type.
   - Static
   - RSA
   - Combined

See also

Review member design (page 849)
Review foundation and pile design (page 850)
Review slab and mat design (page 851)
Design review filters (page 853)

Review member design

Review member design status

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Status**.

   Color codes display the design status of each member.

   Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The member has passed all design checks.
Fail	The member has failed one or more design checks.
Warning	Although the member has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.

Status	Description
Error	The member currently cannot be designed because an error has occurred.
Beyond Scope	The member cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The member has not been designed.

Review member design ratios

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Ratio**.

   Color codes display the design ratio of each member. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates members that either are beyond scope or have yet to be designed.

Review member depth ratios

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Depth Ratio**.

   All steel and concrete beams are color coded to indicate their span to depth utilization ratios.

See also

Design review filters (page 853)

Review design summary tabular results (page 898)

Review foundation and pile design

Review foundation or pile status

1. Open a view and change the view regime (page 280) to **Review View**.
2. Go to the **Review** ribbon.
3. Either in the **Foundations** group or the **Piles** group, click **Status**.

   Color codes display the design status of each foundation or pile.

   Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The foundation or pile has passed all design checks.
Fail	The foundation or pile has failed one or more design checks.
Warning	Although the foundation or pile has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Error	The foundation or pile currently cannot be designed because an error has occurred.
Beyond Scope	The foundation or pile cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The foundation or pile has not been designed.

Review foundation or pile ratios

1. Open a view and change the view regime (page 280) to **Review View**.
2. Go to the **Review** ribbon.
3. Either in the **Foundations** group or the **Piles** group, click **Ratio**.

   Color codes display the design ratio of each foundation or pile. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates foundations and piles that are either beyond scope or have yet to be designed.

See also

Design review filters (page 853)

Review design summary tabular results (page 898)

Review slab and mat design

Review slab and mat design status

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Slab/Mat Design** group, click **Status**.

Color codes display the design status of each slab and mat.

Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The slab item has passed all design checks.
Fail	The slab item has failed one or more design checks.
Warning	Although the slab item has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Error	The slab item currently cannot be designed because an error has occurred.
Beyond Scope	The slab item cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The slab item has not been designed.

Review slab and mat design ratios

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, in the **Slab/Mat Design** group, click **Ratio**.

   Color codes display the design ratio of each slab item. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates slab items that are either beyond scope or have yet to be designed.

Filter slab and mat design information

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click the list in the **Slab/Mat Design** group.
3. In the list that appears, select one of the following filtering options according to your needs:

Filter	Description
Overall	The governing Top X, Top Y, Bottom X, Bottom Y, or span depth result is displayed.
Reinforcement	The governing Top X, Top Y, Bottom X, or Bottom Y result is displayed.

Filter	Description
Top X	The result is only displayed for top reinforcement in the X direction.
Top Y	The result is only displayed for top reinforcement in the Y direction.
Bottom X	The result is only displayed for bottom reinforcement in the X direction.
Bottom Y	The result is only displayed for bottom reinforcement in the Y direction.
Span Depth	The span to depth result is displayed.
Bearing Pressure	The bearing pressure result is displayed.

See also

Design review filters (page 853)

Review design summary tabular results (page 898)

Design review filters

Design review filters help the user to focus on specific results.

The filters are displayed in the **Properties** window when **Status** or **Ratio** are selected from any group on the **Review** ribbon.

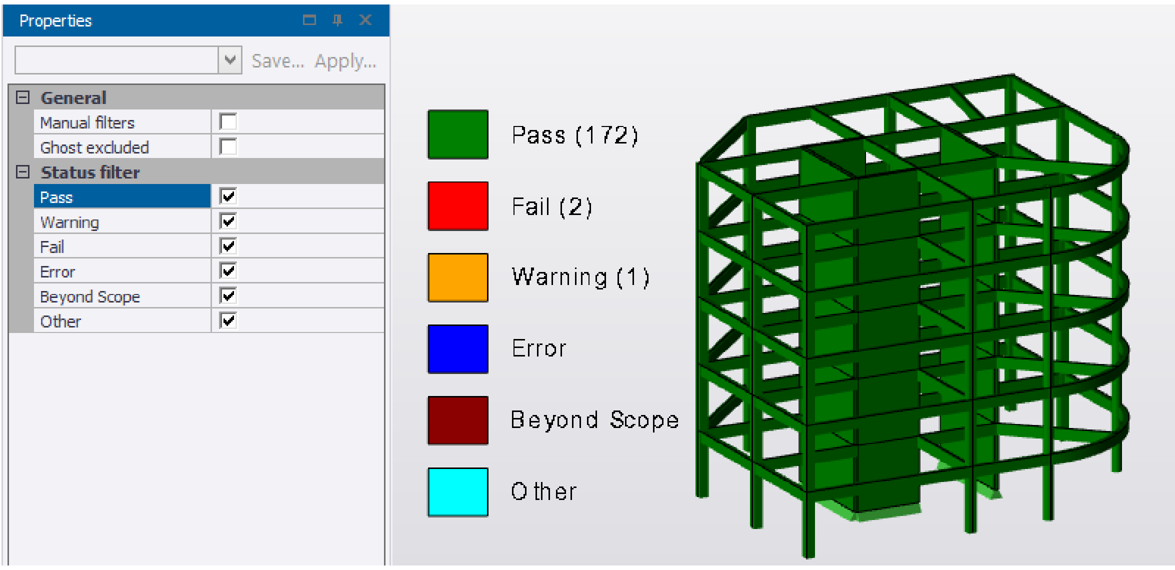
The following filter types are available and can be used individually or in combination:

- **Status filter**
- **Utilization ratio filter**
- **Entity type filter**

TIP These filters can be particularly useful when working with large models.

Working with the Status filter

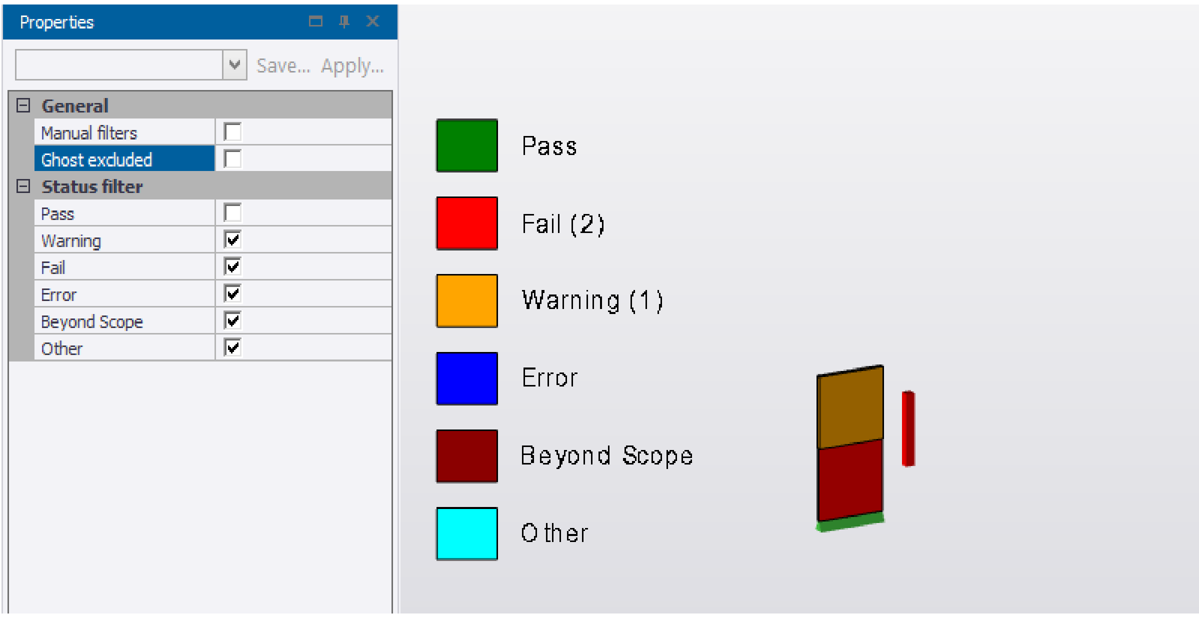
The **Status filter** is always active in the **Properties** window when **Status** is selected from the **Review** ribbon.



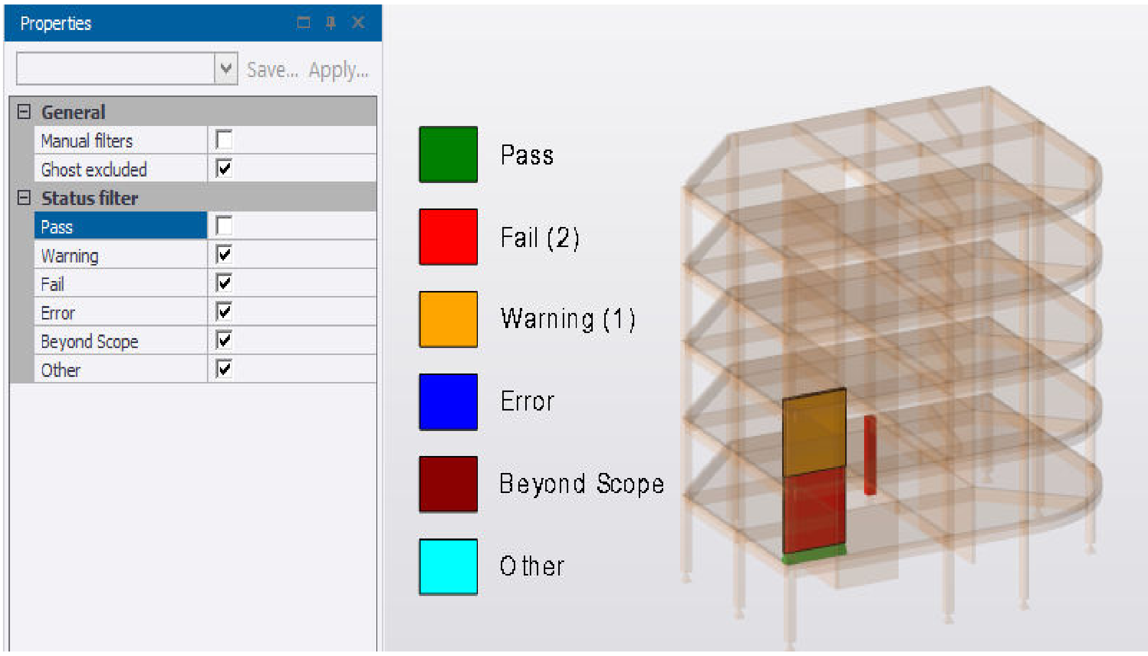
All entities are assigned a design status from the below list:

- **Pass** - The entity has passed all design checks.
- **Warning** - Although the entity has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
- **Fail** - The entity has failed one or more design checks.
- **Error** - The entity currently cannot be designed because an error has occurred.
- **Beyond Scope** - The entity cannot be designed because its design is beyond the scope of Tekla Structural Designer.
- **Unknown** - The entity has not been designed.

In the above model the legend indicates that two members are failing and another has a warning but their locations are not immediately apparent. Deselecting a status in the **Properties** window causes entities of that status to be hidden in the **Review View**, so in this instance deselecting the Pass status helps to focus on where the problems are occurring.



By also selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

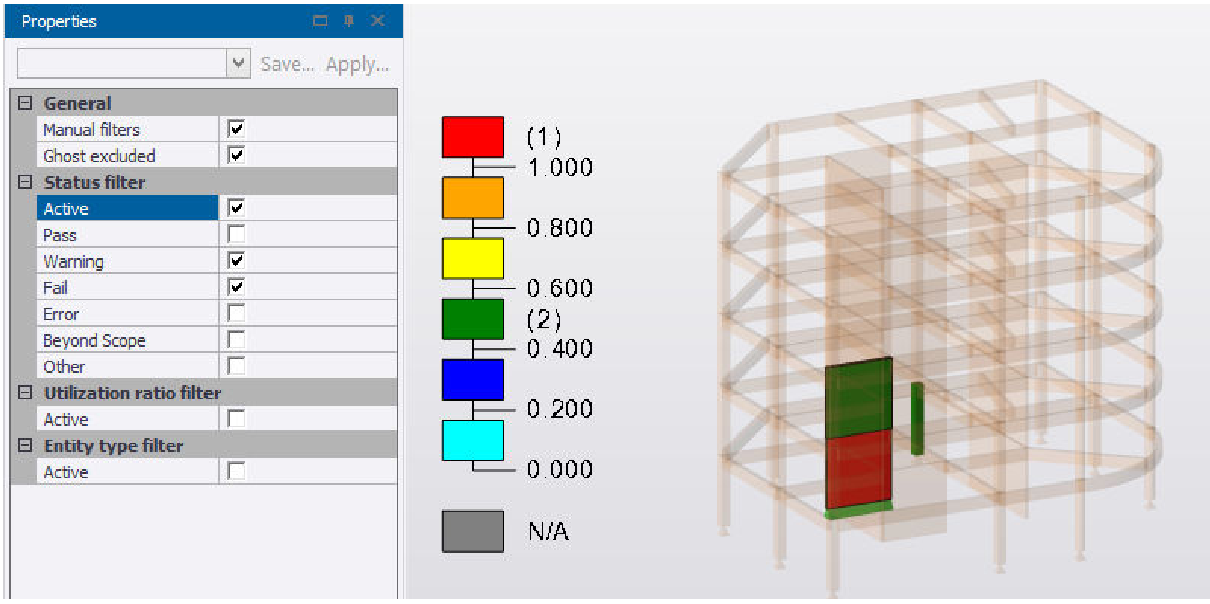


If required, you can also apply the **Status filter** when investigating ratios. This is achieved as follows:

1. From the **Review** ribbon select **Ratio**
2. Select **Manual filters** in the **Properties** window.

   The **Status filter** group is displayed.
3. Select **Active** and choose the statuses by which to filter the display of ratios.

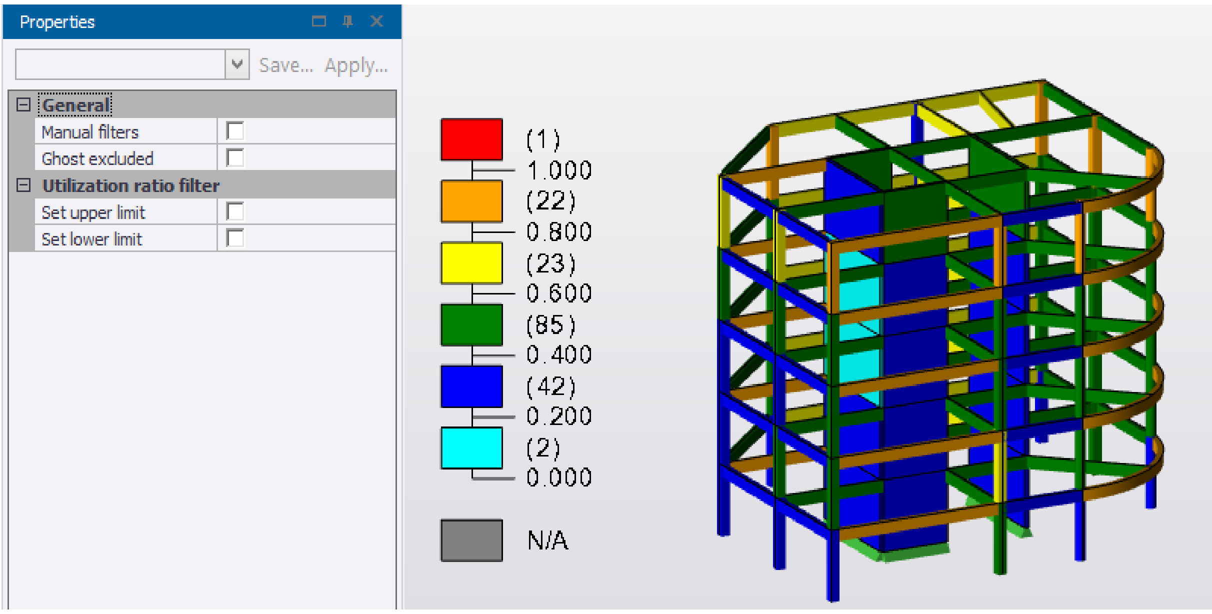
   The result is that ratios are then only displayed for those members in the chosen **Status filter** categories.



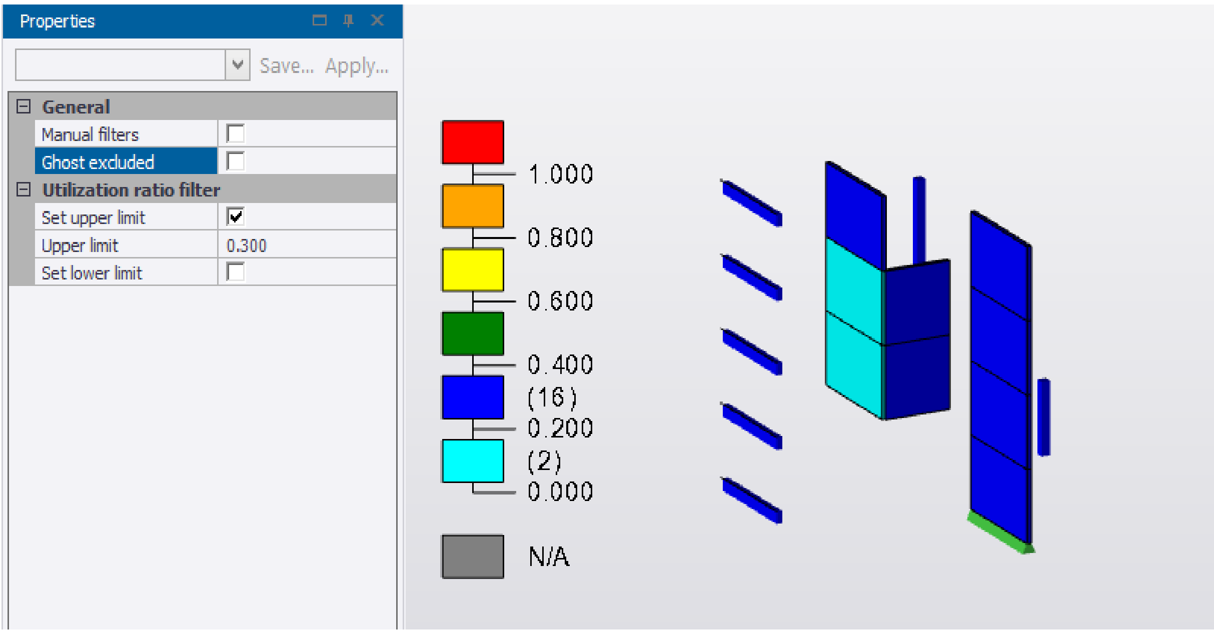
Working with the Utilization ratio filter

All entities that have been designed are assigned a utilization ratio. The **Utilization ratio filter** allows you to display only those entities within a given utilization range.

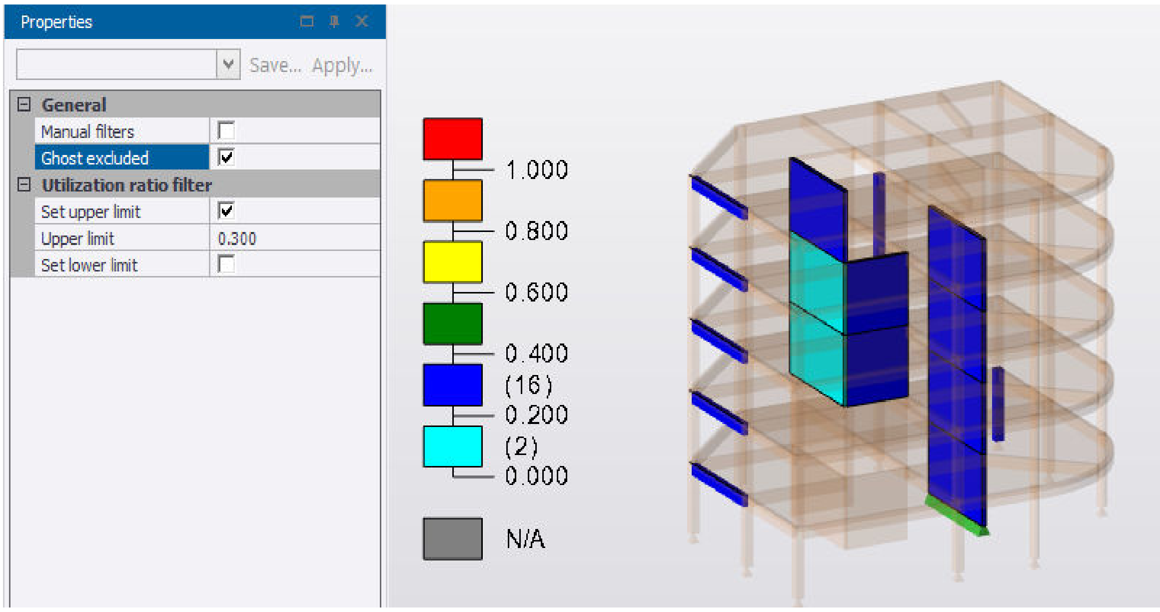
This filter is always active in the **Properties** window when **Ratio** is selected from the **Review** ribbon.



Initially entities with any ratio are displayed, but by selecting upper and/or lower limits, entities outside the defined range are hidden in the **Review View**.



By selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

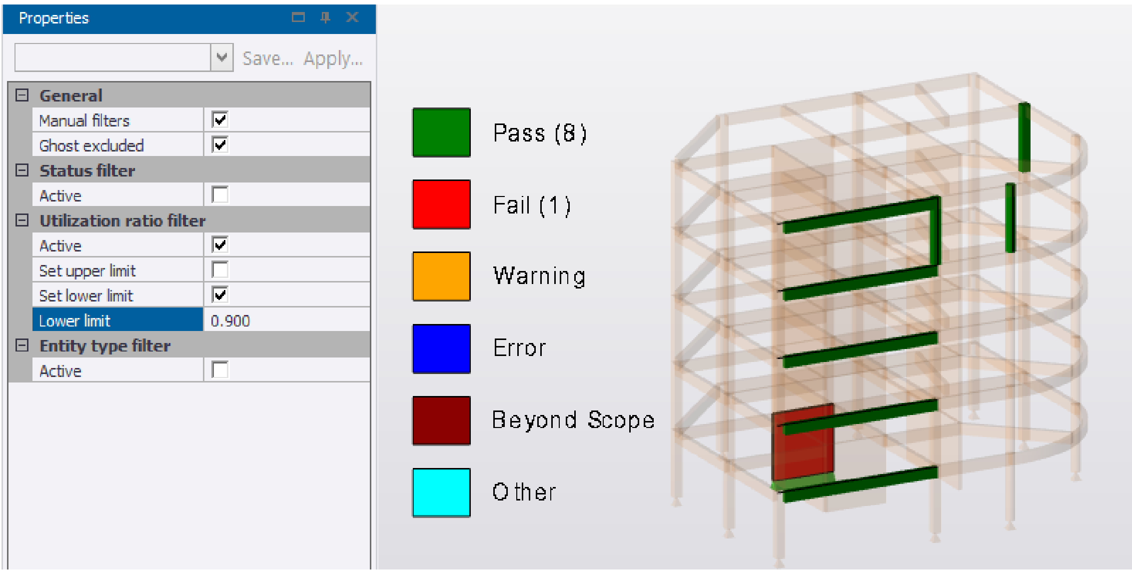


If required, you can also apply the **Utilization ratio filter** when investigating statuses. This is achieved as follows:

1. From the **Review** ribbon select **Status**
2. Select **Manual filters** in the **Properties** window.

   The **Utilization ratio filter** group is displayed.
3. Select **Active** and set the upper and/or lower limits by which to filter the display.

   The result is that statuses are then only displayed for those members within the specified utilization ratio limits.



Working with the Entity type filter

Unless they have been unselected in **Scene Content**, all relevant entity types are initially displayed when either **Status** or **Ratio** are selected from the **Review** ribbon.

You can use the **Entity type filter** to focus on specific types only, it is made active as follows:

1. Select **Manual filters** in the **Properties** window

   The **Entity type filter** group is displayed.

2. Select **Active**

A list of designable entity types is displayed in the **Properties** window.

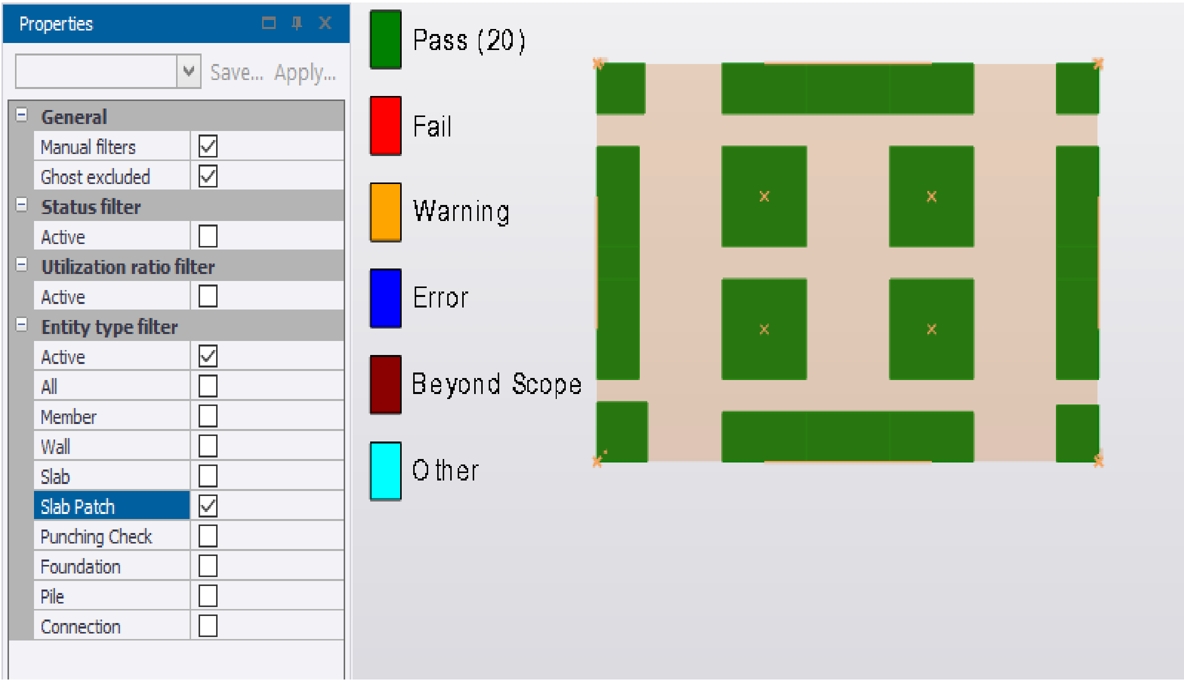
Deselecting an entity type from the list causes entities of that type to either be hidden in the **Review View**, or displayed as ghosted if **Ghost excluded** is selected.

NOTE Clicking 'All' toggles between all entity types being selected, or unselected.

Example:

The below view shows slab status at a particular level. Initially, the **Entity type filter** is not active and as a result the **slab**, **slab patch**, and **punching check** statuses are all displayed making interpretation difficult. Note that a Fail is reported, but it is not immediately apparent where it is.

By activating the **Entity type filter** you can investigate the status for specific entity types. The below view is filtered for patches only, which are all passing.



The below view is filtered for punching checks only and clearly reveals where the Fail occurs.

9.2 Review model properties (show/alter state)

The **Show/Alter State** group of commands on the **Review** toolbar allow you to graphically review and modify various model properties. For each of the commands, entities are color coded in accordance with a legend, and the entity color changes to reflect any changes as they are made.

Those **Show/Alter State** commands that can be accessed directly from the toolbar are as follows:

- Auto/Check Design (page 865) - to graphically review or modify autodesign settings of a member or slab
- Diaphragm On/Off - to graphically review or modify diaphragm settings
- Fixed/Pinned (page 867) - to graphically review or modify member end fixities
- BIM Status (page 867) - to graphically assess the BIM Status, and also to exclude members and panels from the import/export process
- Slab / Foundation Reinforcement - to graphically review and edit or rationalize panel, patch or foundation reinforcement
- Section/Material Grade (page 870) - to graphically copy section sizes, or, (by changing the Attribute), material grades (page 870)

- Copy Properties (page 871) - to copy a specified element parameter (such as web openings, shear connectors, or transverse reinforcement) from a source member to valid target members.
- Report Filter (page 871) - to graphically review and modify report filters
- Sub Structures (page 872) - to graphically review and modify sub structures
- Concrete Beam Flanges (page 872) - to graphically set whether flange are considered
- Column Splices (page 872) - to graphically review and modify splice positions within steel columns.
- Property Sets - to graphically review or apply property sets to members, slab items and roof panels.
- UDA - to graphically review and modify the values of user defined attributes that have been attached to the model.
- Show/Alter State (page 873) - to review and modify various model properties

Modify autodesign settings

You can graphically modify autodesign settings of members and slabs from a **Review View**.

To modify the autodesign settings of a member or slab, do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:
    a. Click **Auto/Check Design**, or,
    b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

    Members and slabs are color coded according to their auto design setting.
3. In the **Properties** window, select the appropriate mode.
4. Do one of the following:
    - Click an individual entity to modify its auto design setting.
    - Hold down the left mouse button and drag a box from left to right to modify the auto design setting of all members within the box.
    - Hold down the left mouse button and drag a box from right to left to modify the auto design setting of all members within the box or cut by the box.

Review and modify diaphragm settings

The **Diaphragm On/Off** command on the **Review** tab allows you to select whether the slab items, roof panels, and nodes in the model are included

Review the diaphragm settings

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.

Modify the diaphragm settings of slab items or roofs

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.
3. In the **Properties** window, set **Entity type** to **Slab Item** or **Roof**.
4. Do one of the following:
   - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
   - Hold down the left mouse button and drag a box from left to right to switch between including and excluding slab items and roof panels within the box from the diaphragm.
   - Hold down the left mouse button and drag a box from right to left to switch between including and excluding slab items and roof panels within the box or cut by the box from the diaphragm.

Include or remove solver nodes from the diaphragm

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or

b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.

3. In the **Properties** window, set **Entity type** to **Solver Node**.
4. Do one of the following:
   - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
   - Hold down the left mouse button and drag a box to switch between including and excluding the nodes within the box in diaphragm.

Modify end fixity

The **Fixed / Pinned** command allows you to graphically review and modify the end fixity of all members in the model.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Fixed / Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Fixed / Pinned** is selected.
5. With the **Mode** set as **Toggle**, do one of the following:
   - Click an individual member to switch its end fixity between the valid end fixity types.
   - Hold down the left mouse button and drag a box from left to right to alter the fixity settings of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to alter the fixity settings of all members within the box or cut by the box.
6. Alternatively, with the **Mode** set as **Set On**, select the **FixityType** required and then click an individual member, or drag a box, as above to apply the chosen fixity.

NOTE If the end fixity of a member is labeled **Cantilever**, or **N/A**, the end fixity at end 1 of the element is different from the fixity at end 2. You can only edit a mixed end fixity by right-clicking on the member and editing its properties in the Properties dialog.

Modify BIM status

The **BIM Status** command allows you to graphically modify the BIM status and to exclude members and panels from the BIM import or export process.

To include or exclude members and panels from the BIM process, do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **BIM Status**.
3. Do one of the following:
   - Click an individual member or panel once to exclude it or twice to re-include it.
   - Hold down the left mouse button and drag a box from left to right to modify the exclude setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the exclude setting of all members within the box or cut by the box.

Copy or modify slab and foundation reinforcement

You can graphically review and modify the bar/mesh size and spacing applied in each layer and direction for slab items, mats, patches and isolated foundations. Tekla Structural Designer also allows you to copy reinforcement from one slab item, mat, patch or isolated foundation to another.

TIP If you open two **Review View**s side by side, you can then use the first one to modify the reinforcement while displaying the design status in the second one. This way, each change you make to the reinforcement immediately updates the design status.

Copy reinforcement

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:

   a. Click **Auto/Check Design**, or,

   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.
3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.

5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to copy.
7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. Click the slab item or patch whose reinforcement you want to copy.
10. Click the slab items or patches to which you want to copy the reinforcement.

    The slab reinforcement is copied to the selected slab items or patches.

    TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Modify reinforcement

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, either:
    a. Click **Auto/Check Design**, or,
    b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

    Members and slabs are color coded according to their auto design setting.
3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.
5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to modify.
7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. If you want to modify bars, in **Bar Parameters**, select the properties that you want to modify.
10. In **Apply**, select the bar properties that you want to apply to the slab.
11. Click the slab item or patch whose reinforcement you want to update.

    TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Copy section sizes

If necessary, you can graphically copy the section size applied to a member to other members in the model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Section**.
5. Set **Mode** to **Copy**.
6. In **Material type**, select the desired material type.

   NOTE You can use a characteristic filter to reduce the elements displayed in the model.

7. In the model, click the element from which you want to copy a section size.
8. Click the elements to which you want to copy the section size.

   The section size is copied to the selected elements.

Copy material grades

If necessary, you can graphically review the material grade applied to the members in your model. Tekla Structural Designer also allows you to copy the grade of one member to another member with the same material type . For more information, see the following paragraphs.

RESTRICTION You can only copy material grades to and from steel, cold formed, cold rolled, concrete, or timber members.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Material Grade**.
5. In **Material Type**, select the desired material type.

   TIP If necessary, use a characteristic filter to reduce the elements displayed in the **Review View**.

6. Set **Mode** to **Copy**.

7. Click the element whose grade you want to copy.
8. Click the elements to which you want to copy the grade.

   The material grade is copied to the selected elements.

Copy properties

The **Copy Properties** command allows you to graphically copy a specified element parameter (such as web openings, shear connectors, or transverse reinforcement) from a source member to other valid target members. To copy properties from one member to another, see the following instructions.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Copy Properties**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select the parameter that you want to copy.
5. In the model, click the member from which you want to copy the parameter.
6. Do one of the following:
   - Click an individual member to which you want to copy the parameter.
   - Hold down the left mouse button and drag a box around multiple members to which you want to copy the parameter.

   Tekla Structural Designer copies the parameter to the selected members.

Review and modify member filters

The **Report Filter** command allows you to graphically review and modify report filters that you have defined for the members in your model. The **Report Filter** command remains unavailable until you have defined a member filter.

TIP To define a report filter, do the following:

1. Go to the **Report** tab.
2. In the **Filters** group, click **Members**.
3. Define the member filter according to your needs.

---

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Report Filter**.
3. Do one of the following:

- Click an individual member to switch between including and excluding the member from the selected member filter.
- Hold down the left mouse button and drag a box from left to right to switch between including and excluding the members within the box from the selected member filter.
- Hold down the left mouse button and drag a box from right to left to switch between including and excluding the members within the box or cut by the box from the selected member filter.

Review sub structures

The **Sub Structures** command allows you to graphically create, review and modify sub structures for modeling purposes. The command can be especially useful in large models, as individual sub structures can then be differentiated by color and worked on in separate views.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Sub Structures**.

   The elements in the model are color coded based on whether they belong to sub structures.

See also

Manage sub structures (page 1031)

Review concrete beam flanges

The **Concrete Beam Flanges** command allows you to graphically review the flanges of concrete beams. See detailed instructions on how to review conrete beam flanges in the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Concrete Beam Flanges**.

   Each concrete beam is color coded to indicate if its flanges are considered and flange widths determined.

Use of beam flanges

Review and modify column splice positions

The **Column Splices** command allows you to graphically review and modify splice positions within steel columns. For detailed instructions on how to review and modify splice positions, see the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Column Splices**.

   All potential splice locations are color coded to indicate if they are on or off.
3. Do one of the following:
   - Click a potential splice location to alter its setting between on or off.
   - In a 2D view, hold down the left mouse button and drag a box to alter the setting of multiple splices simultaneously.

Review and apply property sets

The command provides a means to graphically review or apply property sets to entities.

- Review where property sets have been applied (page 1024)
- Apply property sets to existing entities (page 1023)

Copy or modify user-defined attributes

If necessary, you can graphically review and modify the values of attributes that have been attached to the model.

Show/alter state

The **Show/Alter State** commands allow you to efficiently review and modify various model properties. For each of the commands, entities are color coded in accordance with a legend, and the entity color changes to reflect any changes as they are made.

In the **Properties** window, you can use filters for most **Show/Alter State** commands. The filters allow you to control both the visibility of entities by their current setting and their adjustability by their type and characteristic. Inactive but visible entities are shaded a light transparent grey whose opacity you can also control, if necessary.

To access the **Show/Alter State** commands, do the following:

1. Open a **Review** view.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. Choose the command required from the **Attribute** list:

Option	Description

Active (page 875)	set single span members to be active/inactive
All properties (page 871)	copy all properties between members
Assume cracked (page 875)	set concrete members as either cracked or uncracked
Autodesign (page 865)	review or modify autodesign settings
BIM Status (page 867)	modify BIM status, or exclude members/panels from BIM import/export
Bracing/Braced (page 878)	modify slenderness settings in each direction for concrete columns and walls
Cantilever end (page 878)	mark beam and column ends as cantilevers
Deflection Limits (page 879)	review and copy specified deflection limits between steel beams
Diaphragm (page 865)	select the slab items, roof panels, and nodes to be included in diaphragms
Drift check	review check status, or specify members to be checked
Fixed/Pinned (page 867)	review and modify member end fixity
Gravity only (page 880)	review and modify the gravity only setting of beams and columns
Material Grade (page 870)	review and copy the material grade between members
Punch check position (page 880)	modify assumed punching shear check positions
Copy quick connector layout (page 881)	copy the connector layout from one beam to others
Restraints	review and modify lateral restraint settings of steel beams, columns, trusses and portal frames
Rotational stiffness (page 890)	review and modify rotational stiffness settings of beam ends
Section (page 870)	review and copy section sizes between members
SFRS Type and Direction	review or modify SFRS type and direction
Shear Connectors (page 891)	copy shear connector properties between composite beams
Size Constraints	review and copy specified size constraints for steel beams, columns, and braces
SidePlates (page 892)	review the status and locations where SidePlate connections have been applied to steel beams
Stud auto layout (page 893)	modify the stud auto layout setting of composite beams
Sway check	review check status, or specify members to be checked
Transverse reinforcement (page 894)	modify the transverse reinforcement of composite beams
UDA (page 873)	review and modify user defined attributes applied to members
User defined U/R	review and modify user defined utilization ratios
Web Openings (page 896)	copy web opening properties between members
Westok Openings (page 896)	copy westok opening properties between members
Wind drift	review check status, specify members to be checked and set the limit

TIP You can use **Ctrl** + **A** as a shortcut to quickly scroll the **Attribute** list.

Modify active / inactive settings

Initially all members are active. If a member is set inactive it does not participate in the analysis but any load applied to (or decomposed) to it is still accounted for.

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each therefore have an **Active** setting in the properties which defaults to 'on', but can be unchecked. In the **Review View** these members are color coded allowing the same setting to be toggled graphically.

NOTE Members cannot be inactive if:

- They are multi-span
- They support another member (active or inactive)

To modify the active setting of a member graphically do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State** .
3. Go to the **Properties** window.
4. In the **Attribute** list select **Active**.

   Members are color coded according to their active setting.
5. In the **Properties** window, select the appropriate mode.
6. Do one of the following:
   - Click an individual entity to modify its active setting.
   - Hold down the left mouse button and drag a box from left to right to modify the active setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the active setting of all members within the box or cut by the box.

Modify assumed cracked settings

You can graphically review and modify the assumed cracked setting of the concrete beams, columns and walls in a Review View using Show/Alter State - *Assume cracked*.

Set or toggle assumed cracked settings

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to one of the following:
    - **Set cracked** - click or box around concrete member(s) to set to cracked.
    - **Set uncracked**- click or box around concrete member(s) to set to uncracked.
    - **Toggle** - click or box around concrete member(s) to toggle between cracked or uncracked.

    The member color coding updates to reflect the cracked or uncracked status.

Review Wall Stress

NOTE The following limitations and assumptions apply to **Review Wall Stress**

1. Only meshed wall panels are considered, mid-pier walls are excluded.
2. The analysis type specified in **Design Settings>Analysis** determines which set of analysis results are used for the review.
3. For the above analysis type, the loadcases and combinations considered are those from the **last performed building analysis of that type.**
    - This gives you the flexibility to review the stresses for a subset of loadcases/combinations if required.
    - By running Analyse All you can ensure all active gravity, lateral and seismic (not seismic RSA) combinations have been considered.
        - Stresses are available for seismic but not for seismic RSA combinations - these are beyond scope. (Review Wall Stress is more particularly aimed at determining the cracked status of panels under wind loading).
4. Chasedown results are **not** considered.
5. Axial load reductions are always taken into account.
6. The maximum stress value from all nodes for all analysed cases/combinations is reported in a tooltip for each panel. This value is used to determine if the stress threshold has been exceeded.
    - The reinforcement content in the panel is not considered in determining the stress value.
7. The loading droplist in the status bar does not filter the status for individual cases/combinations.

To review/update the cracked status of all wall panels to be compatible with the stress levels for the loadcases/ combinations that have been analysed, proceed as follows:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Review Wall Stress**.
6. Check the yellow command prompt to confirm which analysis type is being used for the review, (this will be the type currently specified in **Design Settings>Analysis**).

Review Wall Stress Assume cracked (First-order linear): Select entity to Set compatible

7. Set the **Result type** to control which factors to use in combinations for the review.
    - **Strength** factors, or
    - **Service** factors
8. Set the **Stress type** to be considered.
    - **Max tension - Y**, or
    - **In-plane tension - Y**
9. Set the **Stress threshold** above which walls should crack. The existing cracked/uncracked status of each panel is displayed, along with an indiation of whether the stress threshold has been exceeded.
10. To make panel status compatible with the stress threshold for one or more panels:
    a. click an individual panel, or
    b. box around multiple panels, or
    c. to make all panels compatible, click then click the Auto update button

    Panel status is made compatible, i.e.

    - If uncracked and the stress threshold has been exceeded, the status is changed to cracked.
    - If cracked and the stress threshold has not been exceeded, the status is changed to uncracked.

See also

Concrete member cracked or uncracked status (page 1286)

Modify slenderness settings

The **Braced/Bracing** command provides a means to graphically review and modify the slenderness settings in each direction for concrete columns and walls.

To modify slenderness settings, do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Braced/Bracing**.
5. Select the direction under consideration.

   Members are color coded according to their slenderness setting in the selected direction.
6. Do one of the following:
   - Click an individual column or wall to modify its slenderness setting between braced and bracing.
   - Hold down the left mouse button and drag a box from left to right to modify the slenderness setting of all concrete columns and walls within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the slenderness setting of all concrete columns and walls within the box or cut by the box.

Apply cantilever ends

The **Cantilever end** command allows you to graphically apply cantilever ends where required to beams and columns.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Fixed / Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Cantilever end**.
5. Set **Mode** to **Set On**, **Set Off**, or **Toggle** as required.
6. Do one of the following:
   - Click an individual member end to set/unset a cantilever end as per the **Mode**.
   - Hold down the left mouse button and drag a box from left to right to alter the cantilever end settings of all members entirely within the box.

- Hold down the left mouse button and drag a box from right to left to alter the cantilever end settings of all members within the box or cut by the box.

Review and copy deflection limits

You can graphically review specified deflection limits and copy the limits from one steel beam to others.

Review deflection limits

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Review**.
6. Set **Load type** to the load type that you want to review.

Copy deflection limits

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Copy**.
6. In the model, click the beam from which you want to copy the deflection limits.
7. Click the beam to which you want to copy the deflection limits.

   The deflection limits are copied to the selected beam.
8. Continue copying the limits by clicking the desired beams or press **Esc** to finish.

Review and modify drift checks

You can graphically review drift check results and set the members to be checked for columns and walls.

Review drift checks

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Review**.

Set drift checks on or off

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Modify gravity only settings

If necessary, you can review and modify the gravity only setting of the beams and columns in your model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Gravity only**.

   All members are color coded to indicate their gravity only setting.
5. Do one of the following:
   - Click a steel beam or column to set its gravity only setting on or off.
   - Hold down the left mouse button and drag a box from left to right to set on or off the gravity only setting of all steel beams and columns within the box.
   - Hold down the left mouse button and drag a box from right to left to set on or off the gravity only setting of all steel beams and columns within the box or cut by the box.

Modify punching shear check position

If necessary, you can graphically modify the assumed punching shear check position. This allows you to determine the loaded perimeter when the perimeter is close to a free edge.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Punch check position**.
5. To change the punching shear check position, do one of the following:

To	Do this
Switch the punching shear check position between Edge Z and Internal	• Click the Z edge of a punching check item.
Switch the punching shear check position between Edge Y and Internal	• Click the Y edge of a punching check item.
Switch the punching shear check position between Corner, Edge Z, Edge Y and Internal	• Click a corner of a punching check item.

NOTE Clicking an internal punching check item does not change the punching shear check position.

TIP You can also box around multiple punching check item to switch their position. Do one of the following:

- Hold down the left mouse button and drag a box from left to right to switch the position setting of all punch check items within the box.
- Hold down the left mouse button and drag a box from right to left to switch the position setting of all punch check items within the box or cut by the box.

Copy quick connector layout

You can graphically copy the connector layout from one beam to another. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Quick Connector Layout**.
5. Click on the beam with the layout that you want to copy (must be valid).

   The composite beams in the model are color coded to indicate the source and valid targets.

6. To select the beams to apply the layout to, do one of the following:
    - Click an individual valid target beam.
    - Hold down the left mouse button and drag a box from left to right to totally enclose the target beams.
    - Hold down the left mouse button and drag a box from right to left to select target beams within the box or cut by the box.

    The connector layout is copied from the source to the target beam(s).

Review and modify restraints

By selecting the **Restraints** attribute in **Show/Alter State** you can graphically review and modify continous and discrete restraints for beams, columns, trusses and portal frames.

TIP Graphical editing of restraints will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review restraints

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Review**. For details of the restraint type choices etc. that are presented in Review mode, see the **Options overview** topics below.
6. Set the **Continuous** box as required:
    - Checked: shows continuous restraints, each sub-member being colour coded in accordance with the legend.
    - Unchecked: shows discrete restraints, each internal node at which other members connect being colour coded in accordance with the legend.
7. Set the Restraint Type that you want to review.
8. Set the Entity type that you want to review.

Toggle restraints

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.

5. Set **Mode** to **Toggle**. For details of the restraint type choices etc. that are presented in Toggle mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.
7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. In the model:
   - click the member at the location where want to toggle the restraint.
   - drag a box around multiple members to toggle the restraint at all valid locations within the box.

   The type of restraint at the selected location(s) changes to the next rerstraintr type shown in the legend. (Simply click again as necessary until the desired restraint type has been applied.)
10. Continue toggling the restraints by clicking the desired members or press **Esc** to finish.

Set restraints

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Set**. For details of the restraint type choices etc. that are presented in Set mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.
7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. Set the **Restraint** as required.
10. In the model:
    - click the member at the location where want to set the restraint.
    - drag a box around multiple members to set the same restraint at all valid locations within the box.
11. Continue setting the checks by clicking/boxing the desired members or press **Esc** to finish.

Options overview (US headcode)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description	
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints. • **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.	
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.	
Restraint Type	The restraint type determines which page(s) of properties are being edited. • **LTB** - enables the review/setting of the check-boxes located on the **Restraints (LTB)** property page. • **Compression** - enables the review/setting of the check-boxes located on the **Restraints (Comp)** property page. • **LTB & Compression Out of Plane** - enables the simultaneous review/setting of check-boxes on both the the **Restraints (LTB)** and **Restraints (Comp)** property pages.	
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.	
	For **LTB** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked.	

Command or option	Description	
	• **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.	
	For **LTB** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked. • **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.	
	For **Compression** restraint types: • **Out of Plane** - Out of plane check-box is checked, In plane check-box is unchecked. • **In Plane** - In plane check-box is checked, Out of plane check-box is unchecked. • **Out of Plane & In Plane** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.	
	For **LTB & Compression Out of Plane** restraint type: • **Top & Bottom & Out of Plane** - Top flange, Bottom flange, and Out of Plane are checked. • **Top & Out of Plane** - Top flange and Out of Plane are checked, Bottom flange is unchecked. • **Bottom & Out of Plane** - Bottom flange and Out of Plane are checked, Top flange is unchecked. • **Unrestrained** - Top flange, Bottom flange, and Out of Plane are checked.	

Command or option	Description	
	NOTE For each of the above, the In Plane check-box is unaffected, retaining its existing setting.	

Options overview (other head codes)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description	
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints. • **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.	
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.	
Restraint Type	The restraint type determines which page(s) of properties are being edited. • **Lateral** - enables the review/setting of the check-boxes located on the **Lateral restraints** property page. • **Strut** - enables the review/setting of the check-boxes located on the **Strut restraints** property page • **Lateral & Strut Minor** - enables the simultaneous review/setting of check-boxes on both the the **Lateral restraints** and **Strut restraints** property pages.	
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.	

Command or option	Description	
	For **Lateral** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked. • **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.	
	For **Lateral** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked. • **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.	
	For **Strut** restraint types: • Major - Major check-box is checked, Minor check-box is unchecked. • Minor - Minor check-box is checked, Major check-box is unchecked. • Major and Minor - both check-boxes are checked. • Unrestrained - both check-boxes are unchecked.	
	For **Lateral & Strut Minor** restraint type: • **Top & Bottom & Minor** - Top flange, Bottom flange, and Minor are checked. • **Top & Minor** - Top flange and Minor are checked, Bottom flange is unchecked. • **Bottom & Minor** - Bottom flange and Minor are checked, Top flange is unchecked.	

Command or option	Description	
	• **Unrestrained** - Top flange, Bottom flange, and Minor are checked. **NOTE** For each of the above, the Major check-box is unaffected, retaining its existing setting.	

Portal Frame options (US headcode)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description	
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.	
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Out of Plane are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked. • **Unrestrained** - Top flange, Bottom flange, and Out of Plane are unchecked. Columns: • **Torsional** - Face A, Face C, and Out of Plane are checked. • **Outer** - Face A is checked, Face C is unchecked. • **Inner** - Face A is unchecked, Face C is checked.	

Command or option	Description	
	• **Unrestrained** - Face A, Face C, and Out of Plane are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.	

Portal frames options (other head codes)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description	
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.	
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Minor are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked. • **Unrestrained** - Top flange, Bottom flange, and Minor are unchecked. Columns: • Torsional - Face A, Face C, and Minor are checked. • Outer - Face A is checked, Face C is unchecked. • Inner - Face A is unchecked, Face C is checked.	

Command or option	Description	
	• Unrestrained - Face A, Face C, and Minor are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.	

Related video

Show/Alter State Restraints

Apply rotational stiffness to a beam end

The **Rotational stiffness** command allows you to graphically apply partially fixity/linear springs to the ends of beams.

Related video

Automatic modelling of partial fixity

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Fixed / Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Rotational stiffness**.
5. Set **Mode** to **Set On**.
6. Set **Direction** to **Major**, or **Minor** as required.
7. Set **Rotational stiffness** to **Partially fixed**, or **Spring linear** as required.
8. If **Partially fixed**, set the percentage fixity value; or if **Spring linear** set the spring stiffness value as required.
9. To apply the new setting do one of the following:
    - Click an individual beam end.
    - Hold down the left mouse button and drag a box from left to right to apply to all beam ends entirely within the box.
    - Hold down the left mouse button and drag a box from right to left to apply to both ends of all beams within the box or cut by the box.

Review and modify SFRS settings

If necessary, you can graphically review and modify the the types and directions of seismic force resisting systems (SFRS) in the model. For more information, see the following paragraphs.

Review SFRS type and direction settings

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS type and direction settings.

Modify SFRS type & direction settings

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS direction setting.
5. Set **Mode** to one of the following:
   - **Update Type & Direction**: includes a member in the selected SFRS type & direction.
   - **Remove from SFRS**: removes the member from SFRS.
6. Click the desired member.

   Depending on the set mode, Tekla Structural Designer includes the member in the SFRS type & direction, or removes the member from SFRS.

Copy shear connectors

If necessary, you can copy the composite beam shear connector properties from a source beam to other composite beams. Do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Shear Connectors**.
5. In the model, click the composite beam from which you want to copy the shear connector properties.

6. Click the composite beams to which you want to copy the shear connect properties.

   The properties are copied to the selected beams.

Modify SidePlates

The **SidePlates** command allows you to graphically review and modify the SidePlate connection status.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **SidePlates**.
3. Do one of the following:
   - Click on the first half of a beam to reset end 1, the second half to reset end 2 and the middle to reset both ends simultaneously.
   - Hold down the left mouse button and drag a box from right to left to reset all the beam ends within the box or cut by the box.

See also

SidePlate connections (page 825)

Review and copy size constraints

If necessary, you can graphically review specified size constraints for steel beams, columns, and braces, and copy the constraints from one steel member to others. For more information, see the following paragraphs.

Review size constraints

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.
5. Set **Mode** to **Review**.
6. In **Size constraint**, select the size constraint that you want to review.

Copy size constraints

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.

5. Set **Mode** to **Copy**.

   TIP Select the **By constraint type** option if you want to only copy an individual constraint type.

6. In the model, click the member from which you want to copy the size constraints.
7. Click the member to which you want to copy the size constraints.
8. Continue copying the size constraints by clicking the desired members or press **Esc** to finish.

Modify stud auto layout

If necessary, you can graphically modify the stud auto layout setting of composite beams in the model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Stud auto layout**.

   The composite beams in the model are color coded according to their auto layout setting.
5. Do one of the following:
   - Click an individual composite beam to alter its auto layout setting.
   - Hold down the left mouse button and drag a box from left to right to alter the auto layout setting of all composite beams within the box.
   - Hold down the left mouse button and drag a box from right to left to alter the auto design setting of all composite beams within the box or cut by the box.

Review and modify sway checks

You can graphically review sway check results and set the members to be checked for columns and walls.

Review sway checks

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Review**.

Set sway checks on or off

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Copy transverse reinforcement

If necessary, you can copy beam transverse reinforcement from one composite beam to others in your model. Do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Transverse reinforcement**.

   The composite beams in the model are color coded based on whether you can copy transverse reinforcement properties from them.
5. Click the composite beam from which you want to copy transverse reinforcement properties.
6. Click the composite beams to which you want to copy the transverse reinforcement properties.

   The properties are copied to the selected beams.

Review and modify user defined U/R

From a **Review View** you can graphically review user defined utilization ratios, set the members to which they are applied, and set whether they apply to autodesign alone, or both autodesign and check.

Review user defined U/R

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.

3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.

Apply a user defined U/R for autodesign only

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.
9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.
10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Apply a user defined U/R for autodesign and check

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign+Check**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.
9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.
10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Turn off user defined U/R

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Set Off**.
6. If required, set **Entity type** and **Material** to make the command easier to apply.
7. In the model, click those column/wall stacks, beam spans, or objects for which you want to turn off user defined U/R.
8. Continue by clicking the desired stacks/spans/objects or press **Esc** to finish.

See also

Apply user defined utilization ratios (page 786)

Copy web openings

If necessary, you can copy web opening properties from one member to others. Do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Web Openings**.

   The members in the model are color coded based on whether you can copy web opening properties from them.
5. In the model, click the member from which you want to copy web opening properties.
6. Click the members to which you want to copy the web opening properties.

   The properties are copied to the selected members.

Copy westok openings

If necessary, you can copy westok opening properties from one member in your model to others. Do the following:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Westok Openings**.

   The members in the model are color coded based on whether you can copy westok opening properties from them.
5. In the model, click the member from which you want to copy westok opening properties.

6. Click the members to which you want to copy the westok opening properties.

   The properties are copied to the selected members.

Review and modify wind drift checks

You can graphically review wind drift check results, set the members to be checked and set the check limits for columns and walls.

Review wind drift checks

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Review**.

Set wind drift checks on or off

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Set wind drift checks on and set the ratio limit

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Set On**.
6. Select **Set limit ratio**.
7. Specify the required **Wind drift ratio limit**

8. In the model, click the column or wall for which you want to set the limit.
9. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

9.3 Review tabular data

The **Tabular Data** command allows you to view design summaries, sway & drift results, material lists, connection resistances and more in tabular views. Tabular data can be filtered, sorted and exported to Excel.

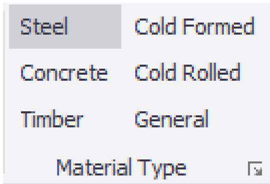
- Review design summary tabular results (page 898)
- Review sway check tabular results (page 900)
- Review story shear tabular results (page 900)
- Review drift check tabular results (page 901)
- Review wind drift check tabular results (page 902)
- Review connection resistance tabular results (page 817)
- Review material list tabular results (page 902)
- Review floored area tabular results (page 929)
- Filter tabular data (page 929)
- Export tabular results to Excel (page 930)

Review design summary tabular results

To review a tabular design summary:

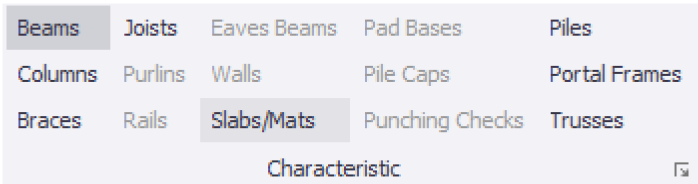
1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Design Summary**
4. In the **Result Type** group, select the analysis type for which the design summary is required.
5. In the **Material Type** group, select the material.



The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.

6. In the **Characteristic** group, select the required characteristic.



The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.

7. If applicable, in the **Construction** group, select the construction.

The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

8. In the **Fabrication** group, select the fabrication.

The tabular design summary is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 929)

See also

Export tabular results to Excel (page 930)

Review member design (page 849)

Review sway check tabular results

For those stacks to which the check has been applied, the elastic critical load factor is calculated in both directions.

NOTE Sway results are not relevant to models that use the ACI/AISC head code.

To review tabular sway check results:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Sway**
4. In the other groups of the **Review Data** ribbon, make the necessary selections to filter the results.

TIP If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the α_{Dir1} header until the columns are arranged in increasing order, then repeat for α_{Dir2} .

See also

Review and modify sway checks (page 893)

The sway check (page 1152)

Review story shear tabular results

To create a tabular floored area summary:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Inter-story Shear** or **Cumulative Story Shear** as required.

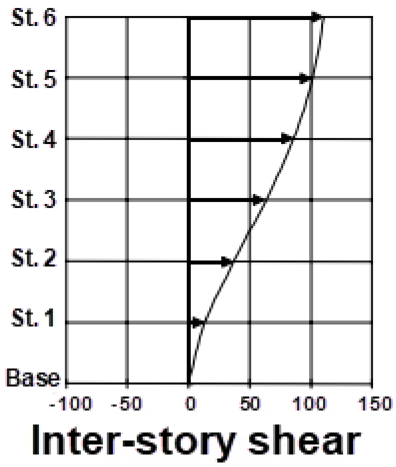
A table of story shears is displayed.

See also

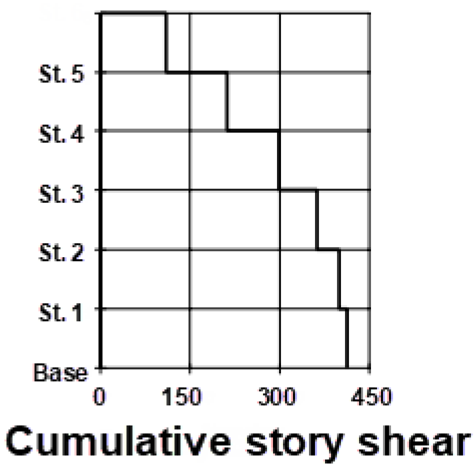
Inter-story shear and cumulative story shear (page 901)

Inter-story shear and cumulative story shear

The Inter-story Shear (also known as Floor Forces) table displays the sum of the lateral forces applied at each level in the structure.



Cumulative story shears are calculated by summing the floor forces from the top to the bottom of the structure, consequently they should get bigger as you move down.



Review drift check tabular results

NOTE Drift results are only relevant to models that use the ACI/AISC head code.

To review tabular drift check results:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Drift**
4. In the other groups of the **Review Data** ribbon, make the necessary selections to filter the results.

See also

Review and modify drift checks (page 879)

The drift check (page 1154)

Review wind drift check tabular results

To review tabular wind drift check results:

1. Open a view and change the view regime (page 280) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Wind Drift**
4. In the other groups of the **Review Data** ribbon, make the necessary selections to filter the results.

See also

Review and modify wind drift checks (page 897)

The wind drift check (page 1156)

Review material list tabular results

The material content of the model can be readily assessed in the form of a **Material List**. This can be created and viewed either in a *tabular data view* or a *report*, according to your needs.

- Material List tabular data view:
  - created for a specific material and entity type
  - displayed on screen (not suitable for direct printing)
  - can be sorted in ascending/descending order of a column by clicking a column heading
  - row content can be located within a 3D view by double-clicking a row
  - can be exported to a spreadsheet
- Material List report:
  - created for *multiple* materials and entity types
  - displayed in a format suitable for printing
  - can be exported to a spreadsheet

Two levels of content can be generated:

- **Summary Only** - this is the default level and is very condensed, for example:
  - total quantities only for each beam/ column section size
  - total quantities for each slab
  - total weight of reinforcement (by mass, and by mass/unit volume)
- **Detailed** - this is a longer report, for example:
  - quantities for each span/ stack/ panel for each concrete beam / column / wall
  - quantities for each section size of a given length for beams and columns in other materials
  - quantities for each slab item
  - quantities for individual reinforcement bar/mesh types

The topics in the section all relate to the tabular data view. To find out how to create a material list report, see: Material Listing report (page 959)

Create material list tabular results

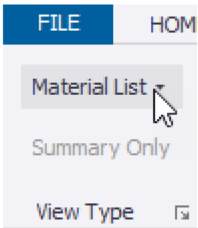
To create a material list as tabular data:

1. Open a view and change the view regime (page 280) to **Review View**.

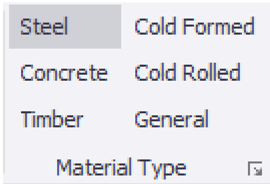
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Material List**



4. In the **Material Type** group, select the material.

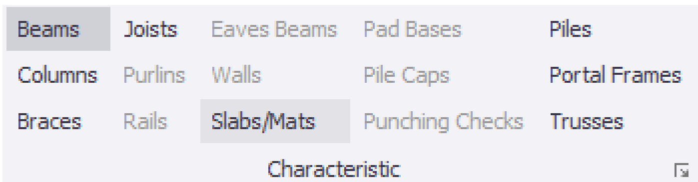


   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 907)
- Concrete material lists (page 913)
- Timber material lists (page 924)
- Cold formed material lists (page 926)
- General material lists (page 927)

5. In the **Characteristic** group, select the required characteristic.



   The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.

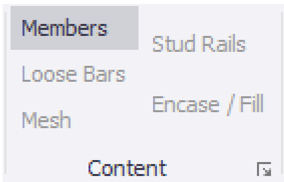
6. If applicable, in the **Construction** group, select the construction.

The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

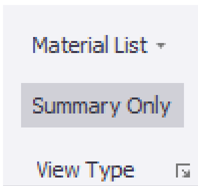
7. In the **Fabrication** group, select the fabrication.

The available selections in the **Content** group are reduced to those appropriate to the selected fabrication.

8. In the **Content** group, select the content.



9. If you have selected **Members** in the **Content** group, you have the option to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular material list is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 929)

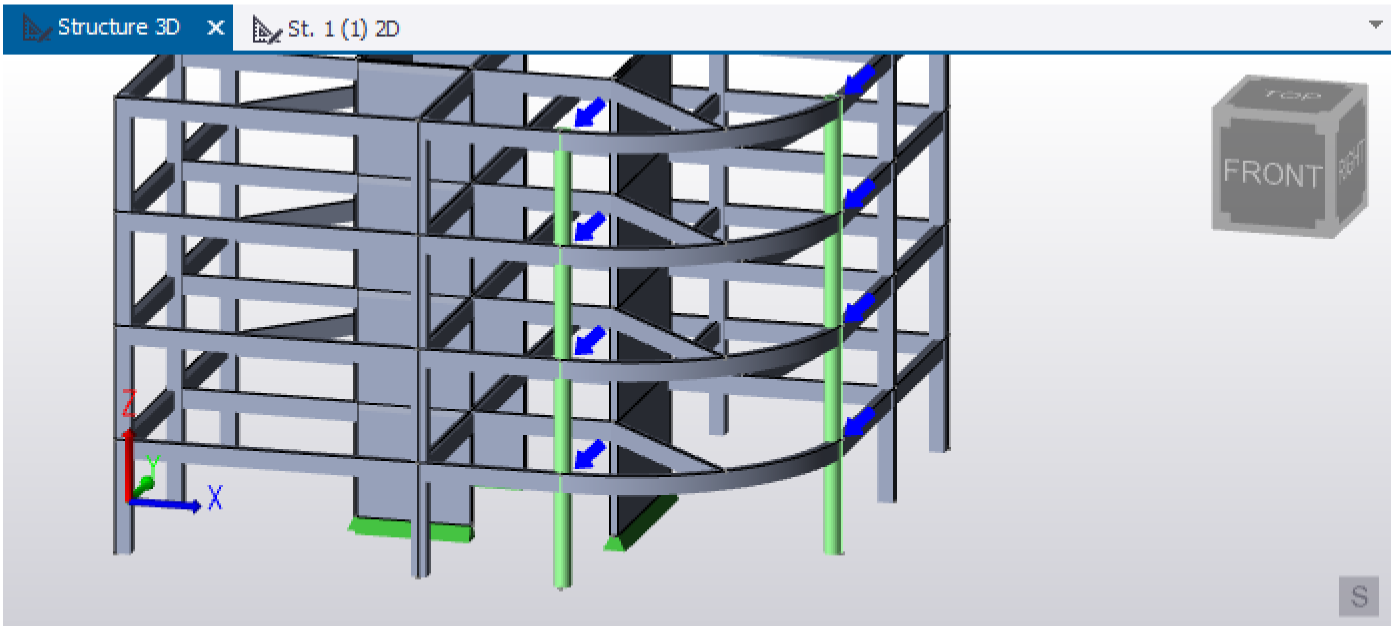
See also:

- Steel material lists (page 907)

- Concrete material lists (page 913)
- Timber material lists (page 924)
- Cold formed material lists (page 926)
- General material lists (page 927)

Locate material list rows in a 3D view

Double-clicking on a specific row in the material list tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Material List

Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m^2]	Volume [m^3]	Reinforcement [kg]	Reinforcement [kg/m^3]
Rectangular	400x400	C32/40	32	96.000	38400.00	153.6	15.4	10315.35	672
Circular	450 Dia	C32/40	8	24.000	9542.59	33.9	3.8	1334.72	350
			40	**120.000**	**47942.59**	**187.5**	**19.2**	**11650.07**	**608**

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Export material list to Excel

1. Create the material list and ensure that it contains the necessary information.
2. On the **Review Data** ribbon tab, click **Excel**.
3. Click **Yes** to confirm exporting the report.

   The material list opens in Excel

Material lists for steel

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length
- Composite beam reports also include the number of connectors, the transverse reinforcement associated with the beam (at detailed level only), and the weight of reinforcement

Construction	Fabrication	Content	View Type	Description
Composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, mass, surface area, reinforcement
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, mass, surface area, reinforcement

Construction	Fabrication	Content	View Type	Description
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, gross mass, net mass, gross surface area, net surface area
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass

Construction	Fabrication	Content	View Type	Description
		Mesh	Mesh	One row for each type: type, unit mass, total area, total mass
Non-composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, gross mass, net mass, gross

Construction	Fabrication	Content	View Type	Description
				surface area, net surface area

Columns, Braces, Joists

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Fabrication	Content	View Type	Description
Columns	• Rolled • Plated	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	• Concrete Filled • Concrete Encased	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size &

Characteristic	Fabrication	Content	View Type	Description
				length), length, mass
		Encase/Fill	Detailed	One row for each size:
				size, grade, number (of that size), length, mass
Braces	Rolled	Members	Summary	One row for each size:
				size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length:
				size, grade, number (of that size & length), length, mass, surface area
Joists	N/A	Members	Summary	One row for each size:
				size, grade, number (of that size), total length, mass
			Detailed	One row for each size & length:
				size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Portal Frames

Characteristic	Content	View Type
Portal Frames	Members	Detailed

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for concrete

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column. This can be useful for example when wanting to sort members by how much reinforcement they contain.

Beams and Columns

NOTE
- Summary reports provide total quantities for each section size
- Detailed reports provide quantities on a span/stack basis
  - When group design is active this reduces to every detailing group span/stack
- Reinforcement running between spans/stacks is shared
- No additional detailing allowance has been applied
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Fabrication	Content	View Type	Description
Beams	Cast-in-place	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each span: Group/ Member ref., section geometry, size, grade, number (of that ref), length, mass,

Characteristic	Fabrication	Content	View Type	Description
				surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume

Characteristic	Fabrication	Content	View Type	Description
	Post-tensioned	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume
Columns	Cast-in-place	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each size: section geometry, size, grade, number (of that size), total length,

Characteristic	Fabrication	Content	View Type	Description
				mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each stack: Member reference, section geometry, size, grade, number, length, mass, surface area, volume

Walls

NOTE
- Summary reports provide total quantities for each wall
- Detailed reports provide quantities on a wall panel basis
- Reinforcement running between panels is shared
- No additional detailing allowance has been applied
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Fabrication	Content	View Type	Description
Walls	Cast-in-place	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass

Characteristic	Fabrication	Content	View Type	Description
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
			Detailed	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

- Patch and punching reinforcement is approximately distributed by sharing it equally between slab items it touches - it is not split based on covered areas
- Reinforcement allows for laps at the slab item boundary
- An approximate user defined additional detailing allowance (specified in Design Settings for cast-in-place slabs) is applied to all reinforcement defined in any slab type.
- Punching reinforcement weight is approximated (based on ACI guidance)
- Precast and post-tensioned slabs are not designed, but the reinforcement that is specified in the slab properties is reported. The post-tensioned tendon information is not reported and must be allowed for in some other way.

Construction	Fabrication	Content	View Type	Description
Composite	N/A	Members	Summary	One row for each slab: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
			Detailed	One row for each slab item: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total

Construction	Fabrication	Content	View Type	Description
				mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
Non-composite	Cast-in-place	Members	Summary	One row for each slab: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
			Detailed	One row for each slab item: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement

Construction	Fabrication	Content	View Type	Description
				(mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
	Precast	Members	Summary	One row for each slab: level, slab, depth, manufacturer, reference, unit depth, topping, grade, mass, gross surface area, net surface area, volume, reinforcement
			Detailed	One row for each slab item: level, slab, depth, manufacturer, reference, unit depth, topping, grade, mass,

Construction	Fabrication	Content	View Type	Description
				gross surface area, net surface area, volume, reinforcement
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
	Post Tensioned	Members	Summary	One row for each slab: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
			Detailed	One row for each slab item: level, slab, type, depth,

Construction	Fabrication	Content	View Type	Description
				grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance

Punching checks

NOTE The weight of punching shear rails is approximated based on ACI guidance.

Content	View Type	Description
Stud Rails	Detailed	One row for each rail ref: ref., diameter,stud height, studs per rail, stud spacing, rail length, total no. of rails, approx. mass

Pad Bases, Pile Caps and Piles

NOTE
- Reports provide quantities for each individual base
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Content	View Type	Description
• Pad Bases • Pile Caps	Members	Detailed	One row for each base: member reference, grade, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
	Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
	Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Material lists for timber

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	View Type	Description
• Beams • Columns • Braces	Summary	One row for each size: size, grade, number (of that size), total length, mass
	Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for cold formed

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
		Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for general materials

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
		Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area

Walls

Characteristic	Content	View Type	Description
Walls	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
		Detailed	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Review floored area tabular results

To create a tabular floored area summary:

1. If necessary, change the view regime (page 280) to a **Review View**.
2. On the **Review** ribbon tab, click **Tabular Data**.

   A **Review Data** tab opens on the ribbon and a **Review Data View** is displayed.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Floored Area**.

A table of floored areas is displayed.

Filter tabular data

You can apply model filters to enable selective display of tabular data. The filters available depend on the view type displayed.

Create and apply filters

1. In the **Filters** group of the **Review Data** tab, select the desired filter type.

   The **Select filter items** list opens.
2. In the **Select filter items** list, select the necessary options to define the filter requirements.
3. Click **OK**.

   Tekla Structural Designer applies the filter.

Edit filters

NOTE If a filter type is not applicable for the current tabular data view type, it will be unavailable.

1. On the **Review Data** tab, click either **Filter Items...**

   The **Select filter items** list opens.

2. In the **Select filter items** list, select the necessary options to define the filter requirements.

3. Click **OK**.

   Tekla Structural Designer applies the filter.

Export tabular results to Excel

1. Create the tabular data view and ensure that it contains the necessary information.

2. On the **Review Data** ribbon tab, click **Excel**.

3. Click **Yes** to confirm exporting the report.

   The tabular data opens in Excel

10 Calculate slab deflections

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

- To begin, see Get started with slab deflection analysis (page 931)
- To define event sequences in Tekla Structural Designer, see Work with event sequences (page 932).
- To set up the deflection checks and place check lines, see Work with check lines (page 934).
- To perform the analysis, see Run a slab deflection analysis (page 936)
- To investigate the results, see Slab deflection results and reports (page 937).

Related video

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.1 Get started with slab deflection analysis

Two different ways of checking slab deflections in Tekla Structural Designer are available. Either deemed-to-satisfy checks, or a rigorous theoretical deflection estimation. These are introduced in the topic Slab deflection methods (page 1382).

If you decide to adopt the latter approach the video links at the bottom of this page are a good start point, and the Rigorous slab deflection workflow (page 1384) topic also provides an overview of the steps required.

Many topics are then discussed in more detail in the Slab deflection handbook (page 1381), which also provides Tutorial worked examples for you to download and work through.

Related video

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.2 Work with event sequences

Event sequences are edited in the (page 2410).

Add an event to the end of the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Click **Add** to Add the Event as the last event in the sequence.
4. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
5. Click **OK**.

Insert an event within the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event above which you want to insert the new event.
4. Click **Insert** to insert the event.
5. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
6. Click **OK**.

Re-order events in the event sequence

You can Add, Insert, Remove and Re-Order Events from the Model Event Sequence page.

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. In the **Drawing Variant** list, select the Event you want to move.
4. Click**Move Up** or **Move Down** until the Event is in the required position.
5. Click **OK**.

Remove an event from the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event which you want to remove.
4. Click **Remove** to remove the event.
5. Click **OK**.

Edit event parameters

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.
3. Edit the event parameters according to your needs. If editing the Model Event Sequence and you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
4. Click **OK**.

Edit event loadcases

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.

3. In the left hand pane, expand the Event Sequence if required so that you can select the Event.

   The available and included loadcases are displayed.
4. Move loadcases between the available and included lists as required. If editing the Model Event Sequence and you want the same loadcases to apply to custom events, ensure the **Update custom event sequences** box is selected.
5. Click **OK**.

Create a custom event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Event Sequences** sub page.
3. Click **Add** to add a new Custom Event.
4. In the right hand pane click on the event name to rename it.
5. In the left hand pane click on the event name in order to define the event parameters in the sequence.
6. Click **OK**.

Apply a custom event sequence to a submodel

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Submodels** sub page.

   The table displays the event sequence assigned to each submodel.
3. Use the droplist to change the Event Sequence assigned to a specific submodel as required.
4. Click **OK**.

10.3 Work with check lines

User defined Check Lines can be placed across 2D element meshes, They are similar to 2D results strips, (but with zero width and different properties).

From these check lines, deflection results are determined from the shell/plate/ membrane nodal analysis results - these can then be checked against specified limits for slab design purposes.

Engineering judgment is required when positioning the check lines to ensure worst case deflections are obtained.

Create the deflection checks to be applied to check lines

1. On the **Slab Deflection** tab, click **Deflection Checks**

   The Slab Deflection Check Catalogue (page 2425) is displayed, containing any checks that have been defined.
2. Click **Add**

   A new check is created in the table.
3. Select the name in order to give it a more descriptive title.
4. Choose the check **Type**: (Total, Instantaneous, or Differential).
   a. If the Type is Differential, select the **Start Event** from the droplist.
5. Select the **Event** to check from the droplist.
6. Enter the **Deflection Limit** to be checked.
7. Select **Use in new Check Lines** if you want this check to be performed in each check line that is created.

Create a check line

Check lines can only be created while in the **Slab Deflections View** regime.

1. Open a 2D view of the FE mesh where the check line is to be placed.
2. On the **Slab Deflection** tab, click **Create**

   The Check Line properties are displayed in the **Properties** window.
3. Adjust the properties to specify:
   a. The number of stations
   b. The upper and lower Flat Zone Reductions
   c. Up to 6 deflection checks to be performed (these can either be selected from the predefined Deflection Checks Catalogue, or a new check can be defined)
4. Click a point where the strip is to start.
5. Click a 2nd point where the strip is to end. (Neither start or end points have to match nodes in the mesh.)

   A check line is created between the points that you identified.

6. Either continue to place further check lines, or if done, press **Esc** to exit.

Delete a check line

To be able to delete a check line you must first ensure that Slab Deflection Check Lines are switched on in Scene Content.

1. Open a view displaying the check lines to be deleted.
2. Click or press **Delete**
3. Click on the check lines to be deleted.

10.4 Run a slab deflection analysis

The same basic analysis process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

To run the analysis for specific or all sub models see the following instructions.

Run a slab deflection analysis for the current sub model

To analyze a specific sub model:

1. Open a view of the required sub model from the Structure Tree in the Project Workspace.
2. On the **Slab Deflection** tab, click **Analyze Current** At the end of the analysis the active sub model view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for all sub models

To analyze all sub models:

1. On the **Slab Deflection** tab, click **Analyze All** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for selected sub models

To analyze selected sub models:

1. On the **Slab Deflection** tab, click **Analyze Selected** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

10.5 Slab deflection results and reports

Click the links below to find out more:

- Display slab deflection analysis results (page 937)
- Display check line results (page 939)
- Display slab deflection status and utilization (page 940)
- Slab deflection optimization (page 942)
- Slab Deflection Reports
- Slab Reinforcement (page 868)

Display slab deflection analysis results

Once a slab deflection analysis has been performed analysis results can be reviewed for the chosen level (sub-model), or the entire structure - dependent upon your chosen analysis.

The following results are available for display in the **Slab Deflections View** regime by making appropriate selections from the **Results** group of the **Slab Deflection** ribbon.

- **Deflections**.
- **Extent of Cracking** at any load event.
- **Relative Stiffness** in a particular result direction for any specified event.
- **Effective Reinforcement** for a chosen result direction for each FE element.

Display deflection contours

Three deflection types are available for review:

- **Total** deflection at the end of any event.
- **Differential** deflection between any two events (Start of Event and End of Event).

  For further details, see: Interrogating slab deflection calculations (page 1408)
- **Instantaneous** deflection (not actually needed for TR 58).
  - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
  - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

To display deflection contours:

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Deflections**
2. Then from the same group:
   a. Make your selection from the **Result Direction** list
   b. Make your selection (Instantaneous, Total, or Differential) from the **Result Type** list
3. From the **Results** group select the **Event**, and if viewing differential deflections also select the **Start Event**.

   NOTE Hover the cursor over a specific solver node in order to see the nodal displacement at that location displayed in a tooltip.

Display extent of cracking

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Extent of Cracking**
2. Make your selection from the **Result Direction** list
3. From the **Results** group select the **Event**.

   NOTE Hover the cursor over a specific solver element in order to see the detailed cracking calculation parameters for that element displayed in a tooltip.

Display relative stiffness

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Relative Stiffness**
2. Make your selection from the **Result Direction** list
3. From the **Results** group select the **Event**.

NOTE Hover the cursor over a specific solver element in order to see the detailed relative stiffness calculation parameters for that element displayed in a tooltip.

Display effective reinforcement

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Effective Reinforcement**
2. Make your selection from the **Result Direction** list

NOTE Hover the cursor over a specific solver element in order to see the detailed effective reinforcement calculation parameters for that element displayed in a tooltip.

Display check line results

Once a slab deflection analysis has been performed, the Check Line results are available for display in the **Slab Deflections View** regime.

Display deflections along all check lines

1. From the **Slab Deflection** ribbon **Check Lines** group click **Deflections**

   The deflected shape of each check line and an accompanying legend is displayed in the current view.

   For clarity you may want to switch off other results to obtain a clearer view.

Display detailed deflections and average slopes along an individual check Line

1. Right click on the check line you want to view and select Open deflections check view from the context menu that is displayed.
2. Select the Result Type and the Event from the Loading Analysis toolbar.

   The deflections and average slopes are displayed in a deflections view.

Display check line status and utilization

Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio

Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization.

To view check line status or utilization:

1. From the **Slab Deflection** ribbon **Check Lines** group
    - Click **Status**
    - Click **Utilization**

    Color codes are used to graphically display the status or utilization of each Check Line.

Display slab deflection status and utilization

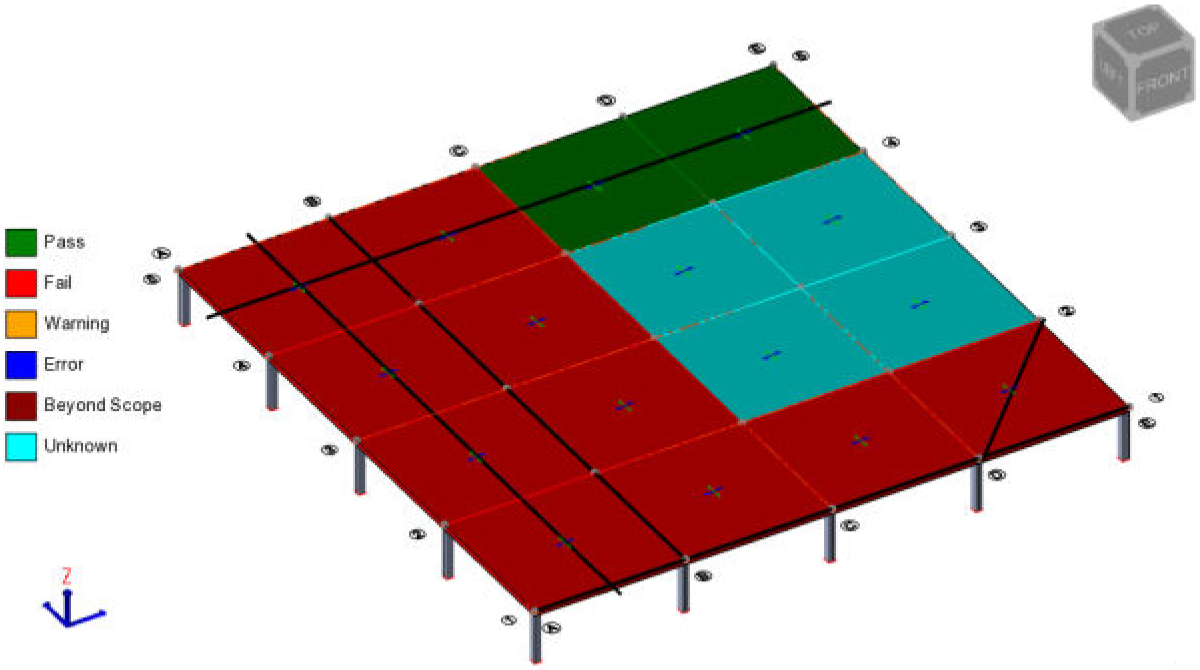
Once a slab deflection analysis has been performed, the Slab deflection status and utilization are available for display in the **Slab Deflections View** regime.

Display slab deflection status

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Status**

    Color codes are used to graphically display the status of each slab item.

In the screenshot below;



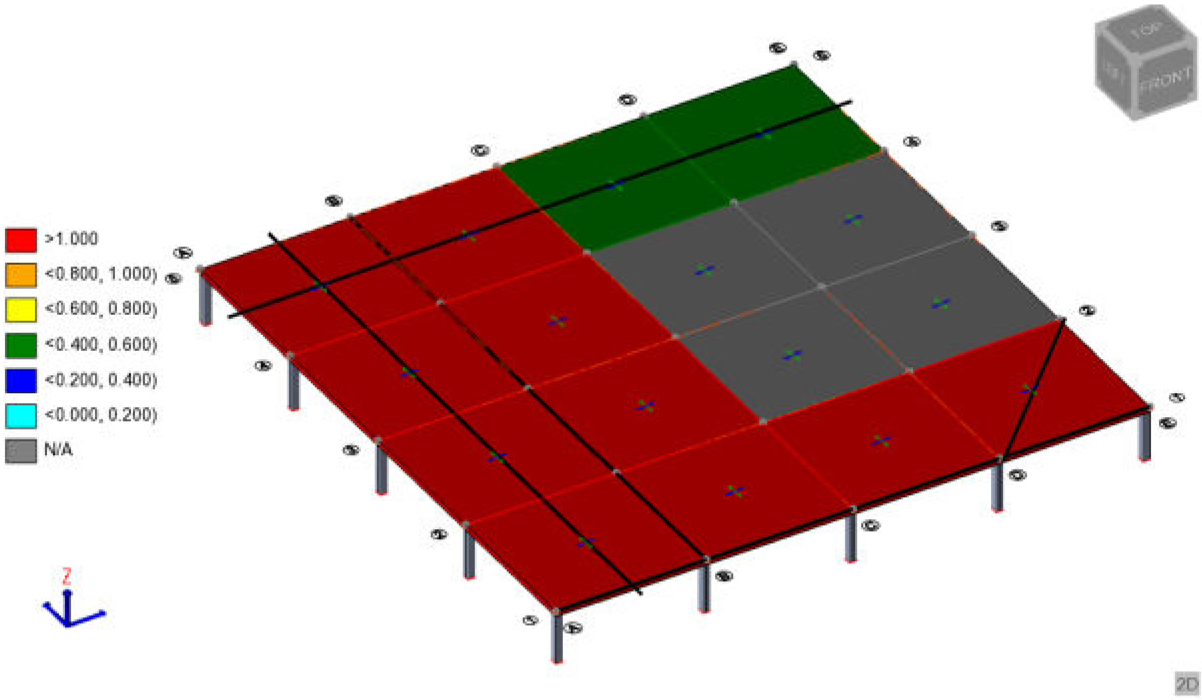
- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Display slab deflection utilization

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Utilization**

   Color codes are used to graphically display the utilization of each slab item.

The worst utilization from all associated check lines is displayed.



Slab deflection optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

View slab deflection reports

A Slab Deflection Check Lines report can be generated for multiple check lines from the Reports toolbar. The same report can also be generated for individual check lines from the right click menu.

View an individual check line report

1. Right click on the check line you want the report for and select **Report for Member** from the context menu that is displayed.

   A report is displayed consisting of a defection check summary table.

   NOTE Optionally the report can be configured to include a picture showing the Check Line location within the slab, (by clicking Member Report on the Reports ribbon and choosing the Slab Deflection Check Line as the Member Type.)

View all/multiple check line reports

1. From the **Report** tab click **Model Report...**

   A report is displayed consisting of a defection check summary table.

2. Create a report that includes the Slab Deflection Check Lines chapter (located under Concrete>Slab/Mat Design per Plane).
3. Apply Model filters to the report as required.

View an effective modulus report

A slab Effective Modulus Report can be exported to Excel for an individual slab panel, or all panels.

1. Right click on a slab panel and select **Export Eff. Modulus Report to Excel** from the context menu that is displayed.
2. Choose whether to export for the current slab item, or all slab items from the sub menu that is displayed.

   The report opens in Excel, (assuming Excel is installed).

11 Create reports and drawings

Having designed your model in Tekla Structural Designer you can then output the results in the form of reports and/or drawings.

11.1 Create and modify reports

You can tailor a wide range of different reports according to your needs by using the **Report** toolbar.

NOTE Initially, only the **Contents** group and **Filters** group are displayed on the **Report** toolbar. Only when a report is displayed and the **Report View** is active, do the **Appearance** group, **Navigation** group, and **Export** group become available.

Report terminology

The following terms are used to described specific aspects of the report creation process in Tekla Structural Designer.

Model reports

Model reports are used to set up the printed output for either the entire structure or a part of it. Model reports are configured by selecting specific output categories (referred to as *chapters*) in a list of all the available output categories. You can include entire chapters, or just the headings within the chapter you require.

The combination of selected chapters is referred to as the *report structure*. If necessary, you can apply filters to individual headings in the report structure to limit the output that is produced.

Model report chapters include:

- Structure:

  Structure data (under headings such as **Loadcases** or **Wind Data**).
- Analysis:

  Analysis model properties and results
- Concrete, Steel, Timber, Cold Formed, Cold Rolled, and General Material:

  Member reports for each different member types. Each member report can be configured to contain as much or as little output as needed. You can also decide to include design summary tables for the members that Tekla Structural Designer can design.
- Beam End Forces, Bracing Forces, and Foundation Reactions:

  Chapters for specific sets of analysis results.
- Picture:

  A 3D view of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary, you can also include applied loads in the picture by applying a load filter.
- View:

  The current display in any 2D or 3D view can be saved as a view configuration, and then, included in the model report as a View chapter.
- Analysis Diagram:

  A 3D diagram of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary, you can also display the analysis results by applying an analysis method filter.
- Material Listing:

  Tabulated quantities of materials.
- Revision History:

  The revision history that has been recorded in the **Project Wiki**.

Member reports

Member reports are only output as part of a created model report or by right-clicking on an individual member and selecting **Report for Member**.

Member report chapters include:

- Picture:

  The 3D view that you receive by right-clicking a member and selecting **Open [element name] view**.
- Drawing:

  The DXF file that you receive by right-clicking a member and selecting **Generate Detailing Drawing...**

- Loading:

  The table that you receive by right-clicking a member and selecting **Show Member Loading**.
- Different force diagrams and Deflected Shape Diagram:

  The diagrams that you receive by right-clicking a member and selecting **Open Load Analysis View**.

Active model report

The active model report is the default report displayed in the list on the left corner of the **Report** toolbar, unless another report view is already active.

To specify the active model report, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active member report

The active member report is the report that is generated when you right-click a member in a 2D or 3D view and select **Report for Member**.

To specify the active model report, do the following:

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active and inactive chapters

In the **Report Contents** dialog box, each chapter in the report structure is marked as either *active* or *inactive*. Only the chapters that are marked as *active* are included in the report that you generate.

Report filters

You can use filters to limit the amount of output in the report.

Tekla Structural Designer offers the following filters for different data types:

- Model filters:

  If necessary, you can filter the report for selected levels, frames, or planes. You can also filter certain data types for selected beams, columns, or walls.

If you have applied user-defined attributes to your model, you can also use them to filter the report output.

- Loading filters:

  If necessary, you can filter the report for selected loadcases and combinations.

Available styles

A number of sample reports are available in the **Available Styles** list of the **Report Contents** dialog box. The reports serve as templates and can be modified to suit the model in question. If you cannot find a report that can be customized to meet your needs, you can add further reports to the list by clicking **Add**.

If you start a new project, the same default reports are available. However, they are reset to the default report structures. It is not currently possible to save customized reports from the **Report Contents** dialog box to apply them to other projects.

Create reports

Tekla Structural Designer allows you to create two kinds of reports: you can either create a report based on the entire model or an individual member in the model.

Configure and display model reports

To create model reports and display them, see the following instructions.

Configure a model report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. Select the report:

   - To select an existing report, click the report name in the **Available Styles** list.
   - To add a report, click **Add** and type the report name in the **Active Style** field.

3. Review the report structure and modify it (page 950) according to your needs.
4. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 951).

TIP You can further limit the output of **Loadcases** and **Combinations** sub chapters by applying a loading filter (page 951).

5. If you have created specific view configurations for the model and want to include them in the report, include a separate **View** chapter for each view configuration.
6. If the report structure includes any member chapters (such as beams, columns, or walls), select an appropriate member report style for each member chapter.
7. Click **OK**.

NOTE If you have included a drawing in the report structure, specify appropriate settings for the drawing as follows:

   a. Right-click the drawing in the report structure.
   b. Click **Settings...**
   c. Define the drawing settings according to your needs.
   d. Click **OK**.

Display a model report

1. In the list on the far left of the **Report** tab, select the report that you want to view.
2. On the **Report** tab, click **Show Report**.

   Tekla Structural Designer displays the report in a new window. If the report contains loading analysis views of individual members (such as force diagrams or pictures), the views are displayed for each member type according to the selected member report style.

See also

Export reports (page 957)

Print reports (page 957)

Format reports (page 952)

Configure and display member reports

To create member reports, see the following instructions.

Configure a member report

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Member Type** list, select the desired member.

3. Select the report:
   - To select an existing report, click the report name in the **Available Styles** list.
   - To add a report, click **Add** and type the report name in the **Active Style** field.
4. Review the report structure and modify it (page 950) according to your needs.
5. To control the level of output, modify the report structure (page 953) or loading filters according to your needs.
6. Click **OK**.

NOTE If you have included a drawing in the report structure, specify appropriate settings for the drawing as follows:

a. Right-click the drawing in the report structure.

b. Click **Settings...**

c. Define the drawing settings according to your needs.

d. Click **OK**.

Display a member report

1. Ensure that you have selected and activated the member report style (page 949).
2. Hover the mouse pointer over the desired member.
3. Once the outline of the member is high-lighted, right-click the member.
4. In the context menu, select **Report for Member**.

   Tekla Structural Designer opens a report for the selected member in a new window.

See also

Format reports (page 952)

Export reports (page 957)

Select the member report style

Different member report styles produce different levels and types of output both in model reports and member reports. To select the member report style, see the following instructions.

Select the member report style used in a model report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. Select the desired report type.
3. Right-click the member chapter in the report structure.
4. In the context-menu, go to **Style**.
5. Click the desired member report style.

Select the member report style used in an individual member report

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Member Type** list, select the desired member.
3. In the **Available Styles** list, select the desired member report style.
4. Click **>> Active**.

See also

Configure and display member reports (page 948)

Configure and display model reports (page 947)

Format reports (page 952)

Export reports (page 957)

Modify the report structure

Once you have selected the report type, you can modify the report structure according to your needs. For more information, see the following instructions.

To aid the report structuring process, an option is provided to display the structure as either a Flat, or a Hierarchical layout.

1. According to your needs, click either **Model Report...** or **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Report Contents** dialog box, do one or more of the following:
   - To include chapters and options in the report, drag them from the left column to the right column.
   - To remove unwanted chapters and options permanently from the report, drag them from the right column to the left column.
   - To exclude a specific chapter from the report but maintain the current report structure, right-click the chapter and clear the **Active** option.

- To re-arrange the report order, drag and drop chapters within the right column.
- To apply a filter to a chapter, right-click the desired chapter. For more information, see Filter reports (page 951).

TIP In **View Mode**, you can select if you want to display the structure as flat or hierarchical.

3. Click **OK**.

See also

Configure and display member reports (page 948)

Filter reports

Tekla Structural Designer contains two main categories of filters that you can apply to chapters in the report structure. The categories are model filters and loading filters. For more information, see the following paragraphs.

You can apply model filters to enable selective output based on selected:

- Levels
- Frames
- Planes
- Groups
- Members
- Trusses
- Portal frames
- UDAs

Using loading filters enables selective output based on selected:

- Load cases
- Combinations
- Envelopes

See also

Configure and display model reports (page 947)

Create filters

1. In the **Filters** group of the **Report** tab, select the desired filter type.

   The **Edit filters** dialog box opens.

2. Click **Add**.
3. Type a name for the new filter.
4. In the **Selected items** list, select the necessary options to define the filter requirements.
5. Click **OK**.

   Tekla Structural Designer creates the filter. You can now apply the filter to a specific report.

   TIP If you create a member filter, you can later review and modify it graphically by clicking **Report Filter** on the **Review** tab.

Apply filters

NOTE If a filter type is not applicable for the selected chapter, it will be unavailable.

1. On the **Report** tab, click either **Model Report...** or **Member Report...**

   The **Report Contents** dialog box opens.
2. Select the report that you want to filter.

   In the right column, any chapters that can be filtered are displayed in blue text.
3. Right-click a chapter.
4. In the context menu, select the desired filter type and filter name.

   TIP If you cannot find an appropriate filter, do the following:

   a. In the context menu, go to the filter type and click **Edit\New...**
   b. In the **Select filter** dialog box, click **Add**.
   c. Type the name of the filter in the **Active filter** field.
   d. In the **Selected items** list, select the necessary options to define the filter requirements.
   e. Click **OK**.
5. Click **OK**.

   Tekla Structural Designer applies the filter to the selected report chapter.

Format reports

Tekla Structural Designer allows you to customize the appearance of reports in a number of ways. You can, for example, configure paragraphs styles and tables, and adjust header and footer information.

Adjust and apply report settings

Report settings allow you to modify the appearance of reports. You can adjust, for example, paragraph styles, page margins and numbering, tables, and headers or footers.

1. On the **Report** tab, click **Settings**.

   The **Settings** dialog box opens.

2. In the **Settings** dialog box, do one or more of the following to adjust the report settings:

To	Do this
Customize the paragraph styles in the different areas of the report	a. Go to the **Styles** page. b. Adjust the settings according to your needs.
Adjust the page margin width, page numbering, and margin frame	• Go to the **Page Options** page. • Adjust the settings according to your needs.
Adjust the appearance of tables	• Go to the **Table Options** page. • Adjust the settings according to your needs.
Define whether headers and footers are displayed, adjust image width and paragraph spacing, and control page breaks	• Go to the **Document Options** page. • Adjust the settings according to your needs.
Adjust the appearance of fonts used in pictures and force diagrams	• Go to the **Picture Fonts** page. • Adjust the settings according to your needs.

3. Click **OK**.

   Provided that you adjusted the report settings of an active settings set, the new report settings are applied to the report.

See also

Report settings (page 2394)

Adjust report headers and footers

Tekla Structural Designer provides you with default page header and footer layouts for your reports. However, you can also create, use and save different layouts according to your needs.

Enter company details in the header

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.

3. Go to the **Headers** page.
4. In the **Available layouts** list, select an existing layout that contains an **Address** cell.
5. Go to the **Fields** page.
6. In the **Available fields** list, select **Address**.
7. In the list on the right, select the necessary company details and click **<<** to include them in the report.
8. To define the text displayed in each included field, select the field in the left **Available fields** list and type the desired text in the **Displayed value** field.
9. To insert the company logo in the header, select **Company Logo** in the left **Available fields** list and click **...** to browse to the desired image.

   TIP
   - Large logos are automatically resized to within the column width in which they are placed. However, using smaller images in PNG format saves memory and speeds up exporting reports.
   - To include more images in the header, click **Add Image Field**, type a name for the field and selected the desired image.

10. Click **OK**.

    The company details are displayed in the report header.

Enter project-specific details in the header

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.

2. On the **Project Summary** page, type the project details in the available fields.
3. Click **OK**.

   The project details are saved.

   NOTE When you go back to the **Report** tab, remember to update the report to display the project details in the header.

Create a new header or footer layout

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.
3. According to your needs, go to the **Headers** or **Footers** page.
4. In the **Available layouts** section, click **New**.
5. Type a name for the new layout in the **Name** field.
6. According to your needs, do some of the following to adjust the header layout:

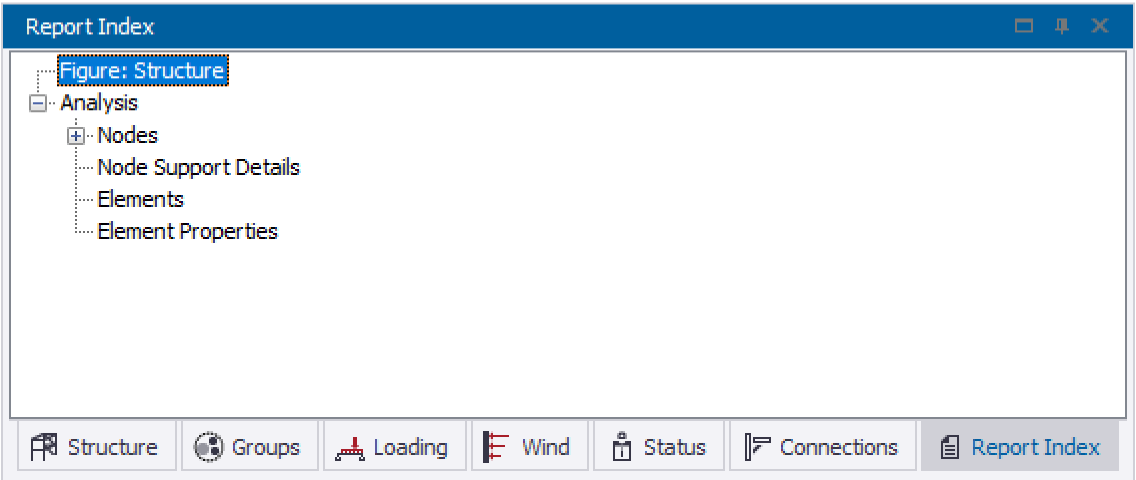
To	Do this
Remove an existing field and create an empty shell	a. Hover the mouse pointer over the field in the **Current layout** section until the field name is highlighted. b. Click the field that you want to remove it and replace it with an empty cell.
Place a field into an empty cell or replace an existing field	• Hold down the left mouse button and drag the field from the **Available fields** list into the desired cell.
Merge cells	• Hold down the left mouse button and drag over the cells that you want to merge. **NOTE** To unjoin previously merged cells, right-click the merged cell and in the context menu, select **Unjoin**.
Modify the field alignment within a shell	a. Right-click the required cell in the layout. b. In the context menu, set the horizontal and vertical alignment options as required.
Insert or remove rows or columns or change their alignment	a. Right-click the layout. b. In the context menu, select the desired command.
Change column type, width and alignment in the layout	a. Under the **Current layout** section, click **Edit...** b. Adjust the column properties according to your needs. c. Click **OK**.

7. Click **OK**.

   The new layout is created.

Navigate reports

Use the **Report Index** to quickly navigate to a specific section in a report.



You can also use the buttons in the Navigation group of the **Report** toolbar to move through the report pages.

Navigation using the Report Index

Open a report view then:

1. In the **Navigation** group of the **Report** toolbar, select **Report Index**

   The **Report Index** tab is displayed in the **Project Workspace**.
2. Click the **Report Index** tab in the **Project Workspace**.
3. Click on a specific heading or sub heading Report Index to move to that section in the report.

Navigation buttons in the Report toolbar

Open a report view then:

- Click  to move to the first page
- Click  to move to the previous page
- Click  to move to the next page
- Click  to move to the last page
- Click  to open the **Report Index** in the **Project Workspace**

Export reports

Tekla Structural Designer allows you to export your reports to PDF format, Microsoft Word, Excel, and Tekla Tedds. For detailed instructions, see the following paragraphs.

Export a report to PDF

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **PDF**.
3. Click **Yes** to confirm exporting the report.

   The report opens as a PDF file.

Export a report to Microsoft Word

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Word**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Microsoft Word.

Export a report to Excel

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Excel**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Excel

Export a report to Tekla Tedds

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Tedds**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Tekla Tedds

Print reports

To print any reports that you require, see the following instructions.

1. Create the report and click **Show Report**.

   Tekla Structural Designer displays the report.

2. On the **File** menu, go to **Print** --> **Print...**
3. Adjust the printing settings according to your needs.
4. Click **OK**.

Example reports

Tekla Structural Designer provides a number of standard reports that are installed by default. Even though the standard reports may not match your exact requirements, they can often serve as templates that you can modify according to your needs.

Solver model data report

By default, the report contains a picture of the model and the following tables of analysis model input data:

- Nodes
- Node support details
- Elements
- Element members
- Element properties
- Contains to PDF

To display a solver model data report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Solver Model Data**.
2. Click **Show Report**.

 TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Building loading report

By default, the report contains a picture of the model and the following tables of loading related input data:

- Action codes
- Resistance codes
- Combinations
- Wind data

To display a building loading report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Loading**.

2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Building analysis checks report

By default this report contains the following tables:

- Load case summary
- Analysis drift results
- Analysis sway results

To display the building analysis checks report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Analysis Checks**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Building design report

By default, the report contains the design results for the building at a summary level.

To display a building design report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Design**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Material listing report

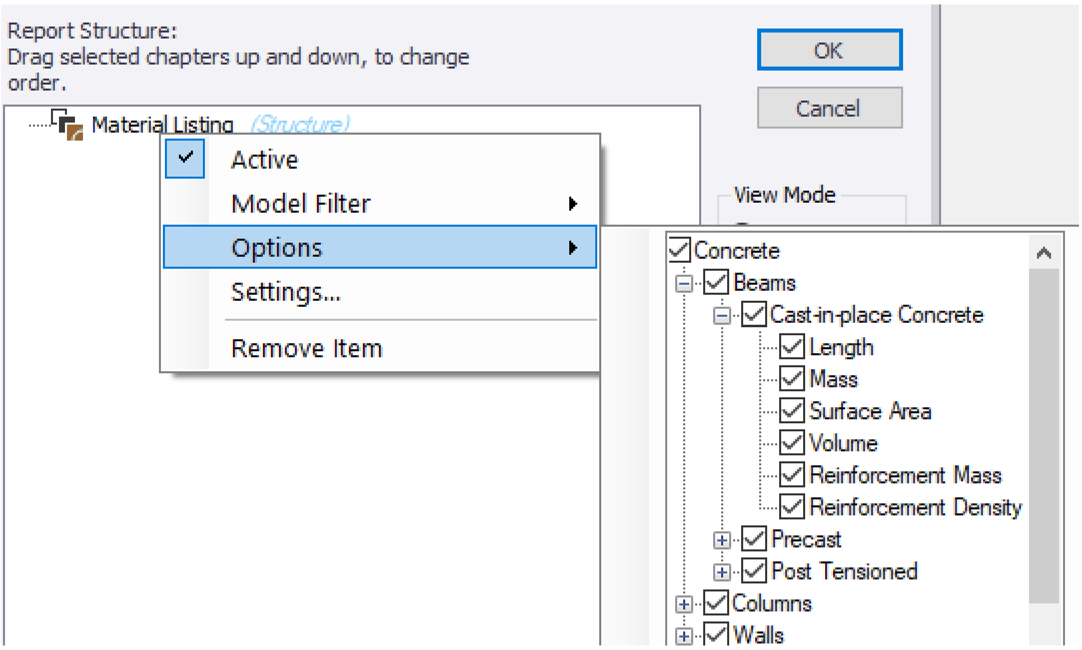
To configure and then display a material listing report, do the following:

1. On the **Report** ribbon, select **Material Listing** from the list in the **Contents** group.
2. Click **Model Report...**

   The **Report Contents** dialog box opens.
3. The default report is created for all materials and entity types in the model and includes all associated properties. If required you can

configure which materials/entity types are to be included in the report and also choose to exclude properties that you don't require. This is done as follows:

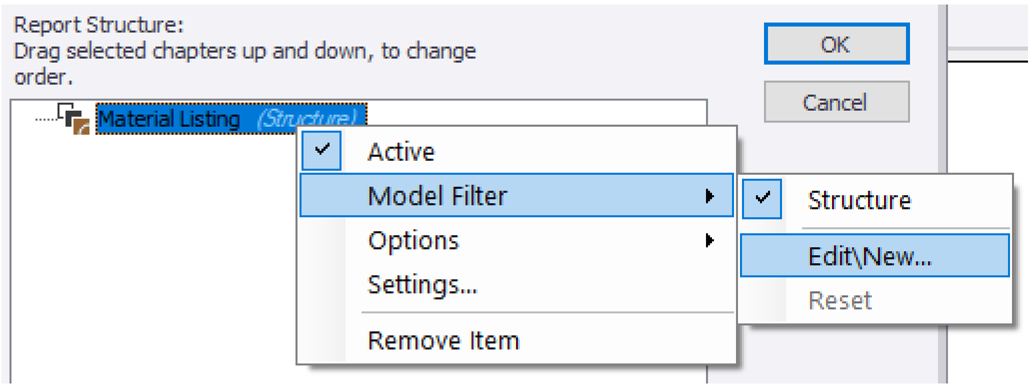
a. Right-click on the **Material Listing** chapter in the Report Structure and choose **Options** from the context menu.



b. Uncheck those materials/entity types and any properties that you don't want to include in the report.

4. The default report is created for the whole structure. If required you can configure a filter to limit the report to selected floor levels only, or other criteria as follows:

a. right-click on the **Material Listing** chapter once again and choose **Model Filter > Edit/New**



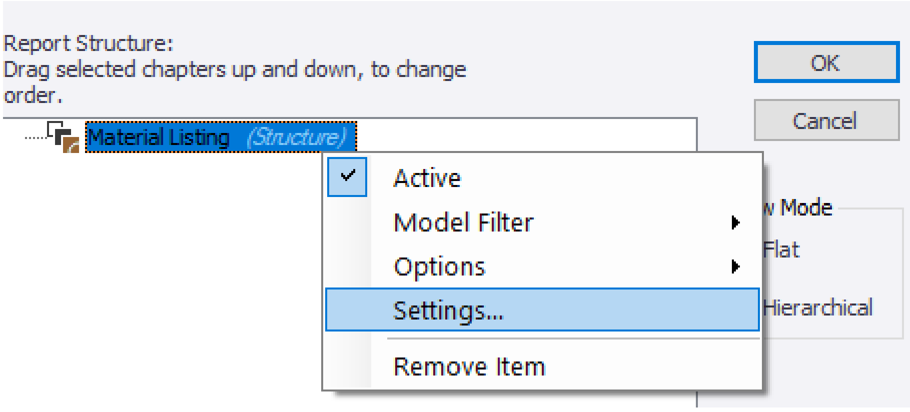
b. Click Add

c. Select the type of filter

d. Select the items to be included

e. Click OK

5. To choose between a summary or detailed report, right-click on the **Material Listing** chapter once again and choose **Settings**



- Uncheck **Show summary only** for a detailed report
- Leave it checked for a summary report

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 907)
- Concrete material lists (page 913)
- Timber material lists (page 924)
- Cold formed material lists (page 926)
- General material lists (page 927)

6. Click **OK** to close the settings dialog.

7. Click **OK** to close the **Report Contents** dialog. The report is displayed according to the configuration choices that you have applied.

Beam end forces report

The report is typically generated for steel or timber beams in order to provide connection design forces in one of the following configurations:

- End 1/End 2:

  6 forces are output in the support's local co-ordinate system for the selected load cases, combinations, or envelopes. For each end of each beam span, a single row of data is output for load cases, combinations, or

envelopes in the selected loading filter (Fx, Fy, Fz, Mx, My, Mz). Select this option if you intend to design the connections at both ends of each beam independently.

- End 1/End 2 + Coincident:

  For each end of each beam span, a separate row is output for each load case, combination, or envelope according to the applied loading filter for the maximum and the minimum of each of the 6 forces that are output. In addition, the report contains the coincident forces in the other directions.

  Up to 12 rows of data can be output for a load case, combination, or envelope as follows:

  - maximum: Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum: Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - maximum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - minimum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - maximum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - minimum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - maximum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - minimum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - maximum: My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - minimum: My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - maximum: Mz (+ coincident Fx, Fy, Fz, Mx, My)
  - minimum: Mz (+ coincident Fx, Fy, Fz, Mx, My)

  The load case, combination, or envelope names in which the maximum or minimum values occur are also reported.

- Higher End Only:

  For each beam span, two rows of data are output for the chosen loading filter: maximum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz; minimum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz. This configuration does not contain load case, combination, or envelope names because they could vary for each of the 6 values.

- Higher End Only + Coincident (initial default option):

  A separate row is output for the maximum and the minimum of each of the 6 forces, along with the coincident forces in the other directions. For each beam span, up to 12 rows of data can be output for the applied loading filter as follows:

  - maximum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - maximum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)

- minimum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)
- maximum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
- minimum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
- maximum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
- minimum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
- maximum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
- minimum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
- maximum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)
- minimum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)

The load case, combination, or envelope names in which the maximum or minimum values occur are also reported.

TIP To change the default beam end forces report configuration, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select **Beam End Forces**.
3. In the **Chapters and Options** list, expand the **Member End Forces** group.
4. Hold down the left mouse button and drag the desired configurations into the report structure.
5. Click **OK**.

To display a beam end forces report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Beam End Forces**.
2. Click **Show Report**.

   TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Bracing forces report

Tekla Structural Designer provides the bracing design force data for all load cases and combinations in steel buildings.

To display a bracing forces report:

1. In the list on the left side of the **Report** toolbar, select **Bracing Forces**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Foundation reactions report

The report summarizes the foundation design forces and is available in two different configurations:

- Foundation Reactions (initial default option):

  6 forces are output in the support's local co-ordinate system for the selected load cases, combinations, or envelopes.

- Foundation Reactions + Coincident

  The maximum and minimum value of each of the 6 forces in the support's local co-ordinate system are output for the selected loading type. In addition, the report contains the coincident forces that exist in the other directions for the loading type in which the maximum or minimum value occurs. The maximum values are highlighted in red, whereas the minimum values are highlighted in blue.

  NOTE The table for the Foundation Reactions + Coincident option never contains more than 12 rows, irrespective of the number of selected load cases or combinations.

To display a foundation reactions report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Foundation Reactions**.
2. Click **Show Report**.

   TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

   In addition to model and loading filters, you can also reconfigure settings for each of the two configurations. These settings provide options for:

   - choosing strength or service factors
   - choosing to display all rows, or non-zero rows only
   - excluding reactions from columns assigned to cores
   - excluding reactions from walls assigned to cores

Seismic design report

By default, the report contain the following tables:

- Seismic loading summary
- Analysis seismic drift results

To display a seismic design report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Seismic Design**.
2. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

Member design report

By default, the report contains the design results for each member at a summary level.

To display a member design report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Member Design Calcs**.
2. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report.

See also

Configure and display member reports (page 948)

11.2 Create drawings

You can create drawings of individual levels and frames by using the commands on the **Draw** toolbar. As for single-member drawings, they are created by right-clicking the desired object and selecting the correct option in the context menu that appears. In addition, you can create drawings in batches using the **Drawing Management...** commands.

Drawing categories

In Tekla Structural Designer there are five main drawing categories. Each of the categories contains different drawing variants, and each variant has a number of specific options that allows you to configure their appearance. In addition, each variant contains a set number of drawing layers that are switched on and off according to the selected layer configuration. The appearance of each layer is controlled by the layer style. Drawing options, drawing layers, and layer styles can be managed in drawing settings.

See the different drawing categories and variants in the following table:

Category	Drawing variant	Description
Planar drawings	General arrangement	General arrangement (GA) of 2D levels and frames.
	Foundation reactions	GA that also contains support reactions in order to assist foundation design.
	Loading plan	GA that also contains applied loads for the selected load case.
	Beam end forces	GA that also contains forces at the ends of steel beams in order to assist steel connection design.
	Column splice loads	Frame GA that also contains the splice loads at splice locations in steel columns in order to assist steel connection design.
Member details	Concrete beam detail	Beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.
	Concrete column detail	Column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.
	Concrete wall detail	Wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.
	Non-concrete beam detail	Individual details for non-concrete beams.

Category	Drawing variant	Description
	Non-concrete column detail	Individual details for non-concrete columns.
Concrete member schedules	Concrete beam schedule	Tabular data created by building, by floor, or by selected beams. The information shown in the schedule is based on the design groups.
	Concrete column schedule	A table containing cross sections through each stack for the selected columns. If necessary, you can also include a reinforcement quantity table.
	Concrete wall schedule	A table containing cross sections through each stack for the selected walls. If necessary, you can also include a reinforcement quantity table.
Slabs and mats	Slab/mat layout	Slab item, patch and punching shear reinforcement requirements. If necessary, you can also include a reinforcement quantity table for the reinforcement displayed with an added detailing allowance.
	Punching check detail	Punching shear reinforcement details for individual punching check locations.
Foundations	Isolated foundation detail	Individual foundation details in plan. If necessary, you can also include the detail in cross section and a reinforcement quantity table.
	Foundation layout	GA at foundation level displaying isolated

Category	Drawing variant	Description
		bases, pile layouts, and mats. If necessary, you can also include isolated foundation details, a reinforcement quantity table, and an isolated foundation schedule.

Adjust and apply drawing settings

To adjust the drawing settings of the current project or set them as default settings for future projects, see the following instructions.

Adjust drawing settings in the current project

1. On the **Draw** toolbar, click **Settings**

   The **Model Settings** dialog box opens.

2. Review and modify the drawing settings according to your needs.
3. Do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes to the selected settings set, click **Save...**
   - To revert to the drawing settings specified in the selected settings set, click **Load...**

Adjust drawing settings in future projects

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.

2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. Review and modify the drawing settings according to your needs.
5. To save the settings as defaults for future projects that use the selected settings set, click **OK**.

See also

Drawing settings (page 2352)

Create drawing scales

You can only create new drawing scales by accessing the **Settings** dialog box via the **Home** tab. For more information on creating scales, see the following instructions.

1. On the **Home** toolbar, click **Settings**.
2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. In the **Drawing Variant** list, select the drawing category that you want to configure.
5. Type the desired scale in the **Scale** field.
6. Click **Add**.

   The new scale is added to the **Available Styles** list.
7. Click **OK**.

   TIP If necessary, you can also adjust the distance between independent drawing blocks by typing a value in the **Minimum Text Block Spacing** field. However, note that if the value is too great, text labels can end up far from the objects to which they refer.

Create, modify, or delete layer configurations

Layer configurations control which layers are displayed when the drawing is created. Tekla Structural Designer contains several default layer configurations that you can modify. If necessary, you can also create your own layer configurations.

Create a new layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. Click **Add**.
5. Name the new layer configuration.
6. Select or clear the layers according to your needs.
7. Click **OK**.

Modify a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Configurations** sub page.
3. In the **Available Configurations** list, select the configuration that you want to modify.
4. Select or clear the layers according to your needs.
5. Click **OK**.

Copy a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose properties you want to copy.
4. In the **Available Configurations** list, select the configuration that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.

6. Select the properties that you want to copy.
7. Name the copy.
8. Select or clear the layers according to your needs.
9. Click **OK**.

Delete a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. In the **Available Configurations** list, select the configuration that you want to remove.
5. Click **Remove**.

   The selected layer configuration is deleted for the selected drawing category.

6. Click **OK**.

Create, modify, or delete layer styles

Drawing styles control how a drawing is displayed, for example, what line types, fonts, and colors are used. Tekla Structural Designer contains several default layer styles that you can modify. If necessary, you can also create your own layer styles.

Create a new layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. Click **Add**.

   A new layer style is added to the **Available Styles** list.

5. Name the new layer style.
6. Adjust the layer properties according to your needs.
7. Click **OK**.

Modify an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Styles** sub page.
3. In the **Available Styles** list, select the layer style that you want to modify.
4. Adjust the layer properties according to your needs.
5. Click **OK**.

Copy an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.

6. Select the drawing item that you want to copy.
7. Click **OK**.
8. Name the copied layer style.
9. Adjust the layer properties according to your needs.
10. Click **OK**.

Delete an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to delete.
5. Click **Delete**.

   The selected layer configuration is deleted for the selected drawing category.
6. Click **OK**.

Create planar drawings

Planar drawings cover general arrangement drawings, beam end force drawings, column splice load drawings, foundation reaction drawings and loading plan drawings.

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** toolbar, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to **Options**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

Create general arrangement drawings

General arrangement drawings display 2D levels and frames, and can only be created in 2D views. To create general arrangement drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. On the **Draw** toolbar, click **General Arrangement**.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create drawing scales (page 968)

Create beam end force drawings

Beam end force drawings are general arrangement drawings that also contain support reactions. They are typically created to assist connection design in steel structures and can only be created in 2D views. To create beam end force drawings, do the following:

NOTE	Beam end forces are not displayed for concrete beams.

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. In the **Loading** list, select the load case or combination that you want to display.
3. On the **Draw** toolbar, click **Beam End Forces**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:

- Accept the automatic file name.
- Clear the **Use automatic file name** option and name the file yourself.

8. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create drawing scales (page 968)

Create column splice load drawings

Column splice drawings are general arrangement drawings that also contain applied loads for the selected load case. Column splice drawings are typically created to assist connection design in steel structures. To create column splice load drawings, do the following:

NOTE Column splice loads are not relevant in concrete structures, and therefore, they are not displayed for concrete beams.

---

1. Open a frame view containing the steel frame for which you want to see the column splice loads.
2. On the **Status bar**, click **Results View**.
3. In the **Loading** list, select the load case or combination that you want to display.
4. On the **Draw** toolbar, click **Column Splice Loads**.

   TIP If the **Column Splice Loads** command is not active, ensure that the current view is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create drawing scales (page 968)

Create foundation reaction drawings

Foundation reaction drawings are general arrangement drawings that also contain support reactions in order to assist foundation design. Foundation reaction drawings can only be created in 2D results views. To create foundation reaction drawings, do the following:

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   Typically, the part is be the base construction level.
2. In the **Loading** list, select the load case or combination that you want to display.
3. On the **Draw** toolbar, click **Foundation Reactions**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create drawing scales (page 968)

Create loading plan drawings

Loading plan drawings are general arrangement drawings that also contain applied loads for the selected load case. Loading plan drawings can only be created in 2D scene views. To create loading plan drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. In the **Loading** list, select the load case or combination that you want to display.

3. On the **Draw** toolbar, click **Loading Plan**.

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create drawing scales (page 968)

Create member detail drawings

Member detail drawings include concrete beam detail drawings, concrete column detail drawings, concrete wall detail drawings, non-concrete beam detail drawings and non-concrete column detail drawings. For instructions on how to create different member detail drawings, see the following paragraphs.

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Options** --> **Member Details**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

See also

Create drawing scales (page 968)

Create concrete beam detail drawings

Concrete beam drawings display the beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create concrete column detail drawings

Concrete column drawings display the column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the column that you want to detail.
2. When the desired column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create concrete wall detail drawings

Concrete wall details drawings display the wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the wall that you want to detail.
2. When the desired wall is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create non-concrete beam detail drawings

Non-concrete beam detail drawings display individual steel beam details.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create non-concrete column details drawings

Non-concrete column detail drawings display individual steel beam details.

1. Hover the mouse pointer over the column that you want to detail.
2. When the desired column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create slab and mat drawings

Slab and mat drawings refer to slab and mat layout drawings and punching shear check detail drawings. Slab and mat detail drawings are used to convey slab reinforcement and patch reinforcement requirements, and contain a quantity table for the reinforcement displayed with a detailing allowance added. Punching check detail drawings display an individual check detail in, and also contain the option to display the reinforcement quantities table.

See also

Create drawing scales (page 968)

Create slab or mat layout drawings

1. Open a 2D scene view displaying the slabs or mats that you want to include in the drawing.
2. On the **Draw** tab, click **Slab/Mat Detailing**.

   NOTE If the **Slab/Mat Detailing** command is not active, ensure that you are in a 2D scene view.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.

4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

Create punching shear check detail drawings

1. Hover the mouse pointer over the punching shear check item that you want to detail.
2. When the desired punching shear check item is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create foundation drawings

Foundation drawings refer to isolated foundation detail drawings and foundation layout drawings. Isolated foundation detail drawings display an individual foundation detail in plan. If necessary, you can also display the detail in cross section and a reinforcement quantity table. As for foundation layout drawings, they display the piling layout and the layout of isolated foundations. If necessary, you can also display isolated foundation details, a reinforcement quatity table, an isolated foundation schedule, and a pile location table.

See also

Create drawing scales (page 968)

Create isolated foundation detail drawings

NOTE Before you create a base cap detail drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Options** --> **Isolated Foundation Detail**.
3. Adjust the drawing options on the **Content** and **Isolated Foundation Detail** sub pages according to your needs.
4. Click **OK**.

---

1. Hover the mouse pointer over the base cap or pile cap that you want to detail.
2. When the desired base cap or pile cap is highlighter, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create foundation layout drawings

1. Open a 2D scene view displaying the piles, bases or pile caps that you want to include in the drawing.
2. On the **Draw** tab, click **Foundation Layout**.

   NOTE If the command is not active, ensure that you are in a 2D scene view displayed in 2D.

   The **DXF Export Preferences** dialog box opens.

3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.

5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

Create concrete member schedule drawings

Concrete member schedule drawings include concrete beam schedules, concrete column schedules and concrete wall schedules. In beam schedule drawings, the reinforcement is listed on a span by span basis, and contractors or specialist detailing firms can use the information to produce the necessary bar bending schedules. As for concrete column schedule drawings, they display a cross section through each stack of the selected columns, and if necessary, allow you to include a reinforcement quantity table. Concrete wall schedule drawings display a cross section through each stack of the selected walls, and if necessary, also allow you to include a reinforcement quantity table

---

NOTE Before you create a schedule drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to **Options**.
3. Adjust the drawing options on the sub pages according to your needs.
4. Click **OK**.

See also

Create drawing scales (page 968)

Create concrete beam schedule drawings

You can create beam schedule drawings by building, by floor or by selected beams. The information shown in the schedule is based on the different design groups. Beam schedule drawings are created in dxf format even though they do not include graphical information, so that you can add them to beam detail drawings.

1. Open a 2D scene view of a part displaying the beams that you want to include in a schedule. The part can be, for example, a construction level, frame, or sub model.

2. On the **Draw** tab, click **Beam Schedule**.

   NOTE If the **Beam Schedule** command is not active, ensure that you are in a 2D scene view that is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

Create concrete column schedule drawings

1. Open a 3D view or a frame view displaying the columns that you want to include in the schedule.
2. On the **Draw** tab, click **Column Schedule**.

   The **Select column schedule content** dialog box opens.
3. Select the columns to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Create concrete wall schedule drawings

1. Open a 3D view or a frame view displaying the walls that you want to include in the schedule.

2. On the **Draw** tab, click **Wall Schedule**.

   The **Select wall schedule content** dialog box opens.

3. Select the walls to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

Manage drawings in batches

Although you can create individual drawings according to your needs, creating a batch of drawings at once is often more efficient, especially when working with larger models.

The **Drawing Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.
- Either add a drawing sheet manually or generate a series of drawing sheets.
- Select the frames, levels, or members for which you want to create drawings.
- Arrange the drawings on the drawing sheet either in a linear or a grid arrangement.
- Select the load cases or combinations for load-dependent drawing variants.
- Create drawing revisions.
- View the revision history of drawings.
- Reset reinforcement marks on concrete detail drawings in order to remove gaps in the bar mark numbering.

Create or generate drawings in batches

To create multiple drawings easily or to automatically generate concrete detail drawings, see the following instructions:

See also

Specify the drawing layout (page 985)

Specify the loading for load-dependent drawings (page 986)

View drawings (page 987)

Review drawings (page 987)

Reset reinforcement marks in concrete detail drawings (page 986)

Create new drawings manually

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.
6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.

Generate concrete beam and column detail drawings

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.
3. Click **Generate**.

   New drawings are created with automatically generated content. One drawing contains typical beams or columns, wheras additional drawings can contain any ungrouped beams or columns.
4. If necessary, click the available drawings to rename them, and type a new name.

Specify the drawing layout

To specify the layout of the drawings that you have created, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Layout...**

   The **Drawing Layout** dialog box opens.
5. Define the direction and arrangement of the layout.
6. Click **OK**.

See also

Specify the loading for load-dependent drawings (page 986)

Specify the loading for load-dependent drawings

To specify the loading for load-dependent drawings, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Loading...**

   The **Select loading** dialog box opens.
5. Select the desired load cases and combinations.
6. Click **OK**.

See also

Create or generate drawings in batches (page 984)

Reset reinforcement marks in concrete detail drawings

Each bar geometry used in the model has an associated reinforcement mark. However, if a model is designed and some of the bars fall out of use, the mark assignment is still retained. As a result, there may be gaps in the marks and marks starting at high numbers. To avoid this, you can reset all the reinforcement marks in the model. Do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.
3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

See also

Create or generate drawings in batches (page 984)

View drawings

To view the drawings that you have created in the **Drawing Management** dialog box, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

See also

Create or generate drawings in batches (page 984)

Review drawings

To apply a revision to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.
5. Name the revision and type a revision note.

6. Click **OK**.

   The drawings open in an available DXF tool.

See also

View the revision history of drawings (page 988)

View the revision history of drawings

To view the formerly applied revisions to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

See also

Review drawings (page 987)

Manage schedule drawings in batches

Although you can create individual schedule drawings according to your needs, creating a batch of schedule drawings at once is also possible. For more information, see the following instructions.

The **Schedule Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.
- Create new drawing sheets.
- Select the members that you want to include in the drawing.
- Arrange the drawing layout.
- Create drawing revisions.
- Reset reinforcement marks in order to remove gaps in the bar mark numbering.

Create new schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.
6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.

Specify the schedule drawing layout

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. Click **Layout...**

   The **Drawing Layout** dialog box opens.
3. Define the direction and arrangement of the layout.
4. Click **OK**.

View schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

Reset reinforcement marks in schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.

2. In the **Drawing Variant** list, select **Concrete Beam Schedule**, **Concrete Column Schedule**, or **Concrete Wall Schedule**.

3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

Review schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.

3. In the **Available Drawings** list, select the desired drawing.

4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.

5. Name the revision and type a revision note.

6. Click **OK**.

   The drawings open in an available DXF tool.

View the revision history of schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.

3. In the **Available Drawings** list, select the desired drawing.

4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

12 Manage models

Click the links below to find out more various model management tasks:

- Define and modify head codes and design codes (page 992)
- Define and modify units (page 993)
- Manage object references (page 995)
- Create and manage user-defined attributes (page 1026)
- Manage settings sets (page 1000)
- Manage material databases (page 1004)
- Manage properties and property sets (page 1021)
- Manage sub structures (page 1031)
- Working with large models (page 1037)

See also

Model Settings (page 2263)

12.1 Apply and manage model settings

To apply and manage the various defaults and settings that apply to the current project, see the following instructions.

1. On the **Home** tab, click **Model Settings**.
2. Adjust the Model Settings (page 2263) according to your needs.
3. After adjusting the settings, do one of the following:

To	Do this
Apply the changes to the current project	• Click **OK**.
Save the changes back to the active settings set for future use	• Click **Save...**
Cancel the changes	• Click **Cancel**.

Revert to the model settings specified in the active settings set	• Click **Load...** **NOTE** Clicking **Load...** loads all of the model settings from the active settings set, not only those specified on the current page of the **Model Settings** dialog box. Analysis options and design options are not loaded.

See also

Define and modify head codes and design codes (page 992)

Define and modify units (page 993)

Manage object references (page 995)

Create and manage user-defined attributes (page 1026)

Define and modify head codes and design codes

Tekla Structural Designer allows you to select from a range of international design codes of practice. Each new project initially adopts the codes that have been specified in the active settings set. However, you can also change the codes in the middle of a project.

See also

Design code settings (page 2264)

Change design codes in an existing project

WARNING If you change the head code in an existing project, the following will occur in the model:

- Some materials, steel sections, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new head code/unit system.
- Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind load cases will need recreating.
- Seismic loading (if any) will be deleted. The seismic wizard will need re-running.
- All combinations will be deleted.

1. On the **Home** tab, click **Model Settings**.

   The **Settings** dialog box opens.
2. Go to **Design Codes**.
3. Select the head code as required.

   The head code controls which action and resistance codes are available for selection.
4. Set the action and resistance codes according to your needs.
5. Click **OK**.

Define default design codes for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make it active.
3. Go to **Design Codes**.
4. Check that the head code is set as required.

   The head code controls which action and resistance codes are available for selection.
5. Set the action and resistance codes according to your needs.
6. Click **OK**.

   Tekla Structural Designer retains the selected codes to apply them for each new project until you decide to modify them.
7. To use the set design codes in a new project, on the **Home** tab, click **New**. A new project opens with its design codes, (and all of its other model settings) copied from the active settings set.

Define and modify units

Tekla Structural Designer allows you to switch between Metric and US customary units. Furthermore, you can select the units which you want to use in the selected unit system. For example, if you use US customary units, you can input forces in either kip or lb, and lengths in feet, inches, or feet and inches.

Change units and unit precision in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **Units**.
3. Select the unit system.
4. Set the units and unit precision that you want Tekla Structural Designer to use.
5. Click **OK**.

NOTE The length unit can be set appropriate for the type of dimension being input:

- **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
- **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
- **Deflection** units are used for reporting deflection results.
- **Distance** units are used for defining large dimensions.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Define the default units and unit precision for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make the set active.
3. Go to **Units**.

4. Select the unit system.
5. Set the units and unit precision that you want Tekla Structural Designer to use.

NOTE The length unit can be set appropriate for the type of dimension being input:

- **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
- **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
- **Deflection** units are used for reporting deflection results.
- **Distance** units are used for defining large dimensions.

6. Click **OK**.
7. To use the set units in a new project, on the **Home** tab, click  **New**. A new project opens with its units, (and all of its other model settings) copied from the active settings set.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Manage object references

Tekla Structural Designer contains a flexible object referencing system that is specifically designed considering the use of multiple materials within the same model.

You can define the object references to meet your needs on the **References** page of the **Settings** dialog box.

See also

Object reference settings (page 2266)

Basics of object reference formats

Object reference formats are user definable. Each object type has its own reference format that has been built using individual components. You can customize the reference format by adding or removing components as required.

See some of the typical components in the following table:

Icon	Item	Further information
M	Material	Fully user-definable text. For example: • `S` for Steel • `C` for Concrete
C	Characteristic	Fully user-definable text. For example: • `B` for Beam • `C` for Column
L1	Start Level Reference	Not applicable
L2	End Level Reference	
P1	Start Point Reference	Grid or construction point reference. The start and end reference points P1 and P2 work from the grids that you define, and from the construction points that have been created automatically when you place members that do not lie between existing points. **NOTE** When reference points are constructed using two grid reference points, you must always use a separator. For example, type `A/11` to avoid confusion with A1 and 1.
P2	End Point Reference	
D	Direction	Applies for beams only.
X	Count	Tekla Structural Designer keeps a separate count for each level for each object type.

Icon	Item	Further information
		NOTE Tekla Structural Designer does not keep a count by material. For example, steel beams and concrete beams are included in the same count on a level. If the direction prefix is specified, Tekla Structural Designer keeps separate counts for each direction. Direction 1 is defined as objects falling within ± 45 degrees of the horizontal, and Direction 2 ± 45 degrees of the vertical of the global axis.
Not applicable	Reinforcement > Bar Layer: • T1 • T2 • B1 • B2	• T1 is the top surface reinforcement in the 1st (outer) layer. • T2 is the top surface reinforcement in the 2nd (inner) layer. • B1 is the bottom surface reinforcement in the 1st (outer) layer. • B2 is the bottom surface reinforcement in the 2nd (inner) layer.
	Reinforcement > Bar Size	Not applicable
	Reinforcement > Center Spacing	The center spacing between bars.
	Reinforcement > Detailing prefix	An item that is typically added in front of the bar size.
	Custom text	Fixed text that you can include at any position within the reference. For example: • `Block C`
	Separators • backslash • dash • slash	Optional items that you can place between items.

Icon	Item	Further information
	• space • times	

Note that:

- When you create objects of a certain type, the default references from the active settings set are applied.
- You can modify the syntax of the reference format at any time. Objects created after the changes will adopt the new format.
- Once objects have been created, you can edit their references on an object-by-object basis, so that they can be further individualized.
- For members, the object reference does not include the group reference, geometric shape, and section size as part of the reference descriptor. Instead, options are provided in the **Scene Content** settings and the GA drawing's control to show the information.

Modify reference formats and texts in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **References**.
3. Adjust the way in which the references are applied.
4. Click **OK**.

Modify the reference format syntax of an object type

1. Go to **References** --> **Formats** .
2. Click the **...** button on the right side of the desired reference format.
3. Do one of the following:

To	Do this
Add an extra item to the reference format	• Click **Add...** and select the desired item. The selected item appears at the end of the reference format.
Re-order the items in the reference format	a. Select the item you want to move. b. Drag the item to reposition it.
Add custom text to the reference format	a. Click **Add...** and select **Custom Text**. b. Drag the new component to the desired position. c. Click the **Custom Text** component to edit it. d. Type the desired text in the box. e. Click **Set**.

Remove an item from the reference format	a.	Select the item you want to remove.
	b.	Drag the item outside the **Edit Reference Format** dialog box.

4. Click **OK**.

Change the text used for the materials and characteristics in the reference format

1. Go to **References** --> **Texts** .
2. According to your needs, go to the **Characteristics** or **Materials** tab.
3. Modify the desired text according to your needs.
4. Click **OK**.

Renumber members

You can use the **Renumber** command to simultaneously renumber all member types in the model whose reference format includes the count item.

Member types are initially numbered in the order in which they are created. Renumbering makes the members easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

1. On the **Structure** tab of **Project Workspace**, right-click the **Members** branch.
2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all members in the model that include a count in their reference format.

Renumber slabs

You can use the **Renumber** command to simultaneously renumber all slab items in the model.

Slab items are initially numbered in the order in which they are created. Renumbering makes them easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

1. On the **Structure** tab of **Project Workspace**, right-click the **Slabs** branch.
2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all slab items in the number.

Adjust the default references to be applied to new projects

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select a suitable settings set in the list, and click **>> Active** to make it active.
4. Go to **References**.
5. Adjust the way in which the references are applied to each of the object types in your projects.
6. Click **OK**.
7. To start a new project using the new references, on the **Home** tab, click **New**.

12.2 Manage settings sets

The first time Tekla Structural Designer is run after installation you are required to select a country or region. Based on this selection, Tekla Structural Designer creates an initial settings set (the 'active' set), containing defaults for design codes, units, and sections, and settings for drawings, schedules, and reports.

You can edit the settings in this set and also create new sets, but only one set can be designated the 'active' settings set.

Each new project takes a copy of the 'active' settings set to form its own independent model settings.

Model settings can then be edited as required without affecting the active settings set.

Add a new settings set

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.

3. Click **Add Copy**.
4. In the **Settings** dialog box, modify the settings for the new settings sets.
5. Click **OK** to save the settings to the new settings set.

Import a settings set for a different region

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Import...**
4. Select the region for which you want to import the settings.
5. Click **OK**.

NOTE To use these settings in your new projects, you will need to make this the active set (page 1002).

Edit the content of a settings set

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set you want to edit.
4. In the **Settings** dialog box, modify the settings according to your needs. For example:
   - Modify the font used to display results in the **Results Viewer**.
   - Modify the appearance of reports.
   - Set the units that you want to use, and set the desired precision for the units.
   - Set the design codes that you want.
   - Set default section sizes for each of the member types.
   - Set if confirmations are required for specific actions.
   - Set element references.
   - Modify the appearance of schedules.
   - Modify drawing types and styles.
   - Modify the colors used in 2D and 3D views.

5. Click **OK** to save the settings to the selected settings set.

Change the active settings set

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set required from the drop list.
4. To make the set a default set for new projects, click **>> Active**.
5. Click **OK**.

Delete a settings set

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the set you want to delete.
4. Click **Remove**.

Load settings from the active settings set to the current project

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the required settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be updated.

   - On the **Home** tab, click **Model Settings**.
   - On the **Analyze** tab, click **Options**.
   - On the **Design** tab, click **Options**.

- On the **Draw** tab, click **Edit...**

6. Click **Load...**

   A confirmation dialog appears.
7. To update the model settings to match the active settings set, click **Yes**.

Save settings from the current project to the active settings set

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be saved.

   - On the **Home** tab, click **Model Settings**.
   - On the **Analyze** tab, click **Options**.
   - On the **Design** tab, click **Options**.
   - On the **Draw** tab, click **Edit...**
6. Click **Save...**

   A confirmation dialog appears.
7. To update the active settings set to match the current settings, click **Yes**.

Copy a settings set from one computer to another

In order to apply consistent settings on a company wide basis, the xml file of the appropriate settings set should be manually copied to other computers.

1. On the computer containing the settings set, go to the **Home** tab and click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.

3. Click **Open Folder**.
4. Select the xml file of the settings set, and manually transfer it to the **Settings** folder on each destination computer.
5. To view and use the new settings set, restart Tekla Structural Designer on the destination computers.

See also

Settings set settings (page 2386)

12.3 Manage material databases

Material databases contain an extensive range of sections, materials, reinforcement, decking, and connectors for each head code and country. The initial data in the material databases is protected, so that standard items cannot be accidently edited or deleted. However, you can add user data from other sources or suppliers, if needed. All user data is fully editable. You can view the protected data and manage the user data from the Materials dialog (page 2418).

Add, modify and delete user-defined sections

A user defined section can take the form of a custom, or a compound section, a compound section being one that comprises of 2 chords or more connected by battens or lattice or welded.

See also

The Sections dialog box (page 2423)

Add a user-defined custom or compound section to the material database

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.
5. Click **Add...**
6. Enter values for each of the requested variables.
7. Click **OK**.

The new section size is now displayed in the **Item** pane.

Modify a user-defined custom or compound section in the material database

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.
5. In the **Item** pane on the right, select the desired section size.
6. Click **Edit...**
7. Modify the section properties according to your needs.
8. Click **OK**.

Delete a user-defined custom or compound section from the database

NOTE Only user-defined sections, marked with *, can be deleted.

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.
5. In the **Item** pane on the right, select the desired section size.
6. Click **Delete**.

   A confirmation dialog appears.
7. To delete the section, click **Yes**.

Manage design section orders

A design section order is a list of sections you wish to consider for design when the Autodesign property is checked.

The design process commences by starting with the first section in the chosen order file. Any section that fails any of the design conditions is rejected and the design process is then repeated for the next available section in the list.

On completion of the design process, the first satisfactory section from the section designation list is assigned to the member.

View the list of sections in a design section order

You can view the list of sections in a design section order, by following the steps below.

1. Edit the properties of the member.
2. Click the **Design section order** drop list and select **<New\Edit...>**
3. Choose a section order from the available list and then click **Edit...**

The sections contained within the chosen order file appear in the **Sections in use** list on the right of the page.

Specify that a section in the list should not be considered for design

You can control which sections in a design order list are considered by following the procedure below.

Only checked sections within the list are considered during the design process. Uncheck a section and it will no longer be considered.

WARNING Limiting the choice of sections by unchecking a section within an order file is a global change that affects ALL projects, (not just the currently open one). It is typically used to eliminate unavailable or non-preferred sections from the design process. If design requirements for an individual member require section sizes to be constrained, (due to, for example depth restrictions), then the choice of sections should be limited instead by using Size Constraints, (as these only affect the current member).

Sort the listed sections by a different property

You can sort a design section order list by following the steps below.

While viewing the list of sections:

1. Click **Criteria...** to open a dialog for selecting the sort critera.
2. Select a property from the **Available Criteria** list and click **Add**.
3. Choose the order for sorting (Ascending, or Descending).
4. Add further criteria as required.
5. Click **OK** to close the dialog.
6. Click **Sort** to re-order by the chosen criteria.
7. Having sorted, if you don't want to subsequently move individual sections up or down the list, check **Keep sorted** to de-activate **Move Up** and **Move Down**.

NOTE Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Specify that a section is non-preferred

Some sections might be more expensive or difficult to obtain; you might therefore want other sections to be chosen in preference to them, (whilst still keeping them available).

You can achieve this by moving the "non-preferred" sections further down the design order list.

To move a section up or down the list:

1. If **Keep sorted** is checked, you must uncheck it in order to activate **Move Up** and **Move Down**.

2. Highlight the section in the **Sections in use** list and then click **Move Down** or **Move Down** to promote or demote it.

NOTE Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Reset a design section order back to the original default

If you have made changes to a design order list, the following steps you through resetting the order file to the original default values.

1. Edit the properties of the member.
2. Click the Design section order drop list and select **<New\Edit...>**
3. In the Select a Section Order dialog, highlight the section order that you want to reset.
4. Click **Reset**

The highlighted design section order is reset to its default settings.

NOTE The Reset button is only displayed for the pre-installed section orders. (User defined section orders can be deleted but not reset.)

Create a new Design section order

If you want to create a completely new design section order you can do so as follows:

1. Edit the properties of a member.
2. Click the **Design section order** drop list and select **<New\Edit...>**
3. In the **Select a Section Order** dialog, click **Add...**
4. Enter a unique name for the new design section order.
5. Select the **Country** and the **Section Group** required.
6. Either click **Add All** to add all the available sections, or highlight just the sections you require and click **Add Selected**.

   NOTE When adding selected sections you can use the **Top**, **Bottom**, **Above Selected**, **Below Selected** options to specify where they appear in the list. However, these options are ineffective if you have a sort criteria specified and the **Keep sorted** box is checked.

7. Sort individual sections in the **Sections in use** list, using **Move Up** or **Move Down**, as required.
8. When the list of sections in use is as you want it, click **OK**

The new design section order appears on the list of available section orders.

Add simple connection resistances to the database

The materials database contains pre-defined connection types with pre-defined resistances for steel beams to Eurocode[1] and US[2] head codes.

In addition, you can specify user-defined connection types and user-defined resistances for any head code and save them to the section database. Once defined, these can then be used to Check simple connection resistance (page 809) across all projects.

[1] *The Eurocode resistances are derived from SCI publication P358 and make use of UK NA values for partial safety factors, regardless of the Eurocode Country setting.*

[2] *The US resistances are derived from AISC publication Steel Construction Manual 14th Ed and are only given when US Customary units are selected. Separate resistances are given for LRFD and ASD.*

Pre-defined connection types and resistances

For Eurocodes:

With the Member Type set to Simple Beam, three *pre-defined* connection types are available for S355 and S275 grades:

- Fin Plates,
- Full Depth End Plates
- Partial Depth End Plates

These have connection resistance values defined for UB and UKB section beams. The bolts considered are all size M20, of property class 8.8, and ordinary or flowdrill type. The plates considered are all S275 grade. Fin Plates and Partial Depth End Plates have resistances defined for beams with 0, 1 or 2 notches, while the resistances for Full Depth End Plates assume no notches. Fin Plates have resistances for 1 or 2 vertical lines of bolts.

For US (ACI/AISC) and US Customary units:

With the Member Type set to Simple Beam, and the steel grade set to 'Any' one *pre-defined* connection type is available:

- Single Plate

Single Plates have connection resistance values defined for W section beams of any Fy value. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates considered are all of Fy 36 ksi, with thickness ranging from 1/4 in to 9/16 in. Single Plates have resistances defined for beams with no coping and top flange coping only. All resistances assume 1 vertical line of bolts only.

In addition, two further Simple Beam connection types are available for the 50 ksi grade:

- All-Bolted Dbl Angle,
- Shear End-Plate

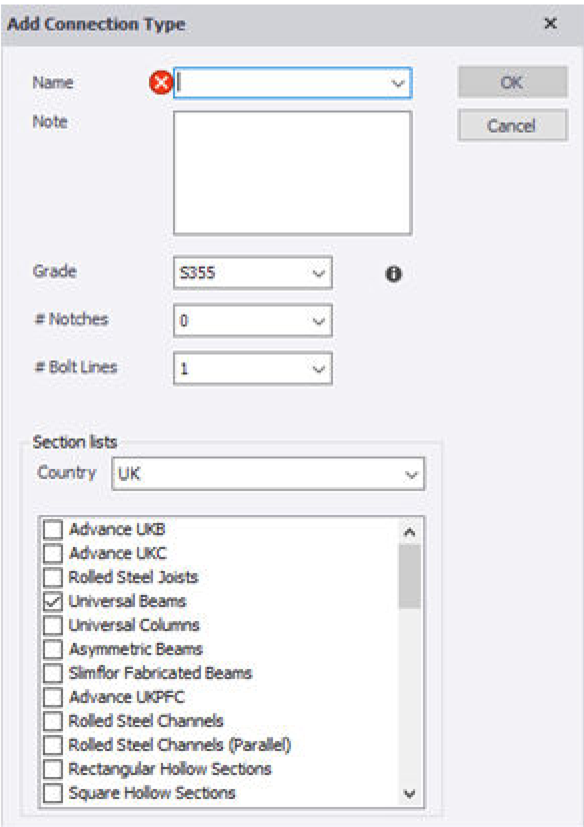
These have connection resistance values defined for W section beams, and for Single Plates with W section beams of F_y 50 ksi. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates and angles considered are all of F_y 36 ksi, with thickness ranging from 1/4 in to 9/16 in. All-Bolted Double Angles have resistances defined for beams with no coping and top flange coping only, while the resistances for Shear End-Plates assume no coping. All resistances assume 1 vertical line of bolts only.

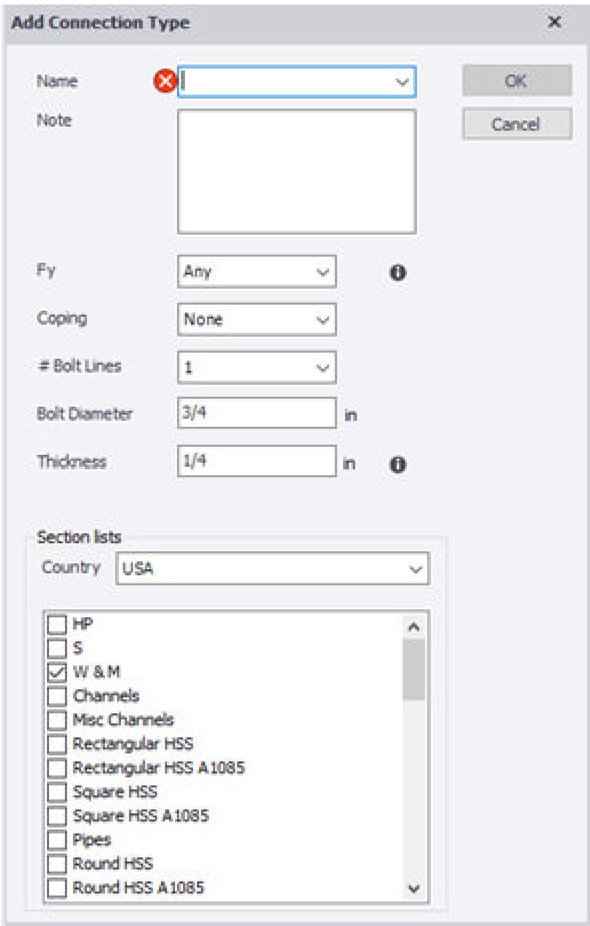
NOTE The pre-defined connection types and resistances can be reviewed in the Connection Resistance dialog (page 2399).

Add user-defined connection types

1. Open the Connection Resistance dialog (page 2399)
2. Under **Connection Types**, click **Add...**

   An **Add Connection Type** dialog appropriate to the current headcode is displayed.





3. In the Name box enter a name for the Connection Type.

   If you want to create a Connection Type with a number of variations of Grade, Notches or Bolt Lines then it is important to enter exactly the same Name each time - e.g. the database will recognise “CT1” and “CT 1” as separate Types simply because of the space in the second name. This has an effect when viewing results in tabular data or reports, where Tekla Structural Designer carries out an optimisation process to find the first passing resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name

   NOTE It is possible to use the same Name as one of the pre-defined Types and it will be recognised in the optimisation process as essentially being the same Connection Type

4. In the Note box enter any descriptive text you would like to see displayed in the dialog’s Info box.

5. Select a Grade or, with US codes, an F_y value.

   Note that *subgrades* like S355 J0 are ignored e.g. a Connection Type with S355 Grade assigned will have its resistances applied to beams or braces of Grade S355 J0, S355 J2, etc.

6. Select # Notches and # Bolt Lines or, with US codes, select Coping, # Bolt Lines, Bolt Diameter and Thickness (of plate or angle leg).
7. Select the section lists that you will want to define resistances for (this list can be added to later on if required, in the Edit Connection Type dialog).
8. Click **OK**

   NOTE **OK** is not active until a Name has been defined. After clicking OK 'Hide undefined values' becomes automatically un-ticked and resistance values can be entered against the required sections. If no resistance values are assigned at this stage for at least one section in a selected Section List, then that List will become de-selected and have to be re-selected later on.

You are now able to **add user-defined connection resistances to the database** against the required sections for this connection type.

Edit user-defined connection types

1. Open the Connection Resistance dialog (page 2399)
2. Under **Connection Types**, click **Edit...**

   An **Edit Connection Type** dialog appropriate to the current headcode is displayed.

3. Edit the values as required.
4. Click **OK**

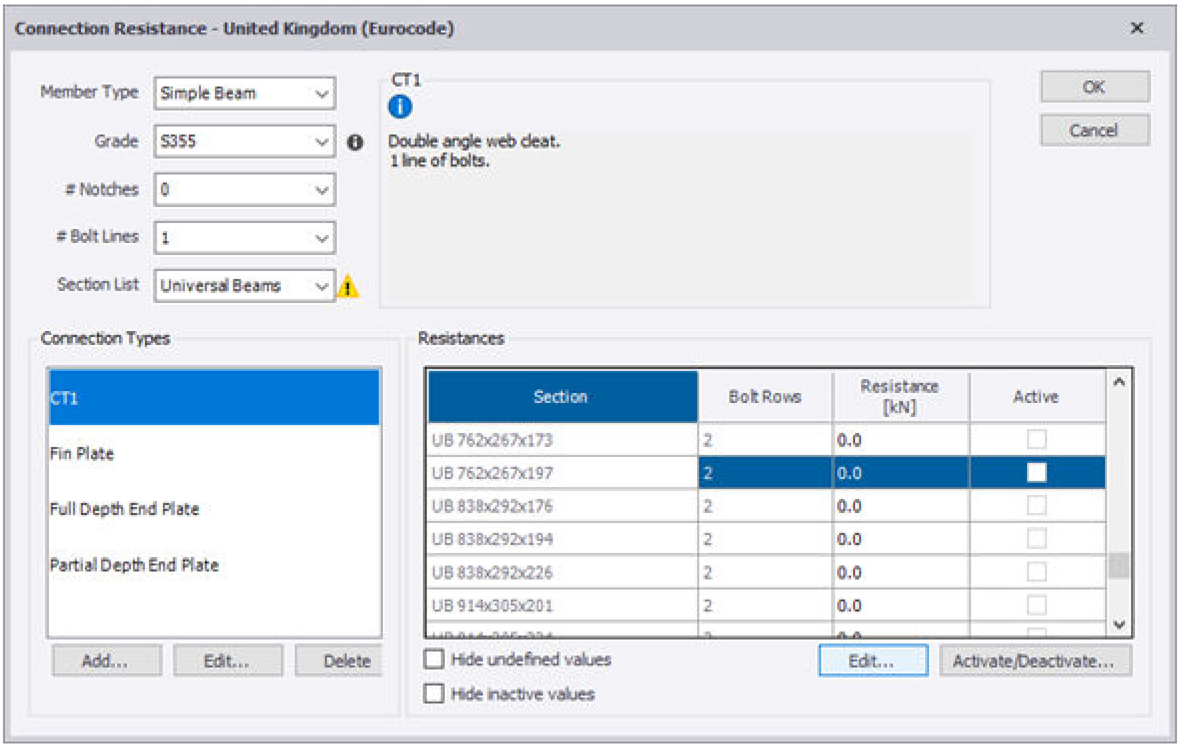
Add user-defined connection resistances

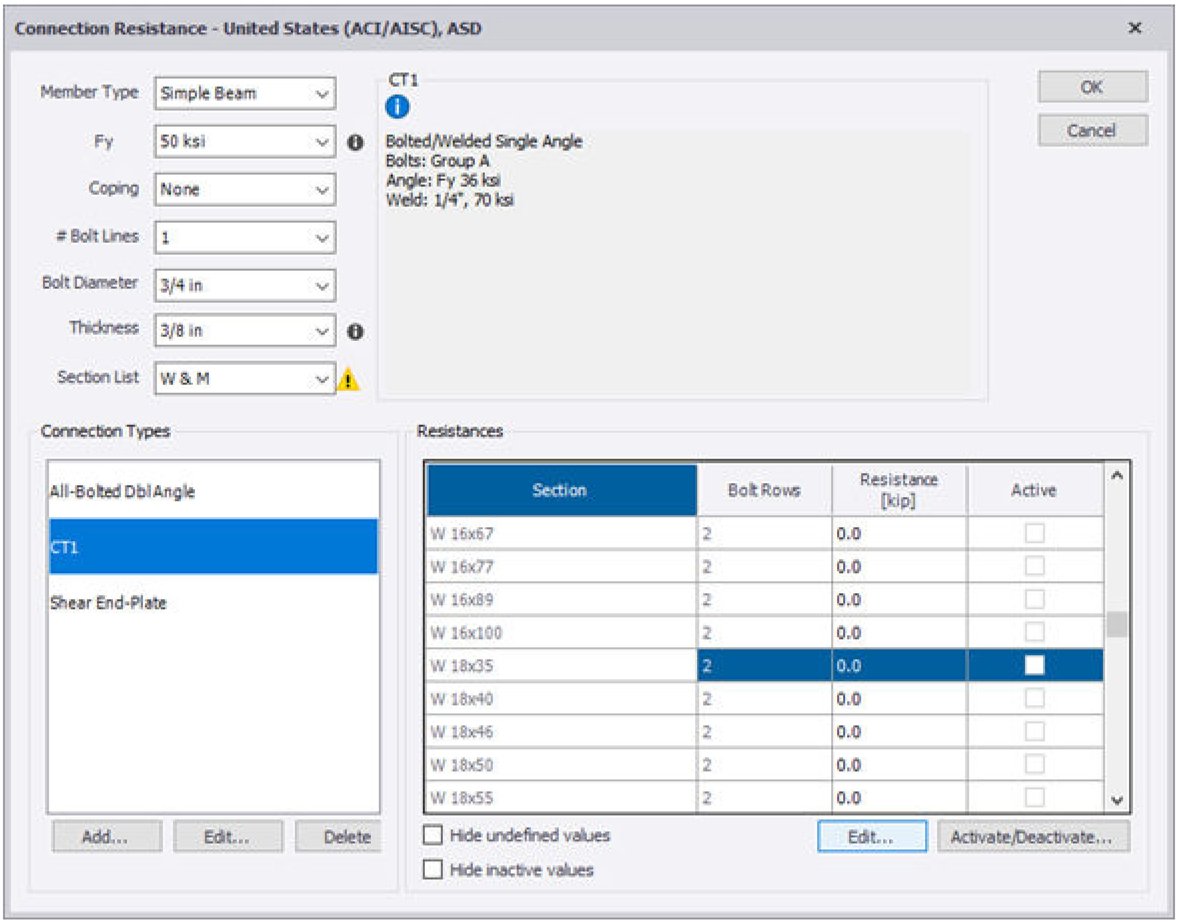
User-defined connection resistances can be added to the database for existing connection types as follows:

1. If it is not already displayed, open the Connection Resistance dialog (page 2399)
2. If necessary, use the filters at the top of the dialog until the required connection is displayed in the Connection Types list.
3. Select the connection in the Connection Types list. The sections are listed in the Resistances table, along with the bolt rows and resistance values for the selected connection type.

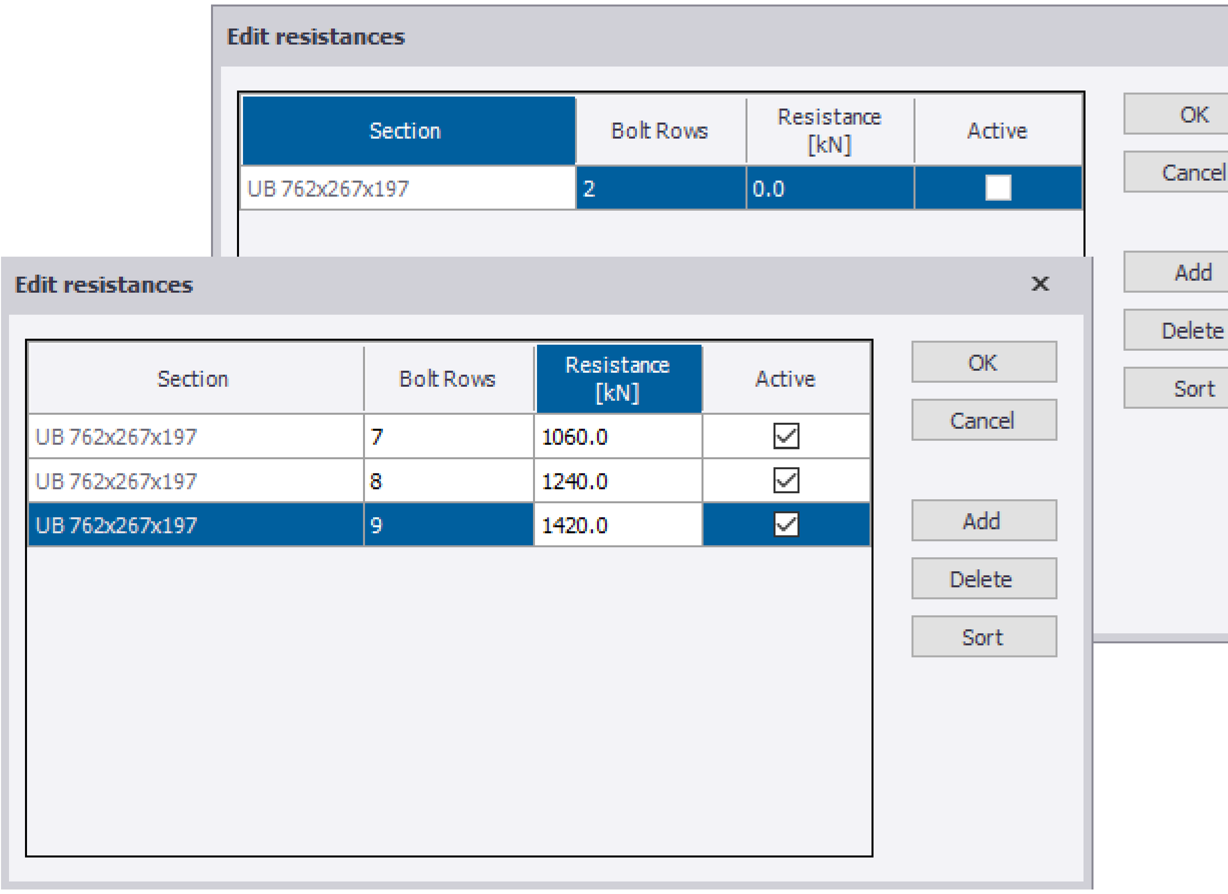
   NOTE If no sections are listed uncheck **Hide undefined values**.

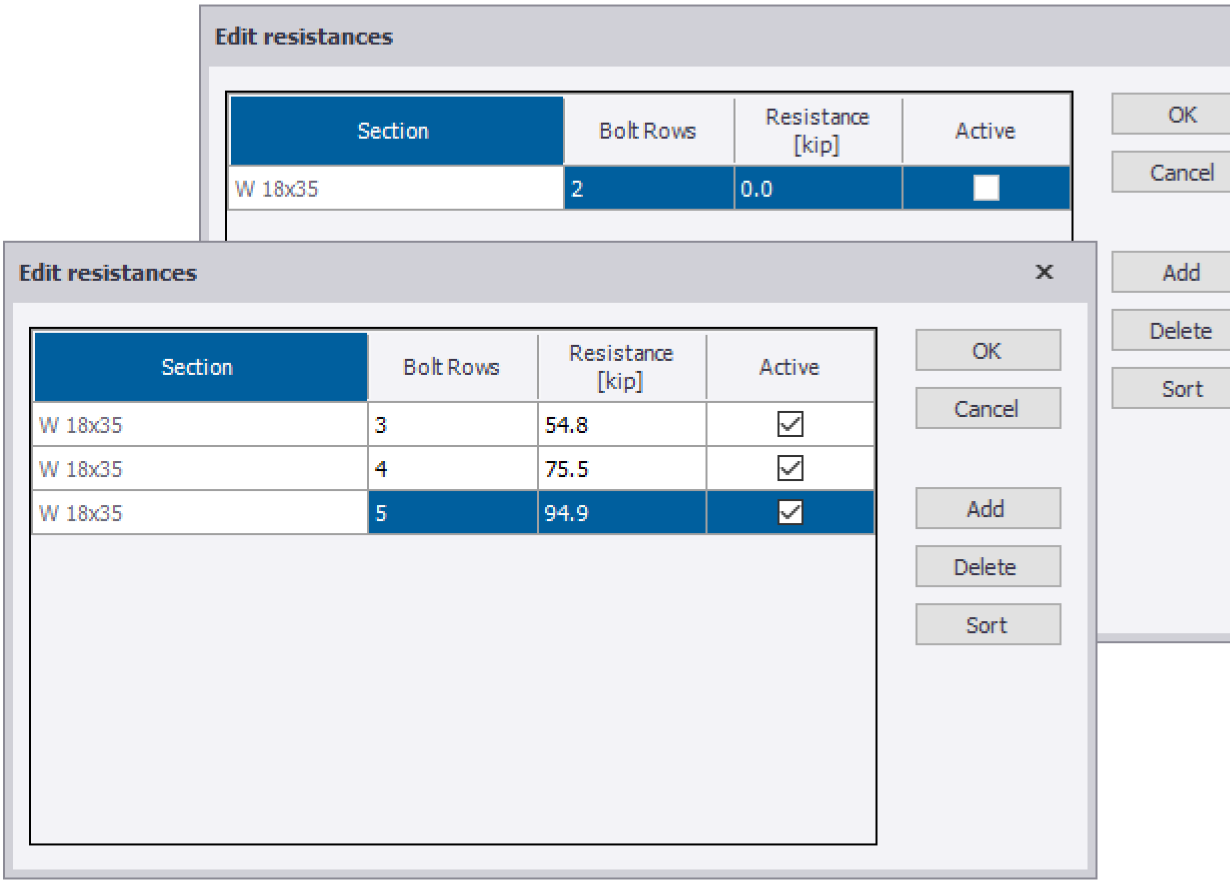
4. Select the section type in the Section List.





5. Resistance values can be entered directly in the table. Alternatively, select a particular section in the table and click **Edit...**. This displays the Edit resistances dialog which allows you to adjust the bolt rows count or to **Add** multiples of the same section size with more than one bolt row count. The bolt row counts can be entered in any order and then use **Sort** to number from lowest to highest. Use the Edit resistances dialog also to **Delete** data rows if no longer required in your database.


Edit resistances
Section
Bolt Rows
Resistance [kN]
Active
UB 762x267x197
2
0.0
OK
Cancel
Add
Delete
Sort
Edit resistances
Section
Bolt Rows
Resistance [kN]
Active
UB 762x267x197
7
1060.0
UB 762x267x197
8
1240.0
UB 762x267x197
9
1420.0
OK
Cancel
Add
Delete
Sort


Edit resistances
Section
Bolt Rows
Resistance [kip]
Active
W 18x35
2
0.0
OK
Cancel
Add
Delete
Sort
Edit resistances
Section
Bolt Rows
Resistance [kip]
Active
W 18x35
3
54.8
W 18x35
4
75.5
W 18x35
5
94.9
OK
Cancel
Add
Delete
Sort

NOTE Connection resistances are made active as the values are entered.

Related video

Predefined connection resistance database for Eurocode and AISC

Add material properties from the model to a material database

When you define a member, its material properties are initially read from the appropriate material database. However, subsequently, the material properties are held with the member itself. This means that you can open and run the model on another computer, even if it does not have a matching material in its database.

When a mismatch between the model's material data and the material database arises, you are not required to add the missing properties to the database. It may however be beneficial to do so if you anticipate that you will need to re-use the property in question in new models.

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.

2. Go to **Model**.
3. Select the desired objects in the **Material data objects in model** list.
4. Click **Add to Database**.

Add materials for a head code

Engineers may need to design using local sections, materials, and reinforcement. These may not be listed in the materials databases for the current head code. However, you can add materials to the material database manually, if necessary.

WARNING If you add data to a material database, you are responsible for both its accuracy and applicability to the selected design had code.

Add a material grade for a head code

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.

2. Go to the **Material** page.
3. Select the required head code.
4. Select the required material type.
5. If the required grade is not listed for the selected head code, click **Add...**

   The **Add Grade** dialog opens.

6. Type in the grade properties.
7. Click **OK** to return the **Materials** dialog box

   The new grade is shown in the list of available grades.

8. To make the new grade the default grade for the material in the selected head code, select the grade and click **>> Default**.

Add a reinforcement class for a head code

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.

2. Go to the **Reinforcement** page.
3. Select the required head code.
4. Select the country to assign the reinforcement to.
5. Select the required type and rib type.
6. If the required class of reinforcement is not listed for the selected head code, click **Add...**

   The **New Reinforcement Class** dialog opens.

7. Type in the reinforcement class properties.
8. Click **OK** to return the **Materials** dialog box

   The new class is shown in the list of available grades.

9. To make the new class the default class for the combination of the selected head code and country, select the class and click **>> Default**.

Add new reinforcement sizes

When you have created a new reinforcement class, the class does not contain any specified bar sizes. Therefore, you must add the sizes you intend to use.

1. On the **Reinforcement** page, click **Add...** on the right side of the **Available sizes** list.

   The **New Reinforcement Bar Size** dialog opens.
2. Type in the properties of the first bar size.
3. Repeat steps 1 and 2 for all bar sizes that you require.
4. When you are finished, click **Close**.

Specify the bar size range to be applied in auto design

After the new reinforcement has been added to the materials database, you can specify the minimum and maximum bar sizes that should apply to the different concrete members when they are auto-designed.

1. On the **Design** tab, click **Options**.
2. Go to the **Concrete** page.
3. Locate the reinforcement of each of the following:
   - **Beam**
   - **Column**
   - **Wall**
   - **Slab** --> **Slab on Beams**
   - **Slab** --> **Flat Slab**
   - **Punching Shear**
   - **Foundations** --> **Pad Base**
   - **Foundations** --> **Pile Cap**
   - **Foundations** --> **Mat Foundations**
4. For each of the above, select the country to which the new reinforcement is registered.
5. For each of the above, select the minimum and maximum bar sizes that apply to auto design.

The new design options now apply to the current project. If you want them to apply to new projects as well, save the changes back to the active set.

Change default design sections for a different head code

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.

2. Go to the **Section Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the **Section Defaults** page, click the currently displayed section for a member to change it.

   The **Select Section** dialog box opens.

5. Select the required country.
6. Select the shape in the **Page** pane on the left.
7. Select the section size in the **Item** pane on the right.
8. Click **Select**.

   The default section has now been updated.

9. Repeat steps 4–8 for all the sections that you want to update.

The new default sections will be applied to new models, provided that the settings set that you modified is the active settings set.

Change default design section orders for a head code

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.

2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.

5. Select the required country.

   NOTE If there are no section orders listed for the selected country, they do not yet exist for the current head code. However, you can still create section orders can still be created by using the **Import...** feature.

6. Highlight an available section order in the list and click **Select**.

   The section order has now been updated.

7. Repeat the steps 4–6 for other members that you want to change.

The new section order will be applied to new models, provided that the settings set that you modified is the active settings set.

Create new section orders for a head code

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.
5. Click **Export...** and save the section order as an XML file.
6. Close the **Select a Section Order** dialog box.
7. On the **Section Order Defaults** page, select the settings set that was originally being modified.
8. In the table, click the section order that you want to change.
9. Select the required country.
10. Click **Import...** and select the XML file that you just created.
11. Select the imported file by clicking **Select**.
12. Repeat steps 4–11 for other member types as needed.

Upgrade material databases

The installed material databases each have their own database version. You can see the version number separately on each page of the **Materials** dialog box.

The original data in each of the material databases is system data, and it cannot be edited. The version number relates specifically to this system data.

From time to time, new system data may become available through an update. When this happens, an **Upgrade** button is displayed next to the current database version on the relevant page of the **Materials** dialog box. You then have the choice to either upgrade the database or retain the old version.

NOTE Updating the database version does not cause you to lose any user data you may have added to the old version, because the user data is automatically copied back in to the new database.

In certain circumstances, an inconsistency can arise between the user data in a model and the installed databases. The inconsistencies can be identified and resolved on the **Model** page of the **Materials** dialog box, either by updating the model data, or by updating the material databases.

1. On the **Home** toolbar, click **Materials**.

   The **Materials** dialog box opens.
2. Click the **Upgrade** button next to the current database version number.

Timber property assumptions

The engineer shall verify that the properties of the selected grades are those that are characteristic of the material, are obtained from up-to-date manufacturer’s data and are applicable to the analysis model. **Analysis Modification Factors** must also be confirmed to be correct.

Head Code Eurocode

The following assumptions have been made:

- Loading will be perpendicular to the wide faces of the laminations (bending about the major axis).
- Moisture content of the sections will be less than 15% - analysis modification factors may need to be applied if this is not the case.
- The glulam grade being used complies with the characteristic properties outlined in BS EN 14080:2013.
- The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
- The section and strength class are compatible.

Head Code ACI/AISC

The following assumptions have been made:

- The moisture content of the structural composite lumber or glulam section is less than 16% and the temperature remains below 100°F - analysis modification factors may need to be applied if this is not the case.
- The glulam section will be loaded perpendicular to the wide faces of the laminations (bending about the x-x axis).
- The glulam grade being used complies with the characteristic properties outlined in AWC NDS supplement 2018.
- The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
- The section and strength class are compatible.

12.4 Manage properties and property sets

When you create a new entity, default properties for it are displayed in the **Properties** window, these can be edited as required before the entity is placed.

Provided the properties are saved to a **property set**, they can be applied to similar entities elsewhere in the current model.

Property sets can also be transferred between models.

In this section we show you how to:

- Save properties to and recall properties from property sets (page 1022)
- Apply property sets to existing entities (page 1023)
- Review where property sets have been applied (page 1024)
- Transfer property sets between models (page 1025)
- Delete property sets (page 1025)

See also

Modify the properties of entities or model objects (page 360)

Save properties to and recall properties from property sets

Saving properties to property sets allows you to re-use them elsewhere.

Save properties from the Properties window to a new property set

NOTE You can only save properties in the **Properties** window to a property set when there are no entities selected. This ensures that each property in the property set has an unique entry.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the entity type whose properties you want to save as a property set.

   The list on the top of the **Properties** window now reads **<unsaved set>**.
2. Specify the properties according to your needs.
3. Click the **Save...** button on the top right corner of the **Properties** window.

   The **Add property set** dialog box opens.
4. Name the property set.
5. Click **OK**.

Save the properties of an existing entity to a named property set

1. Hover the mouse pointer over the desired entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu select **Create property set...** (and if the entity is a beam or column or wall, select the span or stack required).

   The **Add property set** dialog box opens.
4. Name the property set.
5. Click **OK**.

Recall a previously saved property set to the Properties Window

Provided that you have previously saved a property set, you can recall it again later in the **Properties** window, if it is applicable to the current command.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the desired entity type.

   The properties applicable to the selected entity type are displayed in the **Properties** window.
2. Click the list on the top of the **Properties** window.
3. In the list, select the desired property set.

Apply property sets to existing entities

You can apply a property set to existing entities in a **Structural View**, but if you want to make a lot of changes property sets can be applied more easily from a **Review View**.

Apply a property set to an individual entity in a Structural View

1. Hover the mouse pointer over the entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu, select **Apply Property Set...**

   The **Select property set to apply** dialog box opens.
4. Select the desired property set.

5. Click **OK**.

The property set is only applied to the individual entity that you right-click, even if multiple entities were selected. To apply a property set to multiple entities, see the following topic.

Apply a property set to multiple members in a Structural View

1. Select the entities that you want to update.
2. Ensure that the list on top of the **Properties** window shows the correct entity type.
3. On the top right corner of the **Properties** window, click **Apply...**
4. Select the desired property set.
5. Click **OK**.

The property set is now applied to all of the selected entities, provided that they are of the same type.

Apply a property set in a Review View

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, do all of the following:
    a. Set **Review/Update** to **Apply property set**.
    b. Select the entity type.

    NOTE Only those entity types that already have property sets saved will be listed.

    c. If the **Member** entity type was selected, you will also need to select the characteristic.
    d. Select the property set that you want to apply.
4. In the **Review View**, click the entities to which you want to apply the property set.

Review where property sets have been applied

To graphically review where property sets have been applied, see the following instructions.

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, set **Review/Update** as **Review All**.

4. Still in the **Properties** window, select the entity type that you want to review.

   NOTE Only those entity types that already have property sets saved will be listed.

   Entities of the selected type are color-coded to indicate where property sets have been applied.

Transfer property sets between models

Once property sets have been saved, they can be exported from the current model and then imported into another Tekla Structural Designer model.

Export property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to export.
3. Click **Export...**
4. Save the properties as a .tsps file.

Import property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to import.
3. Click **Import...**
4. Select the .tsps file that contains the required property sets.
5. Click **Open**.

   The imported property sets are now listed in the **Manage Property Sets** dialog.

Delete property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to delete.
3. Click **Delete**.
4. Click **OK**.

12.5 Create and manage user-defined attributes

You can define user-defined attributes (UDAs) to save miscellaneous data to files, individual members and panels.

User-defined attributes are flexible, and you can use them for a variety of purposes. For example, you can:

- Apply descriptive labels, such as construction phases.
- Record paint specifications.
- Attach office documents, pictures, or other associated files.
- Link to design files from other applications.
- Filter material lists and member design reports for specific attributes.

Attaching files as attributes

When a file is attached as an attribute, you can embed it within the Tekla Structural Designer file. When embedded, the attached file is included when the model is transferred to another computer.

NOTE The embedded file only gets attached to the Tekla Structural Designer file when the model is saved using **Save** or **Save As**.

This means that the embedded files are not attached when you use **Save Model Only**. Similarly, if you have to revert to an autosaved version of the model, the embedded files will not be attached.

Transferring attributes

Attributes are transferred when models are exported to Tekla Structures and Revit. In the current release, attributes are not yet included for Tekla Structural Designer drawings.

UDA definitions and values

Example UDA definitions are included in the default settings sets. You can easily modify the definitions according to your needs. You can modify the definitions that apply to the current model in **Model Settings**.

You can attach specific UDA values to members and panels according to your needs.

Once a UDA value exists, it can also be assigned to or removed from members graphically by using the **Review View**.

See also

User-defined attribute settings (page 2274)

Create attribute definitions (page 1027)

Attach UDA values to members and panels (page 1028)

Apply attribute filters to material lists and reports (page 1030)

Create attribute definitions

You can create and modify attribute definitions according to your needs. The attributes are defined either in the **Model Settings** dialog box or in the **Settings** dialog box, depending on whether you want the to the attributes be applied to the current model or new models.

Create attribute definitions in the current model

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog opens.
2. Go to the **User Defined Attributes** page.
3. Click **Add**.
4. Define the following properties:
   - Name
   - Type
   - Source
   - Values (value choices that apply when the source is value list)

   NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

   Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Create attribute definitions for new models

1. On the **Home** toolbar, click **Settings**.

   The **Settings** dialog box opens.

2. Go to **Settings Sets**.
3. Select the settings set to which you want the attributes to apply.
4. Go to the **User Defined Attributes** page.
5. Define the following properties:
   - Name
   - Type
   - Source
   - Values (value choices that apply when the source is value list)

   NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

   Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Attach UDA values to members and panels

You can attach office documents, pictures, or other files to the members and panels of your model. When you attach files to members and panels, they are embedded in the file. The embedded files will also be included in the model if the model is transferred to another computer.

See also

Create attribute definitions (page 1027)

Attach a UDA value using the Properties Window

1. Select the members or panels to which you want to attach attributes.

   NOTE If the selection contains different member types, select the first member type in the list at the top of the **Properties** window.

   ---

2. In the **Properties** window, under **UDA**, define the value for each attribute that you want to attach.

3. If the attribute is a file, select the **Embedded** option to save the file inside the model.

   Only embedded files are automatically transferred when the model is copied to another computer.

   TIP For multi-span beams and multi-stack columns or walls, you can attach separate UDAs to individual spans or stacks.

4. Repeat steps 1-3 for additional member types.

Attach an existing UDA value in the Review View

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Set On** or **Toggle**.
3. In the **Properties** window, select the attribute.
4. In the **Properties** window, select a value for the selected attribute.

   NOTE You can use some additional settings in the **Properties** window to control which members and panels are displayed in the **Review View**.

   - Under **Filter**, you can use the **Show** and **Entity type** to select which attribute values and entities are displayed.
   - Select the **Color for other values** box to display members that have an other value of the selected attribute.

5. Click the desired members to add the UDA value.

   TIP To remove the UDA value:

   - If the **[M]ode** is set to **Set On**, change it to **Set Off**.
   - If the **[M]ode** is set to **Toggle**, click the selected members again.

Graphically review the attached UDA values

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.

   The members or panels are color-coded to represent the different values of the selected attribute.

Open a file that has been attached as a UDA

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.
4. Click the member or panel to which the file is attached.

   Provided that the extension has been associated with an appropriate application, the application opens and displays the file.

Apply attribute filters to material lists and reports

User-defined attributes (UDAs) allow you to filter material lists and member design reports for specific attributes. For more detailed information, see the following paragraphs.

See also

Create attribute definitions (page 1027)

Attach UDA values to members and panels (page 1028)

Material list settings (page 2269)

Apply an attribute filter to material list review data

1. On the **Review** tab, click **Tabular Data**.
2. In the **View Type** list, select **Material List**.
3. In the **Filter** list, select **Selected UDAs**.

   The **Select filter items** dialog box opens.
4. Select the required attribute values.
5. Click **OK**.

Apply an attribute filter to a report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the report.
3. In the **Report Structure** list, right-click the chapter or sub-heading that you want to filter.
4. In the context menu, select **Model Filter** --> **Edit\New...**

   The **Select filter** dialog box opens.
5. Click **Add**.

6. In the **Filter properties** list, select **Selected UDAS**.
7. In the **Selected items** list, select the attribute values.
8. Click **OK**.

12.6 Manage sub structures

Sub structures are saved user-defined selection groups of any objects which can be of any size and mix of object types, materials etc. They can be viewed in isolation and used as filters for many areas of program operation. Using sub structures has multiple productivity benefits throughout the Tekla Structural Designer workflow, from model organization and editing through to results review, design and output.

NOTE In Tekla Structural Designer the term *sub structure* is specifically used to refer to a collection of elements, either above or below ground level. It is **not** being used to refer to that part of the structure below ground level (e.g. basements and foundations).

Sub structure characteristics

The basic characteristics of Tekla Structural Designer sub structures are:

- Elements can be a part of more than one sub structure.
- Not every element has to be in a sub structure.
- Deleting a sub structure does not delete the elements within it.
- Sub structures are distinct from, and should not be confused with sub models (page 661).
- Sub structures themselves are never analyzed.
- Sub structure groups are simply collections of other sub structures.

Create a sub structure

1. **To create a sub structure from a Structural View**

1. In the scene view, click or box around the members that you wan to include in the sub structure.
2. Right click and from the context menu select **Add to Sub Structure...**
3. In the left hand pane of the dialog, all the selected objects are listed with on/off check boxes

a. Uncheck any objects that you don't want to be included in the sub structure.

4. In the left hand pane of the dialog, choose an existing or new substructure, and if new, enter the sub structure name.
5. Click **OK**
2. **To create a sub structure from the Project Workspace**
6. In the **Project Workspace** open the **Structure** tree.
7. Expand the **Sub Structures** branch.
8. Right-click the **Sub Structures** sub branch, then in the context menu, select **Create Sub Structure**.
9. Review the sub structure properties in the **Properties** window.
10. In the **Name** field, type the name of the sub structure.
11. Select the color of the sub structure.
12. In the 3D view, click or box around the members that you want to include in the sub structure.
13. Press **[Esc]** when complete.
3. **To create a sub structure from a Review View**
14. On the status bar at the bottom of the window, click **Review View**.
15. On the **Review** tab, click **Sub Structures**.
16. Go to the **Properties** window.
17. Set **Review/Update** to **Update Selected**.
18. Set **Update Sub Structure** to **- - New - -**.
19. In the **Name** field, type the name of the sub structure.
20. Select the color of the sub structure.
21. In the review view, click or box around the members that you want to include in the sub structure.
22. Press **[Esc]** when complete.

Edit a sub structure

1. **To edit a sub structure from the Project Workspace**
1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.

3. Right-click the sub structure that you want to add to, or remove from.
4. in the context menu, select **Edit**.
5. Skip to step 7 below.
2. **To edit a sub structure from a Review View**
6. On the status bar at the bottom of the window, click **Review View**.
7. On the **Review** tab, click **Sub Structures**.
8. Go to the **Properties** window.
9. Set **Review/Update** to **Update Selected**.
10. In **Update Sub Structure**, select the sub structure that you want to modify.
11. In **Selection Mode**, select how you want to modify the structure.
12. In the review view, click or box around the members that you want to modify.

Delete a sub structure

NOTE Deleting a sub structure only removes the association between the selected group of members - the members themselves are not deleted.

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to delete.
4. in the context menu, select **Delete**.

Rename a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to rename.
4. in the context menu, select **Rename**.
5. Enter the new name as required.

Review sub structures

1. On the status bar at the bottom of the window, click **Review View**.
2. On the **Review** tab, click **Sub Structures**.
3. Go to the **Properties** window.
4. Set **Review/Update** to **Review All**.

   Tekla Structural Designer displays each sub structure in a different color.

Create a sub structure group

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Right-click the **Sub Structure Groups** sub branch.
4. In the context menu, select **Create Sub Structure Group**.
5. Select the sub structures to be included in the sub structure group.
6. In the **Properties** window.
   a. In the **User name** field, type the name of the sub structure group.
   b. Select the color of the sub structure group.

Open a 3D view of a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Double-click the sub structure.

   Tekla Structural Designer opens a 3D view of the selected sub structure.

   TIP To open a solver view of the sub structure, right-click the sub structure, and select **Open solver view**.

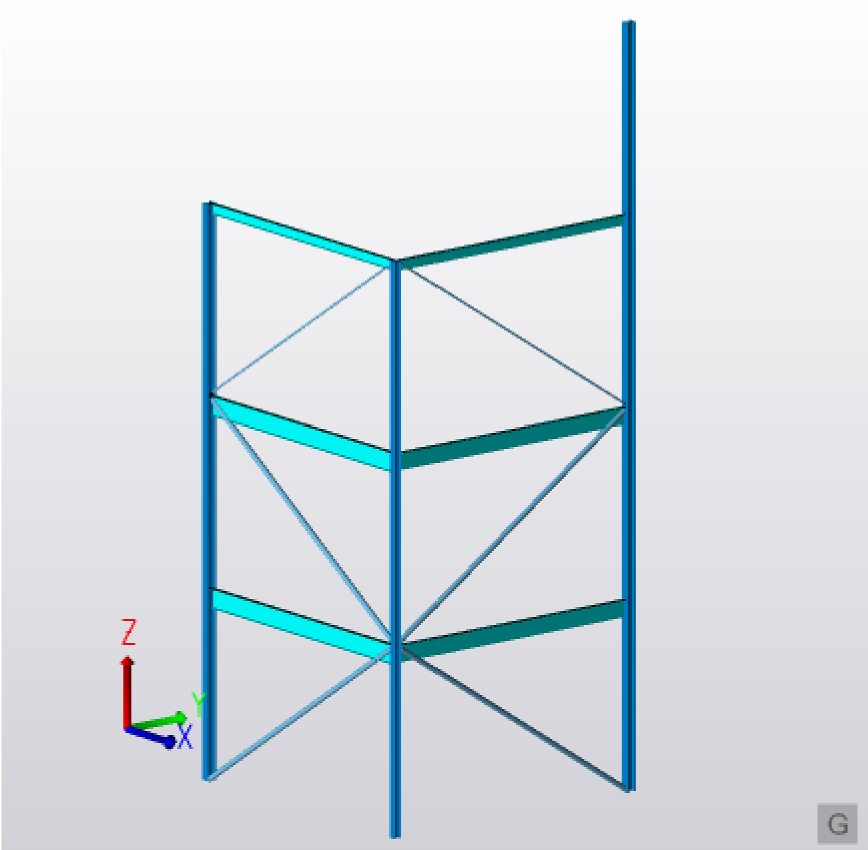
4. Once the view has opened, a (Ghosted) button is displayed in the bottom right corner. Clicking this button toggles the display to either show just the sub structure, or the sub structure with a 'Ghosted On' view of the rest of the model.

Use Ghosted to see the view in the context of the whole model

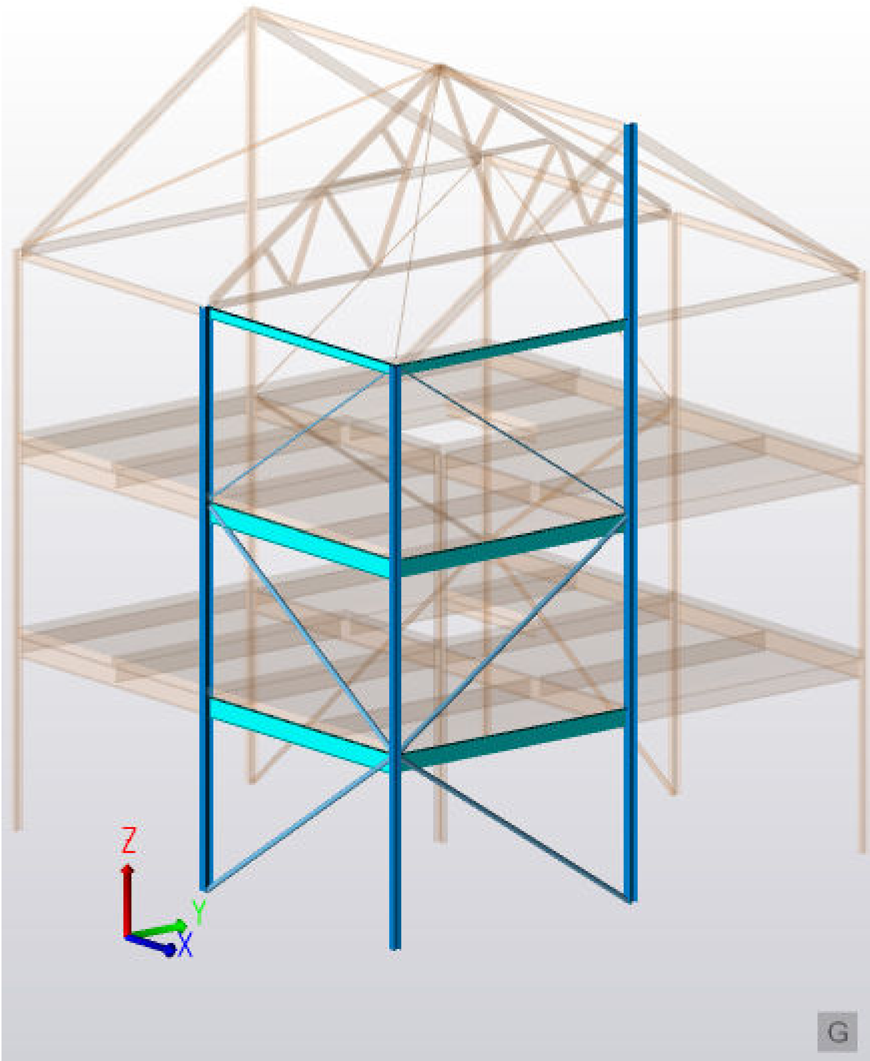
You can toggle **Ghosted** on and off using the G button.

NOTE The G **Ghosted** button is displayed in in the bottom corner of Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button).

- **Ghosted off**: the view is displayed in isolation.



- **Ghosted on**: the view is displayed in the context of the whole structure.



Related video

Ghosted Structure view

Related information

Related videos

How do I use Sub structures?

Sub Structure enhancements

See also

Check all members and walls in a sub structure (page 782)

Design all slab items in a sub structure (page 799)

Check all slab items in a sub structure (page 797)

12.7 Working with large models

Although bigger models will typically need more RAM, you should note that it is the volume of model data and results data created during analysis and design that generates the demand for RAM, so by controlling the volume of data you are able to influence the speed of solution.

Your modelling and design choices can affect the volume of data produced; some of the more significant of these choices are discussed below:

Don't mesh concrete slabs in 3D Analysis

Meshing is not necessary in 3D Analysis for traditional Beam and Slab models as these can use FE decomposition instead.

It is also not necessarily required for Flat Slab models, however, it does become required if you have a transfer slab. It is not critical unless the slab is part of the lateral resistance system.

By default the Mesh 2-way slabs in 3D analysis option that controls this is not checked at any level; you should only check it at a given level when you have good reason to do so.

Using coarser shell meshing in large models

You should review and consider adjusting the mesh parameters as the defaults can often be conservative.

- Using coarser mesh parameters has no impact on sway or modal frequencies.
- If you are not concentrating on slab design you can use very coarse meshes.

Ultimately it is your responsibility to be comfortable with the level of mesh refinement applied to the model. However we would definitely recommend using a courser mesh during design development and then perhaps consider refining a bit more at final design stage.

Using coarser semi-rigid diaphragm meshing in large models

If you have employed semi-rigid diaphragms and are experiencing performance issues you should review and consider adjusting the semi-rigid mesh parameters.

It has been observed that using refined semi-rigid meshes generally has negligible impact on results and only has the effect of slowing down analysis and increasing memory requirements. It is suggested that the default mesh parameters may be more refined that is actually necessary.

Limit the number of load cases and combinations

You can control the number of combinations created when running the Combination Generator.

In particular you should consider limiting the number of wind load cases and combinations.

Also:

- Don't add wind loading during the initial design development.
- Don't activate pattern load cases and combinations until you need to, probably only at the final design stage.

Alternative design approach for large models

For large models, rather than running Design All you may be able to save time by running Analyse All instead and then run a selective design, such as:

- Design by level
- Design by frame
- Design by group
- Design by sub-structure
- Design by member

Effective use of Auto Design

Although for the first design run you might choose to use select bars starting from Minima, on subsequent runs it is generally more efficient to use select bars starting from Current; this will run a check on the current steel provision and if inadequate, it will automatically re-select new steel bars to pass the design.

Check Design can also be very effective - you can turn off the Autodesign and then manually deal with any fails.

Design members for FE chasedown analysis results

This is set in the design options for concrete and by default it is checked on for beams, columns and walls. However in a traditional Beam and Slab model it may not actually be necessary; it is generally not critical unless you have some unusual transfer level challenge. In large models you should therefore consider unchecking it.

Re-design columns (or beams) using previous analysis results

If you change the size of a member you can try out its design without being forced to re-analyse.

Basically you can make any edit you want that does not change the number of stacks in a column, (or spans in a beam); although the analysis results will be marked as out of date, you can still do a design for the changed member based on the old analysis results.

So using the list below as examples design can still be done in all the cases noted:

1. Changing column (or beam) size but retaining shape - designs ok (but see 3 and 4).

2. Changing column (or beam) shape - designs ok (but see 3 and 4).
3. Making column (or beam) smaller so that previously attached members no longer attach - this changes number of stacks - design beyond scope.
4. Making column (or beam) larger so that previously un-attached members now attach - this changes number of stacks (spans) - design beyond scope.
5. Adding / editing / deleting beams that attach to a column - design remains possible up to the point that it affects number of stacks - OK
6. Adding / editing / deleting flat slabs that attach to a column - design remains possible up to the point that it affects number of stacks - OK
7. Adding / deleting stacks or levels (or editing level properties) - Adjusting Levels designs ok because number of stacks is the same. If you add or remove stacks then design is beyond scope.

Model organisation

Tekla Structural Designer has a number of features for organising the model than can each be used to increase efficiency:

- Grouping - one design is applied to all members in the group.
- Sub-structures - allow you to focus on specific areas of interest.
- Duplicate levels - generally save modelling time and reduce the volume of data.

When using duplicate levels, you can achieve further efficiency by designing slabs for a fine mesh at one level only, and then check the slabs at duplicates of the level using a courser mesh result.

NOTE Because meshing parameters are "sub model", rather than "level" based, to achieve this you would set coarse mesh parameters in the structure settings but then override them for an individual sub model.

Our general advice for duplicate levels is:

- For preliminary design set a coarse mesh for entire structure
- For final design where there are a lot of duplicate levels
  - possibly refine the mesh used for the entire structure a little
  - but for each set of duplicate levels, select one and adjust the relevant sub-model parameters to get a finer mesh.

Model complexity

Do not model every little architectural detail - especially not things like small holes in slabs and walls.

Design Options

Take control (get it right at the beginning!)

13 Engineers Handbooks

The **Engineer's Handbooks** provide wider guidance on specific areas of the program, for example, the workflows necessary to achieve specific design objectives.

We recommend you familiarize yourself with the Engineer's Handbook topics from the list below that are relevant to the types of structure you work with:

13.1 Wind modeling handbook

Wind loading can be modeled in Tekla Structural Designer in a variety of ways:

- The Wind Model method (page 1041) requires you to first 'clothe' the structure in wind and roof panels and then run the **Wind Wizard** to create wind zone loads that are subsequently decomposed to the structure during analysis.
- By manually applying wind loads you avoid the requirement to construct a wind model. For this approach, loads can either be applied directly to the structure as Panel, Member, or Structure loads; or they can be applied as

simple wind (page 1119) loads, which are subsequently decomposed to the structure during analysis.

- For tall buildings you may require the assistance of wind specialists to perform wind tunnel testing (page 1130). The information they supply can then be imported in the form of diaphragm loads.

NOTE The Wind Model method is not currently available for the AS:1170.2 wind code variant.

Click the links below to find out more:

- Use of a wind model to create wind loads (page 1041)
- Simple wind and manually applied wind loads (page 1119)
- Wind tunnel testing and diaphragm loads (page 1130)

Use of a wind model to create wind loads

The **Wind Model** method requires you to first 'clothe' the structure in wind and roof panels and then run the **Wind Wizard...**. The wizard creates wind zone loads that are subsequently decomposed to the structure during analysis.

- Overview of the wind model method (page 1041)
- ASCE 7 Wind wizard (page 1044)
- EC1991 1-4 Wind wizard (page 1059)
- BS6399-2 Wind wizard (page 1083)
- IS 875 (Part 3) Wind Wizard (page 1100)
- Wind model loadcases (page 1104)

NOTE The Wind Model method is not currently available for the AS:1170.2 wind code variant.

Overview of the wind model method

This guide provides an outline of the basic steps required to use the wind model method.

The basic steps required to undertake the wind model method are as follows:

Clothe the structure in wind wall and roof panels

The Wind Model calculations depend on the geometry and inter-connectivity of the wall panels and roof panels that envelope the building. You must therefore define the model together with its wall and roof panels before you run the **Wind Wizard...**.

NOTE You can, should you wish, use Tekla Structural Designer purely for wind assessment - by setting up a model consisting only of wall panels and roof panels (no members). Tekla Structural Designercan then determine the wind loading on the building envelope.

In order to get the best results you should ensure that you define the largest possible sizes for the wind wall and roof panels. You may compromise the results if you define many small panels rather than one large one. (The calculation of the reference height in particular can be unconservative.)

Applying Wall Panels

A single wall panel is determined to be a single planar surface. The outward face is vitally important for determining the wind direction relative to the wall, that is windward or leeward.

It is recommended that you check the outward faces are as you intend by ensuring they are all shaded in the same colour (the one assigned to 'Wind Wall - Front' in Settings > Scene). The inward faces will all be shaded in a different colour. To correct any mistakes, choose the **Edit** --> **Reverse** command and then click once on a wall panel to switch its direction. Note that connected wall panels are checked to ensure that the normal directions are consistent whenever automatic zoning is carried out, for example at the end of the **Wind Wizard...**. If there is a problem it is indicated on the **Project Workspace** --> **Wind** tab, with affected panels being marked.

Once a wall panel has been placed the following additional panel properties can be specified:

- **Rotation angle** - defines the span direction, 0° is horizontal and 90° is vertical.
- **Is a parapet wall** - you can indicate whether the wall panel is a parapet or not.

  NOTE If a building face comprises a parapet above a wall, you should not attempt to model this as a single wall panel. It should be input as an upper and lower panel, with the upper panel being set as a parapet.

- **Gap** - (Head Codes: EC and BS only) where the gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funnelling. A zero gap value explicitly means ignore funnelling, for example where this building and the adjacent one are sheltered by upwind buildings
- **Solidity** - (Head Codes: EC and BS only) If you set the wall panel as a parapet, then you also need to indicate the Solidity of the parapet. (Wall panels that are not parapets automatically adopt a solidarity of 1.0).

- **Decompose to** - for wall panels that are not parapets, you can indicate how the wall load is decomposed on to supporting members. See Wind model load decomposition (page 1109)

To set this information as you require, select the wall panels and then use the Properties Window to make changes.

Applying Roof Panels

A single roof panel is determined to be a single planar surface. The orientation of a roof panel is automatically determined when placed based upon the slope vector - the line of maximum roof slope.

Initially the roof type is set to 'Default'. This is interpreted as Flat if the roof slope < 5 degrees, otherwise it is interpreted as Monopitch. You should select the roof panel and then use the Properties Window to adjust the roof type as necessary for all other situations (i.e. For Duopitch, Hip Main, Hip Gable or Mansard).

The span direction is also set in the Properties Window, this is defined as an angle, where 0° is parallel to the X axis and 90° is parallel to the Y axis.

Perform the gravity design

We recommend that you perform an analysis and design at this stage for the gravity loading only, but this is not essential.

Run the wind wizard

Once the model has been 'clothed' in wall panels and roof panels, the Wind Wizard (located on the Load toolbar) guides you through the process of intelligently 'applying' wind to the resulting building envelope.

The wizard uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location.

Having defined the wind directions in which you are interested, on completion of the wizard the appropriate wind zones on the roofs and walls of your structure are automatically calculated.

Related topics

EC1991 1-4 Wind wizard (page 1059)

ASCE 7 Wind wizard (page 1044)

BS6399-2 Wind wizard (page 1083)

IS 875 (Part 3) Wind Wizard (page 1100)

Review the wind zones

The resulting wind model is accessed from the Project Workspace Wind tab. Wind Views can also be opened as required for each wind direction.

From here, you can set the type of each roof to achieve the correct zoning, and can then tailor the zoning to account for particular features in more detail, if you so require.

Define the wind loadcases

The Wind Loadcases dialog (located on the Load toolbar) can then be used to automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases the appropriate wind pressures are calculated on each zone.

NOTE It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

Related topics

Wind model loadcases (page 1104)

Review wind zone loads

Wind zones can be graphically displayed for each wind direction from the appropriate Wind View. Once the wind loadcases have been created you can also display the wind pressures and zone loads for each loadcase.

Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the Design toolbar.

ASCE 7 Wind wizard

This topic will discuss in detail the Wind wizard when using ASCE7

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to ASCE7.

Once the wall and roof panels are in place, you use the Wind Wizard on the Load toolbar to define sufficient site information to calculate the velocity pressures for the required wind directions and heights around the building.

NOTE Unless explicitly noted otherwise, all clauses, figures and tables referred to in the topics in this section are from ASCE 7-10. References (page 1118) 1.

Topics in this section

Scope (ASCE7 Wind Wizard)

The scope of ASCE7-10 Wind Wizard encompasses:

- Choice of method:
    - ASCE/SEI 7-10 - Directional Procedure Part 1 - Rigid Buildings of All Heights
    - ASCE/SEI 7-10 - Envelope Procedure Part 1 - Low-Rise Buildings
- The input of appropriate basic wind data is your responsibility.
- Having defined wall panels and roof panels (defaults are standard wall, flat or pitched roof depending on the slope), you are able to specify the type in more detail e.g. monoslope / mansard etc.
- Wherever possible the wind parameters are determined for you but conservatively, you are able to override the values should you wish to.
- Given the above, zoning is semi-automatic, with full graphical feedback.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).
- There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind load cases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Limitations (ASCE7 Wind Wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind Wizard extensive reference has been made to the References (page 1118) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

The shape of the building must lie within the shapes that are valid according to ASCE7-10 Clauses 27.1.2 and 28.1.2:

- It should be a regular shaped building or structure
- It should not have response characteristics making it subject to across-wind loading, vortex shedding, instability due to galloping or flutter; or have a site location for which channelling effects or buffeting in the wake of upwind obstructions warrant special consideration.

Although the software will generate wind loads for many situations - it is up to you to accept that the loads generated are suitable according to ASCE7-10.

Other documented limitations include:

- Buildings must be enclosed or partially enclosed.
- Open sided buildings are not considered.
- Only rigid buildings are considered, not flexible buildings.
- Zones and load cases are not generated for components and cladding.
- You will need to establish and enter the wind data yourself.
- Roof types must be set manually.
- Barrel-vault and domed roofs are not considered.
- Parapets and free-standing canopies are not considered.
- Roof Overhangs are not explicitly handled.
- There is no special handling for Multi-Bay roofs as they are not covered explicitly in ASCE 7-10.
- There is no special handling for troughed roofs as they are not covered explicitly in ASCE 7-10.

Loaded area

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.
- Orthogonal wind directions at the definition of the user.

Design wind load

The design wind load is not explicitly checked to ensure it is greater than the minimum of 16 psf - (as per Clause 27.4.7 for the Directional method, or 28.4.4 for the Envelope procedure.)

However, the average wind pressure for each of Windward, Leeward and Side directions is provided for you to manually check this is satisfied.

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind load cases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Choice of Method (ASCE7 Wind Wizard)

You can choose between:

- Directional Procedure Part 1 - Rigid Buildings of All Heights (Chapter 27)
- Envelope Procedure Part 1 - Low-Rise Buildings (Chapter 28)

Envelope Procedure Part 1 - Low-Rise Buildings (Chapter 28)

The Low-Rise Building method is explicitly limited to buildings where the mean roof height does not exceed the least horizontal dimension and is less than or equal to 60 ft, (ASCE7-10 Clause 26.2). It is implied that the method should only be used for simple rectangular box-shaped buildings, however, you are given the final responsibility for determining the applicability of this method to your model.

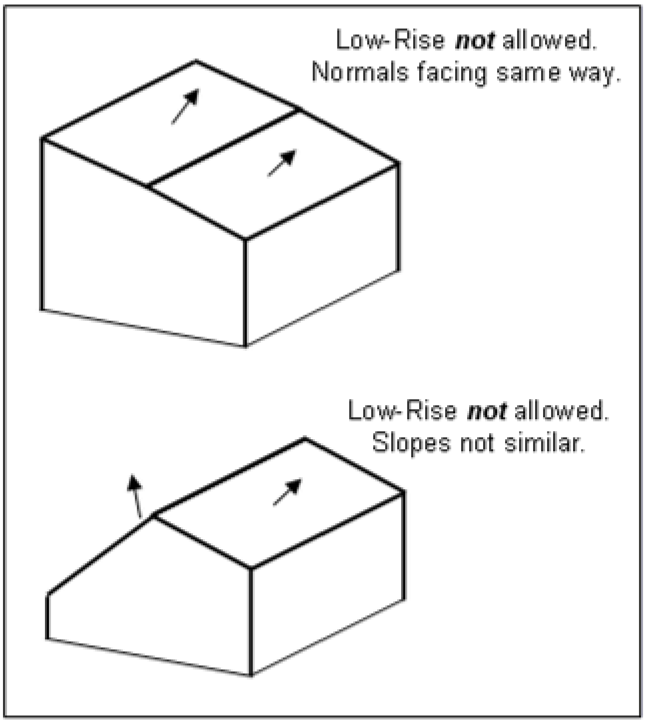
Failure to comply with any of the following conditions will explicitly prevent access to this method:

- There must be 4 walls which must be connected sequentially and form a "simple" quadrilateral in plan form, (all internal corners between 82.5° and 97.5°).
- Each wall must be almost vertical (>80°)

- Each wall must either have 4 sides, forming a "simple" quadrilateral in elevation, or 5 sides forming a convex shape, (allowing for gable ends of buildings).
- The roof system must be one of the following:-
  - Single quadrilateral roof with type "Flat"

    NOTE Where a building has a single roof with a low slope, (e.g. 2°), the default will be "Pitched", but if you consider the building suitable for the Low-Rise method, you can force the roof type to "Flat" rather than "Monoslope".

  - Two quadrilateral "Pitched" roofs which must have a single edge in common. The roofs must face in opposite directions and have similar slopes, (less than 3° difference).
  - Two quadrilateral "Hip Main" roofs which must have a single edge in common and either 1 or 2 hip gable roofs which must be triangular. The "Hip Main" roofs must face in opposite directions and have similar slopes, (less than 3° difference).



  - In particular, the Low-Rise Method is not allowed with Monoslope or multi-span roofs.

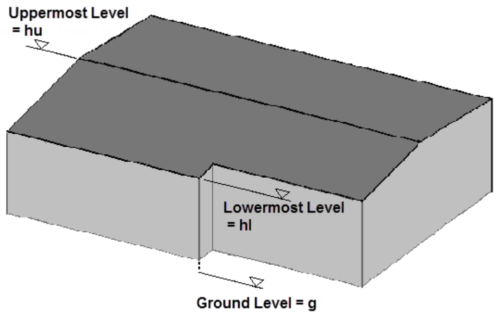
Next

Depending on whether you choose the Low-Rise or the Rigid Buildings method, clicking **Next** takes you to either the Low rise building - Geometry page, or the Rigid building of all heights - Geometry page.

Low Rise Buildings - Geometry (ASCE7 Wind Wizard)

The following geometry items are required for the Low Rise Buildings method. You can override calculated dimensions using your engineering judgement.

Property/Buttons	Description
Property	
Ground level	If for some reason, the level 0.0 feet in the Tekla Structural Designer model does not correspond to the ground level, e.g. you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. The default is zero. The allowed maximum is the minimum wall or roof height. Changing the value in this field will cause the Mean Roof Height to be recalculated, unless you have chosen to override that dimension.
Orientation of Longitudinal Direction relative to axes (Figure 28.4-1)	This information is required to control the orientation of the Transverse and Longitudinal directions, and thus the Tekla Structural Designer Wind Directions. For simple roof shapes, *ASCE7- 10 Wind Wizard* will calculate a default value as below. • Single flat roof - orient along longest side, • 2 Duopitch roofs - orient along common edge, • 2 Hip Main roofs - orient along common edge. • Changing the value in this field will cause either or both of the Longitudinal and Transverse Dimensions to be recalculated, unless you have chosen to override them.
Roof Angle	The field is always visible, even if there are no roof panels in your model. It is calculated as follows: • No roof panels - θ = 0°.

Property/Buttons	Description
	• Else - use maximum angle for all roof panels in the model. You are able to override the calculated value by checking the box; The limits are 0° to 80°.
Mean Roof Height, h (Clause 26.2)	The field is calculated as follows: • No roof panels -h is maximum reference height for walls, • Else if θ <=10° - use maximum eaves height, (h e), for all roof panels in the model, (see Figure 28.4-1) • Else - use maximum Mean Roof height, (h), for all roof panels in the model. In the example below, h = (h_u + h_l)/2 - g  h is limited to a maximum of 60 ft or the Least Horizontal Dimension, whichever is lower. You are able to override the calculated value by checking the box.
Longitudinal and Transverse Dimensions (Clause 26.3 and Figure 28.4-1)	These are similar to the Overall building X dimension and overall building Y dimension (Clause 26.3) for the Rigid Buildings of All Heights Method. These dimensions are calculated from the smallest enclosing rectangle (considered over all roof and walls only), relative to the given orientation of the Longitudinal Direction. You are

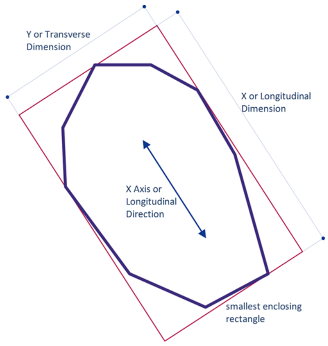
Property/Buttons	Description
	able to override each calculated value by checking its box. These values will then be used to derive the L and B dimensions for each wind direction. For Longitudinal Wind Directions, L = Longitudinal Dimension, B = Transverse Dimension. For Transverse directions, L = Transverse Dimension, B = Longitudinal Dimension.
Buttons	
Next	Clicking **Next** takes you to the Basic wind data page.

Rigid Buildings of All Heights - Geometry (ASCE7 Wind Wizard)

The following geometry items are required for this method. You can override calculated dimensions using your engineering judgement.

Property/Buttons	Description
Property	
Ground Level in Model (Ignore Wind Below)	If for some reason, the level 0.0 feet in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. The default is zero. The allowed maximum is the minimum wall or roof height. Changing the value in this field will cause the Mean Roof Height to be recalculated, unless you have chosen to override that dimension.
Orientation of Principal Axes	This is similar to the Orientation of Longitudinal Direction relative to axes (Figure 28.4-1) (page 1049) for Low-Rise Buildings. Although there is no need to distinguish between the Longitudinal and Transverse axes for this method, the wind X axis will be aligned to this direction with the Y axis at right angles. The vortex view

Property/Buttons	Description
	will be slightly different to the Low-Rise one to reflect that. Due to the potential complexity of the model, the **Wind Wizard...** will only attempt to determine the correct angle for the axes if the building is also suitable for Low Rise, otherwise zero is used as the default.
Mean Roof Height, h (Clause 26.2)	For this method, the actual reference height is only be used for the pressure on windward walls. For all other walls, and roofs, a single height is used to determine the pressure for each direction. By default, the height is calculated as for Low-Rise Buildings except that there is no single value of θ, so the eaves height is only used if all the roof angles are <= 10°. You are able to override the calculated value by checking the box.
Level of Highest Opening in Building, z_i (Clause 27.4.1)	For this method, q_i is evaluated at height h for all cases except for positive internal pressure in partially enclosed buildings, where it should be evaluated at the level of the highest opening. However, the clause allows h to be used even for this case, so the default is for the box to be checked and the level to be automatically updated as the Mean Roof Height changes.
Overall Building X Dimension and Overall Building Y Dimension (Clause 26.3)	These dimensions are calculated from the smallest enclosing rectangle (considered over all roof and walls only), relative to the given orientation of the Principal Axes - see figure below. You are able to override each calculated value by checking its box. These values will then be used to derive the L and B dimensions for each wind direction. For X Axis, L = X Dimension, B = Y Dimension.

Property/Buttons	Description
	For Y Axis, L = Y Dimension, B = X Dimension: 
Design Pressure Factor (Figure 27.4.8 and Clause 27.4.6)	This defaults to 75%, but the commentary suggests that this may not cover all cases so you are allowed to change it. A single factor is used for all Torsional loadcases.
Eccentricity (Figure 27.4.8 and Clause 27.4.6)	This defaults to 15% but again, the commentary suggests that this may not cover all cases so you are allowed to change it. A single factor is used for all Torsional loadcases.
Buttons	
Next	Clicking **Next** takes you to the Basic wind data page.

Basic Wind Data (ASCE7 Wind Wizard)

Once the geometry has been confirmed, (for either method), you are then required to enter the basic wind data. The only difference between the two methods is that the Gust Effect Factor field is only shown for the All Heights Method.

Property/ Buttons	Description
Property	
Basic Wind Speed (Clause 26.5.1)	A strictly positive value is required with 90mph being the default.
Hurricane-Prone Region (Clause 26.2)	The default is cleared, i.e. region not prone to hurricanes.

Property/ Buttons	Description
Directionality Factor, Kd (Clause 26.6, and Table 26.6-1)	The default is 0.85, range 0.85 to 1.0 inclusive. **NOTE** There is no cross-checking to ensure that the model has met the load combination criteria. If that does not apply, then it is your own responsibility to enter the correct value of 1.0.
Enclosure Classification (Clause 26.10)	Options are "Enclosed" and "Partially Enclosed" with default being "Enclosed". **NOTE** "Open" structures are not handled in the current version of the program
Gust Effect Factor (Clause 26.9.1)	This will only be shown for the All Heights Method. The default value is 0.85.
Principal Axes	There are always four directions shown for principal axes, being based on the Orientation of the Longitudinal Direction or Principal Axes depending on the method. You are not able to add, delete or modify any direction.
Exposure Category (Clause 26.7.3)	Options are "B", "C" or "D" with default being "B".
Topographic Feature (Clause 26.8 and Figure 26.8-1)	Options are as follows, with the default being "None": • "None" - no feature, i.e. K_{zt} = 1.0. • "2D Ridge" • "2D Escarp" - 2D Escarpment • "3D Hill" - 3D Axisymmetrical Hill
Crest Height, H (Figure 26.8-1)	Height of the hill or escarpment relative to the upwind terrain. The behaviour of this field depends on the Feature type as follows: • "2D Ridge" - non-zero values allowed, (negative indicates a valley). • "2D Escarp" - strictly positive values allowed • "3D Hill" - strictly positive values allowed
Crest Length, Lh (Figure 26.8-1)	Distance upwind of crest to where the difference in ground elevation is half the height of the hill or escarpment.

Property/ Buttons	Description
Distance to Crest, x (Figure 26.8-1)	Distance upwind or downwind from the crest to the building site. The value may be positive to indicate downwind, or negative to indicate upwind.
Buttons	
Next	Clicking **Next** takes you to the Results page.

Results (ASCE7 Wind Wizard)

The final page of the Wind Wizard is a summary of the velocity pressure results for the principal axes. You are able to use the **Details...** button to obtain additional information, including the values of intermediate factors used in the calculations.

Finishing the Wind Wizard

When you click **Finish**, the **Wind Wizard...** generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 1045) have been encountered.

Related topics

Wind Zones - ASCE7 Low Rise Building Method (page 1055)

Wind Zones - ASCE7 All Heights Method (page 1056)

Wind Zones - ASCE7 Low Rise Building Method

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

Wind Directions

Eight zone directions evenly spaced at 45 degree intervals about the longitudinal direction are defined as follows:

- Long (B) 1 - (longitudinal direction + 22.5°, uses data for +X' principal axis)
- Trans (A) 1 - (longitudinal direction + 67.5°, uses data for +Y' principal axis)
- Trans (A) 1 - (longitudinal direction + 112.5°, uses data for +Y' principal axis)
- Long (B) 4 - (longitudinal direction + 157.5°, uses data for -X' principal axis)
- Long (B) 3 - (longitudinal direction + 202.5°, uses data for -X' principal axis)

- Trans (A) 3 - (longitudinal direction + 247.5°, uses data for -Y' principal axis)
- Trans (A) 2 - (longitudinal direction + 292.5°, uses data for -Y' principal axis)
- Long (B) 2 - (longitudinal direction + 337.5°, uses data for +X' principal axis)

These enable the modeling of two sets of zones per principal axis with the correct reference corner in each case.

Wall Zones

The Wind Wizard automatically generates wall zones, where possible, for each *Direction of MWFRS Being Designed* and *Reference Corner* in accordance with Figure 28.4-1.

Roof Zones

The **Wind Wizard...** automatically generates roof zones, where possible, for each *Direction of MWFRS Being Designed* and *Reference Corner* in accordance with Figure 28.4-1.

For flat roofs and for MWFRS parallel to the ridge line we assume Note 8 is not applicable.

Where Note 7 applies, we assume the dimension to the zone 2/3 boundary is measured horizontally.

Automatic Zoning

Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°.

Wind Zones - ASCE7 All Heights Method

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

Wind Directions

Eight zone directions evenly spaced at 45 degree intervals starting from the longitudinal direction are defined as follows:

- +X - (longitudinal direction, uses qz for 1st principal axis)
- +X+Y - (longitudinal direction + 45°, uses q_z for 1st and 2nd principal axes)
- +Y - (longitudinal direction + 90°, uses q_z for 2nd principal axis)
- -X+Y - (longitudinal direction + 135°, uses q_z for 2nd and 3rd principal axes)
- -X - (longitudinal direction + 180°, uses q_z for 3rd principal axis)
- -X-Y - (longitudinal direction + 225°, uses q_z for 3rd and 4th principal axes)

- -Y - (longitudinal direction + 270°, uses q_z for 4th principal axis)
- +X-Y - (longitudinal direction + 315°, uses q_z for 1st principal axis)

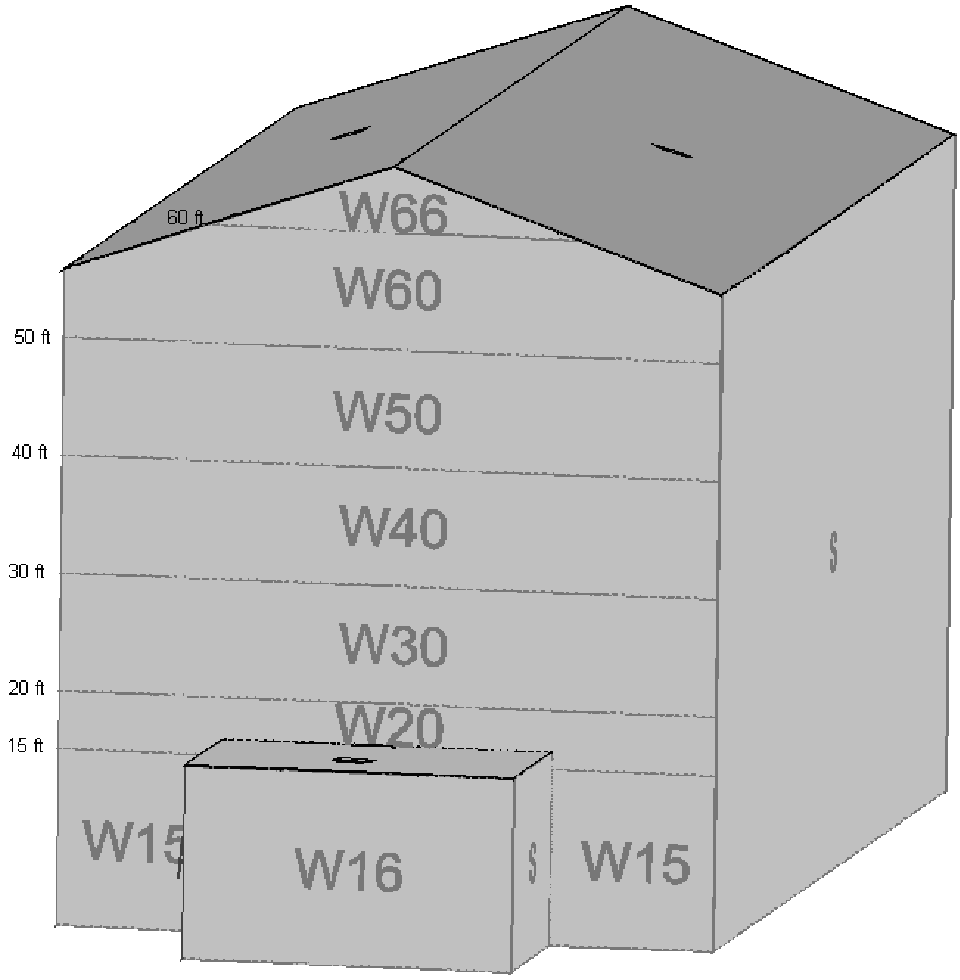
The 4 directions +X,+Y,-X,-Y enable the modeling of design wind load cases 1 and 2 from Figure 27.4-8. The 4 intermediate directions +X+Y etc. enable the modeling of load cases 3 and 4.

Wall Zones

The **Wind Wizard...** automatically generates wall zones, where possible, in accordance with Figure 27.4-1.

Windward Walls are split horizontally over intermediate heights. Zones are labelled with a 'W' followed by the height at top of zone rounded to nearest foot (or meter if using metric units). Each zone uses the same C_p, (0.8 from Wall C_p table in Figure 27.4-1), but a different velocity pressure, q_z, z being determined at the top of the zone.

Complex wall shapes are dealt with by splitting zones intelligently as illustrated below:



Leeward walls have a single zone, 'L', using a Wall C_p from the table in Figure 27.4-1, with interpolation for L/B. The velocity pressure, q_h is used for all such zones.

Side walls have a single zone, 'S', using the same C_p, (-0.7 from the Wall C_p table in Figure 27.4-1). A single velocity pressure, q_h is used for all such zones.

Skew Walls

There is no guidance in ASCE7-10 for walls not orthogonal to the principal axes. However, the **Wind Wizard...** will apply zones in those cases and it is your responsibility to check the wind loads adopted.

Roof Zones

The **Wind Wizard...** automatically generates Windward and Leeward roof zones ('W' and 'L' respectively, where possible, in accordance with Figure 27.4-1. This is possible for Windward and Leeward roof panels with θ >= 10°, because there is only one zone. However, for θ < 10° and Side roof panels, automatic zoning will not be carried out for all cases.

Negative and positive values of C_p are determined for each zone from the Roof C_p table in Figure 27.4-1.

For non-principal axis directions, it is assumed that roof loads are not required and so special zones are created, named "Zero". Such zones will result in zero loading on the relevant roof panels. However, if the you choose to do so you can change these to non-standard zones and enter coefficients to force loads to be applied.

When calculating the area reduction factor we use the slope area not the plan area.

Mansard roofs are not automatically detected, i.e. it is your responsibility to set the Roof Type manually. Generally, Mansard roofs are handled exactly the same as if they are:

- Flat (for slope < 0.1°)
- Pitched (for slope >= 0.1°)

There is no guidance on what to do with other multipitch roofs, so they are treated as pitched roofs, whether they have been flagged as Mansard or not.

Side roofs are split into to 4 zones depending on the size of the roof. See the Roof C_p table in Figure 27.4-1.

- Zone 1 - 0 to h/2, interpolating between 2 values for h/L if necessary.
- **NOTE** Upper value for interpolation can be reduced linearly with the sloped area of this zone, (see ** in Figure 27.4-1).
- Zone 2 - h/2 to h, interpolate between -0.9 and -0.7 for h/L if necessary
- Zone 3 - h to 2h, interpolate between -0.5 and -0.7 for h/L if necessary
- Zone 4 - > 2h, interpolate between -0.3 and -0.7 for h/L if necessary

Automatic Zoning

Automatic zoning will apply to all Windward and Leeward roof panels with θ > 10°, because there is only one zone. However, for θ < 10° and Side roof panels, automatic zoning will not be carried out for all cases.

EC1991 1-4 Wind wizard

This topic will discuss in detail the Wind wizard when using the Eurocode BS EN1991-1-4

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS EN 1991-1-4.

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the peak wind velocity and velocity pressures for the required wind directions and heights around the building, (that is the Reference Heights (z_e and z_i) for each wall panel or roof panel).

The wind velocity calculations are automated, with the data source for the calculations being either:

- Input directly for the worst case,
- Input directly for each direction,
- taken directly from the BREVe database which is based upon the Ordnance Survey data of Great Britain (only available for users working to the UK or Ireland National Annex).

Topics in this section

Design Codes and References

Unless explicitly stated all calculations in the EC1991 1-4 Wind Wizard are in accordance with the relevant sections of EC EN1991 1-4 (Ref. 3) (page 1118) and the chosen National Annex. It is essential that you have a copy of the code and National Annex with you while assessing wind on any structure.

We would recommend having the following books to hand when using the software:

- Designers' Guide to EN 1991-1-4. Euro Code 1 : Actions on Structures, General Actions Part 1-4 : Wind actions. (Ref. 6) (page 1118)
- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. (Ref. 7) (page 1118)

In addition, you may find the following book useful:

- Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009. (Ref. 5) (page 1118)
- Unless explicitly noted otherwise, all clauses, figures and tables referred to in this section of the handbook are from EC EN1991 1-4. (Ref. 3) (page 1118)

Scope (Eurocode EC1991-1-4 Wind wizard)

There is no guidance in the standard for anything other than a cuboid building. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings. It is therefore the user's responsibility to ensure that the wind loading generated by the software meets the needs of any building with a shape that is beyond the scope of BS EN 1991-1-4:2005.

The scope of EN1991 1-4 Wind wizard encompasses:

- Enveloping the building with wall panels and roof panels is undertaken in Tekla Structural Designer in the normal manner. There is only limited validation of the envelope defined (for example connected wall panels must have consistent normal directions). The onus is on you to model the building shape as completely and as accurately as you determine necessary.
- Basic Wind Velocity and Peak Velocity Pressure is determined.
- Having defined wall panels and roof panels (defaults are standard wall, flat or monopitch roof depending on the slope), you are able to specify the type in more detail e.g. multi-bay, monopitch / duopitch etc.).
- The main wind parameters, are calculated for you but conservatively, (for example Crosswind Breadth, b, is determined for the enclosing rectangle of the whole building). Wherever possible other parameters are determined conservatively, but you are able to override the values should you need to.
- Given the above, zoning is semi-automatic, (not attempted for roofs with more than 4 sides which are defaulted to single conservative coefficient), with full graphical feedback.
- The software follows the UK NA (Ref. 3) (page 1118) recommendation that BS6399 roof zones and coefficients are used, including Mansard, Multipitch and Multibay roofs.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).

Limitations (EC1991-1-4 Wind wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind wizard extensive reference has been made to the References (page 1118) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

WARNING	You should seek specialist advice for building shapes that are not covered by the Standard - BS EN 1991-1-4:2005.

- EC1 1-4 does not treat downwind re-entrant corners as special cases - see BS6399 Clause 2.4.3.1 c). So, they are ignored in the software and no warnings are given.
- EC1 1-4 does not handle stepped profiles, or inset storeys - see BS6399 clauses 2.4.4.2 and 2.5.1.7. Hence the software does not handle them automatically, but does generate warnings if such cases are detected - so you can manually edit the zones according to your engineering judgement.
- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered
- Dominant Faces are not explicitly handled - Clause 7.2.9 (5). However, you can use Table 17 to calculate the necessary Cpi value or values and manually apply to a loadcase or individual zone loads.

Loaded areas

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.
- Orthogonal wind directions at the definition of the user.

Overall loads

- Lack of correlation of pressures between the windward and leeward sides. For Overall loadcases, the software automatically reduces the windward and leeward wall pressures only. EC1 1-4 and the UK NA both suggest that the reduction "may" be applied to roofs as well.
- Division by Parts rule for "slender" buildings -Clause 7.2.2 and Figure 7.4 - not applied.

- Friction Forces - Clause 5.3 (3), equation 5.7 and Clause 7.5.

  During the "Update Zones" process, checks are performed to see if the effects can be disregarded, (Clause 5.3 (4)), and a 'Friction needed' warning is generated if not. When they cannot be disregarded you will need to manually model the friction forces as lateral loads in a separate loadcase and include them in your combinations.

Beneficial loads

- Asymmetric and Counteracting Pressures and Forces - Clause 7.1.2 and NA.2.23. Beneficial loads are not automatically removed - instead you are able to flag individual loads to be reduced to zero.

Singapore National Annex - Minimum horizontal loads

- The Foreword to the Singapore National Annex to EN 1991-1-4 Wind Actions has a minimum horizontal load requirement (1.5% characteristic dead weight). Therefore if this National Annex has been applied, it is the users responsibility to check that this requirement has been met (by ensuring that the horizontal component of the factored wind load is greater).

Finland National Annex

- We do not consider thermal inversions for buildings > 100m tall

Norway National Annex

- We do not consider the transition zones between changes in terrain category.

Wind loading on wall panels

Automatic zoning applies to all wall panels subject to the limitations described below:

- Vertical Walls on rectangular buildings -Clause 7.2.2 - the assumption for wall wind forces is that the building is rectangular or close to being rectangular.
- Wall panels that are more than 15° from the vertical are outside the scope.
- Internal Wells are not covered by EC1 1-4 and in any case are not automatically identified but you can manually edit the zones to apply the roof coefficient or otherwise as you see fit - see BS6399 Clause 2.4.3.2a.
- EC1 1-4 does not specify how to treat recesses in side walls - see BS6399 clauses 2.4.3.2 b) and 2.4.3.3 and 3.3.1.5. So, they are ignored but warnings are given.

Wind loading on roof panels

- Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, i.e. all of the internal angles < 180°
- Special care should be taken for winds blowing on duopitch with slopes that differ by more than 5°. If the wind is blowing on the steeper slope (that is that the less steep slope is downwind of ridge), the downwind slope should be set to be a flat roof with mansard at eaves for this wind direction.
- Mansard and Multipitch Roofs are not detected automatically, although certain special cases can be handled if you set the appropriate type manually - see EC1991 1-4 Wind Zones.
- BS 6399 Table 8 curved and mansard eaves - zones start from edge of horizontal roof.
- Roof Overhangs are not explicitly handled. It is suggested that you should define two separate roof panels - one forming the overhang and the other covering the inside of the building. For a small overhang, you can then manually define Cpi values to be the same as Cpe for the adjacent wall panel, (Clause 7.2.1 (3)). Reference 6 (page 1118), p45, implies that larger overhangs can be manually handled by using BS6399, Clauses 2.5.9.3 and 2.6.3, i.e. standard external coefficients for the top surface and Table 18 for the internal coefficients.

NOTE The only slight issue here is that there are two sets of edge zones which will occupy a slightly larger area than strictly necessary.

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind load cases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Using the EC1991-1-4 Wind wizard

This section runs through each page of the wizard and discusses the various options.

Data Source page

You can choose to enter one set of Worst-Case data or different values for each direction to be considered.

TIP Additional options are provided for using BREVe data when working to the UK or Ireland National Annex. For further details see: Using the EC1991 1-4 Wind Wizard with BREVe data (page 1070).

The remaining choices on the Data Source page are:

Property	Description
Consider Orography	If you select this check box, then the orographic data, (manually entered), is used to determine the Orography Factor c_o as clause A.3. When calculating c_{alt}, the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, c_o is 1.0 for all heights and c_{alt} is the same for all directions, using the Site Altitude.
Consider Tall Neighbouring Structures	If the conditions in clause A.4 are met, then the wind loads need to be based on height z_n, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then z_n will be used as the reference height for all wall panels and roof panels in the model. **NOTE** If working to the Sweden NA, Tall Neighbouring Structures are not considered.
Consider Obstructions	With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, h_{dis} as (A15) in clause A.5. Otherwise the obstructions are ignored and h_{dis} is taken as zero. **NOTE** If working to the Sweden NA, Obstructions are not considered.
Next	Clicking Next takes you to the **Basic data** page below.

Basic data page

This page is used you to define the site details.

Property	Description
Air density	You need to enter air density at the site.
Ground level	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a

Property	Description
	building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.
Fundamental Basic Wind Velocity - Clause 4.2 and NA.2.4	Reference should be made to the National Annex being worked to when determining an appropriate basic wind speed.
Season Factor, c_{season}	Valid range 0.01 to 10.0 - default 1.0.
Probability Factor, c_{prob}	Valid range 0.01 to 10.0 - default 1.0.
Default Height for Internal Pressure (z_i)	Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, (z_i), defaulting to the height of the structure. Leaving Use Building Height checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.
Peak Factor, k_p (Sweden NA only)	The Peak Factor was introduced by the Swedish EC NA (EK11) update. Using the default value (3.0) is the equivalent of the old calculation method prior to the update.
Region (Norway NA only)	A Region is needed (Area 1, Area 2 or Area 3) representing three different height zones in the country.
Site Altitude (Norway NA only)	You need to enter the basic altitude that you want to use for the site directly. This is the altitude of your model's base.
Next	Depending on whether you chose worst case data, or data for each direction on the Data Source page, clicking **Next** either takes you to the **Roughness and Obstructions (Worst case)** page or the **Roughness and Obstructions (Data for each Direction)** page.

Roughness and Obstructions (Worst case) page

If you select the Worst Case Data Source, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself.

<table>
<tr><th>Property</th><th>Description</th></tr>
<tr><td>Terrain Category</td><td>Reference should be made to the National Annex being worked to when determining an appropriate Terrain Category.

Depending on the terrain category selected and National Annex being worked to, you may also be required to enter some of the following data:
• Average height of upwind buildings,
• Upwind spacing of surrounding buildings,
• Upwind distance from sea to site,
• Upwind distance from edge of town to site.</td></tr>
<tr><td>Next</td><td>Depending on your selections on the Data Source page, clicking Next either takes you to the Orography (Worst Case) page, the Tall Neighbouring Structure page, or the Results page.</td></tr>
</table>

Roughness and Obstructions (Data for each Direction) page

If you select the **Other - Data for each Direction Data Source**, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself. However, most of the data is then dependent on the wind direction, so you must also make your choice of wind directions on this page.

<table>
<tr><th>Property</th><th>Description</th></tr>
<tr><td>Direction</td><td>Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update these using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined.

Each row of the grid operates in a similar manner to the relevant fields of the Roughness and Obstructions (Worst case) page.</td></tr>
<tr><td>Next</td><td>Depending on your selections on the Data Source page, clicking Next either takes you to Orography (Data for each Direction) page, the Tall Neighbouring Structure page, or the Results page.</td></tr>
</table>

Orography (Worst Case) page

If Consider Orography was checked, then the next page of the Wizard for the Worst Case Data Source allows you to enter the data for Orography.

Property	Description
Orographic Feature (Clause A.3)	Options are: • None - no feature, i.e. c_o = 1.0. • 2D Escarp - Cliffs and Escarpments, • 3D Hill - Hills and Ridges.
Altitude of Upwind Base of Feature, A	This value is used to calculate C_{alt} instead of the Site Altitude because the Orography is significant. **NOTE** C_{alt} will be calculated at z_e for each wall and roof panel, not z_s.
Effective Crest Height, H (Figures A.2 & A.3)	Effective height of the feature.
Length of Upwind Slope, L_u (Figures A.2 & A.3)	Actual length of the upwind slope in the wind direction.
Length of Downwind Slope, L_d (Figures A.2 & A.3)	Actual length of the downwind slope in the wind direction.
Horizontal Distance to Crest, x (Figures A.2 & A.3)	Distance upwind or downwind from the crest to the building site.
Orography factor, $c_o(z)$ and Turbulence factor, k_l (Norway NA only)	When working to the Norway NA you are not required to enter the above factors; instead you enter the orography factor and turbulence factor directly for the defined 3D or 2D orographic feature.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall**

Property	Description
	Neighbouring Structure page , otherwise it takes you to the **Results** page.

Orography (Data for each Direction) page

The wind directions defined on the previous page are maintained and you are not able to update them.

Property	Description
	Each row of the grid operates in a similar manner to the relevant fields of the Orography (BREVe) page (page 1075) page.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall Neighbouring Structure** page , otherwise it takes you to the **Results** page.

Tall Neighbouring Structure page

For all methods, if **Consider Tall Neighbouring Structure** was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)

Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, h_{high}, Largest Horizontal Dimension of Tall Neighbour, d_{large} and Distance to Tall Neighbour, x are all as described on Figure A.4 of the code.

Property	Description
Average Height of Neighbours , h_{ave} (Figure A.4)	The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If **Override calculated dimension** is cleared, then the value will be updated whenever the wizard is run, otherwise the user-value is used.
Height of this structure, h_{low} (Figure A.4)	The field is for information only - difference between top of highest wall / roof panel and ground level in the model.
Next	Click Next to go to the **Results** page.

Results page

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but except for the **Other - Data for Each Direction method**, you are able to update this using the **Dir.** buttons and / or changing the direction value as required. (Note : Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the **<<Details>>** button to obtain additional information, including the values of intermediate factors used in the calculations.

Property	Description
Other - Worst Case Data	The calculation of q_p is very similar to the BREVe Method, (see above), except that the worst case data has been entered by you, and this page allows you to enter your own values for C_{dir}. As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each direction.
Other - Data for each Direction	The calculation of q_p is very similar to the BREVe Method, (see above), except that the data has been entered by you for each direction only so a direct calculation can be performed instead of taking the worst case over a range of sectors. Also this page allows you to enter your own values for C_{dir}. As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each direction.
Finishing the Wind Wizard	When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions. **NOTE** Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 1061) have been encountered.

Using the EC1991 1-4 Wind Wizard with BREVe data

This page steps you through the EC1991-1-4 Wind wizard when using BREVe data

NOTE This option is only available when either the UK or Ireland National Annex has been selected.

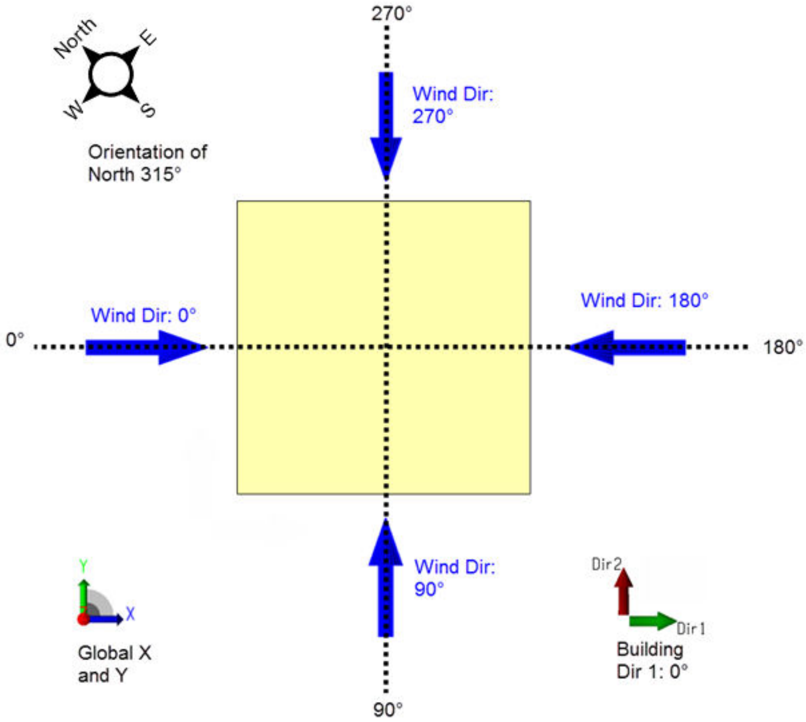
Data Source (BREVe) page

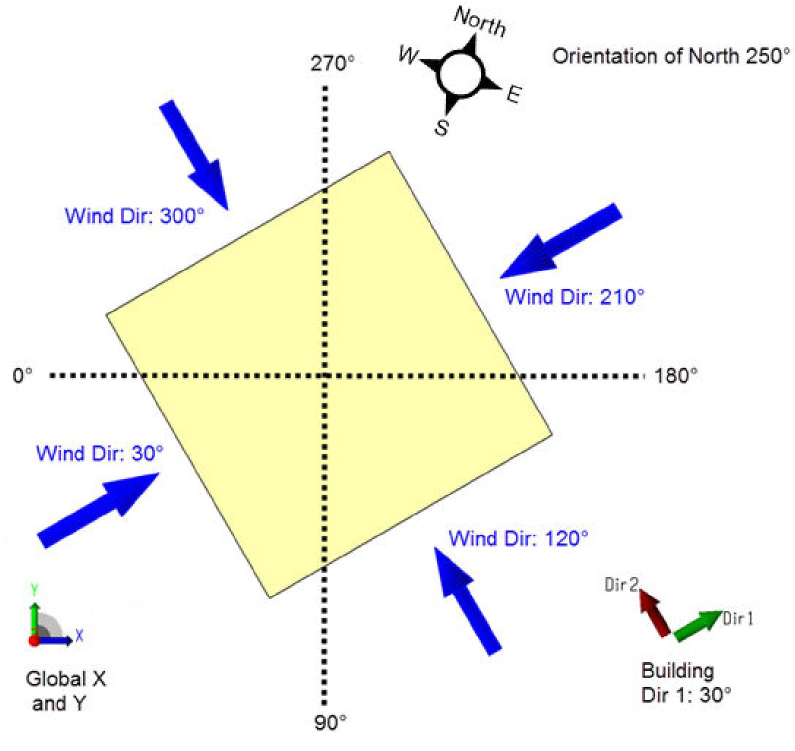
Assuming you choose to specify the site data using BREVe Grid Ref data the remaining choices on the Data Source page are:

Property/ Buttons	Description
Property	
Consider Orography	If you select this check box, then the orographic data, (either recovered by BREVe for the site or manually entered), is used to determine the Orography Factor c_o as clause A.3. When calculating c_{alt}, the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, c_o is 1.0 for all heights and c_{alt} is the same for all directions, using the Site Altitude.
Consider Tall Neighbouring Structures	If the conditions in clause A.4 are met, then the wind loads need to be based on height z_n, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then z_n will be used as the reference height for all wall panels and roof panels in the model.
Consider Obstructions	With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, h_{dis} as (A15) in clause A.5. Otherwise the obstructions are ignored and h_{dis} is taken as zero.
Buttons	
Next	If you have chosen to use BREVe Grid Ref data, clicking **Next** takes you to the **BREVe location** page , otherwise it takes you to the **Basic Data** page.

BREVe location page

Property/ Buttons	Description
Property	
Grid Ref.	This shows the grid reference of the site which you have picked through BREVe, irrespective of the method you use to define the site location.
Orientation of building known	If you know the orientation of the building with respect to North, then you can define this information by checking this box. You can then define a value which relates the building direction axes of your Tekla Structural Designer model to geographic north.

Property/ Buttons	Description
Orientation of North	The orientation of North is defined using the same convention as is applied to the orientation of the Building Direction Arrows. This can best be understood by reference to a couple of examples: In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°. The resulting relation between the building axes and North is as shown below:  In the second example the building direction has been input with Dir 1 = 30° and the orientation of North has been set to 250° In this case the building axes are related to North as shown below:

Property/ Buttons	Description
	
Buttons	
	Using BREVe, there are 2 methods available for you to define the site location:
Site By Ref...	You can define the grid reference of the site. You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid reference from a postcode or given location, for example www.streetmap.co.uk or www.multimap.co.uk
Site By Map...	You can pick the site from a Land / Town Map, • You can pick the site from a Orography Map. • You can pick the site from a ground roughness Category Map, The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.

Property/ Buttons	Description
Next	Click **Next** to move to the **Basic Data (BREVe)** page.

Basic Data (BREVe) page

This page allows you to review the site details taken from the BREVe database.

Property/ Buttons	Description
Property	
Site Altitude, A	The basic site altitude of your model's base.
Air Density	Air density at the site.
Ground Level	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.
Fundamental Basic Wind Velocity ($v_{b,map}$) - Clause 4.2 and NA.2.4	The value required is defined as "the characteristic 10 minutes mean wind velocity, irrespective of wind direction and time of year, at 10m above ground level in open country terrain with low vegetation such as grass and isolated obstacles with separations of at least 20 obstacle heights", but is the value before the altitude correction is applied. (Valid range 1.0 to 1000 m/s).
Season Factor, c_{season}	Valid range 0.01 to 10.0 - default 1.0.
Probability Factor, c_{prob}	Valid range 0.01 to 10.0 - default 1.0.
Default Height for Internal Pressure (z_i)	Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, (z_i), defaulting to the height of the structure. Leaving Use Building Height checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Roughness and Obstructions (BREVe)** page.

Roughness and Obstructions (BREVe) page

This page of the Wizard automatically defaults the data for ground roughness and obstructions for you.

Property/ Buttons	Description
Property	
Terrain Category	Options available are: • **Sea** - this setting is for sites where the distance to sea is between 0 and 1 km, not for offshore sites. As the worst case must be for wind blowing across the sea, there is no need to specify data for upwind buildings or distance in town. • **Country** - the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town, • **Town** - for this category you need to specify data for upwind buildings and distance to the edge of the town, so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for h_{ave} For this category, the Upwind distance from edge of town to site cannot be greater than the Upwind distance from sea to site.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Orography (BREVe)** page.

Orography (BREVe) page

NOTE When using Breve Data you should really leave it up to the system as to whether orography is significant or not. If you chose not to consider orography, the actual factor is not applied; however the site data is still displayed to allow you to check that your decision to ignore it is reasonable.

Property/ Buttons	Description
Property	
Orographic Feature (Clause A.3)	Options available are: • **None** - no feature, i.e. c_o = 1.0. • **2D Escarp** - Cliffs and Escarpments,

Property/ Buttons	Description
	• **3D Hill** - Hills and Ridges.
Altitude of Upwind Base of Feature, A	This value is used to calculate C_{alt} instead of the Site Altitude because the Orography is significant. **NOTE** C_{alt} will be calculated at z_e for each wall and roof panel, not z_s.
Effective Crest Height, H (Figures A.2 & A.3)	Effective height of the feature.
Length of Upwind Slope, L_u (Figures A.2 & A.3)	Actual length of the upwind slope in the wind direction.
Length of Downwind Slope, L_d (Figures A.2 & A.3)	Actual length of the downwind slope in the wind direction.
Horizontal Distance to Crest, x (Figures A.2 & A.3)	Distance upwind or downwind from the crest to the building site.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall Neighbouring Structure**, otherwise it takes you to the **Results (BREVe)** page.

Tall Neighbouring Structure page

For all methods, if Consider Tall Neighbouring Structure was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)

Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, h_{high}, Largest Horizontal Dimension of Tall Neighbour, d_{large} and Distance to Tall Neighbour, x are all as described on Figure A.4 of the code.

Property/ Buttons	Description
Property	
Average Height of Neighbours, h_{ave} (Figure A.4)	The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If Override calculated dimension is cleared, then the value will be updated whenever the wizard is run, otherwise the user-value is used.
Height of this structure, h_{low} (Figure A.4)	The field is for information only - difference between top of highest wall / roof panel and ground level in the model.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to got to the **Results (BREVe)** page.

Results (BREVe) page

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update this using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the **<<Details...>>** button to obtain additional information, including the values of intermediate factors used in the calculations.

BREVe Data

BREVe determines the parameters required to calculate $q_p(z)$ for each height in the building at 30° intervals, (0° to 330°).

For each required wind direction the worst case q_p is used for each height, based on splitting the difference to the next direction, with a maximum of ±45 degrees. Within these ranges q_p is not interpolated.

Theoretically, it is possible for a quadrant to use different 30° directions for each height, so the critical wind direction is not displayed in the summary.

The Vortex view shows the Peak Velocity Pressures calculated for each reference height for each 30° sector.

Finishing the Wind Wizard

When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 1061) have been encountered.

EC1991 1-4 Wind Zones

Enter a short description of your topic here (optional).

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

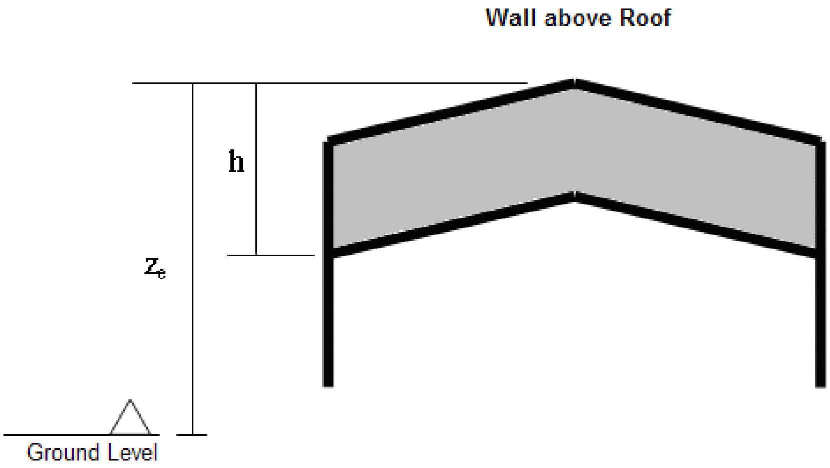
If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

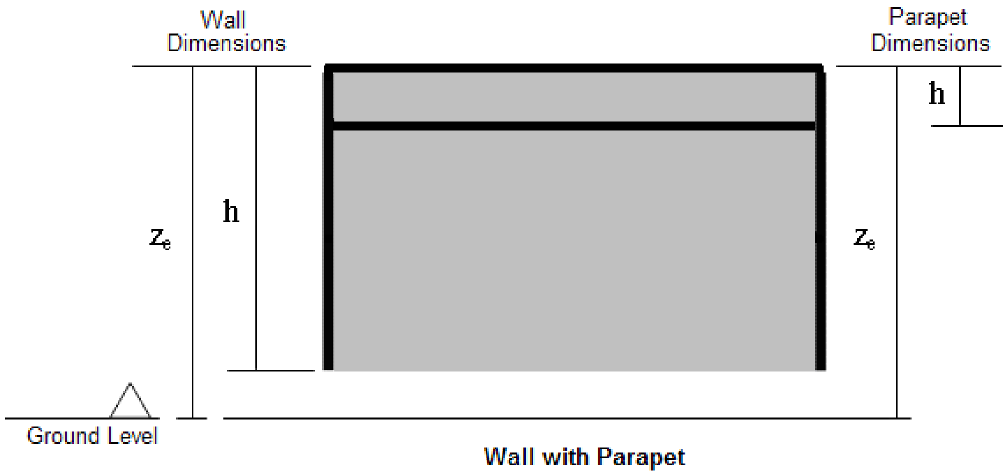
Basic Geometry

The basic building geometry is assessed as follows:

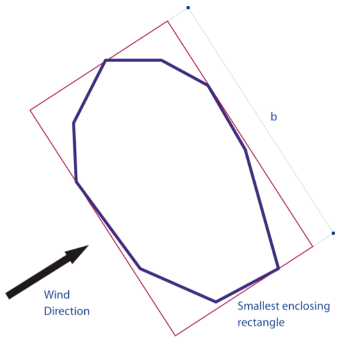
- Reference Height (z_e) - is taken as the difference between highest point on wall or roof panel and ground level.
- Wall height (h) - is taken as the difference between highest and lowest points on the wall panel.

  These definitions apply to wall panels without parapets and the actual parapets. Wall panels with parapets above them will take their highest point from the parapet. See the diagram below.





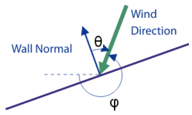
- Roof height (h) - as z_e - taken as the difference between highest point on wall or roof panel and ground level. This definition does not handle the upper roof of inset storey but is conservative.
- The Building Breadth, b is calculated from the smallest enclosing rectangle around the whole building (considered over all roof and wall panels only) for the given direction. You can override the calculated value in case the Tekla Structural Designer model does not include the whole building.



Wall Zones

Wall Type

We assess each wall panel to determine if it is a windward, leeward or side wall. We classify the type of wall dependent on q:



- θ ≤ 60 deg - Windward,
- θ ≥ 120 deg - Leeward,
- Other walls are classed as Side.

Windward walls	Windward walls have a single zone and Table 7.1 is used with interpolation for h/d.
Leeward walls	Leeward walls have a single zone and Table 7.1 is used with interpolation for h/d.
Side walls	In all cases, side walls have the relevant number of zones from Figure 7.5 and Table 7.1 is used. There is no guidance in the standard for Irregular Flushed Faces, Recesses and Downwind Re-entrant Corners that are covered in BS6399. However, it is reasonable to automatically detect Irregular Flushed Faces and process them as described in BS6399 Clause 2.4.4.1 and Figure 14. The program also detects potential Recesses but only generates a warning and no special handling occurs. Downwind re-entrant corners are conservatively ignored.

Parapets	Parapets are assessed for return corners and then classified as windward, windward oblique, leeward or leeward oblique dependent on q. Depending on the classification, parapets will either have a single zone, or up to seven zones (determined by extrapolation from Figure 7.19). Table 7.9 is used with interpolation for Solidity and l/h. **NOTE** An "r" suffix on zone name indicates return corners have been detected. Side Parapets are Special Zero zones, i.e. no nett pressure.

Roof Zones

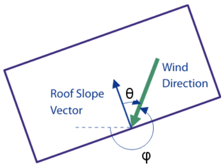
Roof zones are automatically generated where possible for each wind direction. In essence each roof panel is assessed in its own right based on its properties. The interconnectivity of touching roof panels is not generally considered.

NOTE The Advisory note on page 34 of the UK NA is followed so that zones and coefficients are generated according to BS6399.

Direction

Internally the roof slope vector (line of maximum slope) is determined from the normal vector, with its direction always giving a positive slope angle, i.e. the roof slope vector must always point up the slope.

We calculate the angle between the wind direction and projection of roof slope vector onto horizontal plane (q in range -180° to +180°).

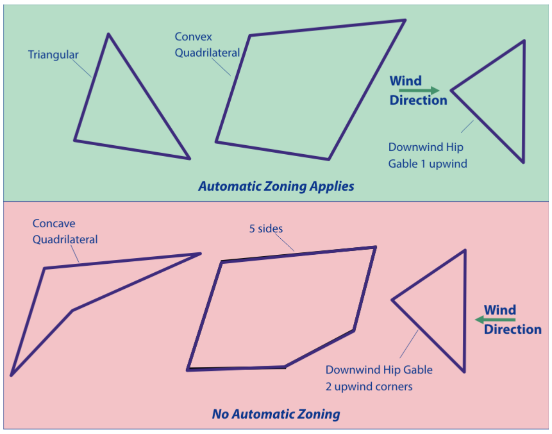


Scaling Dimension, e

The scaling dimension e = min(b, 2h)

Automatic Zoning

Automatic zoning normally only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°. However, additionally, it only applies to Hip Gable roofs if they are triangular, and Hip Main roofs if they are quadrilateral. Further, Downwind Slope Hip Gables must not have 2 upwind corners.



Dimensions	All zone dimensions are specified in plan.
Flat Roofs	See BS 6399 Clause 2.5.1, Figure 16 and Table 8.
Monopitch Roofs	See BS 6399 Clause 2.5.2.3, Figure 19 and Table 9.
Duopitch Roofs	See BS 6399 Clause 2.5.2.4, Figure 20 and Table 10.
Hip Gable	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Hip Main	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Mansard Roofs	If you manually set the connected roof types to Mansard, then the program will correctly identify the special cases in BS6399 Figures 17c, 22a and 22b, and use the correct tables and values. See BS 6399 Clauses 2.5.1.6.2 & 2.5.4
Multi-bay Roofs	We allow you to interpret BS 6399 Clause 2.5.5 and Figure 23 as you think appropriate and manually define the roof types and sub-types accordingly. You also have the ability to manually set the multi-bay position for each roof panel for each wind direction: • Not Multi-Bay - for this wind direction (conservative default), • Upwind Bay - first bay of many for this wind direction, • Second Bay - for this wind direction, • Third or more Bay - for this wind direction. Where the reduction applies, the values of all coefficients are reduced according to Table 12.

Non-Automatic Zoning

Where automatic zoning does not apply, the system creates a single zone covering the entire roof as follows:

- Hip Gable - B for upwind, B for downwind, D for side,
- Flat - B,
- Monopitch - B,
- Duopitch - B for upwind, A for downwind, B for side,
- Hip Gable - B for upwind, B for downwind, D for side,
- Hip Main - B for upwind, A for downwind, D for side.

BS6399-2 Wind wizard

This topic will discuss in detail the Wind wizard when using the British Standard BS 6399-2.

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS 6399-2.

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the effective wind speeds and dynamic pressures for the required wind directions and heights around the building, (that is the Reference Height (H_r) for each wall panel or roof panel).

The wind speed calculations are automated, the data source for the calculations is either:

- taken directly from the BREVe database, which is based upon the Ordnance Survey data of Great Britain,
- input directly.

It should be noted that BS6399-2:1997 recommends that the Standard Method requires assessment of orthogonal load cases for wind directions normal to the faces of the building. The wizard permits you to create wind load for any wind direction and thus it is up to you to create those loads for the directions most appropriate to your structure.

Topics in this section

Design Codes and References

Unless explicitly stated all calculations in the BS 6399-2 Wind Modeller are in accordance with the relevant sections BS 6399-2:1997 incorporating Amendment 1 and corrigendum No. 1. (Ref. 4) (page 1118) It is essential that you have a copy of this code with you while assessing wind on any structure.

Your attention is particularly drawn to BS6399-2:1997 - Clause 1.1. **For building shapes which are not covered by the Standard you will need to seek specialist advice.**

We would recommend having the following books to hand when using the software:

- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. (Ref. 7) (page 1118)
- Wind and Loads on buildings: Guide to Evaluating Design Wind Loads to BS6399-2:1997. (Ref. 8) (page 1118)

Unless explicitly noted otherwise, all clauses, figures and tables referred to in this handbook are from reference 4 (page 1118).

Scope (BS6399-2 Wind wizard)

In the main, BS6399-2:1997 addresses rectilinear buildings. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings using the standard method. For more information, please refer to reference 7 (page 1118) (section 2.5.3.2.4, page 82 and 2.5.4.3.3 pages 89-90).

The scope of BS 6399-2 Wind Modeller encompasses:

- Enveloping the building with wall panels and roof panels is undertaken in Tekla Structural Designer in the normal manner. There is only limited validation of the envelope defined (for example connected wall panels must have consistent normal directions). The onus is on you to model the building shape as completely and as accurately as you determine necessary.
- Choice of method:
  - BS6399-2:1997 - Standard Method - Standard effective wind speeds with standard pressure coefficients,
  - BS6399-2:1997 - Hybrid Method - Directional effective wind speeds with standard pressure coefficients.
- Basic Wind Speed and Dynamic pressure is determined.
- Having defined wall panels and roof panels (defaults are standard wall, flat or monopitch roof depending on the slope), you are able to specify the type in more detail e.g. multi-bay, monopitch / duopitch etc.).
- The main wind parameters, are calculated for you but conservatively, (for example Crosswind Breadth, B, is determined for the enclosing rectangle of

the whole building). Wherever possible other parameters are determined conservatively, but you are able to override the values should you need to.

- Given the above, zoning is semi-automatic, (not attempted for roofs with more than 4 sides which are defaulted to single conservative coefficient), with full graphical feedback.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).

Limitations (BS6399-2 Wind wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind Modeler extensive reference has been made to the References (page 1118) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

DANGER You should seek specialist advice for building shapes that are not covered by the Standard - see Clause 1.1 of BS6399-2:1997.

- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Parapets and free-standing canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered.
- Dominant Openings are not explicitly handled - Clause 2.6.2. However, you can use Table 17 to calculate the necessary C_{pi} values and manually apply to a loadcase or individual zone loads.

Loaded areas

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.

- Orthogonal wind directions at the definition of the user.

Beneficial loads

- No automatic reduction is made for beneficial load. When you edit the Zone Load Data for a wind direction, having generated wind load cases, there is an option to allow for beneficial loads.

Wind loading on wall panels

Automatic zoning applies to all wall panels subject to the limitations described below:

- Wall panels that are more than 15° from the vertical are outside the scope - Clause 2.4.1.5.
- The inset storey clause 2.4.4.2 b) is not implemented. You can edit the zones manually according to your engineering judgement to include zone E if you consider this necessary.
- Wall panels of internal wells are not automatically identified - Clause 2.4.3.2a. You can manually edit the zones to apply the roof coefficient to the wall panels.

Wind loading on roof panels

- Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, i.e. all of the internal angles < 180°
- The inset storey clauses 2.5.1.7 a) and b) are not implemented. In clause a) the software sets H_r and H equal conservatively. You are obviously able to edit the zones manually according to your engineering judgement to include the further zones indicated in Figure 18 should you consider this necessary.
- It should be noted that in Table 8 for curved and mansard eaves, the zones start from edge of horizontal roof and not from the edge of the feature.
- Special care should be taken for winds blowing on duopitch with slopes that differ by more than 5°. If the wind is blowing on the steeper slope (that is that the less steep slope is downwind of ridge), the downwind slope should be set to be a flat roof with mansard at eaves for this wind direction.
- Mansard and Multipitch Roofs are not detected automatically, However, you can manually apply the relevant roof type, apex type and bay position parameters for each appropriate wind direction to match the requirements of Figure 22 and Figure 23 - see BS6399-2 Wind Zones.
- Roof Overhangs are not explicitly handled. It is suggested that you should define two separate roof panels - one forming the overhang and the other covering the inside of the building. You can then define C_{pi} values manually

to either have the same coefficient as the adjacent wall, (Clause 2.5.8.2 Small Overhangs), or as an open sided building (Clause 2.6.3).

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind load cases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Using the BS6399-2 Wind Wizard with BREVe data

Using the BS6399-2 Wind Wizard with BREVe data

Method page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

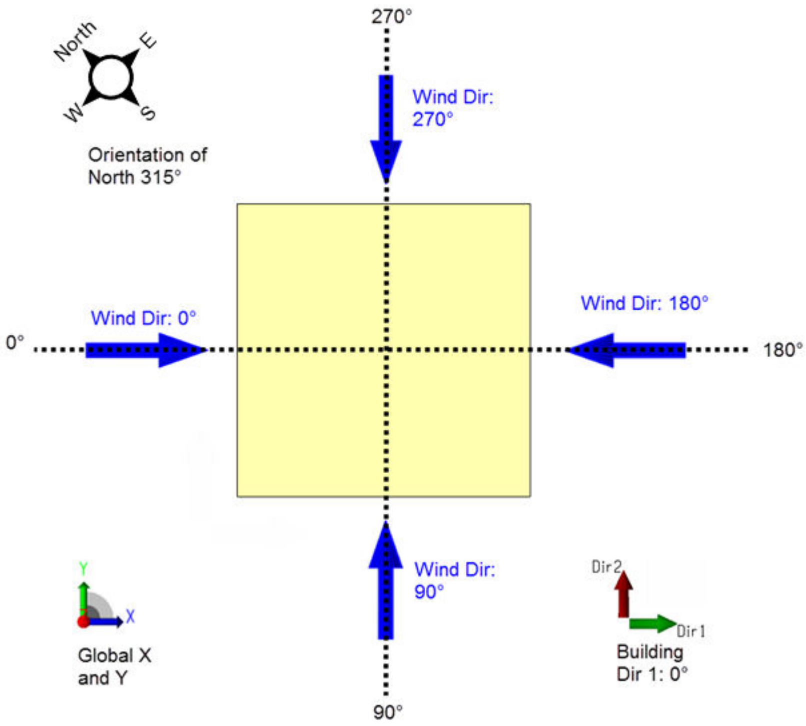
Property/ Buttons	Description
Property	
	There are two calculation methods available:
Standard	Uses standard effective wind speeds with standard pressure coefficients,
Hybrid	Uses directional effective wind speeds with standard pressure coefficients.
	Assuming you have are going to specify the site data using BREVe Grid Ref data there are two options for the source of the wind data: • BREVe - UK National Grid Ref. • BREVe - Irish Grid Ref
Buttons	
Next	If BREVe is the data source, clicking **Next** takes you to the **BREVe location page**; if Other is the data source clicking **Next** takes you to the **Other Location** page.

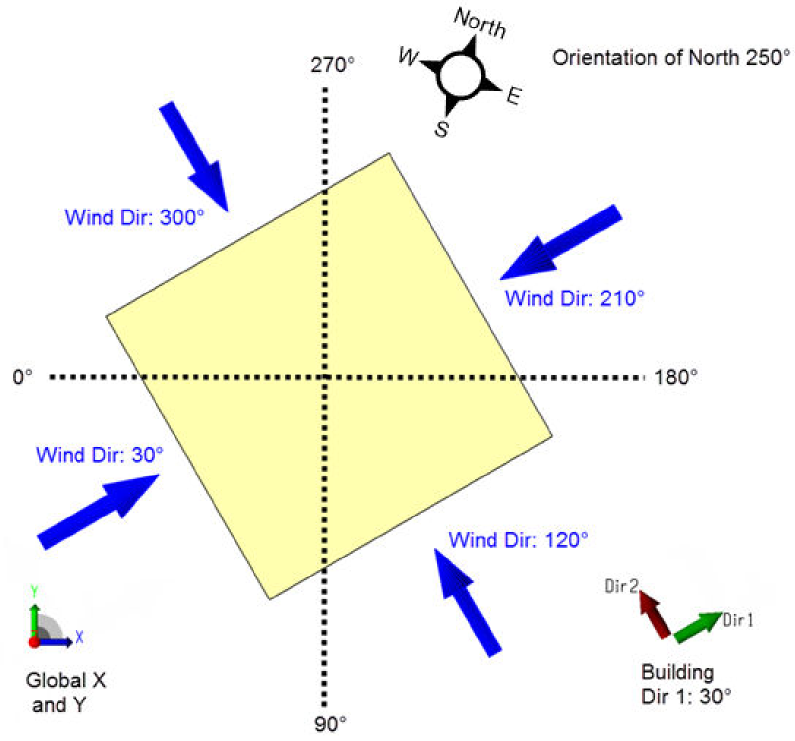
BREVe location page

This page allows you to define the location of the site using the BREVe database, and to define various options to be considered in the wind analysis.

Once you have retrieved the data for a site from the BREVe database you can edit these to take account of your local knowledge of the site.

Property/ Buttons	Description
Property	
Grid Ref.	This shows the grid reference of the site which you have picked through BREVe, irrespective of the method you use to define the site location.
Site Altitude, A	You are able to override the altitude determined by BREVe by entering a value directly here.
Air Density	You need to enter air density at the site.
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly.
Orientation of building known	If you know the orientation of the building with respect to North, then you can define this information by checking this box. You can then define a value which relates the building direction axes of your Tekla Structural Designer model to geographic north. If you want to use the Hybrid method, then you must know and define the building orientation. For the Standard method, the orientation is not essential. If you don't define the building's orientation then North is not shown in graphics views and all the S_d values are set to 1.0.
Orientation of North	The orientation of North is defined using the same convention as is applied to the orientation of the Building Direction Arrows. This can best be understood by reference to a couple of examples: In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°. The resulting relation between the building axes and North is as shown below:

Property/ Buttons	Description
	 In the second example the building direction has been input with Dir 1 = 30° and the orientation of North has been set to 250° In this case the building axes are related to North as shown below:

Property/ Buttons	Description
	
Consider Topography	If you select this check box, then BREVe uses the topographic data it recovers for the site and determines the Altitude Factor S_a in accordance with Clause 2.2.2.2.3. Otherwise the topographic data is ignored and S_a is calculated in accordance with Clause 2.2.2.2.2. **NOTE** In theory the topography could be significant for some directions and not for others.
Consider Obstructions	With this box checked, BREVe uses the obstruction data it recovers for the site and determines the Effective Height H_e as defined in Clause 1.7.3.3. Otherwise the obstructions are ignored and H_e is taken as H_r - see Clause 1.7.3.2.
Buttons	
	Using BREVe, there are 2 methods available for you to define the site location:
Site By Ref...	You can define the grid reference of the site. You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid

Property/ Buttons	Description
	reference from a postcode or given location, for example www.streetmap.co.uk or www.multimap.co.uk
Site By Map...	You can pick the site from a Land / Town Map, • You can pick the site from a Orography Map. • You can pick the site from a ground roughness Category Map, The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 1094).

Using the BS6399-2 Wind Wizard with other data

This section runs through the wizard when "other data" is specified.

Method page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

Property/ Buttons	Description
Property	
	There are two calculation methods available:
Standard	Uses standard effective wind speeds with standard pressure coefficients,
Hybrid	Uses directional effective wind speeds with standard pressure coefficients.
	The remaining topics in this section assume you have chosen to enter the site data manually (Other).
Buttons	
Next	Assuming Other is the data source, clicking **Next** takes you to the **Other Location** page.

Other location page

This page allows you to define the site details when information is not available from the BREVe database, for instance if it is located outside of the UK.

Property/ Buttons	Description
Property	
Site Altitude	You are able to override the altitude determined by BREVe by entering a value directly here.
Air Density	You need to enter air density at the site.
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Other Standard Wind data** page, or if the Hybrid method was selected to the **Other Hybrid Wind data** page.

Other Standard Wind data page

If you select the Standard Method and Other Data Source, then the next page of the Wizard allows you to enter the wind data yourself.

Property/ Buttons	Description
Property	
Basic Wind Speed	You need to enter the basic wind speed at the site.
Ground Roughness	The following settings are available: • Sea - this setting is for sites where the distance to sea is between 0 and 1 km, (see Clause 1.7.2), it is not for offshore sites., • Country - the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town, • Town - for this category you need to specify data for upwind buildings and distance to the edge of the town,

Property/ Buttons	Description
	so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for H_o For this category, the 'Upwind distance from edge of town to site' cannot be greater than the 'Upwind distance from sea to site'.
Consider Topography / Altitude Factor, S_a	When this box is checked, you need to use your own topographic data and determine the Altitude Factor S_a in accordance with Clause 2.2.2.2.3. Otherwise S_a is calculated in accordance with Clause 2.2.2.2.2 and you are not able to override it.
Season factor	You need to enter the season factor (default 1.0).
Probability factor	You need to enter the probability factor (default 1.0).
Buttons	
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 1094).

Other Hydrid Wind data page

If you select the Hybrid Method and Other Data Source then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself. However, most of the data is then dependent on the wind direction, so you must also make your choice of wind directions on this page.

Property/ Buttons	Description
Property	
Direction	Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update these using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined. Each row of the grid operates in a similar manner to the relevant fields of the Other Standard Wind data page
Consider Topography / Altitude Factor, S_a	Reference 8 (section 4.10, page 26) essentially recommends using the Standard Method approach to topography even for the Hybrid Method. So, when

Property/ Buttons	Description
	calculating the Terrain and Building Factor, S_b, we ignore the effects of topography, that is we take S_h = 0. When the box is checked, you need to use your own topographic data and determine the Altitude Factors S_a as defined in Clause 2.2.2.2.3. Otherwise S_a is calculated as defined in Clause 2.2.2.2.2 and you are not able to override it. **NOTE** In theory the topography could be significant for some directions and not for others.
Season factor	You need to enter the season factor (default 1.0).
Probability factor	You need to enter the probability factor (default 1.0).
Buttons	
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 1094).

Results page (BS6399-2 Wind Wizard)

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

This is the start of your topic.

BREVe Standard Method

Initially this method creates 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North). Except for the Hybrid Method with Other Data, you can update the wind directions either by using the 'Dir.' buttons or by changing the direction value as required.

Separately, for each relevant parameter of the Standard Method, BREVe determines the worst case over all its 30° sectors. If the orientation of the building is not known, then S_d is taken as 1.0 for all directions. Otherwise we determine the worst case S_d for each direction. You cannot override the system value in either case.

The worst case S_d is based on splitting the difference to the next direction, with a minimum of ±15° and maximum of ±45°. Within these ranges S_d is interpolated.

For each reference height in the model, we then calculate the site wind speed (V_s using equation 8) and thus the effective wind speed (V_e using equation 12) and the dynamic pressure (q_s using equation 1) for each direction. When calculating actual loads on walls and roofs, we use the q_s value for the relevant reference height, but the Results page only shows the maximum values for each direction.

The Vortex view shows the effective wind speed calculated for each reference height for each 30° sector. Since a single worst case value is used for each parameter, the speeds for different sectors only differ due to S_d provided that the orientation of the building is known.

BREVe Hybrid Method

In this case, BREVe uses the directional method to determine the parameters required to calculate V_s using equation 8, for each height in the building at 30° intervals, (0° to 330°) taking the diagonal dimension `a' as the default 5.0m. (The size effect factor is applied when determining individual loads). We then use equation 27 to determine V_e and equation 16 for q_s.

For each required wind direction the worst case V_e is used for each height, based on splitting the difference to the next direction, with a maximum of ±45 degrees. Within these ranges V_e is not interpolated.

Theoretically, it is possible for a quadrant to use different 30° directions for each height, so the critical wind direction is not displayed in the summary.

The Vortex view shows the effective wind speed calculated for each reference height for each 30° sector.

Other Standard Method

The calculation of V_e and q_s are very similar to the BREVe Standard Method, (see above), except that the worst case data has been entered by you, and this page allows you to enter your own values for S_d.

As there is no data for each 30° sector, the Vortex view only shows the effective wind speed calculated for each reference height for each direction.

Other Hybrid Method

The calculation of V_e and q_s are be very similar to the BREVe Hybrid Method, (see above), except that the data has been entered by you for each direction only so a direct calculation can be performed instead of taking the worst case over a range of sectors. Also this page allows you to enter your own values for S_d.

As there is no data for each 30° sector, the Vortex view only shows the effective wind speed calculated for each reference height for each direction.

Finishing the Wind Wizard

When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 1085) have been encountered.

BS6399-2 Wind Zones

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

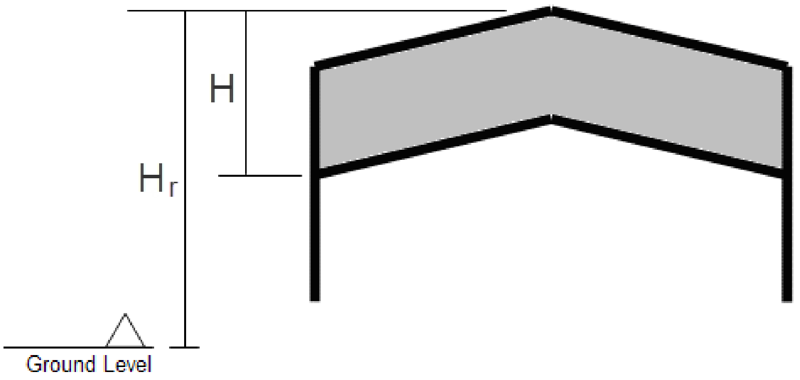
Basic Geometry

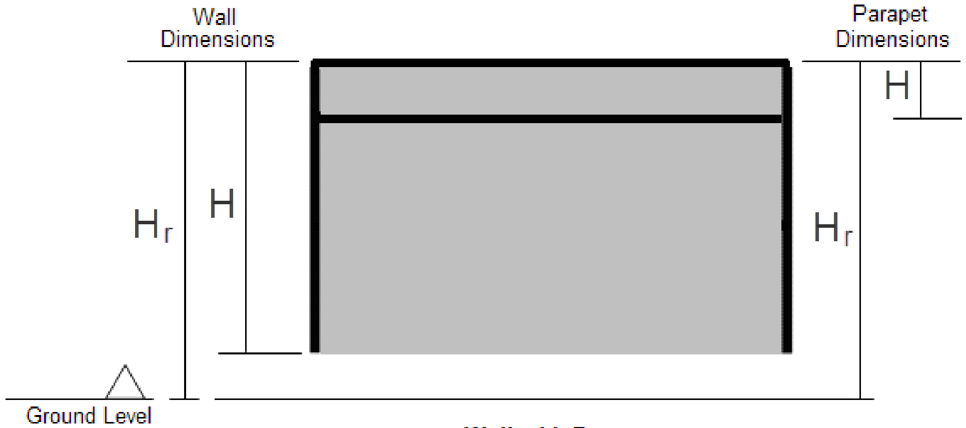
The basic building geometry is assessed as follows:

- Reference Height (H_r) - is taken as the difference between highest point on wall or roof panel and ground level.
- Wall height (H) - is taken as the difference between highest and lowest points on the wall panel.

  These definitions apply to wall panels without parapets and the actual parapets. Wall panels with parapets above them will take their highest point from the parapet. See the diagram below.

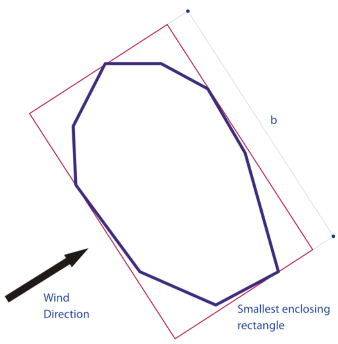
Wall above Roof





Wall with Parapet

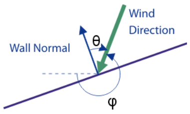
- Roof height (H) - is taken as the difference between highest point on wall or roof panel and ground level. This definition does not handle the upper roof of inset storey but is conservative and only affects the scaling dimension, b - see Clause 2.5.1.7.
- The Building Breadth, B is calculated from the smallest enclosing rectangle around the whole building (considered over all roof and wall panels only) for the given direction. You can override the calculated value in case the Tekla Structural Designer model does not include the whole building.



Wall Zones

Wall Type

We assess each wall panel to determine if it is a windward, leeward or side wall. We classify the type of wall dependent on q:



- θ ≤ 60 deg - Windward,
- θ ≥ 120 deg - Leeward,
- Other walls are classed as Side.

Windward walls	Windward walls have a single zone and Table 5 is used with interpolation for D/H.
Leeward walls	Leeward walls have a single zone and Table 5 is used.
Side walls	Side walls are assessed for recesses (narrow or wide), irregular flushed faces, downwind re-entrant corners. In all cases, side walls have the relevant number of zones. Table 5 is used.

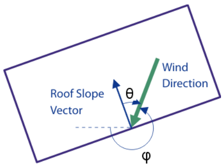
Roof Zones

Roof zones are automatically generated where possible for each wind direction. In essence each roof panel is assessed in its own right based on its properties. The interconnectivity of touching roof panels is not generally considered.

Direction

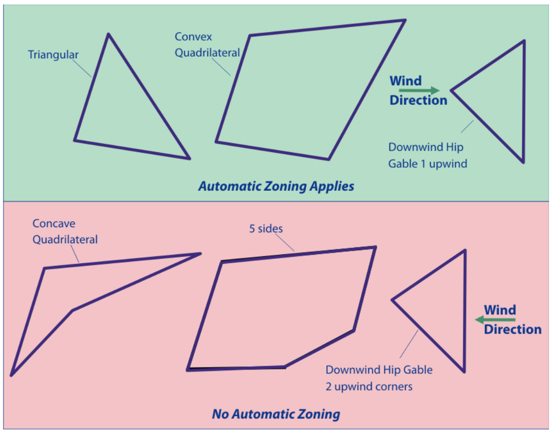
Internally the roof slope vector (line of maximum slope) is determined from the normal vector, with its direction always giving a positive slope angle, i.e. the roof slope vector must always point up the slope.

We calculate the angle between the wind direction and projection of roof slope vector onto horizontal plane (q in range -180° to +180°).



Automatic Zoning

Automatic zoning normally only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°. However, additionally, it only applies to Hip Gable roofs if they are triangular, and Hip Main roofs if they are quadrilateral. Further, Downwind Slope Hip Gables must not have 2 upwind corners.



Dimensions	All zone dimensions are specified in plan.
Flat Roofs	See BS 6399 Clause 2.5.1, Figure 16 and Table 8.
Monopitch Roofs	See BS 6399 Clause 2.5.2.3, Figure 19 and Table 9.
Duopitch Roofs	See BS 6399 Clause 2.5.2.4, Figure 20 and Table 10.
Hip Gable	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Hip Main	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Mansard Roofs	If you manually set the connected roof types to Mansard, then the program will correctly identify the special cases in BS6399 Figures 17c, 22a and 22b, and use the correct tables and values. See BS 6399 Clauses 2.5.1.6.2 & 2.5.4
Multi-bay Roofs	We allow you to interpret BS 6399 Clause 2.5.5 and Figure 23 as you think appropriate and manually define the roof types and sub-types accordingly. You also have the ability to manually set the multi-bay position for each roof panel for each wind direction: • Not Multi-Bay - for this wind direction (conservative default), • Upwind Bay - first bay of many for this wind direction, • Second Bay - for this wind direction, • Third or more Bay - for this wind direction. Where the reduction applies, the values of all coefficients are reduced according to Table 12.

Non-Automatic Zoning

Where automatic zoning does not apply, the system creates a single zone covering the entire roof as follows:

- Flat - B,
- Monopitch - B,
- Duopitch - B for upwind, A for downwind, B for side,
- Hip Gable - B for upwind, B for downwind, D for side,
- Hip Main - B for upwind, A for downwind, D for side.

IS 875 (Part 3) Wind Wizard

This topic discusses the wind wizard when using the IS 875 (Part 3) code.

To access this configuration of the **Wind Wizard...** the Wind Loading Code has to be set to IS 875 (Part 3).

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the effective wind speeds and dynamic pressures for the required wind directions and heights around the building.

The wizard permits you to create wind load for any wind direction, it is up to you to create those loads for the directions most appropriate to your structure.

Unless explicitly stated all calculations in the IS 875 (Part 3) Wind Modeller are in accordance with the relevant sections of IS:875 (Part 3) - 1987 Second Revision, Reaffirmed 2003 and including Amendments 1, 2 and 3.

Wind Options page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

Property/Buttons	Description
Property	
Data Source	The wind velocity calculations are automated, the data source for the calculations is either: • Input directly for the worst case, • Input directly for each direction.
Consider Topography	When this box is checked, you will be asked to enter your own topographic data later in the Wizard.
Buttons	
Next	Click Next to go to the Basic Data page.

Basic Data page

Property/Buttons	Description
Property	
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. This field also gives the lowest level to which wind load is applied in the structure.
Orientation	IS 875 defines the zones assuming the longest principal axis of the building is in the Tekla Structural Designer global Y direction. Due to the potential complexity of the model, Tekla Structural Designer will not attempt to determine the correct angle for the axes, (i.e. default is zero) but you are able to enter a more appropriate value. **NOTE** The convention for Wind Directions is: wind along global +X is 0 degrees; along global +Y is 90 ; along global -X is 180 ; along global -Y is 270.
Building Height Exposed to Wind	Calculated from difference between Ground Level and max level of roof and wind wall panels in the model, with 'override calculated' option to permit user editing.
Building Width and Building Length	The smallest bounding rectangle is determined (considered over all roof and walls only), relative to the given orientation. The building length defaults to the maximum and the width to the minimum of the two dimensions. The 'override calculated' options permit user editing.
Building Type	The following options are available: • Temporary • Low hazard to life • All general buildings • Important buildings
Building Class	This is determined as follows: • A if max(Building Height Exposed to Wind, Building Width and Building Length) < 20m • B if max(Building Height Exposed to Wind, Building Width and Building Length) 50m

<table>
<tr><th>Property/Buttons</th><th>Description</th></tr>
<tr><td></td><td>• C if max(Building Height Exposed to Wind, Building Width and Building Length) > 50m</td></tr>
<tr><td>Basic Wind Speed</td><td>You need to enter the basic wind speed at the site.</td></tr>
<tr><td>Probability factor</td><td>The value of the probability factor is determined from the following table (Table 1):
<table>
<tr><td></td><td colspan="5">Basic Wind Speed</td><td></td></tr>
<tr><td>Building Type</td><td>33</td><td>39</td><td>44</td><td>47</td><td>50</td><td>55</td></tr>
<tr><td>Temporary</td><td>0.82</td><td>0.76</td><td>0.73</td><td>0.71</td><td>0.70</td><td>0.67</td></tr>
<tr><td>Low hazard</td><td>0.94</td><td>0.92</td><td>0.91</td><td>0.90</td><td>0.90</td><td>0.89</td></tr>
<tr><td>All general</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td></tr>
<tr><td>Important</td><td>1.05</td><td>1.06</td><td>1.07</td><td>1.07</td><td>1.08</td><td>1.08</td></tr>
</table></td></tr>
<tr><td>Buttons</td><td></td></tr>
<tr><td>Next</td><td>Click Next to go to the Terrain page.</td></tr>
</table>

Terrain page

The next page of the Wizard allows you to enter the terrain data.

Property/Buttons	Description
Property	
Terrain Category	• Category 1 = Exposed open terrain few obstructions (Default) • Category 2 = Open terrain scattered obstructions • Category 3 = Terrain with numerous closely spaced obstructions • Category 4 = Terrain with numerous large high closely spaced obstructions
Fetch a distance	The velocity profile for a given terrain category does not develop to full height immediately and in this release Tekla Structural Designer only deals with a single terrain profile so you are warned if the velocity profile has not

Property/Buttons	Description
	developed over the fill height of the building for the defined fetch.
Buttons	
Next	Click Next to go to the Topography page.

Topography page

If Consider Topography was checked, then the next page of the Wizard allows you to enter the topography data.

Property/Buttons	Description
Property	• None - no feature (default) • 2D Escarp - Cliffs and Escarpments • 3D Hill - Hills and Ridges
Feature Height, Z	Effective height of the feature.
Site dist from crest, X	Distance upwind or downwind from the crest to the building site (-ve = upwind).
Site height above mean ground level, H	Height above mean ground level.
Upwind Slope Length, L	Actual length of the upwind slope in the wind direction.
Downwind Slope Length	Actual length of the downwind slope in the wind direction.
Buttons	
Next	Click Next to go to the Results page.

Results page

The final page of the Wizard is a summary of the results.

The final page of the Wizard is a summary of the results.

$V_{z\ max} = V_b \times k_1 \times k_2 \times k_3$ (m/s)

p_z max = 0.6 x V_z max2 (N/m^2)

The above is for the highest point in the building only.

Property/Buttons	Description
Buttons	

Property/Buttons	Description
Finish	When you click Finish, the **Wind Wizard...** generates the wind zones for the entire building for each of the specified wind directions. Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no issues have been encountered.

Wind model loadcases

This topic discusses the creation of wind load cases following the **Wind Wizard...** command.

You control the creation of appropriate wind loadcases manually. It is not practical to determine them automatically. The creation of wind loadcases is achieved via the **Load** tab, **Wind Loadcases** command, which opens the **Wind Loadcases** dialog.

NOTE The **Wind Loadcases** dialog is only available once a valid Wind Model has been created using the **Wind Wizard...**

The **Auto** button within the **Wind Loadcases** dialog automatically generates wind loadcases. Alternatively, you can create loadcases manually using the **Add** button.

NOTE The **Auto** button is greyed out and hence unavailable, if wind loadcases have already been defined. These would need to be deleted to use the **Auto** button again.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated, these loadcases are standard load cases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases for design. This ensures that the number of load combinations to be considered are minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Creating loadcases for the Low Rise Buildings method

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate

Wind Loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the **Load** tab).

The **Auto** button on the dialog provides various options to control the number of loadcases before they are generated (its intention being to prevent generation of very large numbers of loadcases). Alternatively you can create loadcases manually using the **Add** button.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard load cases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases for design. This ensures that the number of load combinations to be considered are minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Adding wind loadcases

By clicking the **Add** button you are able to generate loadcases one at a time.

- The loadcase name is auto generated from the other input parameters, but it can be edited if required.
- The direction the loadcase will be created for is selected from the droplist.
- The internal pressure coefficient is entered directly.

NOTE Depending on the wind code you have selected, the default is the negative value depending on the Enclosure Classification from:

- AISC 7-16: Table 26.13-1, p271
- AISC 7-10: Figure 26.11-1, p201
- AISC 7-05: Figure 6-5, p47

- You are able to flag if it is a Torsional Loadcase or not: when flagged, reduced pressure loads are applied to parts of the standard zones (by applying a net pressure factor of 25%).
- You are able to flag if roof loads are to be generated for the loadcase or not.

Auto generating wind loadcases

By clicking the **Auto** button you are able to control the total number of loadcases generated.

Up to 4 loadcases can be generated for each direction, using positive and negative GCpi derived from the Enclosure Classification, (see Figure 26.11-1, p201). If a direction is included, but neither +ve GCpi or -ve GCpi have been checked, then that is interpreted as a loadcase with GCpi of zero. You are also given the option to include the torsional loadcases or not, (default not), but it is assumed that Roof loads are required.

By clicking **OK** the loadcases are generated, but you can then make any changes required before clicking **OK** once more to close the Wind Loadcases dialog.

Creating loadcases for the All Heights method

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the **Load** tab).

The **Auto** button on the dialog provides various options to control the number of loadcases before they are generated (its intention being to prevent generation of very large numbers of loadcases). Alternatively you can create loadcases manually using the **Add** button.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard load cases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized.

Adding wind loadcases

By clicking the **Add** button you are able to generate loadcases one at a time.

- The loadcase name is auto generated from the other input parameters, but it can be edited if required.
- The direction the loadcase will be created for is selected from the droplist.
- The internal pressure coefficient is entered directly.

NOTE Depending on the wind code you have selected, the default is the negative value depending on the Enclosure Classification from:

- AISC 7-16: Table 26.13-1, p271
- AISC 7-10: Figure 26.11-1, p201
- AISC 7-05: Figure 6-5, p47

- You are able to flag if it is a Torsional Loadcase or not: when flagged, reduced pressure loads are applied to parts of the standard zones (by applying a net pressure factor of 25%).
- For torsional loadcases, you are able to flag if positive or negative eccentricity is applied.

- You are able to flag if loads are to be created for roof zones or not.

NOTE This option is provided to allow for the following note, (depending on the wind code you have selected):

- AISC 7-16: Figure 27.3-1, Note 7, p276
- AISC 7-10: Figure 27.4-1, Note 9, p207
- AISC 7-05: Figure 6-6, Note 9, p49

- If loads are created for roof zones, you are able to flag if positive or negative pressure coefficients are to be used for roof zones.

Auto generating wind loadcases

By clicking the **Auto** button you are able to control the total number of loadcases generated.

Up to 8 loadcases can be generated for each direction. You may choose positive and/or negative GCpi derived from the Enclosure Classification, (see Figure 26.11-1, p201). If you include a direction, but have not checked either +ve GCpi or -ve GCpi, then that is interpreted as a loadcase with GCpi of zero. Similarly you may choose positive and/or negative Cp values for roof zones - checking neither indicates Roof Loads to be ignored for this direction. Torsional loadcases will not be included by default, but can be added with positive or negative eccentricity.

By clicking **OK** the loadcases are generated, but you can then make any changes required before clicking **OK** once more to close the Wind Loadcases dialog.

Creating loadcases EC1991 1-4

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the **Wind Loadcases** dialog box (accessed from the Load toolbar).

The **Auto** button on the dialog creates a default set of loadcases in each of the directions, i.e.

- -0.3 for C_{pi} with -ve roof C_{pe}; not Overall
- -0.3 for C_{pi} with +ve roof C_{pe}; not Overall
- +0.2 for C_{pi} with -ve roof C_{pe}; not Overall
- +0.2 for C_{pi} with +ve roof C_{pe}; not Overall
- Overall with zero for C_{pi}; -ve roof Cpe

Alternatively you can create loadcases manually using the **Add** button. You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard load cases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Wind loadcases dialog

As well as specifying which direction the loadcase will be created for, this dialog allows you to set default values for all the zone loads generated in the loadcase.

Item	Description
Fields	
Structural Factor - Automatically calculate seperate c_s and c_d factors	The UK NA states that the Structural Factor, c_sc_d may be separated into a size factor c_s and a dynamic factor c_d, i.e. it is still acceptable to apply Clause 6.2 (1) a) to d) and set c_sc_d = 1, or use Annex D. Where **Structural Factor - Automatically calculate separate c_s and c_d factors is checked**, c_d is calculated using Figure NA.9 and c_s using Table NA.3. **NOTE** The option to automatically calculate c_c and c_d is not available for other National Annexes.
Structural damping	The δ_s value is used to determine the dynamic factor c_d. It is only visible if the **Separate Factors** box is checked. See Table F.2.
Name	The loadcase name is auto generated from the other input parameters, but it can be edited if required.
Direction	The direction the loadcase will be created for is selected from the droplist.
Overall	You are able to flag if the loadcase is to be specifically used for examining the overall behaviour of the structure by checking this box. **NOTE** It may be necessary for you to create a second copy of the loadcase with this check box cleared if the loadcase is also used for designing elements.
b+h	When designing elements, (beams, columns, braces etc), Table NA.3 in the UK NA implies that b and h should be the width and height respectively of an element. Due to the nature of the loads in the program, it is not practical to do this automatically, and so you should specify a

Item	Description	
	value to be used in the loads generated for this loadcase (default 5.0m). If separate factors are not to be used, (i.e. use combined c_sc_d), then this value is redundant. If separate factors are to be used, but **Overall** is checked on the row, then the b+h cell is marked inactive. In this case, for each wall and roof, the program calculates b+h using b & h from the zone properties for the relevant wind direction.	
Use +ve Cpe	Where 2 sets of coefficients are given in a table for roof zones, this field indicates if the negative or positive C_{pe} value is to be used.	
Cpi	Default Internal Pressure Coefficient (-0.3, 0.0, +0.2 or other value) - to be calculated by you from Clause 7.2.9.	
Buttons		
Add	Click this button to add a single wind loadcase.	
Delete	Click this button to delete a wind loadcase.	
Auto	Click this button to create a default set of loadcases in each of the directions.	

Wind model load decomposition

The following topics will discuss the decomposition that occurs when considering wind load applied to your model.

- Roof panel load decomposition (page 1109)
- Wall panel load decomposition (page 1111)
- Alternative wind load decomposition methods for complex models (page 1115)

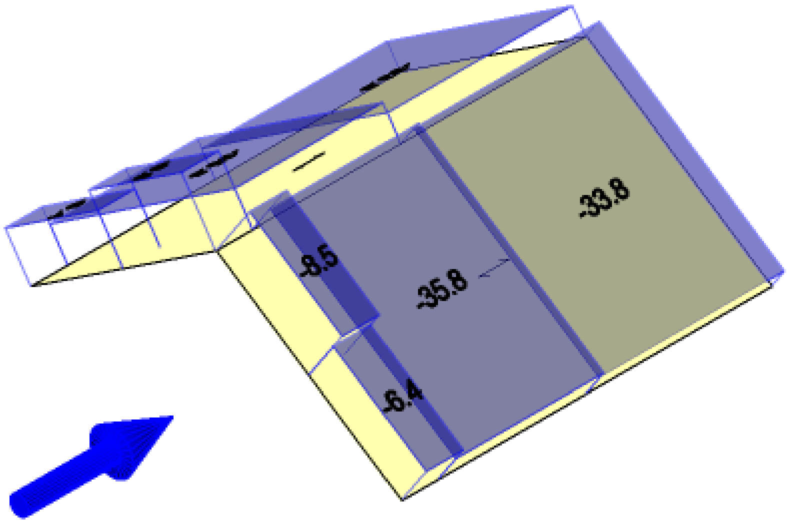
Roof panel load decomposition

This topic will discuss the decomposition that occurs when considering wind load applied to roof panels.

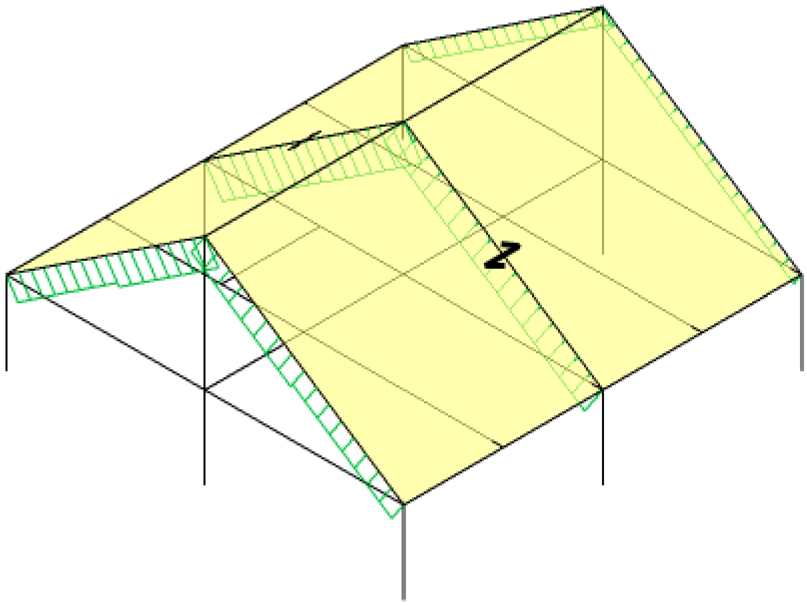
Roof panel load decomposition

The direction of the one way decomposition of the wind zone loads to roof members is determined by the span direction (i.e. rotation angle) of the roof panel.

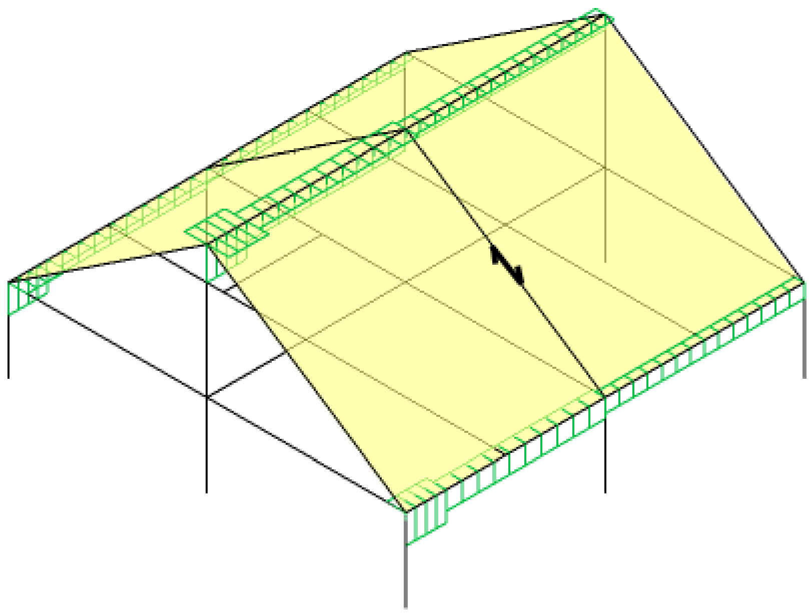
All types of elements within the roof panel plane (except bracing) are considered during the load decomposition.



Zone loads on roof.



Roof panel decomposition (rotation angle 0 degrees)



Roof panel decomposition (rotation angle 90 degrees)

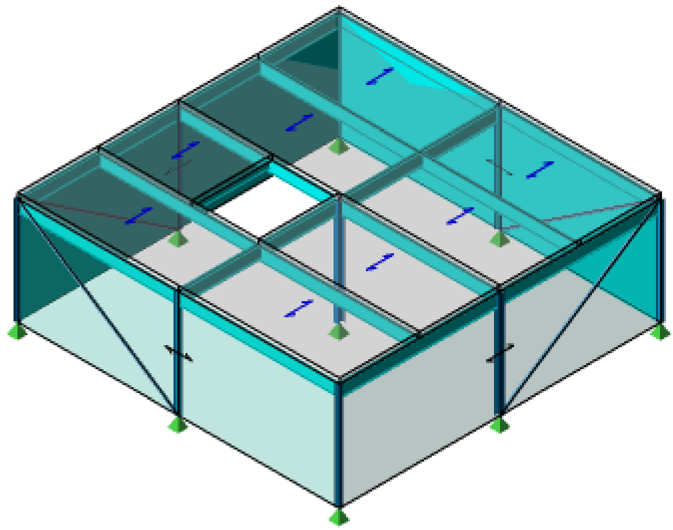
Wall panel load decomposition
This topic will discuss the decomposition that occurs when considering wind load applied to wall panels.

Decomposition options

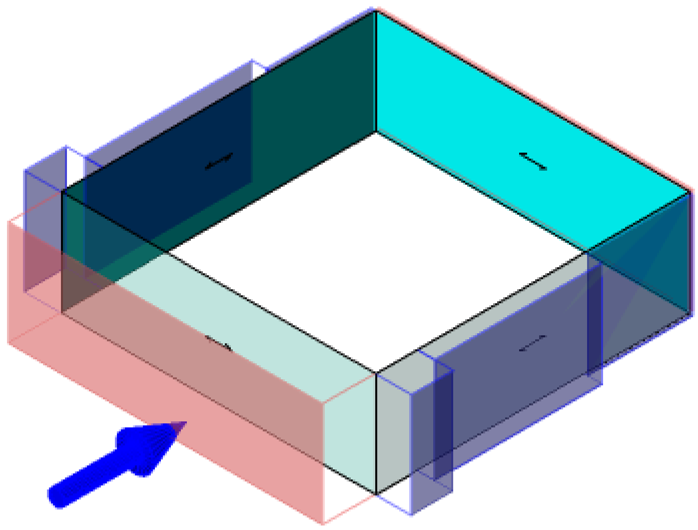
Wind zone loads are decomposed on to the structure according to the **Decompose to** wall panel property. The available options are:

- Members
- Nodes (default)
- Rigid Diaphragms

To demonstrate the effect of the different **Decompose to** values, consider a braced steel frame clad in wind wall panels as shown below:



For wind direction 0, the wind model produces Zone loads as follows:



The topics below illustrate how the zone loads shown above are decomposed using the three different **Decompose to** values.

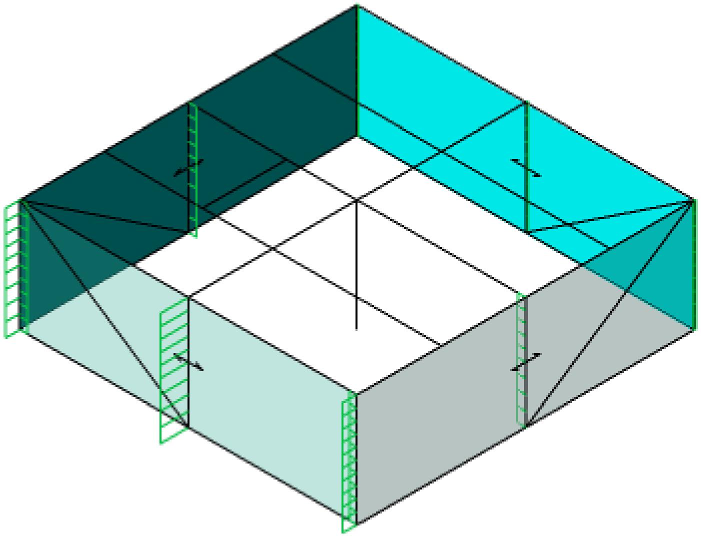
Decompose to Members

Decomposition to members is similar to the roof panel decomposition, the span direction of the wind wall panel determines the direction of the one way decomposition.

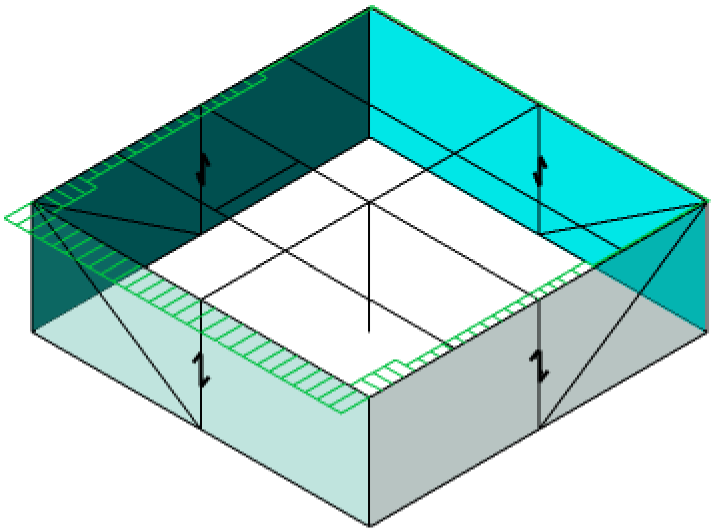
All element types within the wind wall plane are considered with the exception of bracing members.

Decomposition to members allows the generation of UDL's on portal stanchions and gable posts without the need to model side rails.

For the model shown above, choosing **Decompose to Members** produces the following load decomposition:



Decomposition to Members (rotation angle 0 degrees)



Decomposition to Members (rotation angle 90 degrees)

Decompose to Nodes

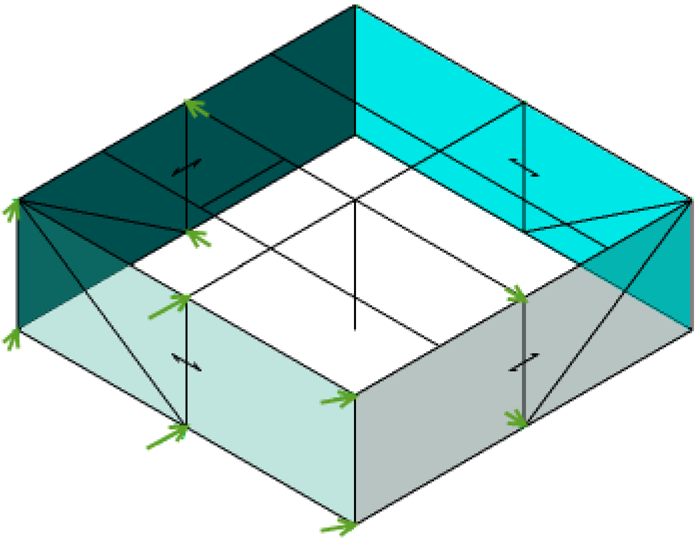
Decomposition to nodes is the **default** setting and results in nodal loads on the supporting members. This setting is typically appropriate to avoid lateral loads on simple beams.

All element types within the wind wall plane are considered with the exception of bracing members.

The initial decomposition is the same as for members, with the direction of the one way decomposition being specified by the span direction of the panel. A second decomposition stage is then undertaken to convert the member loads to nodal loads:

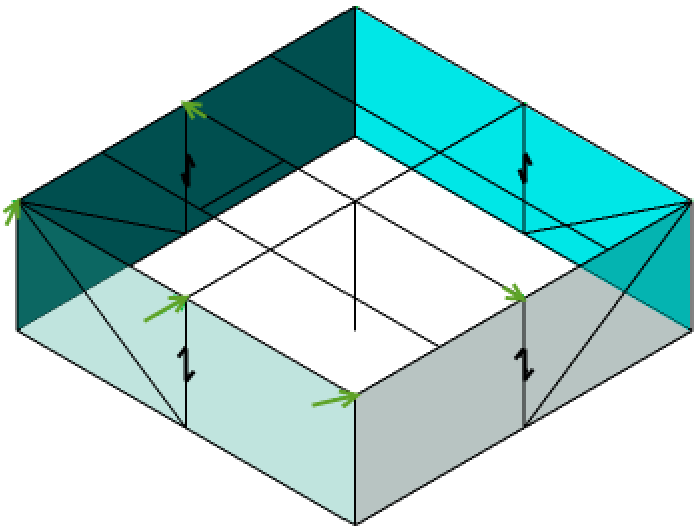
- Full/partial UDLs and VDLs on elements (lengths of beams/columns between nodes) are distributed back to nodes as if the elements were simply supported at either end.
- The final nodal load is the sum of all incoming element loads.

For the model shown above, choosing **Decompose to Nodes** produces the following load decomposition:



collapse

Decomposition to Nodes (rotation angle 0 degrees)



Decomposition to Nodes (rotation angle 90 degrees)

NOTE In the example above, when the rotation angle is 0 degrees, some of the nodal loads are applied directly to supports.

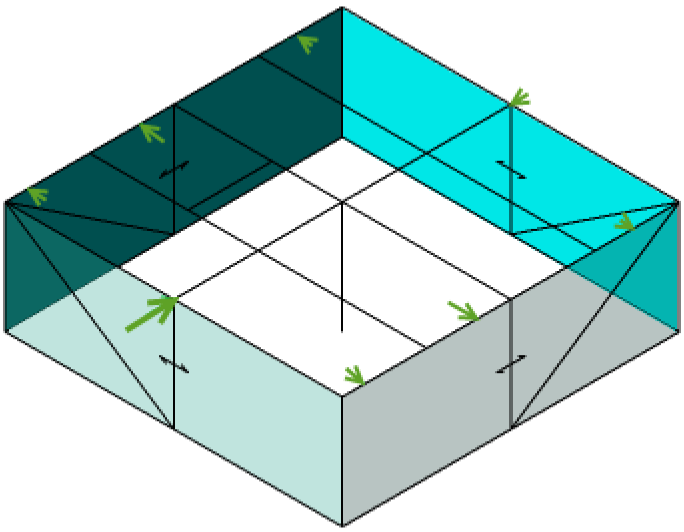
Decompose to Rigid Diaphragms

Decomposition to rigid diaphragms does not consider the span direction of the wall panel (so the rotation angle is irrelevant).

It is particularly useful for flat-slab structures, as the alternative **Decompose to Member** or **Node** decomposition methods require supporting members that may not exist in the model.

NOTE	All rigid diaphragms within the wind wall height are considered for decomposition irrespective of whether they are physically connected to the wind wall.

For the model shown above, choosing **Decompose to Rigid Diaphagm** produces the following load decomposition:



Decomposition to Diaphragms - each Zone load is decomposed as a separate point load on the diaphragm

Validation of Panels set to Rigid Diaphragm Decomposition

The following validation checks are performed on wind wall panels set to **Decompose to Rigid Diaphragms**:

- Each panel must be rectangular
- The top level of every wall panel must align with a rigid diaphragm
- Each panel may be sub-divided into zones, but only by horizontal lines
- Unlike for **Decompose to Member** or **Node**, each wall panel does not need to have supporting members along its edges.

Alternative wind load decomposition methods for complex models

This topic discusses alternative methods of wind decomposition on complex models.

For complex models the placement of wind walls required by the Wind model method can be a time consuming operation.

In such situations (provided the model has suitable rigid diaphragms), the following alternatives offer quicker and simpler ways to apply approximate wind loads:

- Use engineering judgement to clothe the structure with an arrangement of simplified wind wall panels around its bounding box, set the wind walls to decompose to rigid diaphragms, and proceed with the Wind model method
- Do not apply wind walls. Alternatively consider the application of manual wind loads.

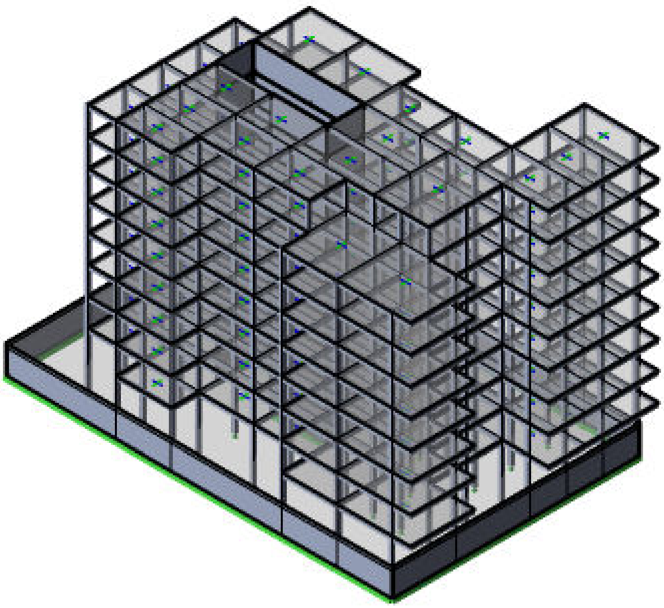
See also

Simplified wind wall panels (page 1116)

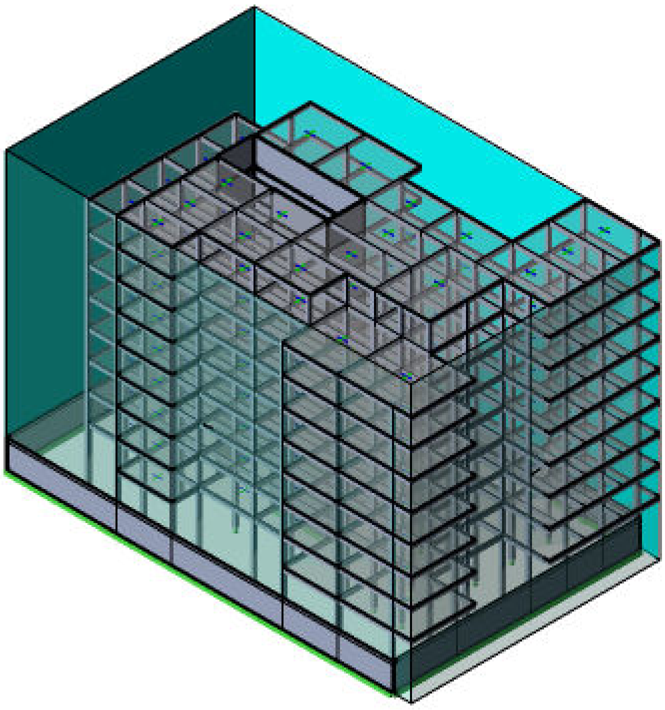
Simplified wind wall panels

This topic demonstrates the simplified wind wall panel approach.

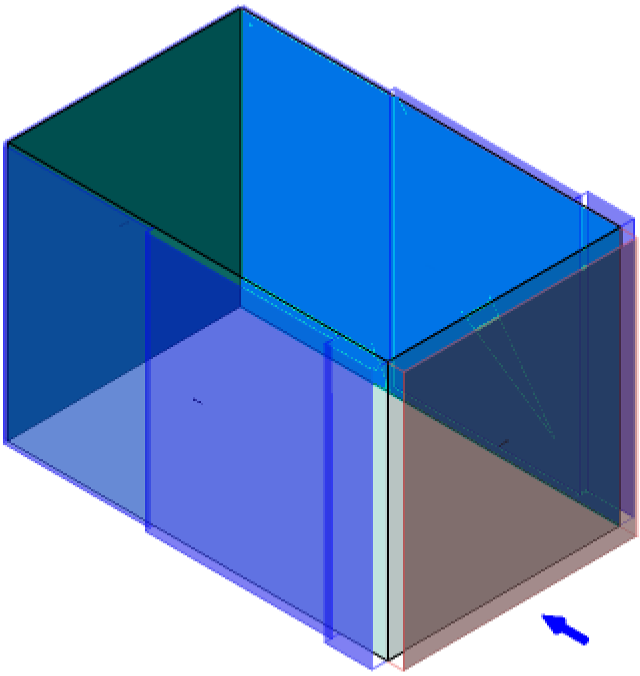
In theory, because wind walls set to **Decompose to Rigid Diaphragms** don't have to physically connect to the diaphragms, even a very complex model could be clothed with just four simplified wind walls defined along its bounding box, as illustrated below:



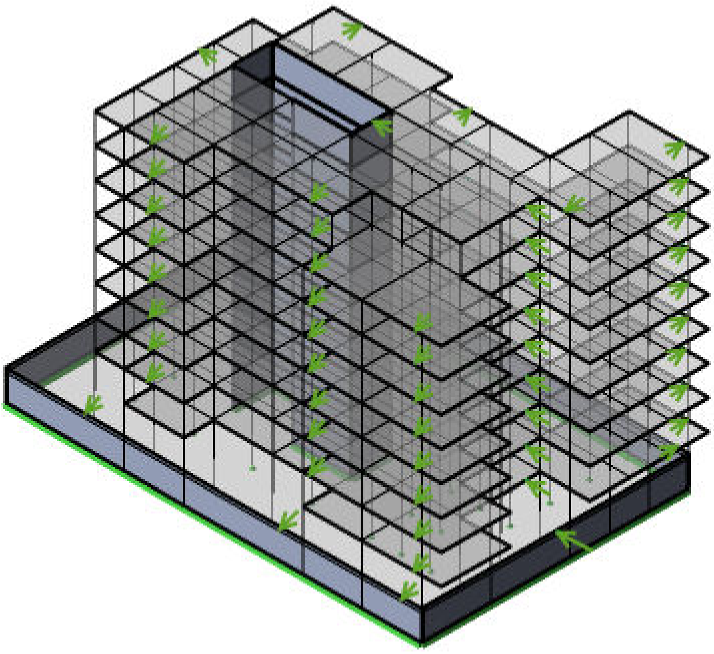
Complex model



Simplified wind walls around the "bounding box"



Zone loads for the simplified wind wall model



Zone loads decomposed to rigid diaphragms

Such an approach could be used to rapidly establish approximate wind loads (based on the rectangular building envelope) which could then be refined at a later stage if necessary.

NOTE If this method is adopted you are strongly advised to review the wind zones that are formed, and the resulting decomposed loads to ensure they meet your expectations - it will not give good results for all models.

If the decomposed loads produced by the above approach are not satisfactory, you might decide to take greater control by applying the loads manually.

References

1. **ASCE/SEI 7-10**. Minimum Design Loads for Buildings and Other Structures. **ASCE, 2010. ISBN: 978-0-7844-1085-1**.
2. **Kishor C. Mehta and James M. Delahay (2004)**. Guide to the Use of the Wind Load Provisions of **ASCE 7-02. ASCE Press. ISBN: 0-7844-0703-7. British Standards Institution (25/04/05)**. Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions. BS EN 1991-1-4:2005.
3. **British Standards Institution (September 2008)**. UK National Annex to Eurocode 1: Actions on structures. NA to BS EN 1991-1-4:2005.
4. **British Standards Institution (July 2002)**. Loading for Buildings - Part 2: Code of practice for wind loads. BS6399-2:1997.

5. **British Standards Institution**. Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009.

6. **Cook, N.J**. Designers' Guide to EN 1991-1-4. Euro Code 1 : Actions on Structures, General Actions Part 1-4 : Wind actions. **Thomas Telford, London.ISBN 978-0-7277-3152-4**.

7. **Cook, N.J. (1999)**. Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. **Thomas Telford, London. ISBN: 0 7277 2755 9**.

8. **Bailey, C.G. (2003)**. Guide to Evaluating Design Wind Loads to BS6399-2:1997. **SCI Publication P286**.

9. BREVe software package version 3. **Copyright © 2009 CSC (UK) Ltd; BRE Ltd; Ordnance Survey.**

Simple wind and manually applied wind loads

This approach provides a quick method of applying wind to the structure without requiring you to create a wind model.

Simple Wind loads can be created which are then decomposed to diaphragms, or you could apply wind loads directly as panel, member, or nodal loads.

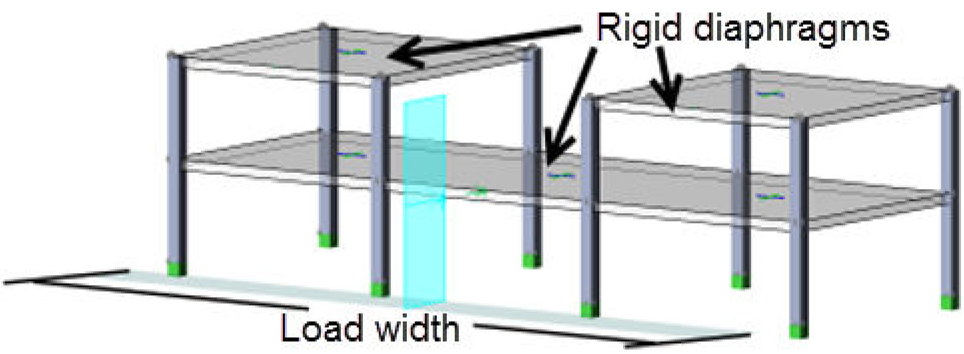
Click the links below to find out more:

- Simple wind overview (page 1119)
- Limitations of wind decomposition to diaphragms (page 1123)

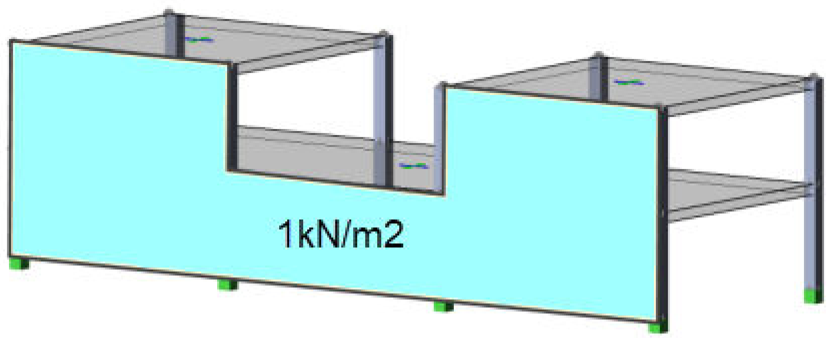
Simple wind overview

Each simple wind load is defined as an area load over a defined width and height.

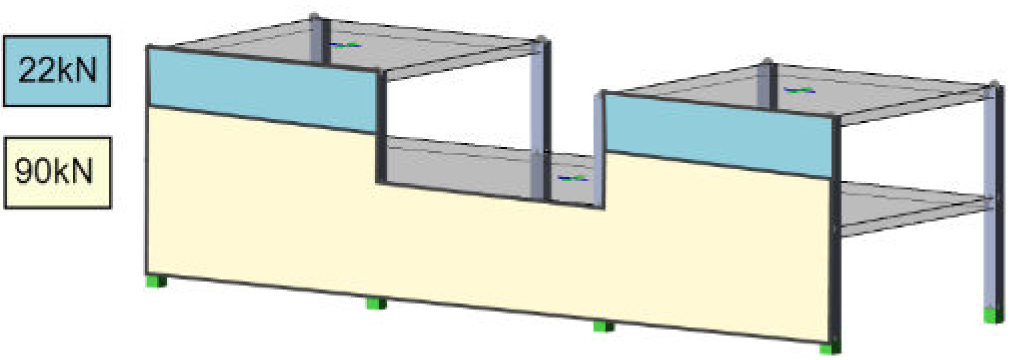
In the two storey example shown below, a 1kN/m^2 load is applied over the full 22.5m width and 6m height of the building.



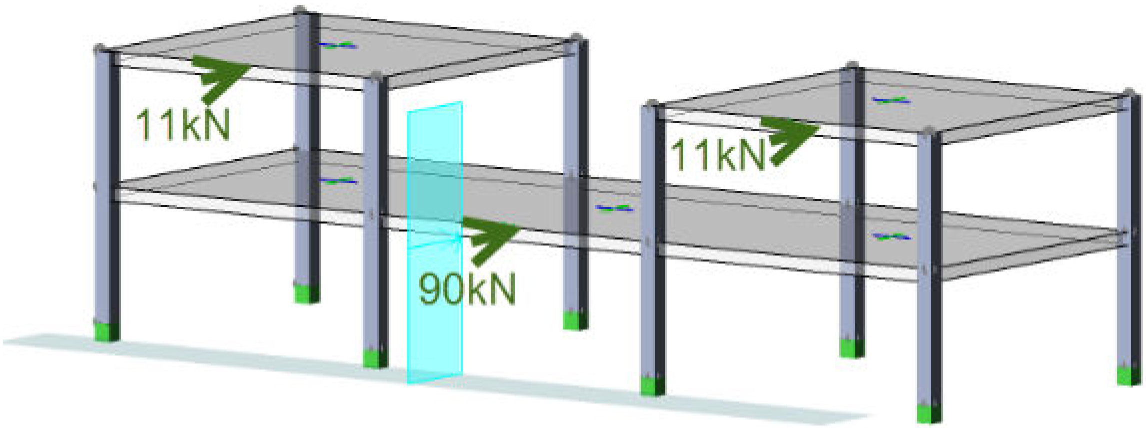
- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation error is generated - this is to ensure the loading is distributed as correctly as possible.
- During decomposition an imaginary 'bounding box' is formed around the building - only the load width within the building profile is considered for decomposition.



- Load within the bounding box that hits the structure (the blue shaded area above) is decomposed to point loads on rigid diaphragms only - it does **not** get decomposed to semi-rigid diaphragms.
- All rigid diaphragms on the top or bottom level or anywhere in-between are considered, with the area load being divided between the levels before it is decomposed. In the example the area load is split between 1st and 2nd floor levels so that a total of 22kN is to be decomposed above and 90kN below.



- The load is then decomposed to the diaphragms at each level in proportion to the width of each diaphragm. Each load being applied as a nodal load in the direction of the simple wind load at the mid point of the projected load.



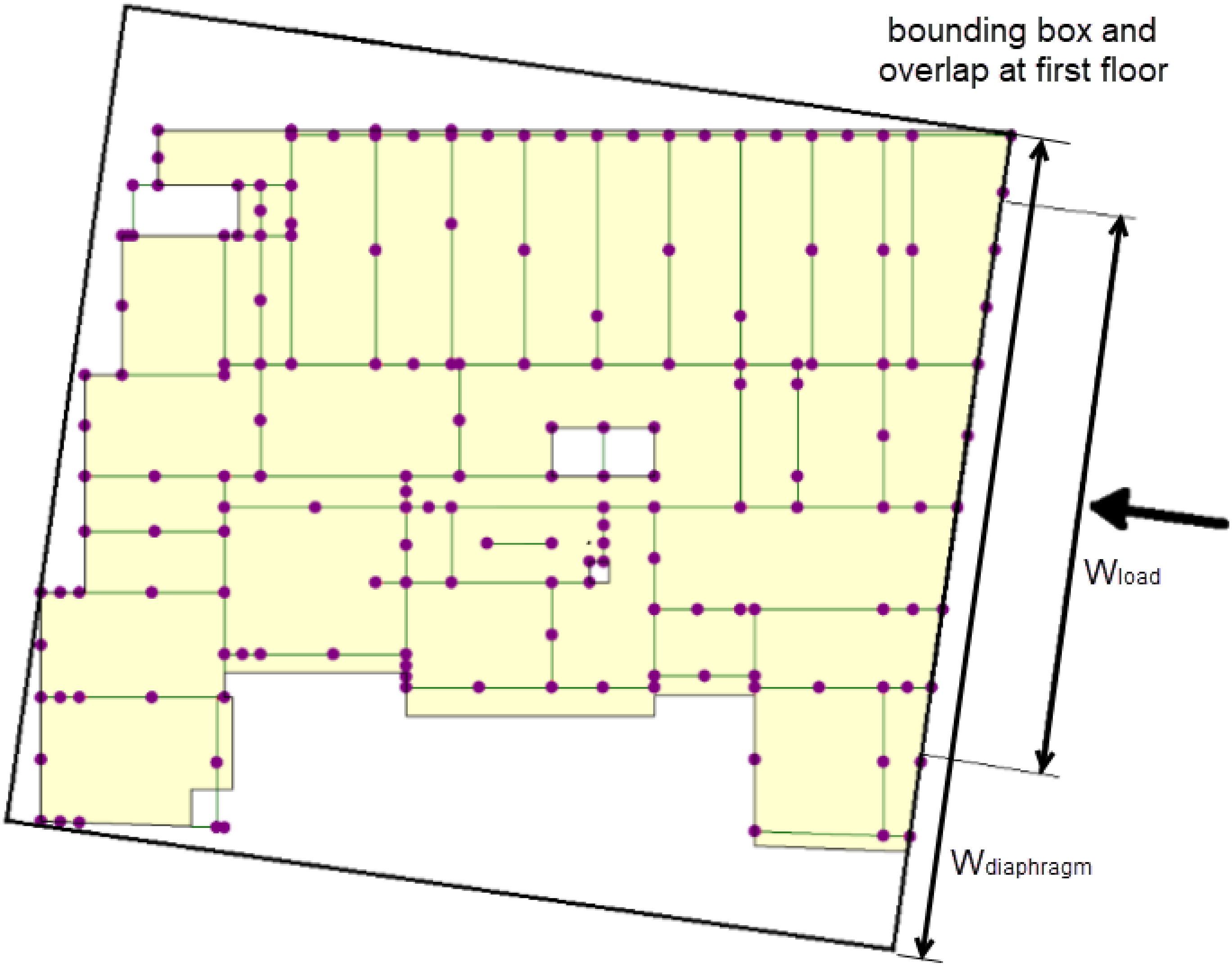
- If there are no suitable diaphragms on the top level, the load is applied at the next level down.
- Similarly if the 'Ignore diaphragms on lowest level' box is checked on the Simple Wind Loading dialog or there are no suitable diaphragms on the bottom level, the load is applied at the next level up.

NOTE If for some reason there are diaphragms at the ground level, then you may decide to check the 'Ignore diaphragms on lowest level' box in order to ensure no load is lost directly to the foundations.

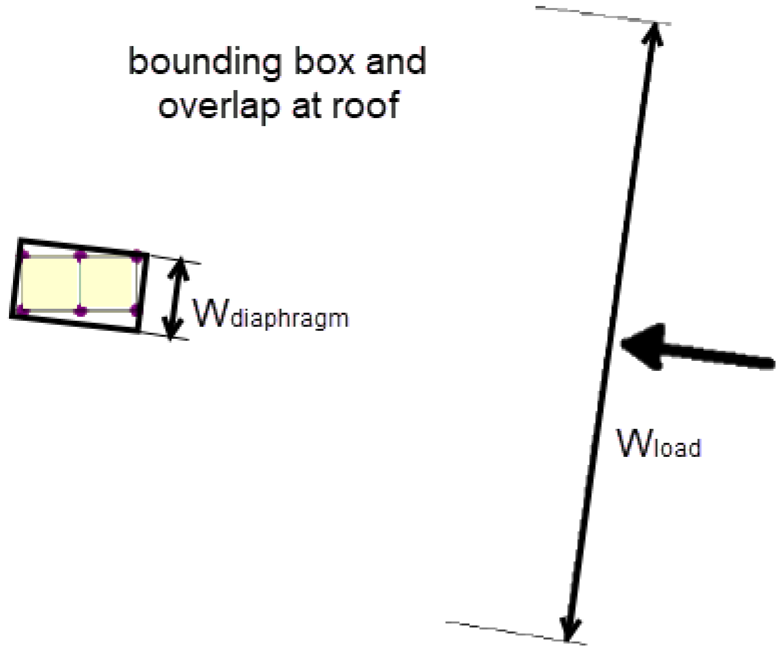
Situations where rigid diaphrams are ignored

If the projected width of a rigid diaphragm in the direction that the load is acting is less than 5% of the simple wind load width it is assumed to be ineffective.

In the below example, at first floor level the width of the diaphragm perpendicular to the load exceeds the width of the load, so the diaphragm is effective.

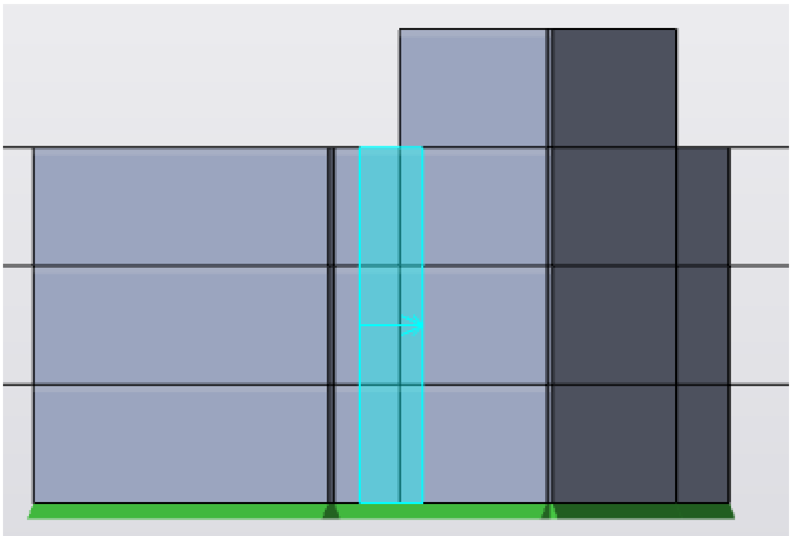


This changes at roof level, as the width of the diaphragm is now significantly smaller than the width of the load. If the diaphragm width is less than 5% of the load width, the diaphragm is ignored.



Furthermore, as this is the only diaphragm at the top level you would be prevented from applying the load, as a "Top level of Simple Wind Load must align with a rigid diaphragm" error message would be displayed.

A workaround would be to reduce the top level of the simple wind load so that it hits a diaphragm at a lower floor of sufficient width, as shown below.



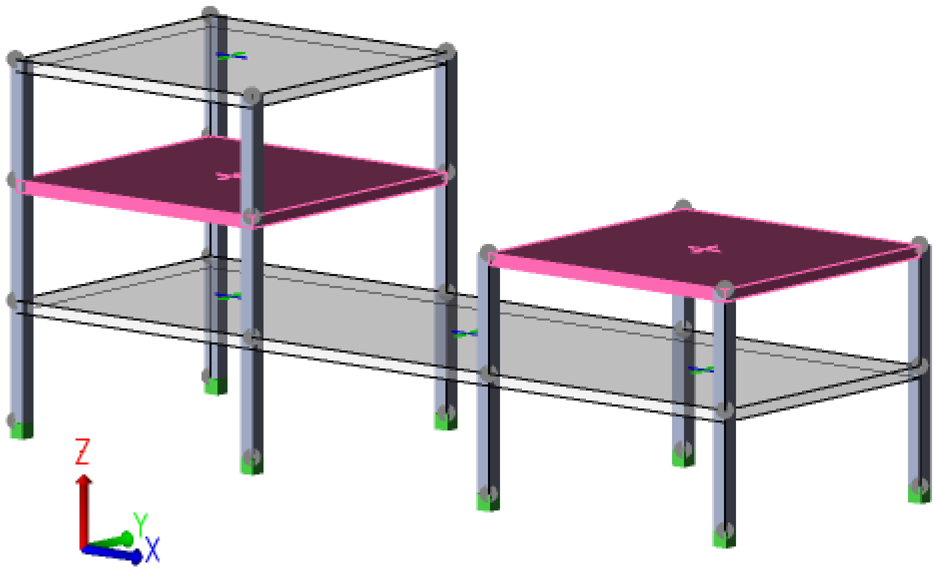
Limitations of wind decomposition to diaphragms

Certain building shapes need extra consideration if simple wind loads have been applied, or if wind loads have been applied to wall panels set to decompose to diaphragms.

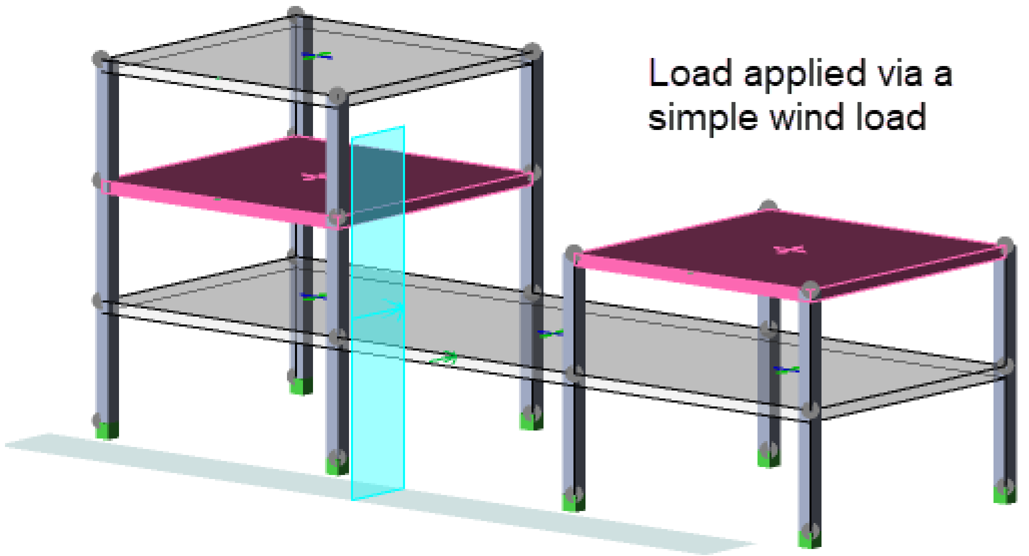
As illustrated by the following examples, buildings containing discrete towers (and thus containing disconnected rigid diaphragms) are a particular concern.

Wind load perpendicular to disconnected diaphragms

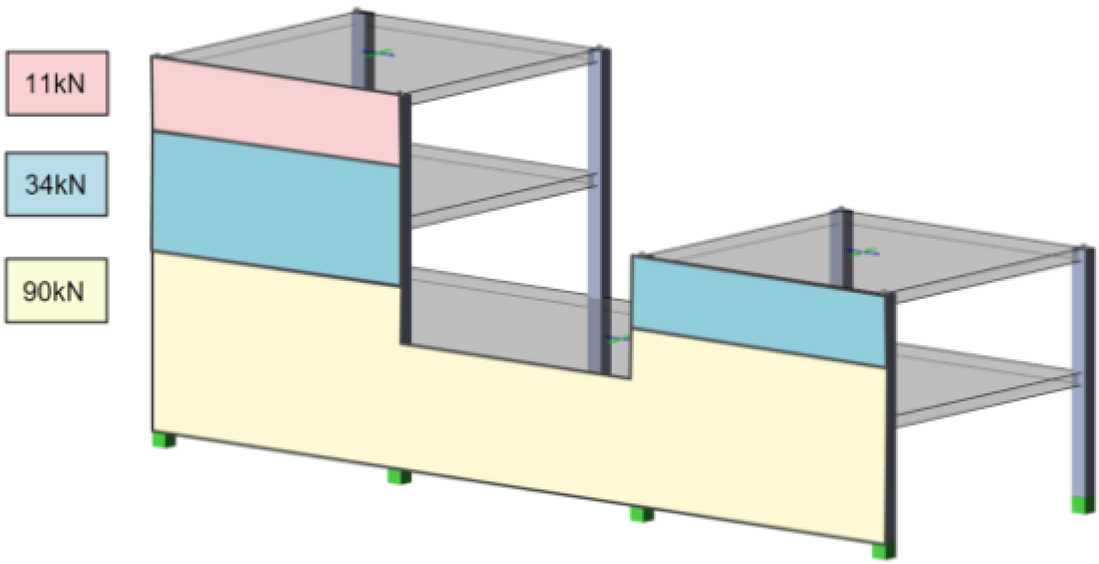
In this example wind load is to be applied in the Global Y direction, perpendicular to the disconnected diaphragms that exist in the highlighted slabs at the second floor level shown below.



An issue arises when the wind load is applied as an area load that has to be decomposed to both diaphragms. This could happen when the load is applied either via a Simple Wind load, or a wind wall panel:

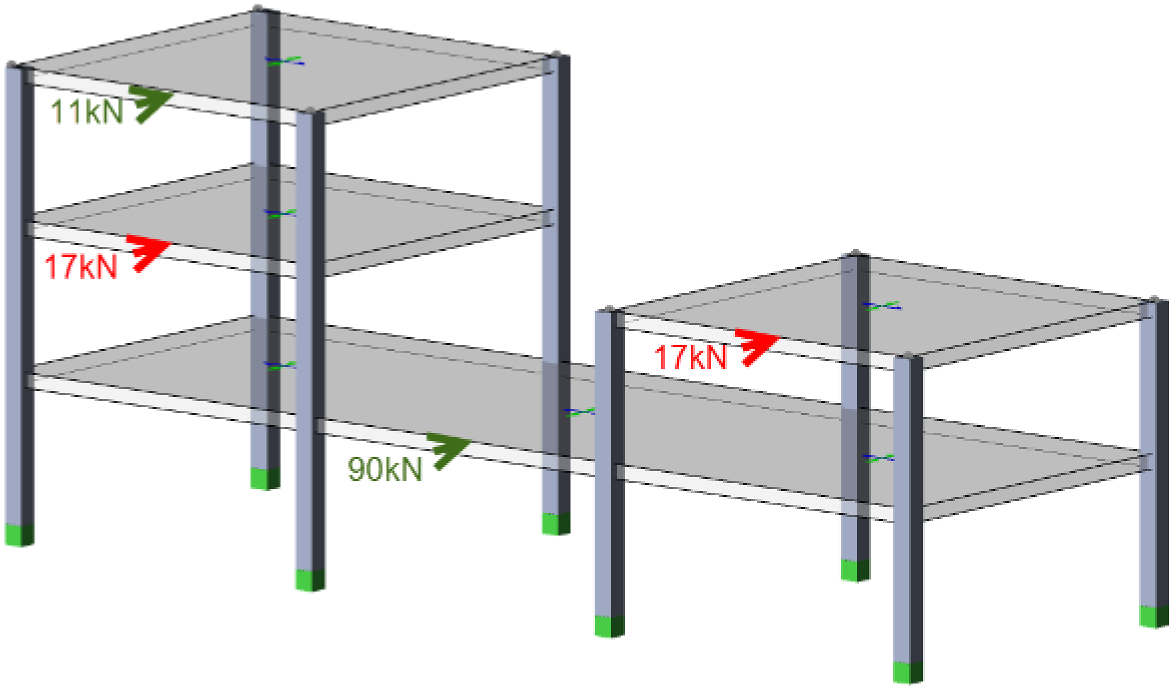


Irrespective of the method used to apply it, the area load within the building profile is shared between levels prior to decomposition.

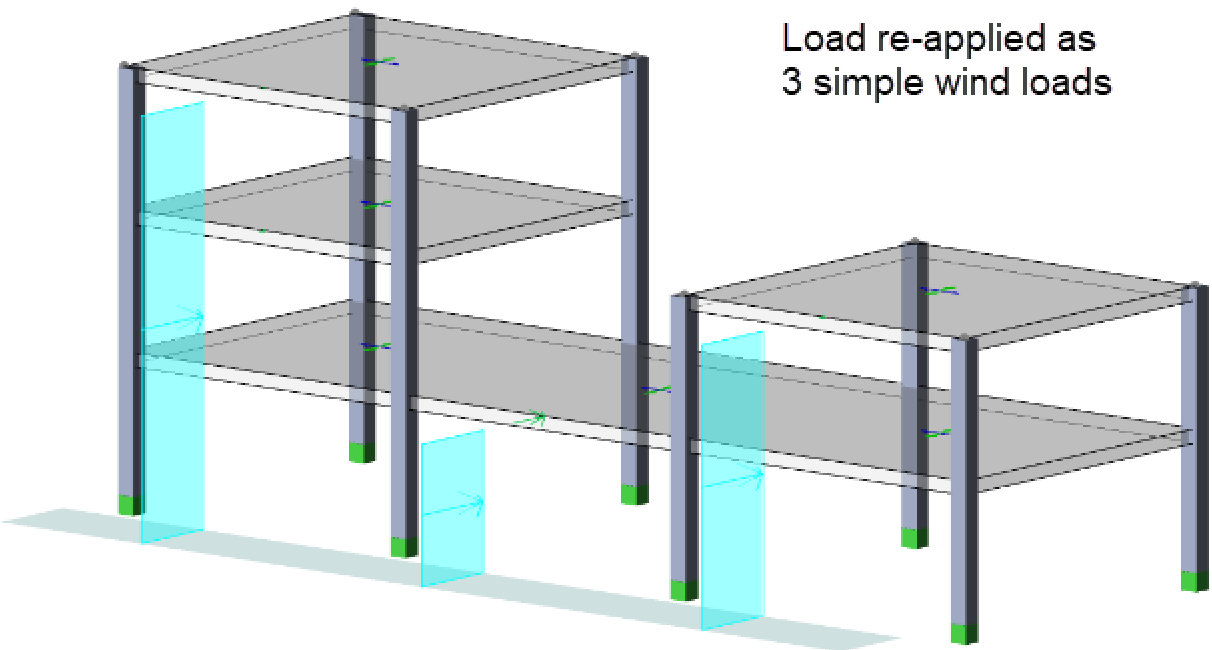


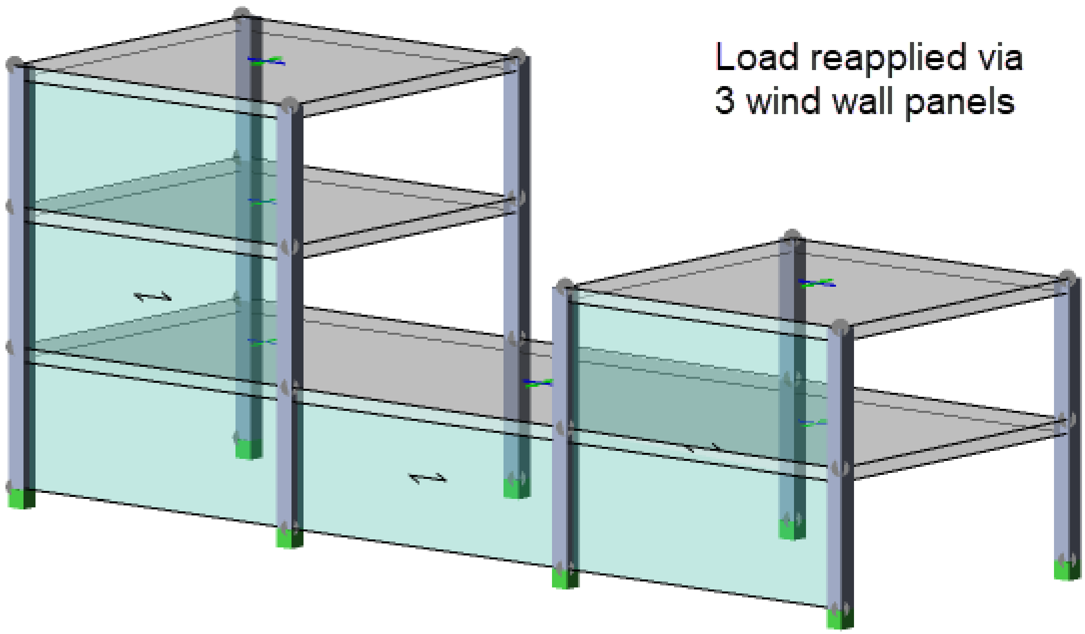
The load is then decomposed to the diaphragms at each level **in proportion to the width of each diaphragm**.

In this case because both diaphragms the second floor level are of equal width, the load is shared equally between them. This is not satisfactory as more of the load should have been applied to the left hand diaphragm in this case.

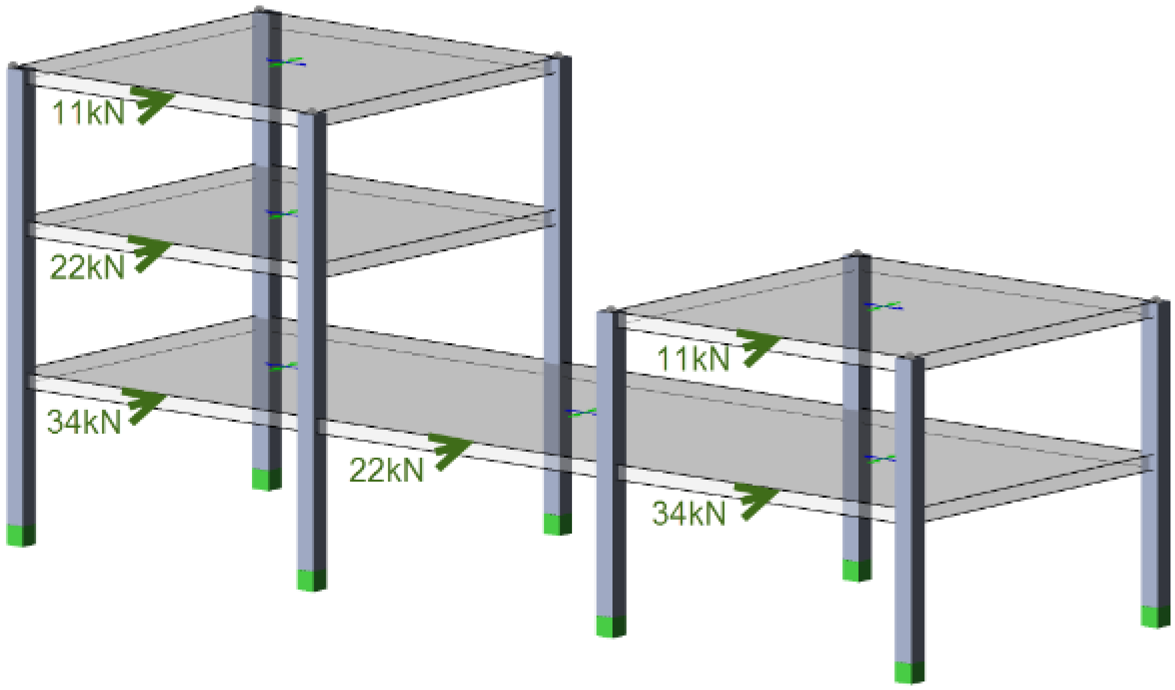


The workaround varies according to the method of loading, but basically involves replacing the original load with separate loads in each bay:



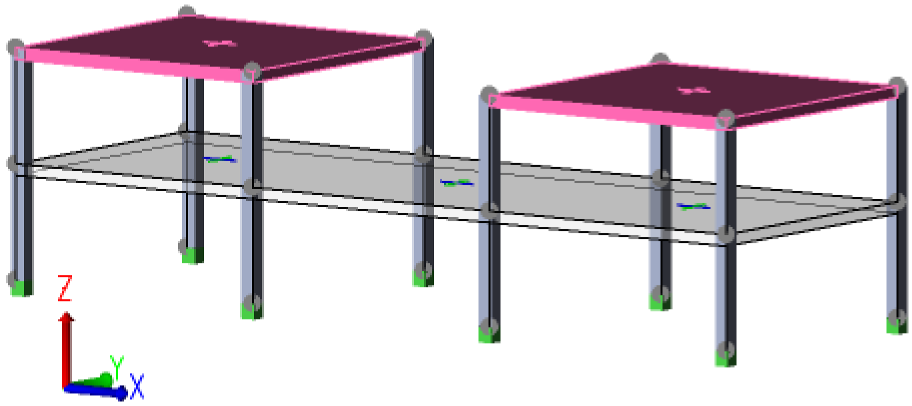


In both the above cases, the load is then decomposed as originally intended.

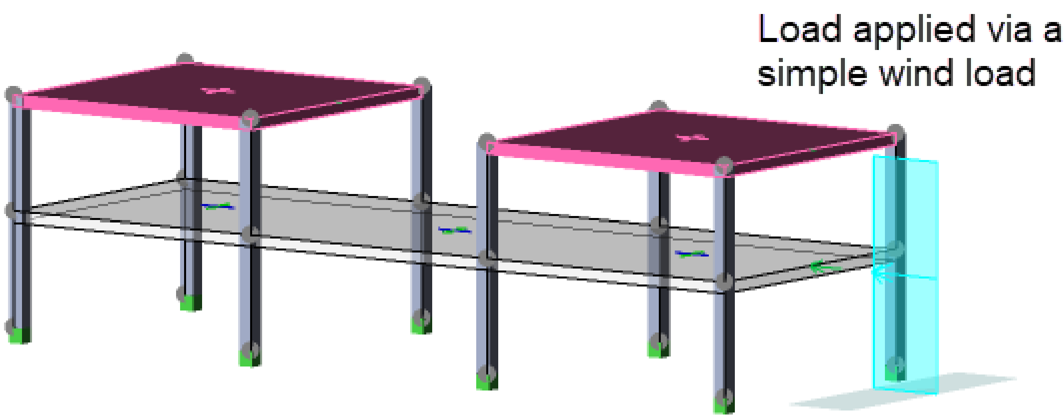


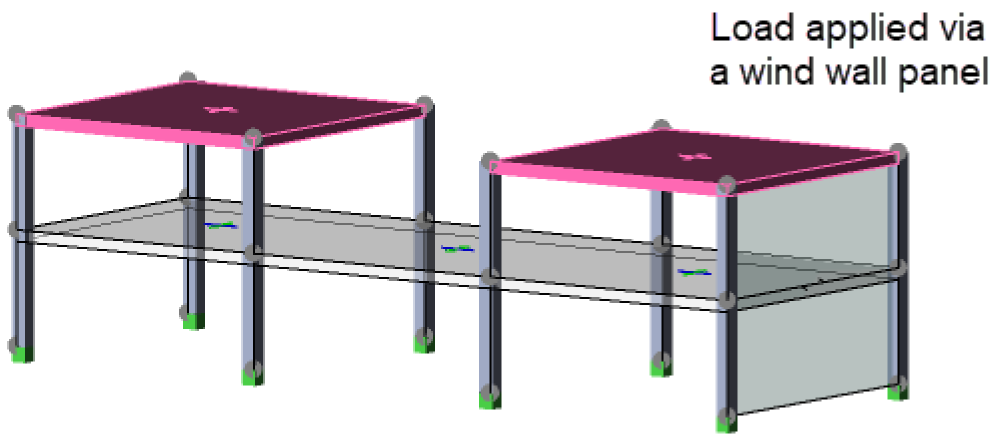
Wind load parallel to disconnected diaphragms

In this example although the two blocks are now the same height, another issue arises when the wind load is applied in the Global X direction, i.e. parallel to the disconnected diaphragms at the second floor level:

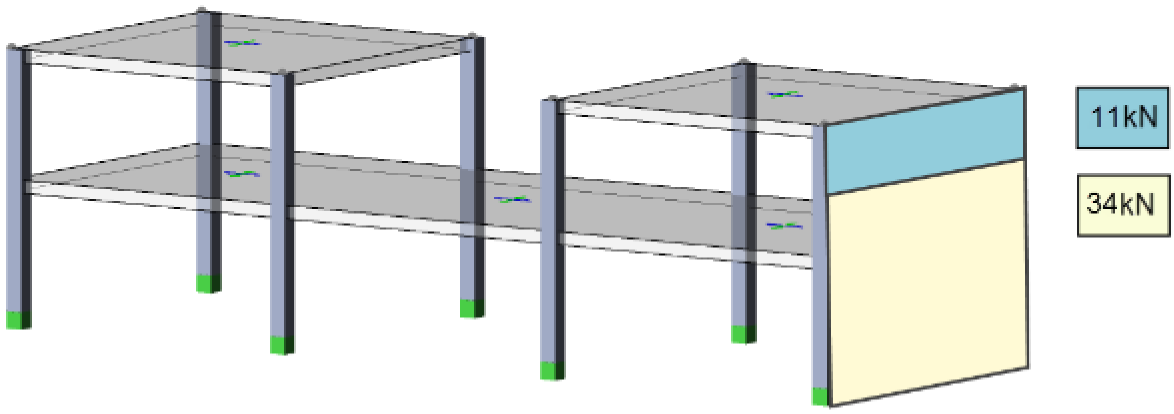


The issue arises because one diaphragm is ‘hidden’ from the applied load by the other diaphragm. The issue occurs irrespective of whether the load is applied via a Simple Wind load, or a wind wall panel:

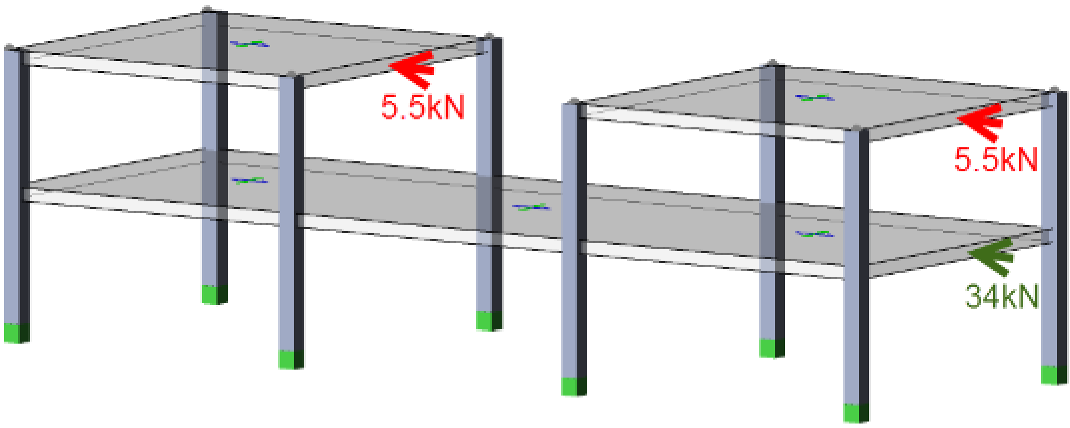




The area load within the building profile is first shared between levels prior to decomposition.



At the second floor level, instead of all the load being decomposed to the diaphragm facing the wind; because it is decomposed **in proportion to the width of each diaphragm at that level,** it ends up being shared equally to both.



To avoid the load being shared equally:

- if using wind wall panels - you would need to decompose to members or nodes instead of to diaphragms
- if using Simple Wind loads - there is no workaround, you would have to manually input the loads as diaphragm loads instead.

Wind tunnel testing and diaphragm loads

Tekla Structural Designer permits the easy flow of information required for wind tunnel testing of tall structures.

This includes:

1. The information out from Tekla Structural Designer to the wind specialists
2. The relevant results from wind specialists back into Tekla Structural Designer

Click the links below to find out more:

- Wind tunnel testing overview (page 1130)
- Exporting wind tunnel data workflow (page 1131)
- Using imported wind tunnel information (page 1133)

Wind tunnel testing overview

There are four steps required in the process of using Tekla Structural Designer design within the process of wind tunnel testing.

1. Tekla Structural Designer creates a “Tekla Structural Designer - Wind Tunnel Report” from a vibration analysis to be used by the wind specialists.

   See: Exporting wind tunnel data workflow (page 1131)

2. The “Tekla Structural Designer - Wind Tunnel Report” is sent to the wind specialists who undertake the wind tunnel tests. At the end of the tests, they create a “Wind Tunnel - Load Report” which they send back to the structural designer.

3. The designer manipulates the “Wind Tunnel - Load Report” to generate 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/level.

   See: Using imported wind tunnel information (page 1133)

4. The 24 wind loadcases are combined in regular combinations to be included as part of the design process in Tekla Structural Designer

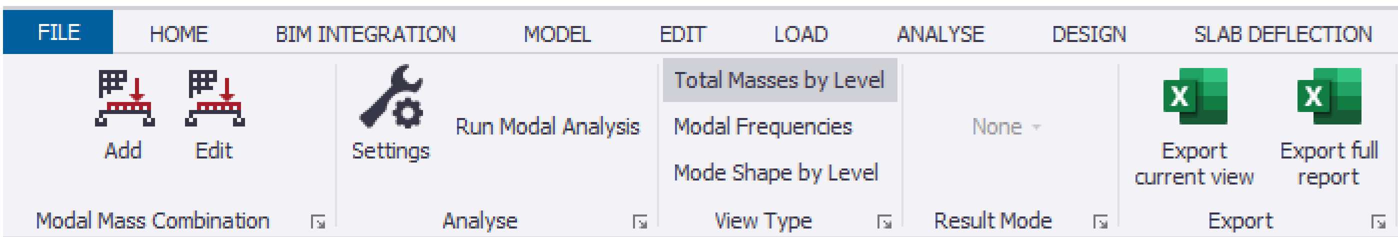
Exporting wind tunnel data workflow

Once the model has been created and had loads applied in Tekla Structural Designer, the basic workflow for generating and exporting wind tunnel data is described below.

Display the Wind Tunnel Data ribbon

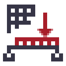
The workflow is initiated by clicking **Wind Tunnel** on the Analyze ribbon.

- This enables the **Wind Tunnel Data** ribbon below:



- You can work through this from left to right...

Add modal mass combinations

1. On the **Wind Tunnel Data** tab, click  **Add**.

   The Loading dialog (page 531) opens on the **Combinations** page with a new **Modal Mass** class of combination added in the left hand pane.

2. Rename the combination as required.
3. Select the load cases that you want to add in the combination, and click **>>**
4. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
5. Click **OK** to save the mass combination.

   Tekla Structural Designer adds the mass combination to the list of combinations in the **Loading** list.

Review the modal analysis settings

1. On the **Wind Tunnel Data** tab, click **Settings**

   The **Analysis Settings** dialog opens on the 1st Order Modal (page 2281) page.

2. Select the total number of modes to be calculated.
3. Review and modify the other analysis settings according to your needs.

Run a modal analysis

1. On the **Wind Tunnel Data** tab, click **Run Modal Analysis**.

   Tekla Structural Designer analyzes the model.

Review the results

After modal analysis the following results are available and can be reviewed as tabular data:

- Masses, Center of Mass, and Mass Moment of Inertia by level,
- Modal Frequencies,
- Mode shape by level

Mode shapes can also be displayed graphically (page 691) if required by switching to a **Results View**.

Export the report

To export all the tabular results that can be viewed in the **Wind Tunnel Data View**, click **Export full report**.

Alternatively, you can export just the tabular results that are displayed in the current view by clicking **Export current view**.

Using imported wind tunnel information

The Tekla Structural Designer workflow for importing and using the wind tunnel information provided by wind specialists is as follows:

Create loadcases for the diaphragm loads

The wind tunnel report typically contains 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/ level.

Before the loads can be imported from the report, the engineer has to manually create wind loadcases to hold them.

Apply the diaphragm loads in each loadcase

Diaphragm loads can either be pasted into Tekla Structural Designer or entered manually.

NOTE Diaphragm loads can only be pasted at floors/levels with one or more horizontal rigid or semi rigid diaphragm.

See: Diaphragm loads and diaphragm load tables (page 546)

Create load combinations

Load combinations are created manually. The Wind Tunnel loadcases behave as any other "Wind" type loadcase when being used in combinations.

Analysis and design

A standard analysis and design process is followed, in which:

- the graphical analysis results for wind tunnel loadcases and combinations being shown in exactly the same way as for any other wind type loadcases and combination,
- wind tunnel loadcases are used in drift checks in the same way as other wind type loadcases,
- wind tunnel loadcases are used in design combinations in the same way as other wind type loadcases.

13.2 Stability and imperfections handbook

This handbook introduces you to the following topics:

- The four sources of stability requirements that may need to be considered, see Overview of stability requirements (page 1134).

- Choosing the correct analysis type for your model and head code, see Allowing for global second-order effects (page 1138).
- Catering for global imperfections, see Allowing for global imperfections (page 1149).
- Catering for member imperfections, see Allowing for member imperfections (page 1151).

Overview of stability requirements

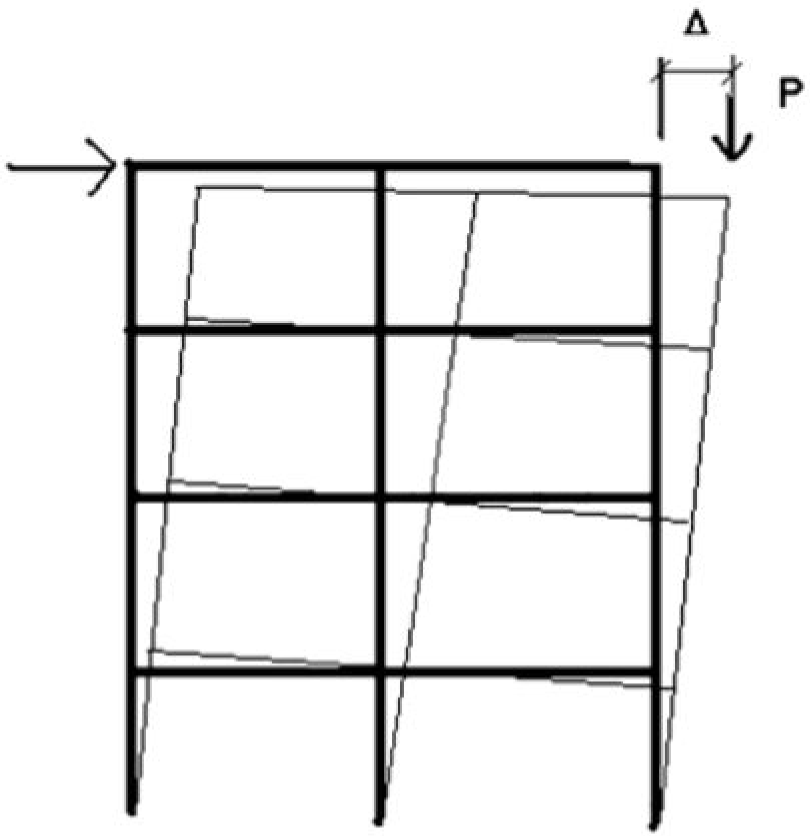
The analysis and design process has to allow for the differences between a theoretical mathematical model of a building and a more realistic representation. For example, buildings are not truly vertical when first built nor do they remain so when subject to load. These are called stability requirements and are from four sources:

1. **Global second-order (P-Δ) effects** to allow for deformation of the structure under load,
2. **Member second-order (P-δ) effects** to allow for deformation of the members under load,
3. **Global imperfections** due to the structure not being built plumb and square,
4. **Member imperfections** due to initial lack of straightness of the member.

There are various methods of allowing for each of these and they can be different for steel and concrete. There is also some variation based on country code.

Global second-order (P-Δ) effects

If gravity loads are applied to the deflected shape of a structure the load P applied at eccentricity Δ generates additional forces.



Provided the deflection is small:

- Structure is 'Non-sway'
- Second order effects can be ignored.

At some level these effects are no longer ignorable:

- Structure is 'Sway sensitive'
- And you have to do something to account for the second order effects.

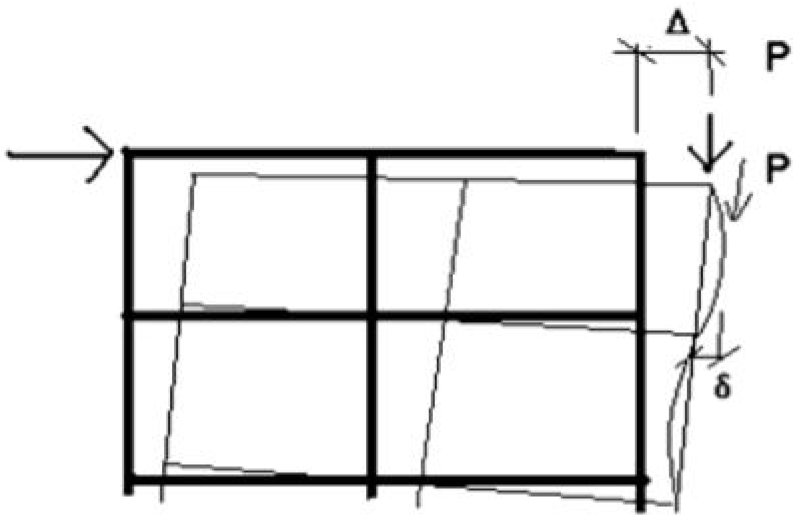
NOTE The terminology: 'Non-sway' and 'Sway sensitive' can vary between codes.

Related concept

Allowing for global second-order effects (page 1138)

Member second-order (P-δ) effects

Under load members will deform between their ends:



- Member curvature introduces a displacement δ between the member ends.
- The member axial loads applied at eccentricity δ generates additional forces.

In concrete structures:

- Where deflections are small :
  - Member is 'Short' or 'Stocky' or 'Non-Slender'
  - member second order effects are considered ignorable
- At some level they are no longer ignorable:
  - Member is 'Slender'
  - effects must be catered for in the design calculations

In steel structures: these effects are intrinsically allowed for in the design equations.

NOTE The terminology: 'Short' , 'Stocky' and 'Non-slender' can vary between codes.

When must global and member second order effects be considered?

Depending on the building's overall sway classification and each member's slenderness, global and member effects must be considered as follows:

Member Effects	Global	Effects

	Non-sway	Sway
Short Member	A	C
Slender Member	B	D

A - All second order effects can be ignored

B - Global effects can be ignored - member effects must be considered

C - Global effects must be considered - member effects can be ignored

D - Global effects must be considered - member effects must be considered

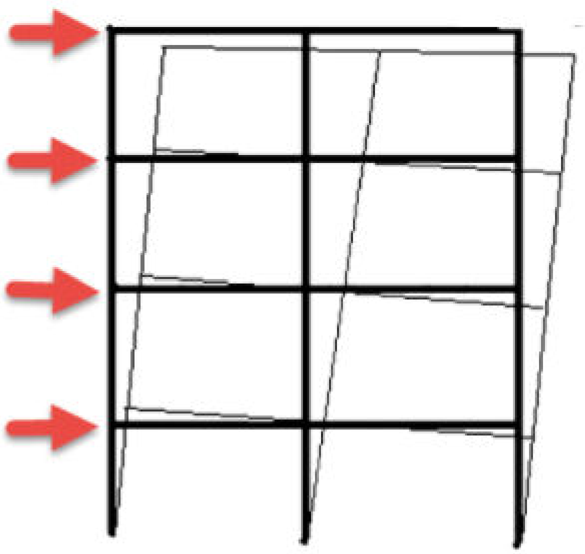
NOTE REMEMBER - every structure and every member has 2 directions!

Global imperfections

When the design code requires it you need to account for some degree of inclination (slope); typically in the range 0.2 to 0.5%

The codes allow you to cater for this in different ways:

- You could build multiple analysis models that are inclined
- You could have a single analysis model where you apply Equivalent Horizontal Forces [Eurocodes] / Notional Loads [AISC/ACI] / Notional Horizontal Forces [AUS/BS/IS] that will induce the same effect. Basically this means applying horizontal forces = 0.2 to 0.5% of the vertical forces in any combination.



In Tekla Structural Designer we use the second option.

NOTE Global Imperfections apply regardless of whether the structure is 'Non-sway' or 'Sway sensitive'.

Related concept

Allowing for global imperfections (page 1149)

Member imperfections

Member Imperfections apply regardless of whether members are slender or not. They are normally dealt with as part of the member design.

Related concept

Allowing for member imperfections (page 1151)

Allowing for global second-order effects

The type of analysis required to meet stability requirements and the checks performed will vary depending on the head code and material.

Choice of analysis type (ACI/AISC)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of two analysis types. These are,

- First-order analysis (not suitable for the final design of steel structures),
- Second-order analysis.

For steel structures: - Unless a number of specific criteria can be met, it is essential that your final design utilizes second-order analysis. However, second order analysis can be more sensitive to parts of your model that lack stiffness. For this reason it is recommended that you initially use first-order analysis to obtain sections and an overall building performance with which you are satisfied before switching to second-order analysis.

For concrete structures: - By choosing second-order analysis the global second-order effects are automatically catered for. However, if you determine that the structure is non-sway (see When should a concrete building be classed as non-sway? (page 1138)) you may instead opt for the design to be based on the first order analysis.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 1149).

When should a concrete building be classed as non-sway?

ACI 318-11 clause 10.10.5.2 states that a story can be considered as non-sway when the stability index Q is less than or equal to 0.05. (Basically, second order effects can be ignored below this value.)

Tekla Structural Designer does not report Q directly, but the Drift report does calculate an indicative stability coefficient (Δ_2/Δ_1) which is effectively the same as a moment magnifier.

From ACI Eq. (10-20):

Moment magnifier, δ_s = 1/(1-Q)

Equating this to the stability coefficient gives

$1/(1-Q) = (\Delta_2/\Delta_1)$

$1-Q = 1/(\Delta_2/\Delta_1)$

$Q = 1 - 1/(\Delta_2/\Delta_1)$

For $Q < 0.05$

$1 - 1/(\Delta_2/\Delta_1) < 0.05$

$(\Delta_2/\Delta_1) < 1/0.95$

$(\Delta_2/\Delta_1) < 1.0526$

Therefore, if the Drift report determines an indicative stability coefficient (Δ_2/Δ_1) less than 1.05 it can be assumed non-sway; whereas if greater than this value a second-order analysis is required.

NOTE The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 1149).

Choice of analysis type (BS)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - see: A practical approach to setting the analysis type (page 1141) below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (regular frameworks with $\lambda_{cr} > 4$).

If the amplified forces method is selected you must also indicate which formula to use for determining the amplification factor,

If the structure is clad and if the stiffening effect of cladding is not taken into account explicitly:

$k_{amp} = \lambda_{cr}/(1.15\lambda_{cr} - 1.5)$

If the structure is unclad or clad with a direct allowance made for the stiffening effect:

$k_{amp} = \lambda_{cr}/(\lambda_{cr} - 1)$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of λ_{cr} for all stacks in the building.

The k_{amp} results are summarised for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review toolbar.
2. Select 'k_{amp}' from the View Type drop list on the Review toolbar.
3. The k_{amp} results in both directions are tabulated for each column in the building.

Second-order analysis

Full second-order analysis is not restricted to regular frameworks, but requires $\lambda_{cr} > 2$.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 1149).

A practical approach to setting the analysis type

Unless λ_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to P-Δ analysis.

Note: If the rigorous second-order (P-Δ) analysis approach is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.
3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, λ_{cr} (See: How do I assess the worst elastic critical load factor... (page 1142))
4. If the λ_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If λ_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
    - Either, refine the design until λ_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces - see below),
    - Or, in order to use the amplified forces (k_{amp}) approach, refine the design further until λ_{cr} is greater than 4.0.
6. When a suitable λ_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate. (If the k_{amp} approach is selected you must also indicate which formula to use for determining the amplification factor, This will depend on whether the structure is clad or not and if the cladding is taken into account explicitly or not.)
7. With the analysis type still set to the full second-order, or the k_{amp} approach perform Design All (Static).

If you use the k_{amp} approach be aware that BS5950-1:2000 classes certain structures outside the scope of this method. e.g. tied portals, or structures containing tension only braces. For such structures, you would need to ensure that λ_{cr} is greater than 2.0, and use the full second-order analysis approach for the static design.

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review toolbar.
2. Select 'Sway' from the View Type drop list on the Review toolbar.
3. The elastic critical load factor in both directions ((λ_{Dir1} & (λ_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest λ value from all of the columns in either direction.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the λDir1 header until the columns are arranged in increasing order of λDir1, then repeat for λDir2.

In BS 5950-1:2000 a building can be considered as 'non-sway' when $\lambda_{cr} \geq 10$ else it is 'sway sensitive' and (global) second-order effects must be taken into account.

Note however that you are not restricted in your choice of analysis type irrespective of the value of λ_{cr} (it is your call, although we will warn you about it).

How is the elastic critical load factor calculated?

In order to determine whether a building is 'non-sway' or 'sway sensitive', Tekla Structural Designer calculates the elastic critical buckling load factor, λ_{cr}. It is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Notional Horizontal Forces (NHF) are determined. It uses 0.5% of the vertical load at the column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the NHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of λ_{cr} for that combination as follows,

λ_{cr} =h/(200 * δ_s)

Where

h= the storey height

δ_s =the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads.

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

Choice of analysis type (Eurocode)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - see: A practical approach to setting the analysis type (page 1144) below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (basically regular frameworks with α_{cr} > 3 - see Validity of the amplified forces method (page 1144) below). Full second-order analysis is preferred for steelwork and since it is not precluded by EC2 it can be used for concrete.

The amplified forces method is described differently in EC3 compared to EC2, whilst the presentations are different, they are both based on the amplifier, k_{amp} given as,

$k_{amp} = 1/(1 - 1/\alpha_{cr})$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of α_{cr} for all stacks in the building.

The k_{amp} results are summarised for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review toolbar.

2. Select 'k_{amp}' from the View Type drop list on the Review toolbar.
3. The k_{amp} results in both directions are tabulated for each column in the building.

NOTE The amplified forces method is not recommended for non-linear structures - a full second-order analysis should be performed instead.

Validity of the amplified forces method

EC3 Clause 5.2.2 (6)B lists limitations on the applicability of the Amp. Forces method. It is therefore your responsibility when selecting this method to ensure all of the following:

- all storeys have a similar distribution of vertical load
- all storeys have a similar distribution of horizontal load
- all storeys have a similar distribution of frame stiffness with respect to the applied storey shear forces

Also according to clause 5.2.1 (4)B limitation:

- roof slope shallow - not steeper than 1:2 (26 degs)
- axial compression in beams or rafters - N_{cr} / N_{ed} <= 11.1

Second-order analysis

Full second-order analysis is more widely applicable for steelwork structures and since it is not precluded by EC2 it can be used for concrete.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 1149)

A practical approach to setting the analysis type

Unless α_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to one of the P-Δ analysis methods.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.

3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, α_{cr} (See: How do I assess the worst elastic critical load factor... (page 1146))
4. If the α_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If α_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
   - Either, refine the design until α_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces),
   - Or, in order to use the amplified forces approach, refine the design further until α_{cr} is greater than 3.0.
6. When a suitable α_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate.
7. With the analysis type still set to the full second-order, or the amplified forces method, perform Design All (Static).

NOTE If full second-order analysis is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

NOTE If you use the 'Second-order analysis - Amp. forces method' be aware that EC3 classes certain structures outside of its scope (see Validity of the amplified forces method (page 1144)). Such structures would need to be refined during gravity sizing until the elastic critical load factor is at least greater than 2.0, so that the full second-order approach can be used for the full design.

When should a building be classed as sway sensitive?

Susceptibility to second order effects is a general characteristic and is not material specific, it has just been presented differently in EC3 and EC2:

- In EC3 a building can be considered as 'non-sway' when the elastic critical load factor $\alpha_{cr} \geq 10$, else the building is 'sway sensitive' and (global) second-order effects must be taken into account.
- In EC2 the definition is slightly different - it does not use the terms 'non-sway' and 'sway sensitive'. Rather it simply defines when second-order effects are small enough to be ignored. The principle is given in Clause 5.8.2 (6) which states that they can be ignored if they are less than 10% of the corresponding first order effects. Because of the way in which the amplification factor, k_{amp} is calculated the change point is at an α_{cr} of 11

not 10. (see: Derivation of the kamp formula for concrete structures (page 1147)).

However, the intent is the same in both cases and so in Tekla Structural Designer $\alpha_{cr} \geq 10$ is taken as the change point. In any event, you are not restricted in your choice of analysis type irrespective of the value of α_{cr} (it is your call, although we will warn you about it)

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review toolbar.
2. Select 'Sway' from the View Type drop list on the Review toolbar.
3. The elastic critical load factor in both directions (α_{Dir1} & α_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest elastic critical load factor from all of the columns in either direction - this is the α_{cr} value for your building.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the αDir1 header until the columns are arranged in increasing order of αDir1, then repeat for αDir2.

Having determined an α_{cr} for your building, you then use it when deciding which is the most appropriate analysis type for design.

How is the elastic critical load factor calculated?

The elastic critical load factor, α_{cr} is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Equivalent Horizontal Forces (EHF) are determined. These consist of 0.5% of the vertical load at each column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the EHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of α_{cr} for that combination as follows,

$\alpha_{cr} = h/(200 * \delta_{EHF})$

Where

h = the storey height

δ_{EHF} = the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

Derivation of the k_{amp} formula for concrete structures

EC2 provides two specific approaches to determine the change point below which second-order effects are small enough to be ignored:

The first specific approach is contained in Clause 5.8.3.3 which provides a pass/fail criterion to check whether the global second-order effects may be ignored. It is given as,

$F_{VEd} = k_1 * n_s/(n_s + 1.6) * S(E_{cd} * I_c)/L^2$

where

F_{VEd} = the total vertical load (on 'braced' and 'bracing' members)

k_1 = a factor that allows for cracking in the concrete of the LLRS and is a Nationally Determined Parameter (NDP)

* n_s = number of storeys

E_{cd} = the design value of the modulus of elasticity of the concrete

I_c = the second moment of area of the uncracked bracing members

L = the total height of the building

However, the above approach has a number of restrictions in its application and as a result it is not applied in Tekla Structural Designer.

The second specific approach is given in Annex H.

The method given in Annex H.1.2 is the background for the more limited method given in Clause 5.8.3.3 as described above, but it does not apply where there is significant shear deformation in the LLRS e.g. for shear walls with significant openings, hence again it is not considered in Tekla Structural Designer.

Instead, recourse is made to determining the level of second-order effect using Annex H.2. Using this approach, by rearranging Equation H.8 it is possible to provide a 'stability coefficient' $1/\alpha_{cr}$ which can be applied as the change point between non-sway and sway sensitive structures.

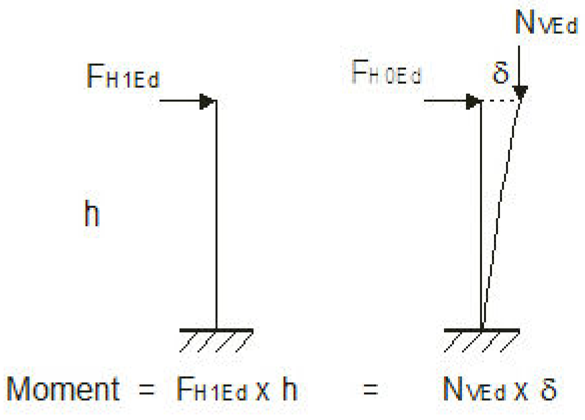
$F_{HEd} = F_{H0Ed} /(1-F_{H1Ed}/F_{H0Ed})$ Equation H.8

Where:

F_{H1Ed} = fictitious horizontal force, giving the same bending moments as vertical load N_{VEd} acting on the deformed structure, with deformation caused by F_{H0Ed}

(first order deformation), and calculated with nominal stiffness values according to 5.8.7.2

Considering how this definition of F_{H1Ed} might apply to an imaginary cantilever of height, h, we arrive at:



1. The moment due to F_{H1Ed} is the same as that due to the vertical load N_{VEd}, so:

$F_{H1Ed} * h = N_{VEd} * \delta$

which can be rearranged to:

$F_{H1Ed} = (N_{VEd} * \delta)/h$

2. Substituting for F_{H1Ed} in Equation H.8, we have:

$F_{HEd} = F_{H0Ed} /(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

3. By defining $k_{amp} = F_{HEd}/F_{H0Ed}$ the above can be rearranged to:

$k_{amp} = 1/(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

4. Now, the EC3 Equation 5.2 for the elastic critical buckling load is:

$\alpha_{cr} = H_{Ed}/V_{Ed} * h/\delta_{HEd}$

which, when re-expressed in the terminology used in H.2 becomes:

$\alpha_{cr} = F_{H0Ed} /N_{VEd} * h/\delta_{HEd}$

and when further rearranged becomes:

$1/\alpha_{cr} = (N_{VEd} *\delta_{HEd})/(F_{H0Ed} * h)$

5. Hence $1/\alpha_{cr}$ can be substituted into the above equation for k_{amp} so that we arrive at the more well-known formula for amplification:

$k_{amp} = 1/(1 - 1/\alpha_{cr})$

NOTE Strictly, the watershed for concrete structures should be at a k_{amp} factor of 1.1 (amplification of no more than 10% due to second-order effects). Setting k_{amp} to be 1.1 and rearranging gives $\alpha_{cr} \geq 11$ i.e. a stability coefficient ≤ 0.0909 not 0.1

NOTE It is important to note that the resulting values of α_{cr} and k_{amp} are very dependent upon the analysis properties that are used and the you therefore need to carefully consider the modification factors you choose to apply via the Analysis Options.

Use of modification factors

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. Consequently design codes can require that analysis stiffness adjustment factors are applied (as the appropriate properties to use in analysis are load and time dependent).

These modification factors can be applied for each of the different materials from the Modification Factors page of the Analysis Options dialog. (which is located on the Analyse toolbar).

For non-concrete members it is also possible that you will want to apply an adjustment to material properties for various other investigations. One suggested example is the assessment of structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

Allowing for global imperfections

These are typically represented by the application of Notional Loads \ Equivalent Horizontal Forces.

Allowing for global imperfections (ACI/AISC)

Steel code

Steelwork design to AISC 360-10 requires an allowance for global imperfections. Columns are assumed to be out of plumb by some amount and this is replicated by applying Notional Loads, (NL), in the analysis. The requirements are given in Clause C2.2b and the value of NL is given as 0.2%. However, to accommodate the requirements of Clause C2.3 an additional 0.1% has been included - so that the actual NL used is 0.3%.

Concrete code

For concrete design to ACI 318-11 there are no requirements. However, given that a building can use mixed materials, (and even for an entirely concrete building), you have the choice to include the NLs via the combinations. By default these are to the AISC requirements - i.e. 0.3%

Allowing for global imperfections (Eurocode)

The formula to calculate the global imperfections (using EHFs) is the same for both steel and concrete, see : EC2 Cl 5.3.2 (3) a) and EC3 Cl 5.2 (5)

$\varphi = \varphi_0\ \alpha_h\ \alpha_m$

Where:

φ_0 is the basic value of inclination.

α_h is the reduction factor for length or height : $\alpha_h = 2/\sqrt{h}$; $2/3 \leq \alpha_h \leq 1$

h is the length or height of the structure.

When height = 9m the maximum reduction of 2/3 will apply.

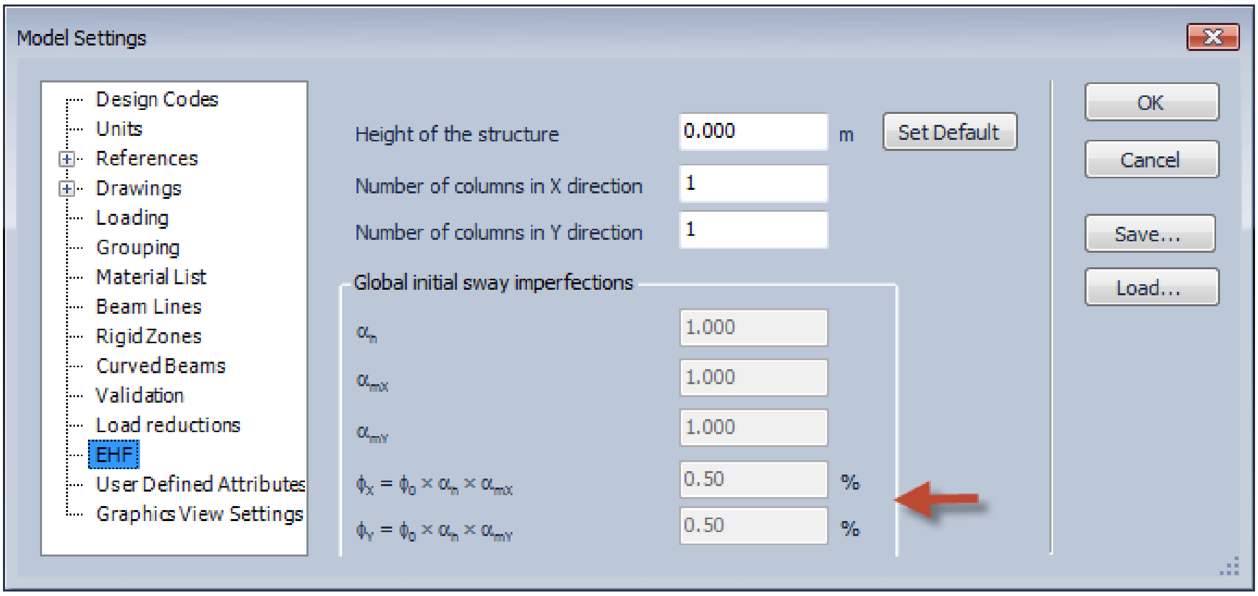
α_m is the reduction factor for number of members (EC2) or columns in a row (EC3), : $\alpha_m = \sqrt{(0.5(1+1/m))}$

m is the number of vertical members contributing to the total effect (EC2), or is the number of columns in a row including only those columns which carry a vertical load not less than 50% of the average value of the column in the plane considered (EC3).

m =	1	5	10	1000
α_m =	1	0.775	0.742	0.707

Guidance on how to count 'm' is vague and varied - however as demonstrated above, once you get above 5 or 10 it starts to make very little difference.

In Tekla Structural Designer the allowance is applied in the same manner for steel and concrete - being controlled in Model Settings as shown below:



It does require some user intervention to provide structure height and number of columns to consider. These user inputs cause adjustment of the default value of imperfection of 0.5% of the vertical load and this can be a different adjustment in the two orthogonal directions (Direction 1 and Direction 2). For example, the adjustment factor might give an EHF of 0.4% in the X-direction and 0.3% in the Y-direction.

Allowing for global imperfections (BS)

Steel code

BS5950 requires up to 0.5% of vertical loading.

Concrete code

BS8110 does not have a requirement (minimum lateral load is not the same thing).

Allowing for member imperfections

Allowing for member imperfections (ACI/AISC)

Steel code

AISC indicates that these are inherently catered for in the strut design.

Concrete code

For concrete design to ACI 318-11 there are no requirements.

Allowing for member imperfections (Eurocode)

Steel code

For steel structures designed to EC3, member imperfections are intrinsically included in the design routines for all members (beams, columns, braces). Apart from one explicit requirement, carrying out the design is all that is necessary.

The explicit requirement is from Clause 5.3.2 (6) in which member imperfection should be included as part of the analysis when the frame is sway sensitive and the axial force in members with moment connections is above a certain limit. If this situation arises Tekla Structural Designer issues a warning.

Concrete code

For concrete structures designed to EC2, explicit calculations which consider an imperfection effect are carried out as part of the design.

In Tekla Structural Designer this is achieved by adding an additional moment to the analysis results before starting design.

Allowing for member imperfections (BS)

Steel code

BS5950 inherently caters for member imperfections in the strut design curves.

Concrete code

BS8110 does not have a requirement (minimum eccentricity is not the same thing).

The sway check

NOTE The sway check is not relevant to models that use the ACI/AISC head code.

By default the sway check is applied to all columns (of all materials), and all walls.

Sway check design options

In **Design Settings** > **Sway & Drift Checks**, there is a Merge short stacks option (default Off) that applies to the check as follows:

- With the setting Off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting On, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged

stack length. The check is not performed on single stack columns/walls that are less than this limit

Manually exclude an entire column or wall or an individual column stack or wall panel from the check

An entire column/wall can be excluded from the check by unselecting **Sway/Seismic drift checks** located under All stacks/panels>Sway and drift checks in the column/wall properties window.

Similarly an individual stack/panel can be excluded from the check by unselecting **Sway/Seismic drift checks** located under Sway and drift checks for the specific stack/panel in the column/wall properties window.

Manually adjust automatically determined stack lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

In order to do this you would have to select Merge with stack below located under Sway and Drift Checks for the specific stack/panel in the column/wall properties window.

Perform sway checks

The checks are performed either by running Analyse All (Static) from the Analyse ribbon, or by running any of the Design (Gravity), or Design (Static) commands from the Design ribbon.

NOTE For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

For those stacks to which the check has been applied, the storey drift in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated, and from this the elastic critical load factor is determined.

Review sway checks

For those stacks to which the check has been applied the elastic critical load factor is calculated in both directions, and the building's overall sway check status is displayed in the Design branch of the Status Tree in the Project Workspace.

- Any column in this branch with a warning status can be double clicked on in order to highlight it in amber in the current view.
- Double click on the Sway main heading in order to highlight all columns with warnings in amber

NOTE Sway warnings remain highlighted in the current view until you press Esc to clear the highlight.

In addition to viewing the summary in the Project Workspace, the sway checks can be interrogated in a variety of ways:

- To display sway values on the structure, see Display sway drift and story shear (page 676)
- To review/modify the checks graphically, see Review and modify sway checks (page 893)
- To review the detailed checks in a table, see Review sway check tabular results (page 900)

Report sway checks

If required, the sway results table can be included in printed output by adding the Analysis>Sway chapter to your model report.

The drift check

NOTE The drift check is only relevant to models that use the ACI/AISC head code.

By default the drift check is applied to all columns (of all materials), and all walls.

Drift check design options

In **Design Settings** > **Drift Checks**, there is a Merge short stacks option (default Off) that applies to the check as follows:

- With the setting Off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting On, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged stack length. The check is not performed on single stack columns/walls that are less than this limit.

Manually exclude an entire column or wall or an individual column stack or wall panel from the check

An entire column/wall can be excluded from the check by unselecting **Drift/ Seismic drift checks** located under All stacks/panels>Drift Checks in the column/wall properties window.

Similarly an individual stack/panel can be excluded from the check by unselecting **Drift/Seismic drift checks** located under Drift Checks for the specific stack/panel in the column/wall properties window.

Manually adjust automatically determined stack lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/ panel. The length of the combined stack/panel is then used in the checks.

In order to do this you would have to select Merge with stack below located under Drift Checks for the specific stack/panel in the column/wall properties window.

Perform drift checks

The checks are performed either by running Analyze All (Static) from the Analyze ribbon, or by running any of the Design (Gravity), or Design (Static) commands from the Design ribbon.

NOTE For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

For those stacks to which the check has been applied, the storey drift in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated, and from this the stability coefficient is determined.

Review drift checks

The building's overall drift check status is then displayed in the Design branch of the Status Tree in the Project Workspace.

- Any column in this branch with a warning status can be double clicked on in order to highlight it in amber in the current view.
- Double click on the Drift main heading in order to highlight all columns with warnings in amber

NOTE Drift warnings remain highlighted in the current view until you press Esc to clear the highlight.

In addition to viewing the summary in the Project Workspace, the drift checks can be interrogated in a couple of ways:

- To review/modify the checks graphically, see Review and modify drift checks (page 879)
- To review the detailed checks in a Review Data table, see Review drift check tabular results (page 901)

In the Review Data table, for those stacks to which the check has been applied, stability coefficients in both directions (RatioDir1,RatioDir2) are calculated as:

Stability coefficient = Δ_2/Δ_1

Where

Δ_2= second order drift

Δ_1= first order drift

A Warning status is displayed against any stack in which the ratio exceeds 1.71 and a Beyond Scope status is applied if it exceeds 2.85 (as this indicates the structure lacks stiffness in whole or in part).

TIP If there are a lot of columns in the building - in order to quickly determine the biggest coefficient in each direction, simply click the RatioDir1 header until the columns are arranged in increasing order, then repeat for RatioDir2.

Stability coefficients can only be calculated accurately by performing a second order analysis, however indicative values can still be displayed when only a first order analysis has been performed. These are determined from the column's elastic critical load factor, λ:

$(\Delta_2/\Delta_1) = 1/(1- 1/\lambda)$

Approximate second order drift:

$\Delta_2= \Delta_1/(1- 1/\lambda)$

Report drift checks

If required, the drift results table can be included in printed output by adding the Analysis>Drift chapter to your model report.

The wind drift check

The Wind Drift check is performed during the structure static design and also when any 3D analysis is run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are considered in the drift checks, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

The check is performed for (wind) combinations using the combination SLS (Service Level) factors - (which can be < 1.0).

By default the check is applied to all columns (of all materials) and all walls.

Wind drift check design options

In **Design Settings** > **Sway & Drift Checks**, there is a 'Check wind cases only' option (default On) that applies to the check as follows:

- With the setting On, the Wind Drift check only considers the effects of the wind load case(s) in wind combinations.
- With the setting Off, the check considers the effects of all load cases in wind combinations (which would include drift induced by gravity loads).

On the same page of the Design Options there is also a Merge short stacks option (default Off) that applies to the check as follows:

- With the setting Off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting On, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged stack length. The check is not performed on single stack columns/walls that are less than this limit

Manually exclude an entire column or wall or an individual column stack or wall panel from the check

An entire column/wall can be excluded from the check by unselecting **Wind Drift check** located under All stacks/panels>Sway and Drift Checks in the column/wall properties window.

Similarly an individual stack/panel can be excluded from the check by unselecting **Wind Drift check** located under Sway and Drift Checks for the specific stack/panel in the column/wall properties window.

Manually adjust the automatically determined stack lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

In order to do this you would have to select Merge with stack below located under Sway and Drift Checks for the specific stack/panel in the column/wall properties window.

Wind drift calculations

For those stacks to which the check has been applied, the lateral drift in each direction (i.e. the difference between top and bottom deflection of the stack) is determined for each wind load case and wind service combination. This drift is then compared against a user-defined limit (the default is 1/300 of the story height, in line with Eurocode 3 recommendations, but you are free to specify a limit of your choice). Different limits can be applied to different stacks if required.

The checks are performed using results from a 1st order linear analysis (with no Reduced stiffness factor), which are generated by running any of the Design (Gravity), or Design (Static) commands from the Design ribbon.

NOTE For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

Review wind drift checks

Any stack failures are flagged in the Design - Wind Drift branch of the Status Tree in the Project Workspace.

- Double clicking on a failing column in this list causes it to be highlighted in red in the current view.
- Double clicking on the Wind Drift heading at the top of the list causes all failing columns to be highlighted in red

---

NOTE Wind drift failures remain highlighted in the current view until you press Esc to clear the highlight.

In addition to viewing the summary in the Project Workspace, the wind drift checks can be interrogated in a variety of ways:

- To display wind drift values on the structure, see Display sway drift and story shear (page 676)
- To review/modify the checks graphically, see Review and modify wind drift checks (page 897)
- To review the detailed checks in a table, see Review wind drift check tabular results (page 902)

Report wind drift checks

If required, the wind drift results table can be included in printed output by adding the Analysis> Wind Drift chapter to your model report.

NOTE The Wind Drift chapter has a Settings option that can be used to dramatically reduce the number of report pages. By default all columns and walls are output, but you can specify instead the most critical 'XX' columns and walls, (where 'XX' is a number of your choice).

The same Settings option also allows you to choose between reporting only the critcal stack/all stacks, and reporting only the critical combination/all combinations.

The seismic drift check

Seismic drift is assessed on a floor to floor horizontal deflection basis and there are limits for acceptability of a structure.

The building's overall seismic drift status is displayed in the Design branch of the Status Tree in the Project Workspace.

Full details for all columns are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the Analysis>Seismic Drift chapter to your model report.

Overall displacement

By expanding the Design branch of the Project Workspace Status Tree, you are able to review the maximum positive and negative overall displacement results from the 3D Analysis for both Strength and Service combinations.

Displacements can also be viewed graphically in the Results View by using the buttons on the Deflections group.

13.3 Static analysis and design handbook

NOTE The aim of this handbook is describe the processes that occur when running one of the combined analysis and design commands, i.e:

- Design Steel (Gravity), Design Concrete (Gravity), Design All (Gravity)
- Design Steel (Static), Design Concrete (Static), Design All (Static)

You can find the following information in this handbook:

Overview of the combined analysis and design processes

Overview of Design Steel (Gravity)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis. **NOTE** For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/or produce unrealistic forces. This is specified via Design Options> General.

Step	Process	Description
4	Member design stage (page 1182)	Steel members are designed or checked for active gravity combinations only. **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.

Overview of Design Steel (Static)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis.
4	Member design stage (page 1182)	Steel members are designed or checked for all active static combinations.
5	Stability checks	Sway (page 1152)/Drift (page 1154) checks and Wind Drift (page 1156) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design Concrete (Gravity)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis. **NOTE** For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/or produce unrealistic forces. This is specified via Design Options> General.
4	FE chasedown analysis (page 1174)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 1174)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.

Step	Process	Description
6	Member design stage (page 1182)	Concrete members and concrete walls are designed or checked for active gravity combinations only. **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.

Overview of Design Concrete (Static)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis. **NOTE** For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/ or produce unrealistic forces. This is specified via Design Options> General.
4	FE chasedown analysis (page 1174)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.

Step	Process	Description
5	Grillage chasedown analysis (page 1174)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 1182)	Concrete members and concrete walls are designed or checked for active gravity combinations only.
7	Stability checks	Sway (page 1152)/Drift (page 1154) checks and Wind Drift (page 1156) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design All (Gravity)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model, with an option to mesh floors.
4	FE chasedown analysis (page 1174)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 1174)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D

Step	Process	Description
		Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 1182)	All steel and concrete members and all concrete walls are designed or checked for all active gravity combinations. **NOTE** Concrete slab design is not performed **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.
7	Stability checks	Sway (page 1152)/Drift (page 1154) checks and Wind Drift (page 1156) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design All (Static)

Step	Process	Description
1	Model validation (page 512)	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 1166)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 1173)	A traditional frame analysis of the entire 3D model, with an option to mesh floors.
4	FE chasedown analysis (page 1174)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 1174)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at

Step	Process	Description
		those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 1182)	All steel and concrete members and all concrete walls are designed or checked for all active static combinations. **NOTE** Concrete slab design is not performed
7	Stability checks	Sway (page 1152)/Drift (page 1154) checks and Wind Drift (page 1156) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

3D pre analysis processes

Pre-Analysis consists of a number of processes, such as:

- Decomposing slab and wall loads
- Preparing loadcases and combinations
- Meshing and diaphragms
- First-order gravity analysis
- Resolving vertical loads for application of global imperfections
- Generation of pattern loading

The actual pre-analysis processes performed will vary depending on the specific model that has been defined.

Overview of slab load decomposition

Decomposition of slab loads on to supporting members is automatically performed where necessary during pre-analysis.

Decomposition is not just performed for beam and slab models, the program may also need to decompose flat slab loads onto supporting columns and walls.

NOTE In Tekla Structural Designer the term "Decomposed Loads" refers to the loading on beams, columns, and walls that comes from slabs

Whether load decomposition is performed or not will depend on the analysis model, the slab properties and the **Mesh 2-way slabs in 3D Analysis** setting as follows:

Decomposition method specified in Slab properties	3D Analysis and Grillage chasedown models	FE chasedown model
Two-way	**Mesh 2-way slabs in 3D Analysis** option **not** selected (default): • loads on two-way slabs decomposed prior to analysis	• it is not necessary to decompose the loads on two-way slabs prior to analysis
	Mesh 2-way slabs in 3D Analysis option selected: • it is not necessary to decompose the loads on two-way slabs prior to analysis	
One-way	• loads on two-way slabs decomposed prior to analysis	• loads on two-way slabs decomposed prior to analysis

Potential Load Decomposition Methods

Traditionally a "tributary area" (sometimes called "yield line") loading approach would have been adopted to determine the decomposed loads, but this has limitations when dealing with complex geometry such as:

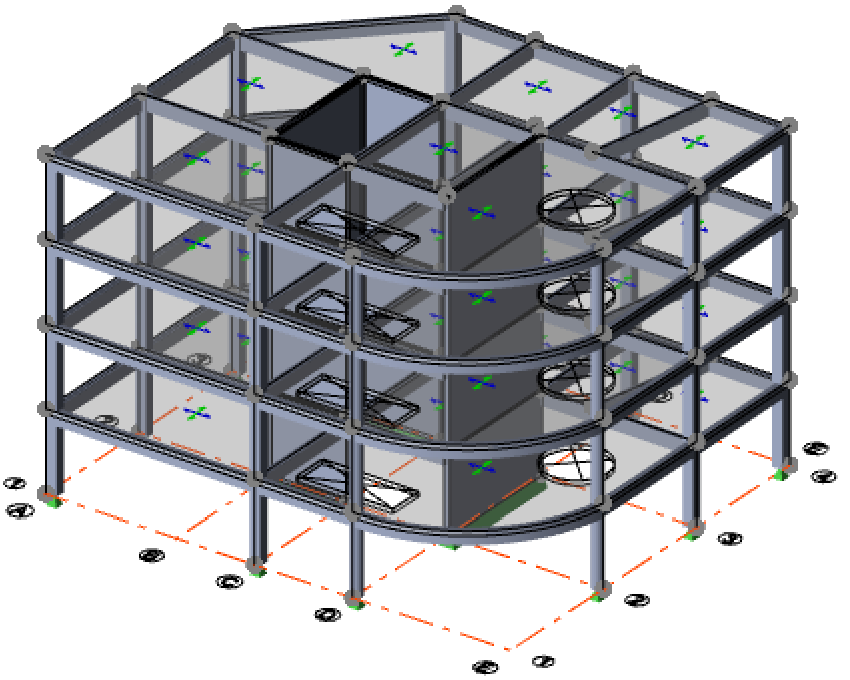
- Slabs not supported on all edges
- Complex panel shapes
- Panels with openings

Also the "tributary area" approach can only approximately handle point, line, and patch loads (by converting them to area loads).

Because of these limitations the "tributary area" method is not used in Tekla Structural Designer - instead a method referred to as FE Decomposition is applied instead. This is based on finite element analysis.

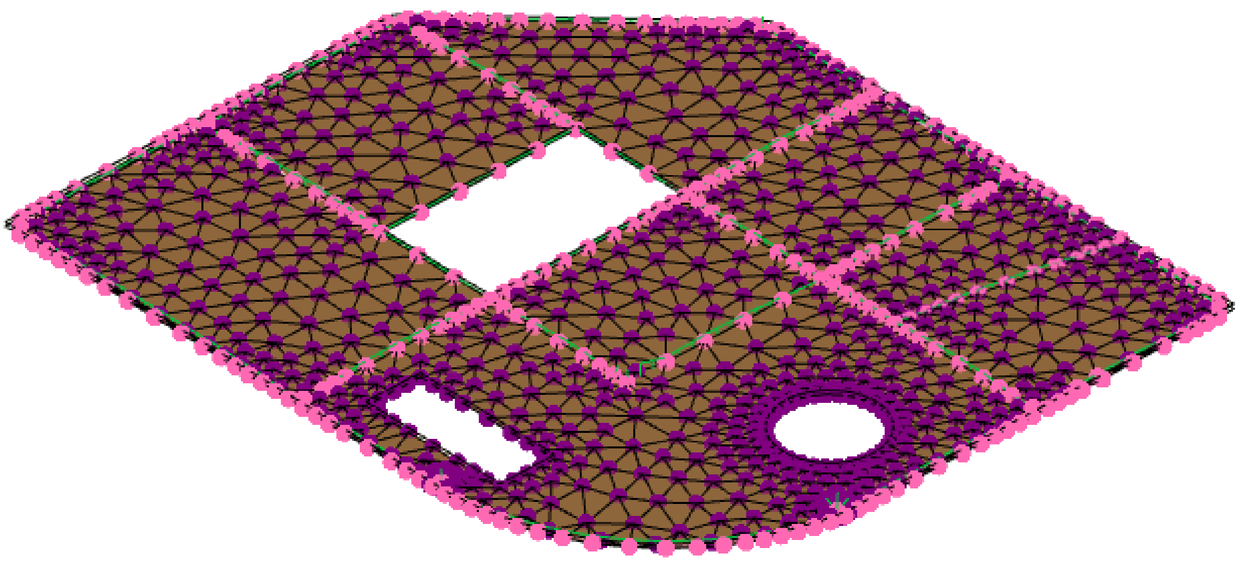
FE Decomposition Model

The FE decomposition model can be demonstrated using the following two-way slab on beams example.

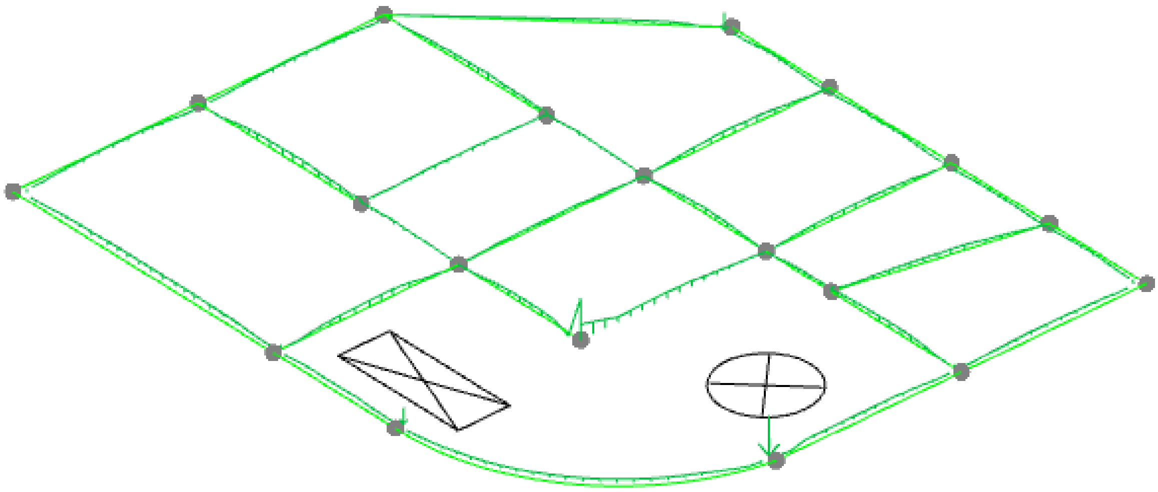


For this type of structure:

- A separate FE decomposition model is created for every floor (or sloped plane).
- Beam column and wall nodes in each FE decomposition model (shown selected in pink below) are all rigidly supported.



• Each FE decomposition model is analysed and the reactions at the rigidly supported nodes are turned back into VDLs along beams and walls, and point loads on columns.

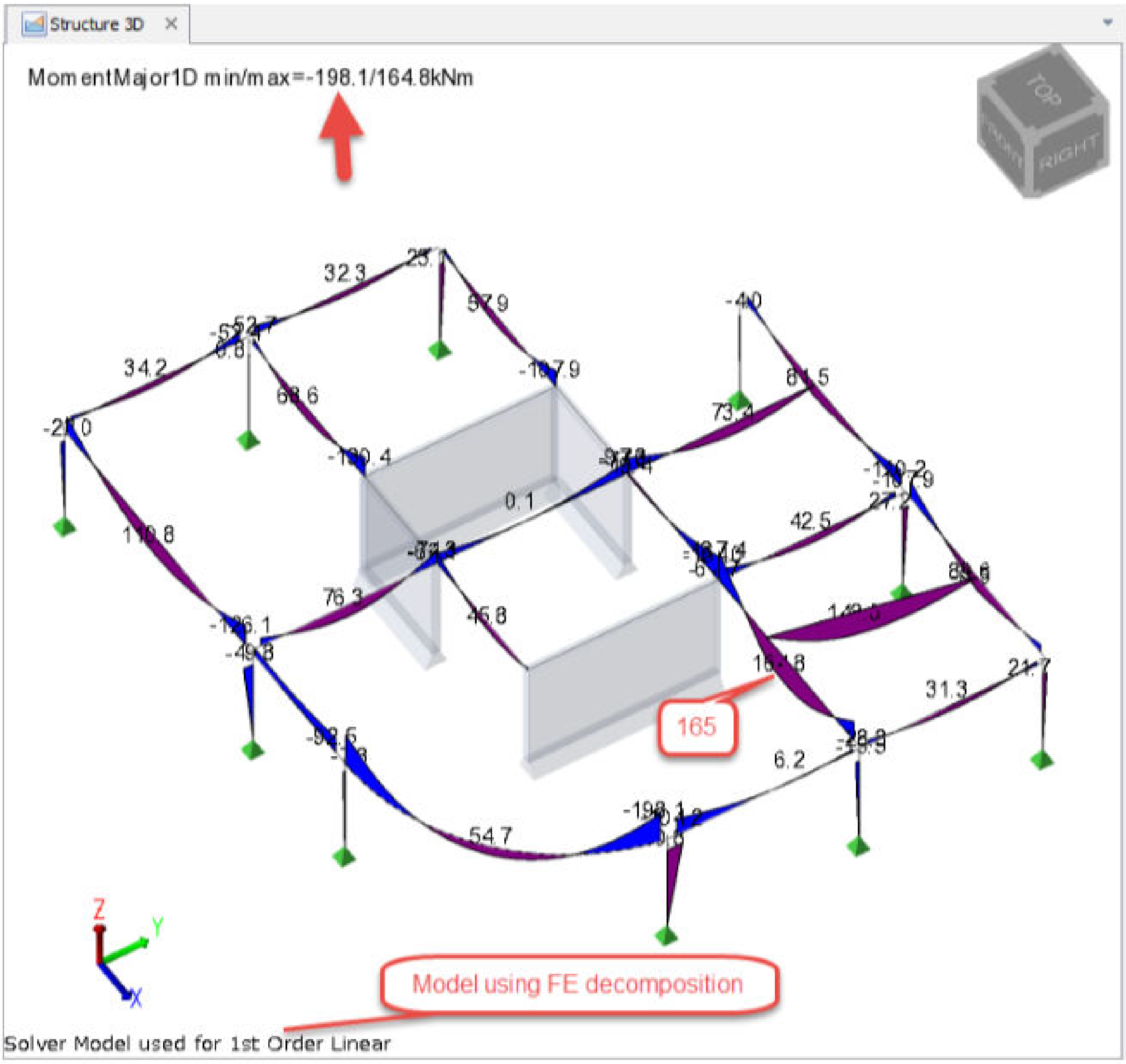


A Common Question

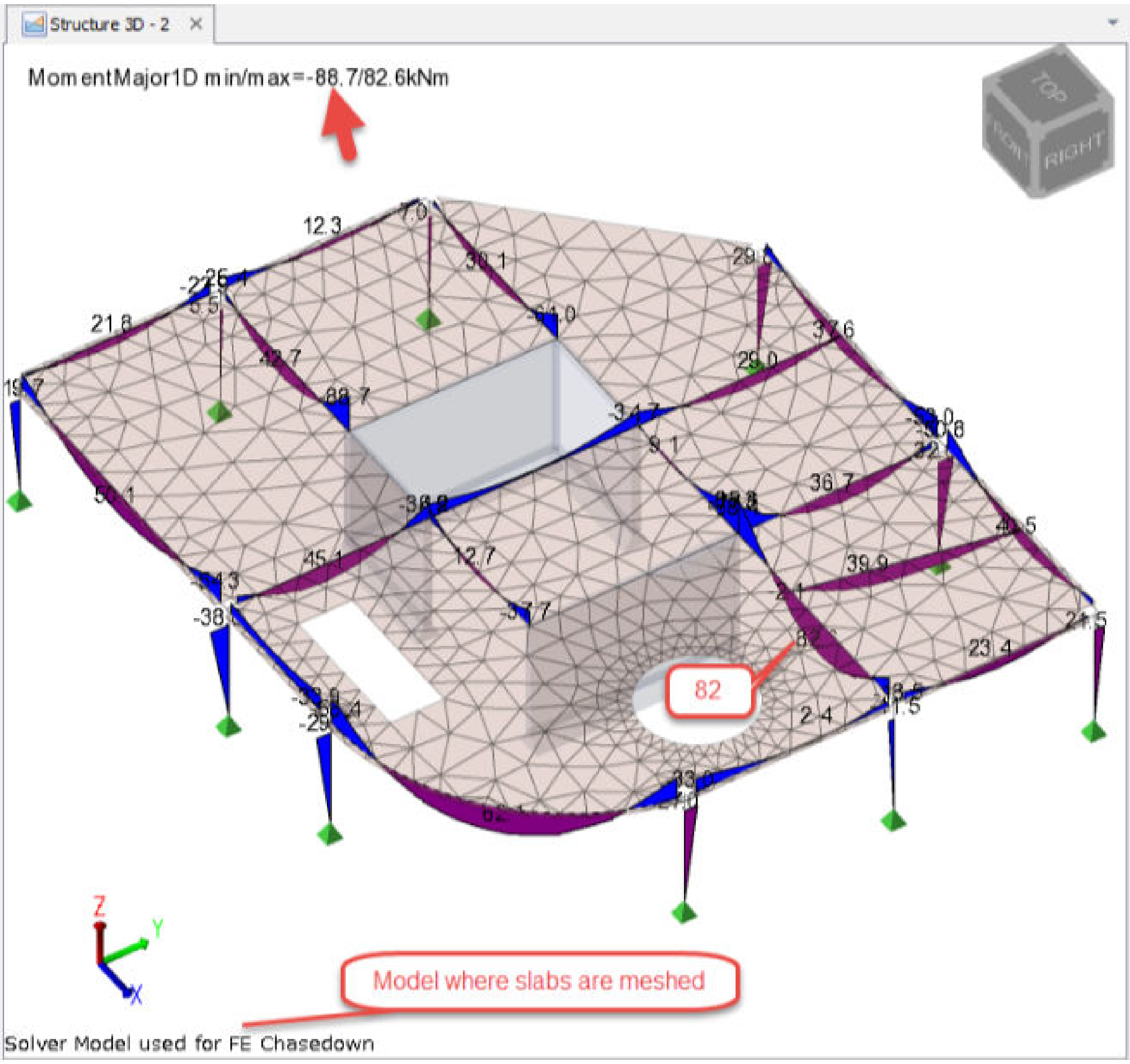
So if the slabs have to be meshed for the FE decomposition during pre-analysis, why not do away with decomposition entirely and just mesh the slabs when performing 3D analysis of the entire model instead?

Because this gives results that you don't like....

In the first run of the model below the slabs are left unmeshed, this requires the loads on the two-way slabs to be FE decomposed prior to analysis. The bending moments from the resulting 3D Analysis are as shown, (max hogging -198, max sagging 165):



The above results can be compared against a second run of the model in which the 2-way slabs are set to be meshed in the 3D analysis. FE decomposition is no longer required and the bending moments from the resulting 3D Analysis are as shown, (max hogging -89, max sagging 82)



In this second run, because the slabs are included in the 3D analysis model, some of the load is being transferred directly to the supporting columns and walls via the slabs themselves. While this is not wrong, it goes against the engineer's expectation - which would be to design the beams on the basis that they transfer all the load to the supporting members.

Overview of global imperfections

Equivalent notional horizontal loads are determined and applied during pre-analysis to cater for global imperfections (additional second order effects due to the structure not being built plumb and square). These loads are also used in seismic design to establish the base shears.

Following a first-order analysis of all gravity loadcases, the forces at the nodes at the top/bottom of each column stack/wall panel are resolved vertically. A proportion of the vertical load is determined which gives the value of the horizontal load at each point. The proportion is code dependent.

These horizontal loads are applied to the nodes in a particular direction (Direction 1 or Direction 2 or both) as specified in an individual design combination.

Overview of load reductions

Live load reductions are established during pre-analysis for use in subsequent column and wall design, and when the head code is set to ACI/AISC - beam design also.

- For head codes other than ACI/AISC, the level of live load reduction to consider for beams can be set manually in the beam properties. This is especially useful for more economic design of transfer beams supporting a number of floors.

Reductions are only applied to those live load cases that have had the Reductions box checked in the Loading dialog.

The reduction percentage to be applied is specified in Model Settings. This percentage can differ depending on the number of floors being supported.

NOTE Reductions are not applied to inclined columns (only vertical ones).

To cater for additional floors that are carried but that have not been included in the model an Assume extra floors supported value can be specified in the column and wall properties.

The methodology for live load reduction differs between national codes of practice:

Head Code: EC or BS

Levels can be designated either as "to be" or "not to be" included in the determination of the load reductions through Count floor as supported check boxes for each level in the column and wall properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular column or wall. The moments from fixed ended beams framing into a column or wall are never reduced.

Head Code: ACI/AISC

Before undertaking member design, Live and Roof Live loads are multiplied by a reduction factor R for roof live loads and other live loads independently. This reduced load is then used in combination to create design forces. The reduction factor is related to the tributary area of load carried by the particular member and also the K_{LL} factor, where K_{LL} comes from Table 4-2 in ASCE7-05/ ASCE7-10.

Essentially:

- Interior and exterior cols (no cantilever slabs) K_{LL} = 4
- Edge and interior beams (no cantilever slabs) K_{LL} = 2
- Interior beams (with cantilever slabs) K_{LL} = 2
- Cantilever beams K_{LL} = 1
- Edge cols (with cantilever slabs) K_{LL} = 3
- Corner cols (with cantilever slabs) K_{LL} = 2
- Edge beams (with cantilever slabs) K_{LL} = 1

As it is not possible to assess where cantilever slabs are and what they are attached to - you are able to change the K_{LL} factor for all column/wall stacks and beam spans as required in the Properties Window.

The default settings are:

- Columns/walls = 4
- Beams = 2
- Cantilevers = 1

Head Code: AS

Before undertaking member design, imposed loads are multiplied by a reduction factor ψ_a.

This reduced load is then used in combination to create design forces. The reduction factor is related to the tributary area of load carried by the particular member.

Overview of pattern loading

If combinations of pattern load exist then the pattern loading is automatically generated prior to analysis. See: Manage load patterns (page 527)

3D analysis

A traditional frame analysis of the complete structure is always the first analysis that Tekla Structural Designerperforms in any combined analysis and design. This analysis generates a first set of results for the design of beams, columns and walls.

First Order or Second Order Analysis?

You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the Analysis page of the Design Options dialog.

The actual options that are presented will vary depending on the design code being worked to.

Linear or Non Linear Analysis?

The choice of linear or non-linear analysis is made automatically:

- if the model has entirely linear properties a linear analysis is performed,
- else if any non-linear properties are detected a non-linear analysis is performed.

Grillage chasedown analysis

We know from experience that 3D Analysis on its own does not give the gravity results engineers have traditionally used or want - staged construction analysis reduces but doesn't resolve this. Therefore, Design All (Static) will also automatically undertake a grillage chasedown analysis provided concrete members exist anywhere in the model (beams, columns, or walls).

The Solver Model used for Grillage Chasedown (page 725) emulates a traditional analysis and establishes an alternative second set of design forces for beams, columns and walls.

See also

Accounting for lateral loading in chasedown results (page 1182)

FE chasedown analysis

The Solver Model used for FE Chasedown (page 725) is generated as part of the combined analysis and design process if the model contains flat slabs, or two way spanning slabs on beams - the results from this analysis being required for the design of these slabs.

By default the same results are also used to generate a third set of design forces for the cast-in-place beams, columns and walls. (Whether the results are used for this purpose is controlled by a setting under **Design > Settings > Concrete > Cast-in-place > Beam, Column, Wall > General Parameters**).

A significant consideration when opting to design for the FE chasedown results is that the slabs will tend to carry a significant proportion of the load direct to the columns.

Consequently, for beam design in particular, it is unlikely that an FE chasedown could result in a more critical set of design forces than would be already catered for by the Grillage chasedown.

NOTE If duplicate levels have been specified in the Construction Levels dialog separate sub models are created and analyzed for the source and every duplicate level. This ensures that the increasing load carried by the vertical members in the lower sub models is catered for. In turn this can cause small differences in the analysis results (and consequently the design) for these sub models.

See also

Accounting for lateral loading in chasedown results (page 1182)

Reasons for performing chasedown analyses

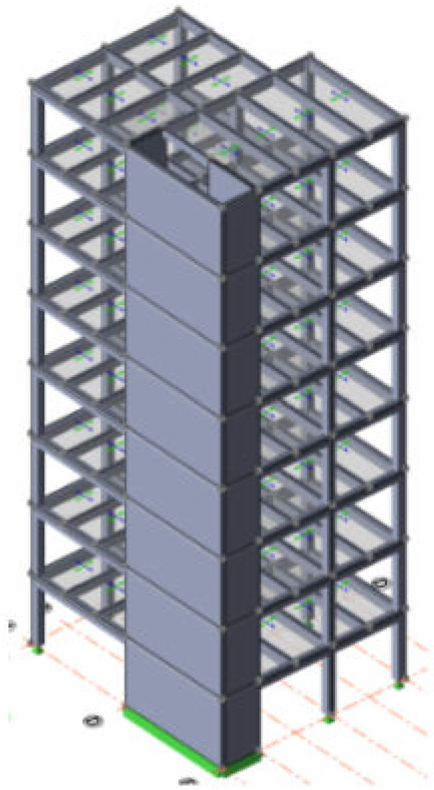
As the 3D Analysis determines a set of design forces which can then be used to design the members - why does Design (All) then carry on and do other "Chasedown" analyses?

The answer relates to expectation of results - in particular with regard to the following:

- Sway Effects under pure gravity loading
- Transfer beam designs
- Differential axial deformation

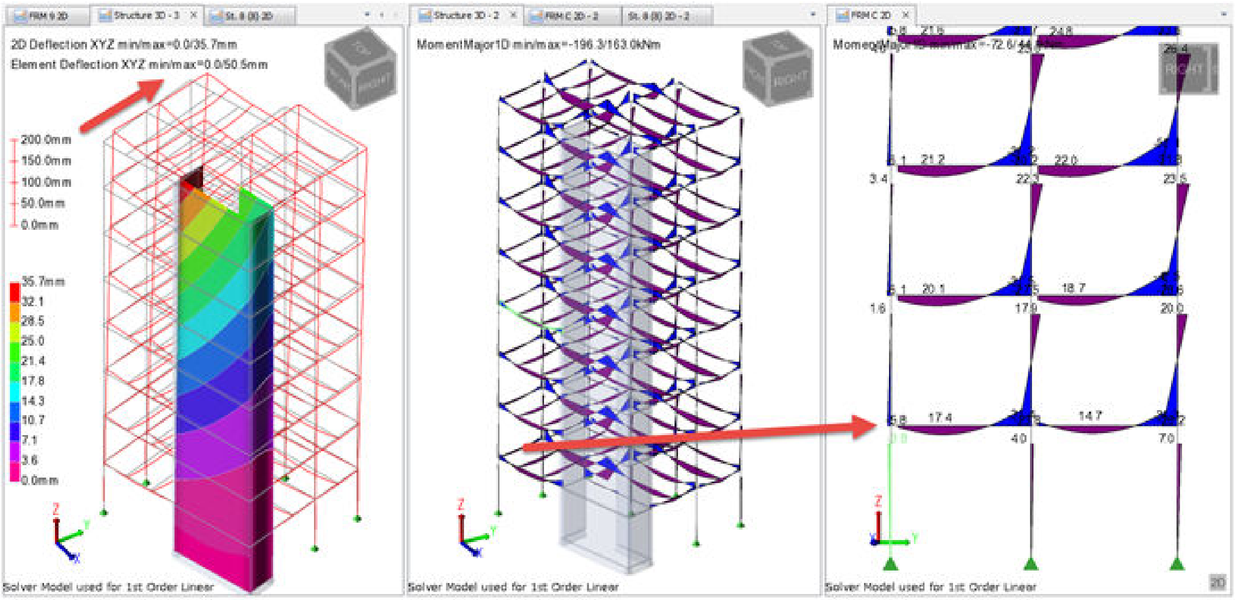
Sway Effects under pure gravity loading

Consider this 8 storey model, when you click Design All, grillage and FE chasedown solver models are created in addition to the 3D Analysis model.

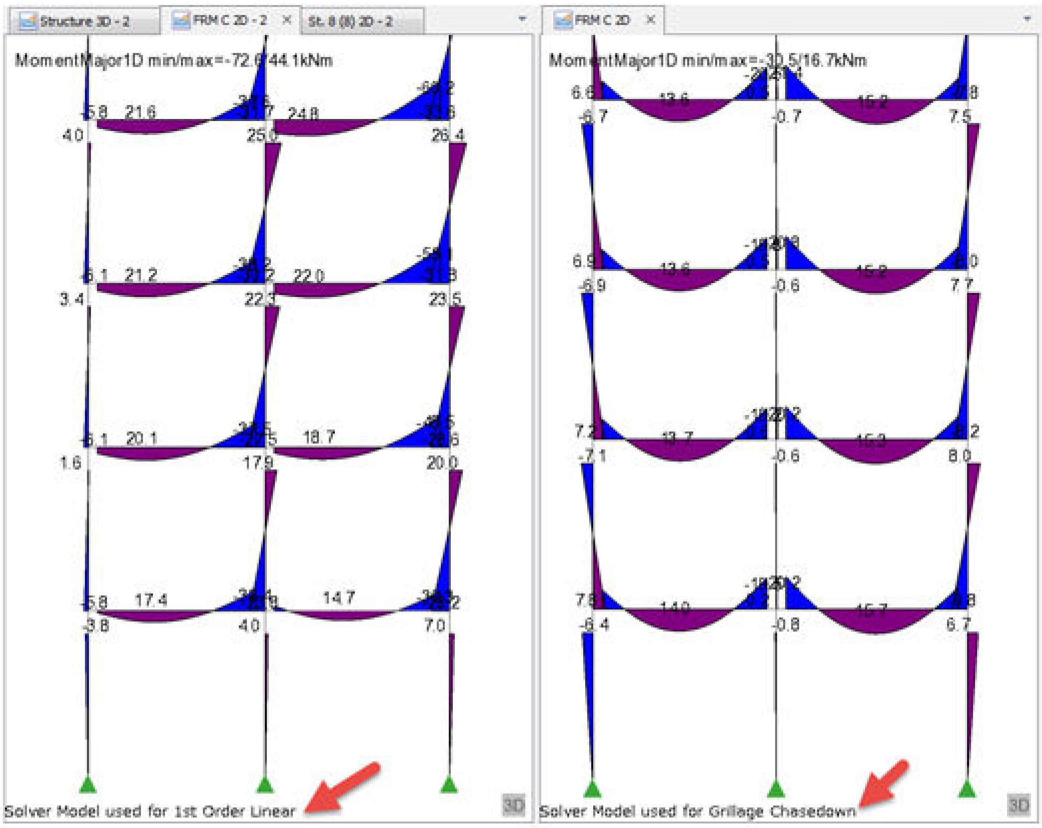


The Identified end frame below is resisting the sway - the design moments shown would be given by any analysis software.

It is an extreme example - but this result does not fit with traditional engineering expectation



The 3D Analysis results (below left) can be compared with those from Grillage Chasedown (below right).



The Grillage Chasedown results are more in line with expectation:

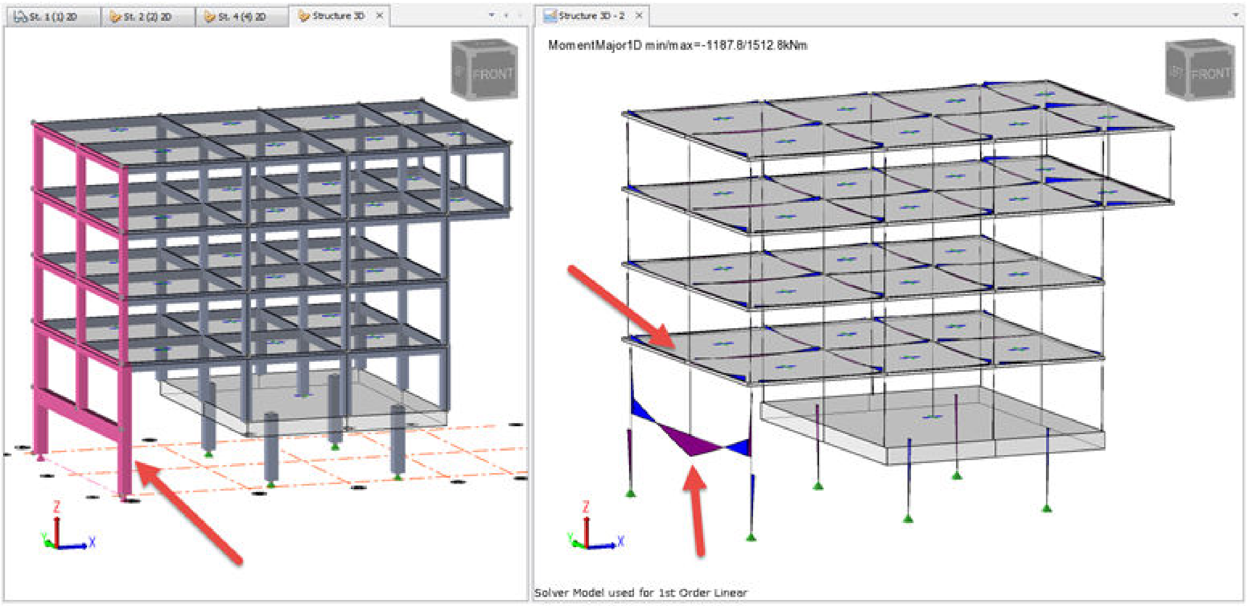
- -ve moment at LH end
- Similar moment profile at every level

Another example of why chasedown is wanted follows below....

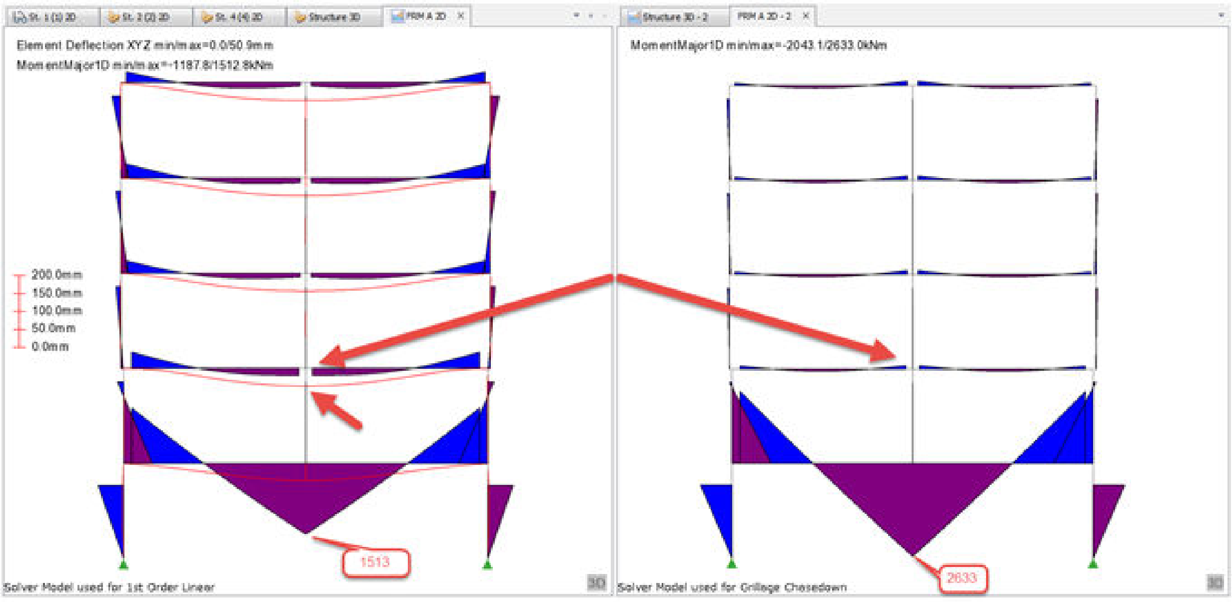
Transfer beam designs

Focus on the transfer beam in the highlighted frame.

At first glance the results look ok, but lets look more closely...



The 3D Analysis results (below left) can be compared with those from Grillage Chasedown (below right).



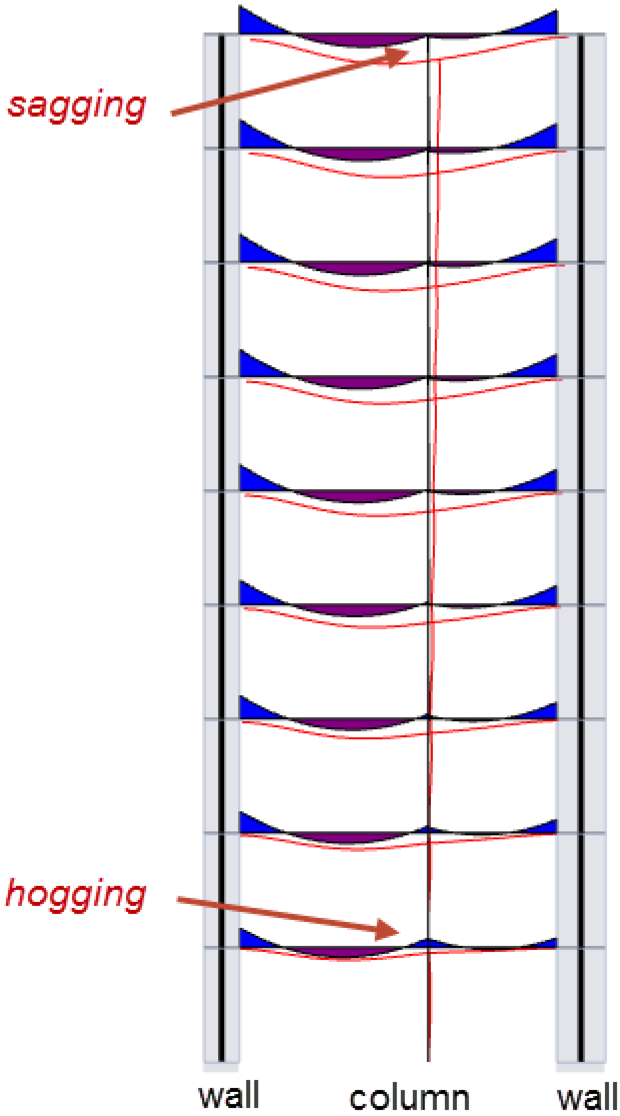
- 3D Analysis - frame deflects and loads are shared according to stiffness.
- Grillage chasedown - loads are collected floor by floor and then applied to the transfer beam - much higher moment in the transfer beam. Once again this is more in line with traditional expectation.

A third example of why chasedown is wanted follows below....

Differential axial deformation (axial shortening)

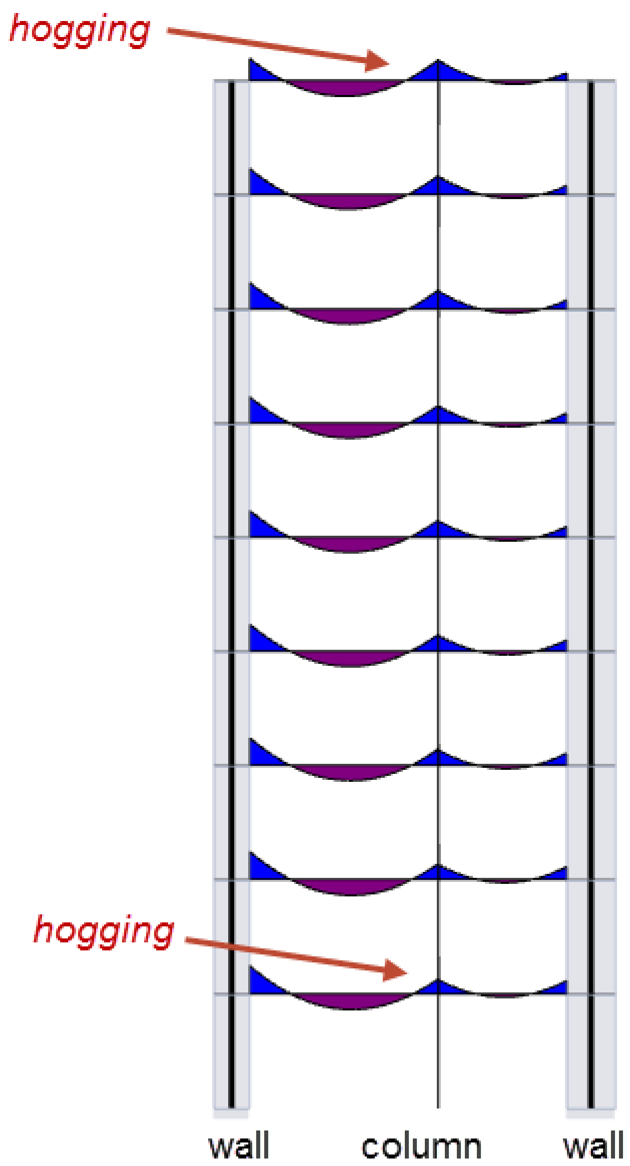
In this case a two span beam sits on walls at each end and a column in the center.

3D Analysis Results:



- Column is more highly stressed than walls and deflects downwards shedding load back to the walls.
- You end up with completely different moment profiles on beams that would traditionally be considered and designed as identical.

Grillage chasedown Results:



• the sub-models at each level are almost identical and you get very similar results over the height of the building.

Findings from the above examples

- Grillage chasedown emulates a more traditional design style where continuous beams or sub-frames are considered in isolation.
- The 3 examples show where 3D Analysis gives results that are not liked (based on traditional design expectation).
- But once you start to think about it you may conclude that actually, the 3D Analysis result should not be ignored.

- By running Design All each analysis type is performed and members are simultaneously designed for each set of results.

NOTE Deliberately extreme examples have been used to illustrate these effects and in real models the differences between the sets of results might not be as dramatic.

Accounting for lateral loading in chasedown results

It is important to note that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loading case (wind / EHF) or from the direct analysis of any combination that includes a lateral loading case are not valid.

Therefore in order to generate the design forces mentioned above, the chasedown analysis results are merged with the 3D building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
    a. Subtract the 1st order linear building analysis result multiplied by the relevant load factor
    b. Add the chasedown result multiplied by the relevant load factor
4. For results with Imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loading cases or combinations is not required.

NOTE This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral load cases.

See also

Select entities (page 350)

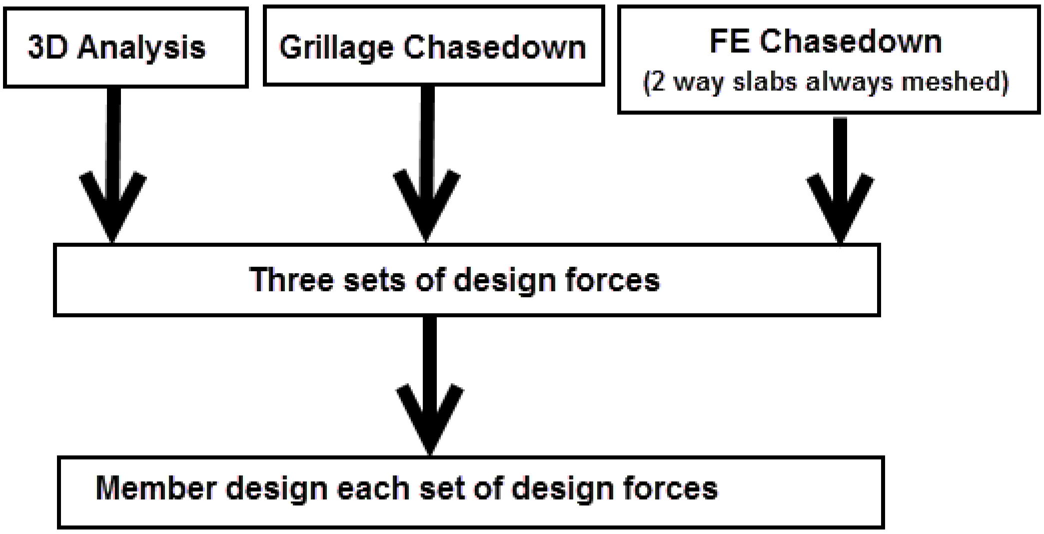
Member design stage of the combined analysis and design process

The final step in the Design All (Static) process is member design for all members for all available sets of design forces.

Steel Member Design Forces

The 3D Analysis results are the only results set used in steel member design.

Concrete Member Design Forces



Up to three sets of analysis results will exist for concrete member design as follows:

- **3D Analysis** results will always be used to design all beams, columns and walls.
- **Grillage Chasedown** results will exist for gravity loadcases if the model contains any concrete beams, in which case they will also be used to design all beams, columns and walls.
- **FE Chasedown** results for gravity loadcases will also exist if the model contains 2-way spanning slabs.

  Concrete beams can be designed for this set of results by checking the 'Design Beams for FE Chasedown analysis results' box under Design > Design Options > Concrete > Beam > General Parameters

  Columns and walls can also be designed for this set of results by checking similar boxes on their respective General Parameters pages.

Reset Autodesign

On completion of your chosen design process, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via Design Options > Autodesign.

Design Review

On completion of the Design All (Static) process the Review View and Review toolbar open automatically.

In this view a color coded version of the model is displayed so that the pass/fail status and utilization ratio of each beam, column and wall can be reviewed graphically. Various other parameters can also be graphically interrogated and/or modified.

See also

Accounting for lateral loading in chasedown results (page 1182)

Features of the three analysis types used for static design

The following table can be used to compare the features of the three analysis types used in the static design process.

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
Examples / When useful?	Gravity and Lateral analysis (Notional/Wind/Seismic)	“Beam & Slab” buildings	Flat slab and “Beam & Slab” buildings
Special Features	• Pattern loading • Automatic EC2 sway sensitivity assessment and sway amplification • Automatically centralised analysis wires (improved rigid offsets / rigid zones) • Option to mesh slabs in the 3D analysis	• Mimics traditional design approach (sub-frame analysis) • Pattern loading	• Mimics traditional design approach (isolated floor analysis) • Slab Pattern loading
Benefits	• Member Design considers sway and differential axial	Member design based on traditional sub frame is considered simultaneously	• Member design based on traditional sub frame is considered simultaneously

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
	deformation effects. • Caters for slabs that contribute to the lateral load resisting system	with that for 3D Analysis	with that for 3D Analysis • Irregular slab panel design automatically catered for
Analysis Model	3D model of entire building: • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • all 2-way slabs meshed
Analysis Method	Whole model in one pass	Each sub model sequentially from top to bottom – chasing member loads down	Each sub model sequentially from top to bottom – chasing member loads down
Analysis Type	• First order • First order - Kamp • Second order - P-D	First order	First order
Supports	External supports as defined by the user	Ends of members above/below each sub model are automatically supported	Ends of members above/below each sub model are automatically supported
Loading	Gravity and Lateral Loads	Gravity Loads only	Gravity Loads only
Forces for design			
RC Slab	Yes– All Combs	No forces	Yes – All Gravity load cases

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
RC Beam	Yes– All Combs	Yes – All Gravity load cases	Optional – All Gravity load cases
RC Column	Yes– All Combs	Yes – All Gravity load cases	Optional – All Gravity load cases
RC Wall	Yes– All Combs	Yes – All Gravity load cases	Optional – All Gravity load cases
Steel/Composite Members	Yes – All Combs except patterns	Not required	Not required
Foundation design	Yes – All Combs except patterns	Yes – All Gravity load cases	Yes – All Gravity load cases

See also

3D analysis (page 1173)

Grillage chasedown analysis (page 1174)

FE chasedown analysis (page 1174)

Reasons for performing chasedown analyses (page 1175)

13.4 Seismic analysis and design handbook

You can find the following information in this handbook:

- Introduction to seismic analysis and design (page 1186)
- Limitations of Seismic Design (page 1194)
- Seismic force resisting systems (page 1196)
- Seismic design methods (page 1199)

See also

Apply seismic loads (page 577)

Introduction to seismic analysis and design

Definitions

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

Various terms used in Tekla Structural Designer's seismic processes are described below:

Code Spectra

The spectra specified in a country's loading and design codes for use in ELF and RSA analysis.

Site Specific Spectra

User defined spectra for ELF and RSA which are required for locations which use another country's loading and design codes where the code spectra are not relevant.

Base Shear Combination

Also referred to as the Effective Seismic Weight Combination (ASCE7/UBC) or the Seismic Inertia Combination (Eurocode). This combination is used for modal analysis, and in the calculation of base shears, during the Seismic Analysis Process. This combination is created and modified by the Seismic Wizard only.

RSA Seismic Combinations

These combinations are created by the Combination Generator at the end of the Seismic Wizard, but can also be modified in the standard Combination dialog. They consist of 3 kinds of loadcases: Static, RSA Seismic and RSA Torsion. The Base Shear Combination is not included in this category of combination.

Static Loadcase

Standard loadcases, e.g. "Self weight - excluding slabs", "Dead", etc., and derived cases for NHF/EHF, but no patterns.

RSA Seismic Loadcase

Two loadcases, i.e. "Seismic Dir1" and "Seismic Dir2", which cannot be edited. These are created at the end of the Seismic Wizard. being derived from information supplied in the Seismic Wizard and the results of the modal analysis. No actual loads are available for graphical display.

RSA Torsion Loadcase

These cases can be generated by the Seismic Wizard and are regenerated whenever RSA Seismic Combinations are modified.

Fundamental Period (T)

Separately for Dir 1 & Dir 2, this is either defined in the Seismic Wizard, (user value or calculated), or determined in the modal analysis for the Base Shear Combination.

Level Seismic Weight

For each relevant level, this is the sum of the vertical forces in nodes on that level, for the Base Shear Combination.

Effective Seismic Weight

This is the sum of the level seismic weights for all relevant levels for the Base Shear Combination.

Seismic Base Shear

The base shear is calculated separately for Dir 1 & Dir 2, for the Base Shear Combination.

NOTE In Tekla Structural Designer the base shear is displayed when you Review story shear tabular results (page 900) for **Cumulative Story Shear**

Square root of Summation of Square (SRSS)

The SRSS formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{k=1}^{n} (\lambda_k^2))}$$

λ = Absolute value of combined "response"

λ_k = "response" value for Relevant Mode k

n = Number of Relevant Modes

Complete Quadratic Combination (CQC)

The CQC formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{i=1}^{n} \sum_{j=1}^{n} (\lambda_i \; \rho_{ij} \; \lambda_j))}$$

λ = Absolute value of combined "response"

λ_i = "response" value for Relevant Mode i

λ_j = "response" value for Relevant Mode j

n = Number of Relevant Modes

ρ_{ij} = Cross modal coefficient for i & j

Cross Modal Coefficient

This co-efficient is used in the CQC method for combining modes in RSA.

$$\rho_{ij} = \frac{8\zeta^2\,(1+\beta)\beta^{1.5}}{(1-\beta^2)^2 + 4\zeta^2(\,1+\beta\,)^2}$$

ζ = modal damping ratio

IBC/ASCE assumed = 5% (ASCE Figs 22-1 to 6)

EC8 assumed = 5% where q accounts for the damping in various materials being different to 5% (EC8 Cl 3.2.2.5)

IS codes the user can define the level of damping and this is accounted for in the above equation.

β = Frequency ratio = ω_i / ω_j

ω_i = Frequency for Relevant Mode i

ω_j = Frequency for Relevant Mode j

Overview

All seismic codes work in a similar manner from the loading view point with relatively minor differences in terminology and methodology.

It is worth noting at the start that seismic analysis determines a set of forces for which it is expected (statistically) that if those forces are designed for and other design precautions taken (additional seismic design) then in the event of an earthquake the structure may well suffer extensive damage but will not collapse and for some categories of building should actually remain serviceable.

In Tekla Structural Designer a seismic wizard gathers all the information together to setup the requirements for a seismic analysis.

From this information a number of things are determined:

- If working to ASCE7 - the seismic design category (SDC) for the building - giving amongst other things the permissible type of analysis
- The Base Shear Combination to determine the seismic base shear in the building
- The natural frequencies of the building in two horizontal directions
- The combination of the gravity and other lateral forces with the seismic load cases

Earthquakes load a building by a random cyclic acceleration and deceleration of the foundations. These are in both horizontal directions (Dir1 and Dir2) but can also be in a vertical direction too. This ground acceleration excites the building in its natural and higher frequencies.

As a result if the building is:

- In an area of low seismic acceleration, low in height and poses limited risk to life then a gross approximation can be used in analysis - assuming a % of gravity loading is applied horizontally to the building to represent the earthquake (US codes 1%, Australian codes 10%).
- In an area of moderate to low seismic acceleration, medium to low in height and does not house a significant number of people - the predominant mode excited is the 1st mode. An equivalent lateral force (ELF) approximation can be used that applies static horizontal loading distributed up the building to mimic the shape of the 1st mode of vibration in a static analysis.
- Anything else, in an area of high acceleration, tall in height and could be holding many people or be critical post-earthquake then a "more representative" analysis method of Response Spectrum Analysis (RSA) should be used. This analysis is based upon a modal analysis considering all mode shapes in the two horizontal directions in which typically 90% of the structure's mass is mobilized.

The results from the chosen method of seismic analysis are used in combination with other gravity and lateral load cases to design both normal members and those members in seismic force resisting systems (SFRS). These latter members need additional design and detailing rules to ensure they resist the seismic forces that they subjected to during an earthquake. If working to ASCE7, the extent of these rules are determined by the SDC noted above (the higher the demand, the ‘better’ the SFRS that will be required).

NOTE The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian Head Codes.

Seismic Wizard

In Tekla Structural Designer the Seismic Wizard sets up the information required for seismic analysis - the main parameters to be input being:

- Ground acceleration - strength of the earthquake
- The Importance Level (or Importance Class) of the building - the use to which the building is being put - typically
    - I= very minor, farm and temporary buildings,
    - II= general buildings occupied by people,
    - III = buildings occupied by a large number of people
    - IV = critical buildings with a post-disaster function eg hospitals, police stations, fire stations and buildings along access route to them)

- The ground conditions upon which the building is founded (typically Hard Rock, Rock, Shallow soil, Deep Soil, Very Soft Soil)
- Building height - for low buildings the first mode is totally dominant in taller buildings other modes become significant
- Plan and vertical irregularities in the building

From this input the Seismic Wizard determines the seismic design category for ASCE7, and also the elastic design response spectrum to be used for the building.

Additionally the Wizard sets up the Base Shear Combination - the combination of loads likely to be acting on the building when the earthquake strikes.

Vertical and Horizontal Irregularities

There are typically 5 types of horizontal irregularity and 5 types of vertical irregularity - all are defined to pick up structures that have lateral framing systems and shapes in plan that will preclude the structure naturally developing a simple first mode of vibration. Since this is a basic assumption of ELF - the presence of these irregularities may preclude the use of ELF.

Torsion

When a structure's center of mass at a level does not align with the position of the center of rigidity then torsion is introduced in the structure at that level when an earthquake excites the structure. To account for this, there are three types of torsion potentially applied to levels with non-flexible diaphragms during a seismic analysis

- Inherent torsion - in a 3D analysis when the center of mass and center of rigidity at a level do not align, this is taken account of automatically
- Accidental torsion - to allow for the "miss positioning" of loads in a structure, an additional eccentricity of usually 5% of the structure's width in all relevant directions - this is accounted for with a torsion load case in the analysis
- Amplified accidental torsion - structures with certain SDCs and certain horizontal irregularities require an amplified accidental torsion to allow for extra effects

Modal Analysis

Using the Base Shear Combination, a modal analysis is run for two purposes:

- the natural frequencies of the building in two directions are determined to assist with the calculation of the seismic base shear that in turn is used to determine the distribution of applied loads up the building for an ELF analysis

- the frequencies and mode shapes of the building are determined that need to be included in an RSA analysis so that typically 90% of the mass in the building is mobilised during the RSA analysis

% of Gravity Load Method

The % of gravity load method is used as a means of a gross approximation of the earthquake and is only used in situations where seismic effects are considered to be low. This load case is combined with the relevant load factor with other gravity and lateral load cases to determine the design forces and moments to be considered in conventional design.

NOTE This method is not applicable when working to Eurocodes.

Equivalent Lateral Force Method

The ELF method assumes that the first mode shape is the predominant response of the structure to the earthquake.

Based on the natural frequency and the Base Shear Combination, a total base shear on the structure is determined and this is then set up as a series of forces up the structure at each level (in the shape of an inverted triangle) and these deflect the structure in an approximation to the shape of the first mode.

The resulting seismic load cases are combined with the correct combination factors with the other gravity and lateral load cases in the **seismic combinations** to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

Response Spectrum Analysis Method

The RSA method uses a set of vibration modes that together ensure that the mass participation is typically 90% in the structure in a particular direction.

The response of the structure is the combination of many modes that correspond to the "harmonics". For each mode, a response is read from the design spectrum, based on the modal frequency and the modal mass, and they are then combined to provide an estimate of the total response of the structure.

Combination methods include the following:

- **Square root of Summation of Square (SRSS)**
- **Complete Quadratic Combination (CQC)** - a method that is an improvement on SRSS for closely spaced modes

As a result of the combination methods (SRSS and CQC), the resulting seismic load cases are without sign and so they are applied with the correct combination factors both + and - around the "static" results of the other gravity and lateral load cases in the **seismic combinations** to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

Summary of RSA Seismic Analysis Processes

RSA Seismic Analysis (1st or 2nd order) is run as a stand-alone analysis from the **Analyze** tab, or as part of the Design (RSA) process. In the latter, the use of 1st order or 2nd order is set for the static analysis is set via **Design Settings > Analysis**.

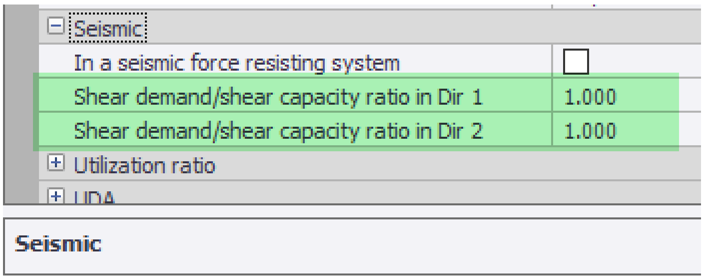
The process consists of the following steps:

Step	Process	Description
1	Model Validation	Run to detect any design issues which might exist. This is similar to standard model validation but also checks: • Base Shear Combination must exist • At least one RSA Seismic Combination must exist including at least one RSA Seismic Loadcase.
2	Modal Analysis	A 1st order modal analysis for the Seismic Inertia Combination only, which returns the standard results for that analysis type, but also the fundamental periods for directions 1 & 2.
3	Pre-Analysis for Seismic	Performs calculations for RSA Torsion Loadcases. The seismic weight and seismic torque are both calculated at this stage.
4	Static Analysis	1st Order Linear or 2nd Order Linear analysis is performed for all RSA Seismic Combinations and all their relevant loadcases, i.e. this includes Static Loadcases, but does not include RSA Seismic and RSA Torsion Loadcases.
5	RSA Analysis	A set of results is generated for a sub-set of vibration modes for each RSA Seismic Loadcase.
6	Accidental Torsion Analysis	Analysis of any RSA Torsion Loadcases that exist.

Seismic Drift

Seismic drift is assessed on a floor to floor horizontal deflection basis and there are limits for acceptability of a structure.

When working to the ASCE7 code, the engineer can directly define the shear demand / capacity ratio (beta) for columns and walls. The default value of 1.0 could be over-conservative. This option is located in the “Seismic” group of member properties, and can be set separately for 'Direction 1' and 'Direction 2' and for each stack/panel, as shown in the picture below.



- The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

The building's overall seismic drift status is displayed in the Design branch of the Status Tree in the Project Workspace.

Full details for all columns are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the Analysis>Seismic Drift chapter to your model report.

Design Coefficients and Factors (ASCE7/UBC)

Typically three factors are determined based on the lateral force resisting systems in the structure and which account approximately for the inelastic response that occurs during the earthquake which is not accounted for directly in the analysis.

- The response modification coefficient which affects the seismic base shear.
- The overstrength factor accounts for the reinforcement steel yielding overstrength and is utilized in the concrete beam and column capacity calculations.
- The deflection amplification factor which is used in the calculation of seismic drift.

Limitations of Seismic Design

The following limitations apply:

- Where seismic design and detailing is required this is only supported in Tekla Structural Designer for the ACI/AISC and Indian Head Codes.
- It is up to the user to assess whether framing is split horizontally or vertically, whether system specific requirements need to be assessed - like mixed system moment frames, whether diaphragms are rigid or flexible - in all instances, the user will need to make the necessary adjustments for the situation in hand. The software does not handle these situations automatically.
- Linear modal analysis with non-linear element properties - currently the modal analysis is limited to a linear model so all non-linear elements are set to be linear.
- ELF can be run as 1st or 2nd order analyses, however if the Fundamental Period is determined using modal analysis the modal analysis is always run as 1st order.
- The RSA analysis itself is a 1st order linear process. For the 2nd Order RSA Seismic analysis, the peak responses are enveloped around the static results for 2nd Order Linear Analysis. Thus when the analysis is set to 2nd order in Design Options, in real terms the results are actually RSA Seismic + 2nd Order.
- Structures with linear members and supports are run using linear analyses. Structures with non-linear supports and /or members are run as non-linear in ELF but linear in modal and RSA.
- We do not consider any of the standard methods for structurally accommodating seismic actions - e.g. base isolators, damping systems
- We do not consider more accurate methods of analysis like time history analysis. As a result there are some situations with very tall buildings and very irregular buildings that Tekla Structural Designer does not cater for.
- Diaphragms - rigid and semi-rigid diaphragms (meshed floors) are available and it is the user's responsibility to ensure they are modelled suitably. Rigid diaphragms are only allowed in limited circumstances and, so called, 'flexible diaphragms' can be modelled as semi-rigid diaphragms with extremely low stiffness. Force transfer into and out of the diaphragm is not checked.
- Collector elements - no checks included.
- Non-structural elements - no checks included.

Specific limitations of steel seismic design

- Coincident V & A braces giving X type are beyond scope
- Various other requirements not checked
    - e.g. V & A braces are restrained at their intersection

    - e.g. tension braces resist between 30% and 70% of total horizontal force
    - e.g. forces in restraining members not checked
- Connections are not designed

A specific limitation of RC seismic design

- While checks can be done in both directions they are direction specific where applicable - there are no biaxial checks.

Seismic force resisting systems

Available SFRS types

The seismic design requirements for a particular member are based upon which Seismic Force Resisting System (SFRS) the member forms part. Hence, in Tekla Structural Designer you can set appropriate members as part of one of the following systems:

SFRS types included for steel members

Moment Frame Systems

- Special Moment Frames (SMF)
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Braced Frame and Shear Wall Systems

- Ordinary Concentrically Braced Frames (OCBF)
- Special Concentrically Braced Frames (SCBF)
- Special Concentrically Braced Frames (SCBF)

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types available for concrete members

Moment Frame Systems

- Special Moment Frames (SMF) - limited design.
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Walls

- Special reinforced concrete structural wall - limited design
- Intermediate precast structural wall - no seismic design performed

- Ordinary reinforced concrete structural wall

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types excluded

Everything else

- e.g. Eccentrically Braced Frames

Members allowed in the SFRS

The following member types are allowed to be part of a SFRS in Tekla Structural Designer

- Steel columns
- Steel beams
- Steel braces
- Concrete columns
- Concrete beams
- Concrete walls

The following member types are not allowed to be part of a SFRS in Tekla Structural Designer

- Any timber, cold-formed, general
- All other “Characteristics”, e.g. steel joists, truss members, purlins
- Composite members
- Plated, Westok, Fabsec, concrete filled, concrete encased – selectable but no design (i.e. only rolled)

Assigning members to the SFRS

The choice of members to be part of a SFRS is entirely the engineer’s responsibility.

- It is expected that all members in a frame are assigned to the SFRS.
- The assigned members should be specified to act in building Direction 1 or Direction 2

Special Moment Frames - assigning connection types at steel beam ends

For SFRS comprising of steel SMF it is necessary to ensure that the beams fail before the columns. To this end, an assessment of plastic moment capacity is made at each floor. The moment capacity is dependent upon the position of the plastic hinge, typically (d_{col} + d_{beam})/2. These locations can be selected appropriate to each beam end either in the beam properties.

Options are provided as follows:

- Plastic hinge position at start
    - Plastic hinge position at end

Either, (d_{col} + d_{beam})/2 (default)

or, d_{col} /2 + L

L = 0 (default)

Validation of the SFRS

There is only a small amount of validity checking for an SFRS that can be performed automatically; it remains in large part the user's responsibility to ensure that each SFRS is defined appropriately.

The following validation conditions are however detected:

1. Any A or V brace in a Seismic Force Resisting System must have the A or V as vertically released. A warning is provided in validation if this is not the case.
2. X type bracing is defined as more than one V or inverted V (A) type brace pair on the beam. When more than one A or V brace pair is detected, the additional checks required by AISC 341-05 and AISC 341-10 given in Section 8.3 are out of scope. This situation is not detected during validation, but it is identified in the seismic design, so that the beam is given a "beyond scope" status.
3. The use of K braces is not allowed in AISC 341. An error is provided in validation.
4. Tension only braces were permitted to the 05 version but had no additional requirements. In the 10 version they are only are allowed for OCBF. Thus, an error is provided in validation when a tension only brace is set as part of a SCBF and the code is the 10 version. (The same validation is also applied to compression only braces.)
5. If seismic loadcases are included in combinations and there is not at least one member assigned to each of Direction 1 and Direction 2 then a warning is issued.

Auto design of SFRS members

All SFRS members can be auto-designed to the conventional design requirements both for seismic and non-seismic combinations;

During the automatic design procedure, besides the conventional auto-design, and for seismic combination results only:

- Steel members in the SFRS are checked for seismic provisions;
- Normal weight reinforced concrete members in the SFRS are auto-designed for seismic provisions

Seismic design methods

To design members and walls for the results of a seismic analysis:

- For geographic regions categorised as "low seismicity" it is acceptable to assume "ductility class low" applies. Under these conditions the seismic analysis results can be fed into "conventional" design, see Seismic analysis and conventional design (page 1199)
- Certain conditions (e.g. "high seismicity") necessitate that a "seismic" design is performed - additional design and detailing requirements have to be applied in this situation, see Seismic analysis and seismic design (page 1200)

NOTE The additional design and detailing requirements of "seismic" design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian Head Codes.

Seismic analysis and conventional design

ELF seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the ELF method is summarised as follows:

1. Modelling
   - No additional seismic modelling requirements
   - There is no need to assign members to a SFRS
2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required

   b. Set the site class (ASCE7), or ground type (Eurocode)

   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method

   d. Set up the relevant seismic combinations
3. Static Design

   Run the Design (Static):

- the results of the ELF seismic combinations are fed into the design and considered in the same way as other combinations.

4. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the RSA method is summarised as follows:

1. Modelling
   - No additional seismic modelling requirements
   - There is no need to assign members to a SFRS
2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE7), or ground type (Eurocode)

   c. Select RSA method of analysis

   d. Set up the relevant seismic combinations
3. Static Design

   Run the Design (Static):
   - Results of the static combinations are fed into conventional design.
4. RSA Seismic Design

   Run the Design (RSA):
   - Results of the RSA seismic combinations are fed into conventional design and considered in the same way as the static combinations.
   - No additional seismic design is required
5. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

See also

Seismic wizard in detail (page 579)

Seismic analysis and design handbook (page 1186)

Seismic analysis and seismic design

NOTE The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian Head Codes.

These requirements vary depending upon the 'sophistication' of the SFRS. For example OMF have less stringent requirements than SMF.

ELF seismic analysis and seismic design

The overall modelling, analysis and seismic design process using the ELF method is summarised as follows:

1. Modelling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type.

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.
   b. Set the site class (ASCE7), or ground type (Eurocode)
   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method.
   d. Set up a modal mass combination
   e. Set up the relevant seismic combinations

3. Static Design

   Run Design (Static) to:

   a. Conventionally design all members for all non-seismic (gravity and lateral) combinations
   b. Conventionally design all members for all seismic combinations in the same way as the other combinations.
   c. Perform additional seismic design for the seismic combinations for those members assigned to a SFRS

4. Calculation Output

   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and seismic design

The overall modelling, analysis and seismic design process using the RSA method is summarised as follows:

1. Modelling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type. These members will be designed and detailed according to the seismic provisions.

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE7), or ground type (Eurocode)

   c. Select RSA method of analysis

   d. Set up a modal mass combination

   e. Set up the relevant seismic RSA combinations

3. Static Design

   Run the Design (Static) to:

   - Conventionally design all members for all non-seismic (gravity and lateral) combinations .

4. Modal Analysis

   At this point it is recommended that you run a 1st order modal analysis in order to confirm the model converges on a solution, (until it is able to do so, it is pointless proceeding with a full RSA Seismic Design).

5. RSA Seismic Design

   Run the Design (RSA) to:

   a. Conventionally design all members for all RSA seismic combinations in the same way as the other combinations.

   b. Perform additional seismic design for the RSA seismic combinations for those members assigned to a SFRS

6. Calculation Output

   - A Seismic Design Report is available

   - Drift limitations are checked

See also

Seismic wizard in detail (page 579)

Seismic analysis and design handbook (page 1186)

13.5 Steel member design handbook

To get started with designing steel structures in Tekla Structural Designer see:

- Combined analysis and design choices for steel structures (page 1203)
- Steel member autodesign (page 1205)
- Steel member design groups (page 1206)
- Steel beam design (page 1208)
- Composite beam design (page 1224)
- Steel column design (page 1249)
- Steel brace design (page 1262)
- Steel joist design (page 1265)
- Steel truss design (page 1270)
- Portal frame design (page 1274)

Combined analysis and design choices for steel structures

Gravity design

For many steel structures a separate gravity design is not necessary and you can simply proceed directly to a Static design (page 1204).

In some circumstances however, you may find it preferable to adopt a two-stage workflow in which gravity members are sized prior to the full design.

The benefits of this are:

- For larger models it can significantly speed up the analysis and design time
- If lateral resisting systems are not modeled, then mechanisms can result and the analysis may not find a solution. Since these lateral resisting systems are perhaps unknown early in the project (or insufficient systems have been provided), running a gravity design has the added benefit of automatically fixing column nodes (not attached to a diaphragm) horizontally, which stabilizes the model.

The gravity design stage enables you to design all members (and hence size those members with the Autodesign property checked), for only the active gravity combinations (this will include the Construction Stage combination). As an example, beams within a rigid floor diaphragm are unaffected by lateral load, and hence they can be sized using only the gravity combinations.

Gravity design is initiated by clicking **Design Steel (Gravity)**.

After the gravity design has been completed, by default all steel members are reset to check design mode.

You may then decide to reset certain members to auto design e.g. columns and beams in 'moment frames', columns and braces in vertical or horizontal braced bays. In such cases, when the full static design is performed member 'pre-sizing' will take place and for members resisting lateral loads the new section size will be used if it is larger than that which resulted from the gravity design.

Static design

Full static design is initiated by clicking **Design Steel (Static)** or **Design All (Static)** on the **Design** toolbar.

All beams, columns and braces that do not have the **Gravity only** property checked, are designed or checked for all active combinations; Gravity only design members are designed or checked for the active gravity combinations only.

As part of the full design process a 3D Analysis is performed, for which you must select (via Design Options) the analysis type. The choice of analysis type will depend on the code being designed to.

Designing individual members for gravity only

By default, when a full static design is run, all members are designed for both gravity and lateral combinations.

You can however tell Tekla Structural Designer to only consider gravity combinations for the design of specific members. This is achieved by checking **Gravity only** in an individual member's properties, i.e.

- **Gravity only** checked - designed for gravity combinations only
- **Gravity only** unchecked - designed for all combinations types: gravity, lateral and seismic

Setting columns that do not help resist lateral loads to be designed for gravity loads only, will reduce the overall design time.

Engineering judgement will be required when identifying members as being 'gravity only'.

For example:

- if an inclined braced member connects to a beam, axial force in the brace (from both gravity and lateral loads) puts the beam into bending and therefore the beam should be designed for both gravity and lateral loads.
- potentially, beams in a sloping roof would also need to be designed for both gravity and lateral load

NOTE If a composite beam is identified as being designed for both gravity and lateral combinations, they are only designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered. Hence a warning is provided, if the ignored loads exceed a preset limit.

Steel member autodesign

The design mode for each member is specified in its properties.

- When **Autodesign** is not selected (i.e. check mode), you assign your desired section size to the member and Tekla Structural Designer determines if the section is sufficient.
- When **Autodesign** is selected the section type to be used is specified from a Design Section Order and Tekla Structural Designer attempts to automatically determine a suitable section.

  The following controls can be applied to further limit the sections considered:

  - Size constraints (page 1205)
  - Design section orders (page 1006)

NOTE If a member type has been set to be designed using Design groups, then if at least one member of the group is set to autodesign, the whole group will be automatically designed.

Size constraints

When undertaking an Autodesign, the Size Constraints property allows you to ensure that the sections proposed by Tekla Structural Designer match any particular size constraints you may have.

For instance:

- Minimum width - you may want to ensure a minimum flange width to satisfy bearing requirements for any supported beams. To achieve this, you would enter a value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than the specified minimum width for the design of the beam.
- Maximum depth - you may want to ensure that the maximum beam depth is specified to ensure that it fits within a floor zone. To achieve this, you would enter a value as the Maximum depth and Tekla Structural Designer would not consider sections with depths greater than the specified value for the design of the beam.

- Max span/depth ratio (beams only) - you may want to set a max span/ depth ratio limit above which autodesign will reject sections that are of insufficient depth.
  - The depth is taken as the depth from the section properties - for single and double angles this is the vertical leg length.
  - Limit is not applied to any member which has rotation.

NOTE Size constraints only work when undertaking an **Autodesign**

Applying a size constraint to an element

You can control which sections in a design order list are considered, by following the procedure below.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window
2. Expand All spans/stacks or the individual span/stack as required to reveal **Size Constraints**
3. Expand **Size Constraints** and set the **Max depth**, **Min depth**, **Max width**, **Min width**, or **Apply max/span/depth ratio** as required.

When an **Autodesign** is performed, the section selected from the section order list will fall within any size constraint limits set.

Steel member design groups

Why use steel design groups?

Steel members are not automatically placed into design groups. They are however, automatically put into groups, primarily for editing purposes. In this way, individual groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

If required, grouping can also (optionally) be utilized in order to design steel member types according to their groups.

In order to use grouping for this second purpose, you should first ensure that your groups are configured so that each group only contains those members that you intend to eventually have the same section size applied.

NOTE Grouped design is optional and can be activated if required, via Design Options> Design Groups

NOTE A fixed set of rules are initially used to determine the **automatic member grouping**: for example beams are designated as either direction 1 or direction 2, are of a similar span, material grade and construction type, columns have the same number of stacks etc. **You can manually edit groups.**

What happens in the group design process?

When the option to design a specific steel member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the section size so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the section size is adequate for each and if this proves not to be the case, it is increased as necessary and the revised section size is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

NOTE When performing the design of a group of composite beams the changes in section size may mean that for some beams the existing stud layout will no longer remain adequate. To ensure the safe and rational design of the studs, the stud design routine is re-run at the end of the group design for all beams with the stud auto-layout active in the beam properties.

Steel design group requirements

Steel member design groups are formed according to the following rules:

Member type	Design group rules
Steel beam	• A beam element may only be in **one** design group • All beam elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled, • All beam elements in the group must have the same material properties Automatic grouping uses the following rules to automatically generate the groups, however, when manually editing the groups these are not enforced: • All beam elements in the group are of a similar span.

Member type	Design group rules
	• All beam elements have a similar direction i.e. beams are grouped as Direction 1 or 2 if they lie +/- 45 degrees to the directional axis
Steel column	• A column element may only be in **one** design group • All column elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled, • All column elements in the group must have the same material properties • All column elements in the group must have the same number of stacks

Group management

Automatic Grouping

Steel members are grouped automatically.

In Model Settings the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for automatic grouping purposes.

Manual/Interactive Grouping

It is recommended that you manually edit the groups for steel members, since members can be placed into any group without the need for similar span lengths or directional requirements. For example, edge beams, primary beams, secondary beams etc. could have different span length and directions, and be placed into the same design group using manual editing.

NOTE A member can only be placed into a single design group.

To manually reassign a member to a different group, locate it in the **Project Workspace**,**Groups tab**and drag it to a different group

NOTE A cross will appear if the element does not meet the Group requirements.

Regroup ALL Model Members

If you have made changes in Design Options that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Steel beam design

Click the following links to find out more:

- Steel beam overview (page 1209)
- Steel beam fabrication (page 1210)
- Steel beam restraints (page 1216)
- Deflection limits (page 1216)
- Camber (page 1217)
- Instability factor (page 1217)
- Beam web openings (page 1218)
- Steel beam torsion (page 1223)

Steel beam overview

Tekla Structural Designer will design steel beams for an international range of doubly symmetric I-sections, C-sections, rectangular and square hollow sections, single angles, double angles and tees for many different countries and also for many specific manufacturers. Plated beam design is also available for some head codes.

Steel beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

In addition to major axis bending, minor axis bending and axial loads are also considered.

In its simplest form a steel non-composite beam can be a cantilever, or a single member between supports to which it is pinned.

It can also be a continuous beam consisting of multiple members that do not, with the exception of the remote ends, transfer moment to the rest of the structure. Each span of a continuous beam can be of different section size, type and grade.

At the remote ends of the beam there are a number of options for the end fixity depending upon to what the end of the beam is connected. These are:

- Free end
- Moment connection
- Pin connection
- Fully fixed
- Partially fixed (% of 4EI/L)
- Partially fixed (linear spring)

The beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Steel beam fabrication

Fabrication types summary - all head codes

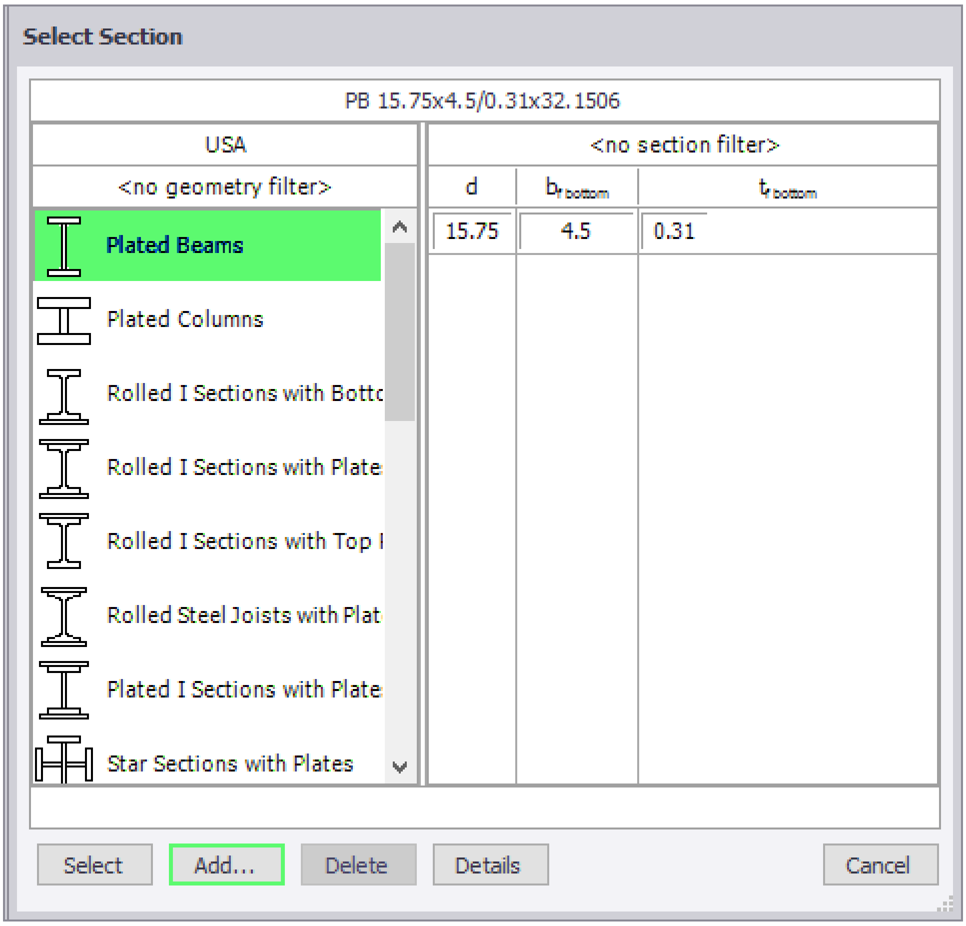
The fabrication types that can be designed as non-composite steel beams in Tekla Structural Designer are dependent on the design code.

Refer to the following table for details.

	Rolled	Plated	Westok cellular	Westok plated	FABSEC	DELTABEAM
AISC	Yes	Yes	No	No	No	No
Eurocode	Yes	Yes	Yes	Yes	No	No
BS	Yes	Yes	Yes	Yes	No	No
IS	Yes	Yes	No	No	No	No
AS	Yes	Yes	No	Yes	No	No

Plated beams - AISC head code

When the Fabrication type is set to Plated, only **Plated Beams** can be designed to the AISC 360-05, 360-10 and 360-16. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.



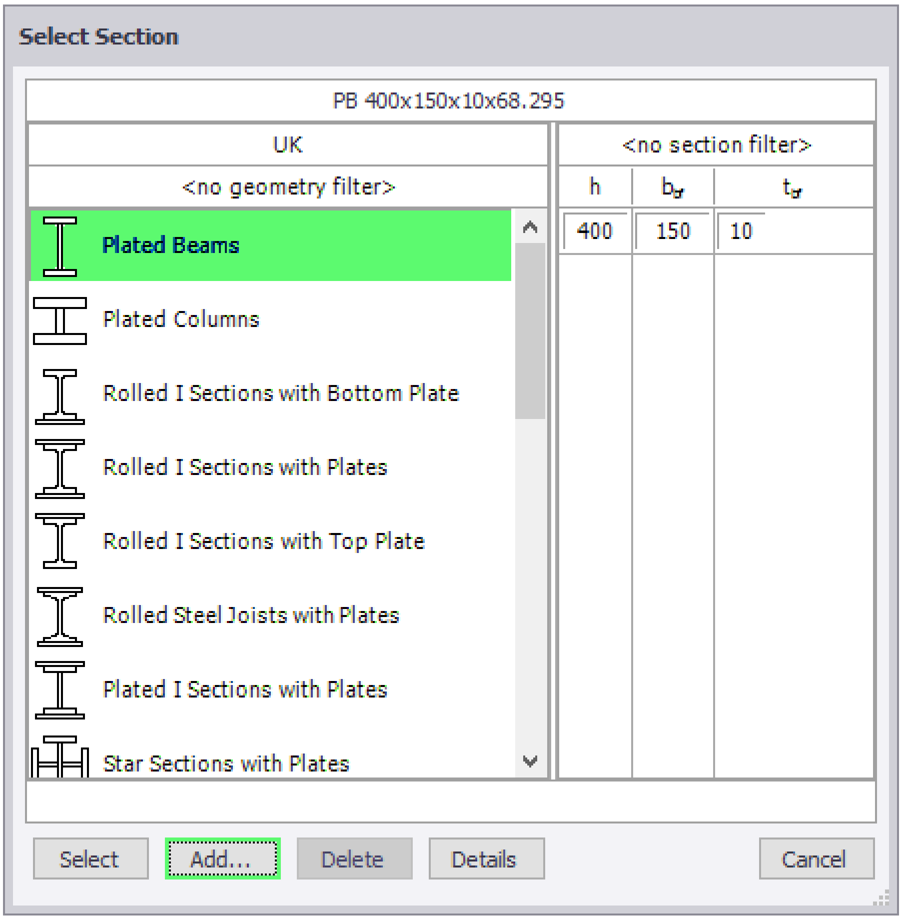
NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- Only parallel flange sections can be used.

Plated beams - Eurocode head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to Eurocodes. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

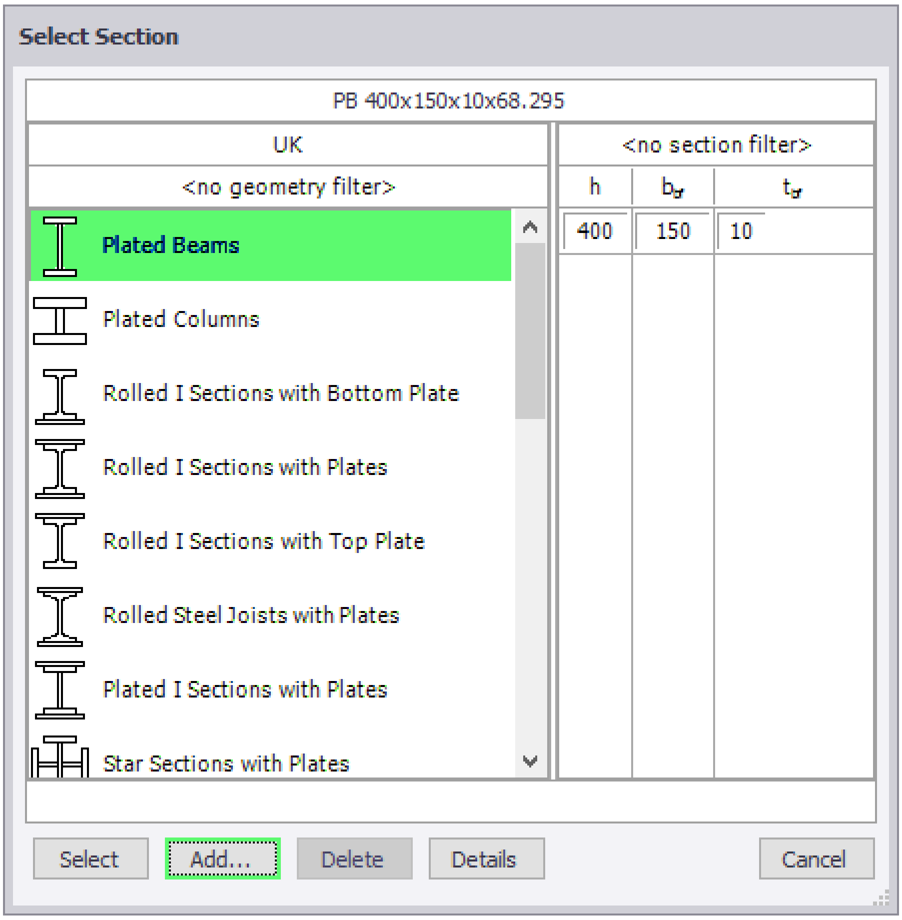


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - BS head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to BS 5950. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

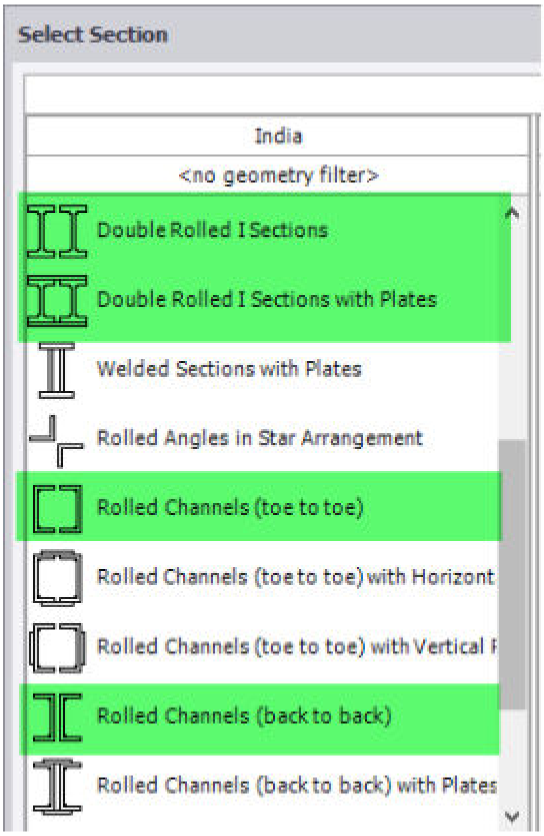


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - Indian head code

When the Fabrication type is set to Plated, a range of steel compound sections can be designed to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections
- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- High Shear case with minor axis moment is beyond scope.

- Design of the lacing or battening system is beyond scope.
- Only parallel flange sections can be used.

Plated beams - Australian head code

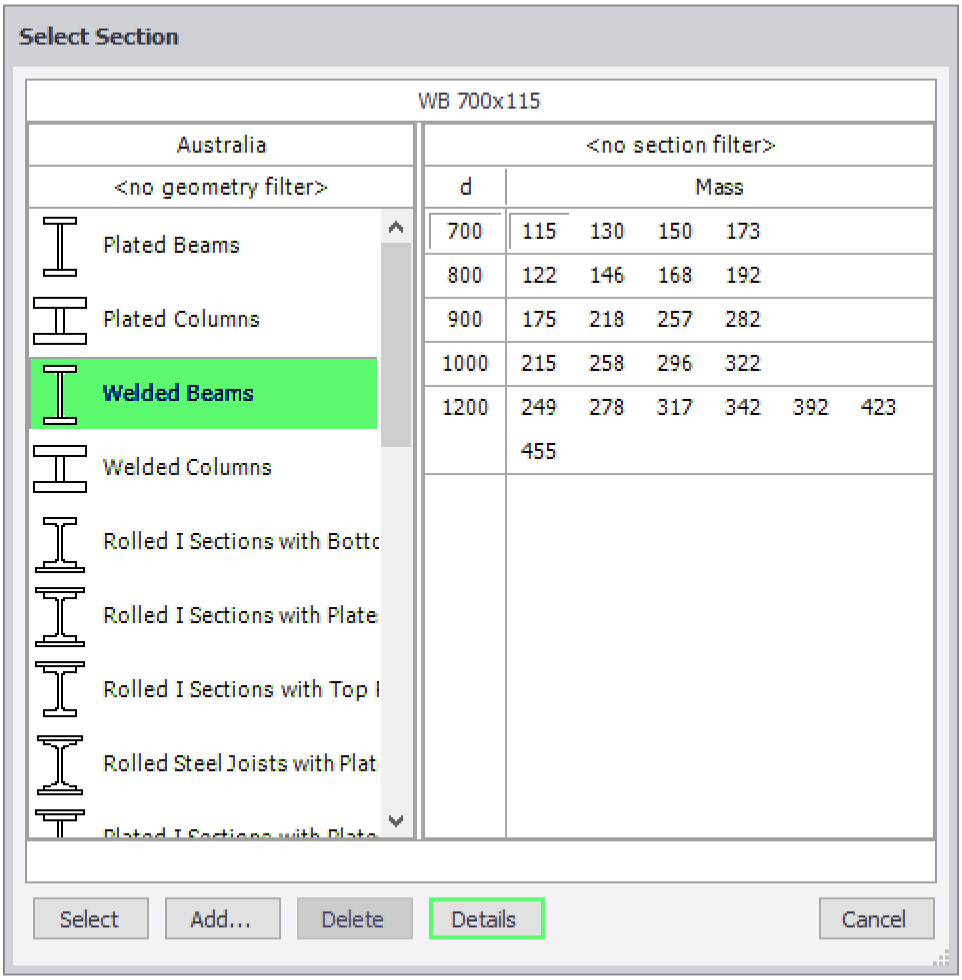
When the Fabrication type is set to Plated, only welded doubly symmetric I-section beams can be designed to the Australian code.

A pre-defined range of these is available from the **Welded Beams** page of the Select Section dialog - these can be both checked and autodesigned.

NOTE Autodesign requires a **Design section order** to be specified, the Welded Beams design section order is provided for this purpose.

While you can add to the range of welded beams the Select Section dialog by clicking the **Add...** button, any such user-defined beams can only be analyzed, but are not designed.

Plated beams available on other pages of the Select Section dialog can be analyzed, but are not designed.



Steel beam restraints

Each beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Conditions of restraint can be defined in- and out-of-plane for compression (strut) buckling and top and bottom flange for lateral torsional buckling (LTB). The effective lengths used in the buckling checks depend on the type of restraint, particularly at supports.

In all cases, the program sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. Any compression or LTB effective length can take the type "Continuous" to indicate that it is continuously restrained over that length.

LTB and Compression restraints

LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model.

The default assumption is that an incoming member connecting to either side of the beam provides lateral restraint to the top flange only, and compression restraint about the minor axis. An incoming member connecting from above or below doesn't provide lateral restraint but provides compression restraint about the major axis.

By right-clicking a member to edit its properties in the Property dialog, you are then able to edit the restraints. You can indicate also continuously restrained sub-beams and edit length factors.

Note that the same level of restraint editing is not provided in the Properties Window (although it does allow you to independently set both the top and bottom flanges as continuously restrained for the entire member length via the Top/Bottom flange cont. rest. properties).

TIP As an alternative to using the Steel Beam Property dialog, restraint settings for steel beams can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

---

Torsion restraints

Torsion restraints are currently displayed in the Property dialog for information only. In design the beam is treated as unrestrained against warping and restrained against torsion.

Continuously Restrained Flanges

In the Properties Window you can independently set both the top and bottom flanges as continuously restrained. By setting 'Top flange cont. rest.' and/or 'Bottom flange cont. rest.' to 'Yes' the relevant buckling checks are not performed during the design process.

Deflection limits

Serviceability criteria often control the design of normal composite beams. This is because they are usually designed to be as shallow as possible for a given span.

Deflection Limits allow you to control the amount of deflection in both composite beams and steel beams by applying either a relative or absolute limit to the deflection under different loading conditions.

A typical application of these settings might be:

- not to apply any deflection limit to the slab loads, as this deflection can be handled through camber,
- to apply the relative span/over limit for live load deflection, to meet code requirements,
- possibly, to apply an absolute limit to the post composite deflection to ensure the overall deflection is not too large.

Camber

Camber is primarily used to counteract the effects of dead load on the deflection of a beam. This is particularly useful in long span composite construction where the self-weight of the concrete is cambered out. It also ensures little, if any, concrete over pour occurs when placing the concrete.

The amount of camber can be specified either:

- As a value
- As a proportion of span
- As a proportion of dead load deflection
  - If this option is selected, the engineer should identify the combination to be used for the calculations by selecting Camber adjacent to the appropriate gravity combination on the Loading combination dialog page.
  - If no combination is selected then the first gravity combination in the combination list is used.

In the latter case, if 100% of the dead load deflection is cambered out, it is also possible to include a proportion of the live load deflection if required.

A lower limit can be set below which the calculated camber is not applied, this ensures that impractical levels of camber are not specified.

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because, for example, there are many smaller members or sheeting, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier can be applied to steel beams, composite beams and steel columns. It is defined in the properties window by selecting Prevent out of plane instability and then entering a suitable value in the Instability factor field.

NOTE This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, there is no amplification of the minor axis inertia in the design of the member.

Beam web openings

You cannot currently automatically design sections with web openings, you must perform the design first to get a section size, and then add and check the openings. This gives you complete control of the design process, since you can add appropriate and cost effective levels of stiffening if required, or can choose a different beam with a stronger web in order to reduce or remove any stiffening requirement.

When openings are added they can be defined as rectangular or circular and can be stiffened on one, or on both sides.

Openings cannot be defined from the Properties Window, they can only be defined from the Properties Dialog, (by right-clicking on the member and selecting Edit...)

For guidance in relation to a specific head code, see:

- Beam web openings to AISC (page 1218)
- Beam web openings to SCI P355 (page 1219)
- Beam web openings to SCI P068 (page 1222)

Beam web openings to AISC

Web Opening design limitations

- Simple beams only (single span, pin ended). Although web openings can be defined on multi-span beams design is beyond scope.
    - Cantilevers are excluded.

- Curved beams are excluded.
- Haunched and tapered members are excluded.
- Section type: rolled symmetrical compact I sections.
  - Plated sections are excluded.
- Steel yield strength is limited to 65 ksi
- Web opening placement and sizes: Opening depth is limited to 70 percent of beam depth. There are also two control parameters which dictate the dimensions of the opening, one being the aspect ratio, ao/ho, of the opening and the other being the opening parameter po. Both should meet the required limits.
- Multiple openings. Checks are performed to ensure that openings are spaced far enough apart so that design expressions for individual openings may be used.
- Openings cannot be placed closer than the section height to a support.
- Openings can be defined as rectangular or circular; circular openings are designed as equivalent rectangular openings.
- Openings can be reinforced on one side or both sides but always top or bottom, but no checks are performed on the reinforcement and its welds.
- No concentrated loads should be placed above an opening. If there is a point load above the opening then that combination gets a Beyond Scope status, (but only if the concentrated load is greater than 25% of the shear strength of the two tee sections).
- The nearest concentrated load should be placed at least d/2 from the edge of the opening.
- A check is performed to determine whether bearing stiffeners are necessary, if so these are not designed but a warning is shown.
- ASD design i.e. LRFD only.

NOTE For details of the web opening design checks performed see Web openings (Beams: AISC 360) (page 1678)

Beam web openings to SCI P355

NOTE When the Head Code has been set to Eurocode, Tekla Structural Designer adopts the following approach to web openings which is specific to the UK National Annex.

As each web opening is added it is checked against certain geometric and proximity recommendations taken from Table 2.1 of SCI Publication P355 (see below).

Guidance on size and positioning of openings

The following general guidance on size and positioning of openings is taken from Table 2.1 Section 2.6 of the SCI Publication P355

NOTE These geometric limits should normally be observed when providing openings in the webs of beams. It should be noted that these limits relate specifically to composite beams and caution should be used in applying these limits to non-composite beams.

Parameter	Limit	
	Circular Opening	**Rectangular Opening**
Max. depth of opening:	<= 0.8h	<= 0.7h
Min. depth of Tee,	>= t_f +30 mm	>= 0.1h
Min. depth of Top Tee:	As above	As above and >= 0.1 l_o if unstiffened
Max. ratio of depth of Tees: h_b/h_t h_b/h_t	<= 3 > = 0.5	<=2 >= 1
Max. unstiffened opening length, l_o Max. stiffened opening length, l_o	- - - -	<= 1.5 h_o high shear* <= 2.5 h_o low shear <= 2.5 h_o high shear* <= 4 h_o low shear
Min. width of web post: - Low shear regions -High shear regions	>= 0.3h o >= 0.4h o	>= 0.5 l_o >= l_o
Corner radius of rectangular openings:	-	r_o >= 2 t_w but r_o >= 15 mm
Min. width of end post, s_e:	>= 0.5 h_o	>= l_o and >= h
Min. horizontal distance to point load: - no stiffeners - with stiffeners	>= 0.5 h >= 0.25 h_o	>= h >= 0.5 h_o

* A high shear region is where the design shear force is greater than half the maximum value of design shear force acting on the beam.

Symbols used in the above table:

h = overall depth of steel section

h_o = depth of opening [diameter for circular openings]

h_t = overall depth of upper Tee [including flange]

h_b = overall depth of lower Tee [including flange]

l_o = (clear) length of opening [diameter for circular openings]

s_e = width of end post [minimum clear distance between opening and support]

t_f = thickness of flange

t_w = thickness of web

r_o = corner radius of opening

In addition, the following fundamental geometric requirements must be satisfied.

$d_o <= 0.8*h$ for circular openings

$d_o <= 0.7*h$ for rectangular openings

$d_o < 2 * (d_{oc} - t_t - r_t)$

$d_o < 2 * (h - d_{oc} - t_b - r_b)$

$d_2 < d_{oc} - d_o/2 - t_t- t_s/2$

$d_2 < h - t_b- d_{oc} - d_o - t_s/2$

$l_o < 2 * L_c$

$l_o < 2 * (L - L_c)$

$L_s < 2 * L_c$

$L_s < 2 * (L - L_c)$

where

d_t= the depth of the web of the upper tee section measured from the underside of the top flange

d_{oc} = the distance to the centre line of the opening from the top of the steel section

d_2 = the distance from the edge of the opening to the centre line of the stiffener

t_s = thickness of stiffener [constrained to be the same top and bottom]

t_t = the thickness of the top flange of the steel section

t_b = the thickness of the bottom flange of the steel section

r_t = root radius at the top of the steel section

r_b = root radius at the bottom of the steel section

L_c = the distance to the centre line of the opening from the left hand support

L = the span of the beam

NOTE Dimensional checks - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udls and in a low shear zone for point loads). This is because for anything other than simple loading the best position becomes a question of engineering judgment or is pre-defined by the service runs.

NOTE Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Beam web openings to SCI P068

As each web opening is added it is checked against certain geometric and proximity recommendations taken from SCI Publication P068.

Guidance on size and positioning of openings

We advise you to comply with the following positional recommendations for web openings:

- Web openings are designed using the bending moment and vertical shear values at the side of the opening where the moment is lower,
- Openings should preferably be positioned at the mid-height of the section. If not, the depth of the upper and lower sections of web should differ by not more than a factor of two,
- Openings should not be located closer to the support than two times the beam depth or 10% of the span whichever is the greater,
- The best location for any opening is between 1/5 and 1/3 of the span from a support in uniformly loaded beams, or in lower shear zone of beams subject to point loads,
- Openings should be not less than the beam depth, D, apart,
- Unstiffened openings should not generally be deeper than 0.6D or longer than 1.5D,
- Stiffened openings should not generally be deeper than 0.7D or longer than 2D,
- Point loads should not be applied at less than D from the side of the adjacent opening.

NOTE Dimensional checks - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udls and in a low shear zone for point loads). This is because for anything other than simple loading the best position becomes a question of engineering judgment or is pre-defined by the service runs.

NOTE Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Steel beam torsion

Torsion can only be checked to the AISC or Eurocode head code and only under the following conditions:

- Section geometry
  - doubly symmetric rolled I-sections
  - closed RHS/SHS/CHS sections (steel and cold formed)
  - closed HSS sections
- pinned
  - single span beams
  - applied torsion moment
  - no web openings

If any of the above conditions are contravened the check is beyond scope.

Only a check design is performed, (no auto-design for torsion).

The procedure and scope are different for Open sections (I's) vs closed sections (HSS's)

- Open Sections:
  - Torsion design and angle rotation check will be carried out for applied torsion forces only. "Inherent" torsion is not checked
- Closed Sections
  - "Inherent" torsion and/or physically applied torsion loads checked
  - Angle of rotation check carried out for applied forces only

Fire check (Eurocode only)

The mechanical resistance of a steel beam in case of fire can be checked in accordance with EN 1993 & national annex for the UK, Ireland, Singapore, Malaysia, Sweden, Norway, Finland or the recommended Eurocode values.

The fire check is activated via the beam properties, after which the user is required to specify:

- Load reduction factor for fire
- Required time of fire exposure
- Exposure
- Protected, or unprotected
  - If the protected option is selected, the engineer is then required to specify the fire protection material details.

The time interval for critical temperature iteration is specified in Design Settings, for both 'unprotected' and 'protected' situations.

The check compares the design shear force against the design resistance at the required time of fire exposure.

For the scope of the check and the limitations that apply, see: Fire resistance check (Beams: EC3 Eurocode) (page 1901)

Composite beam design

Click the following links to find out more:

Composite beam overview

Composite beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

The beams must be simply supported, single span unpropped structural steel beams.

The following are beyond scope:

- continuous or fixed ended composite beams,
- composite sections formed from hollow rolled sections,
- composite sections where the concrete slab bears on the bottom flange,
- the use of fibre reinforcement.

Beams are designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered.

NOTE If either minor axis bending or axial loads exist which exceed a limit below which they can be ignored, a warning is given in the beam design summary.

Profiled steel sheeting can be perpendicular, parallel and at any angle in between relative to the supporting beam web.

Tekla Structural Designer will determine the size of beam which:

- acting alone is able to carry the forces and moments resulting from the Construction Stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to carry the forces and moments at Composite Stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to provide acceptable deflections, service stresses and natural frequency results.

Alternatively you may give the size of a beam and Tekla Structural Designer will then determine whether it is able to carry the previously mentioned forces and moments and satisfy the Serviceability requirements.

An auto-layout feature can be used for stud placement which caters for both uniform and non-uniform layouts.

Composite beam loading

All loads must be positive since the beam is considered as simply supported and no negative moment effects are accommodated.

Construction stage loading

You define these loads into one or more loadcases as required.

The loadcase defined for construction stage slab wet concrete has a Slab wet loadcase type specifically reserved it. Clicking the Calc Automatically check box enables this to be automatically calculated based on the wet density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, this loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the wet concrete. If you do not do this then you effectively would be designing the beam on the assumption that it is propped at construction stage.

It is usual to define a loadcase for Live construction loads in order to account for heaping of the wet concrete etc.

Having created the loadcases to be used at construction stage, you then include them, together with the appropriate factors in the dedicated Construction stage design combination. You can include or exclude the self-weight of the beam from this combination and you can define the load factors that apply to the self weight and to each loadcase in the combination.

NOTE You should include the construction stage slab wet concrete loadcase in the Construction stage combination, it cannot be placed in any other combination since it's loads relate to the slab in its wet state. Conversely, you cannot include the Slab self weight loadcase in the Construction stage combination, since it's loads relate to the slab in its dry state. The loads in the Construction stage combination should relate to the slab in its wet state and any other loads that may be imposed during construction.

NOTE TIP: If you give any additional construction stage loadcases a suitable title you will be able to identify them easily when you are creating the Construction stage combination.

Composite stage loading

You define the composite stage loads into one or more loadcases which you then include, together with the appropriate factors in the design combinations you create. You can include or exclude the self-weight of the steel beam from any combination and you can define the load factors that apply to the beam self weight and to each loadcase in the combination.

The Slab self weight loadcase is reserved for the self weight of the dry concrete in the slab. Clicking the Automatic Loading check box enables this to be automatically calculated based on the dry density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, the Slab self weight loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the dry concrete. For each other loadcase you create you specify the type of loads it contains – Dead, Live or Wind.

For each load that you add to an Live loadcase you can specify the percentage of the load which is to be considered as acting long-term (and by inference that which acts only on a short-term basis).

All loads in Dead loadcases are considered to be entirely long-term while those in Wind loadcases are considered entirely short-term.

Composite beam fabrication

Fabrication types summary - all head codes

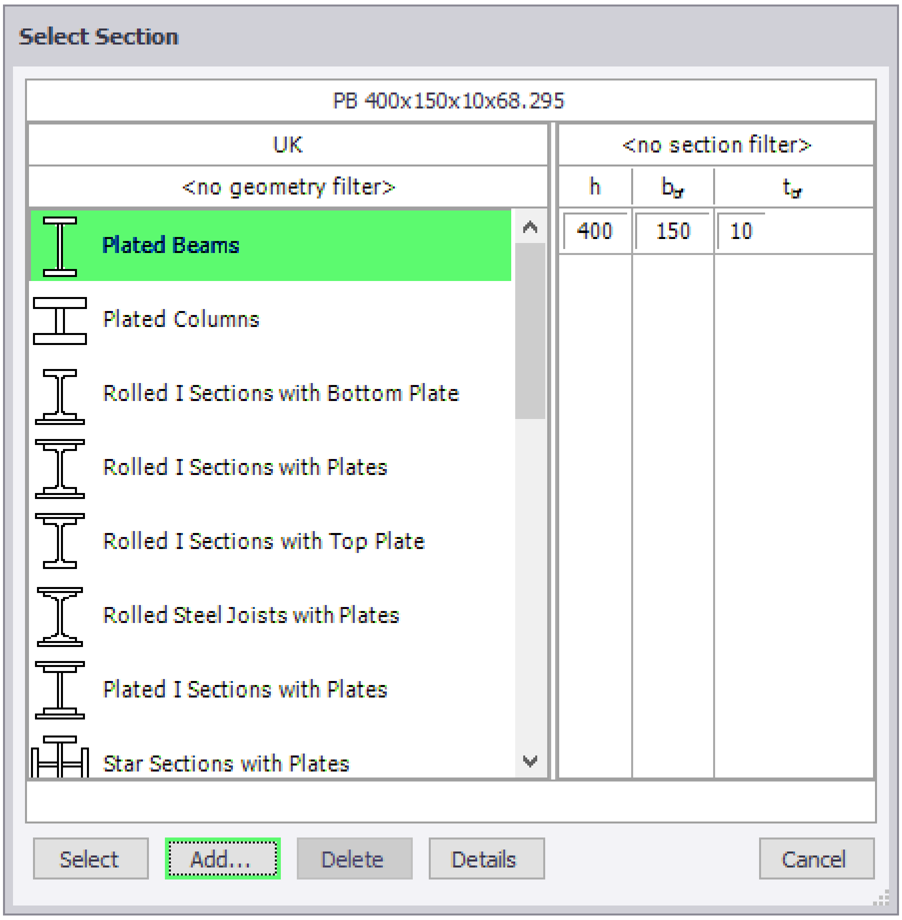
The fabrication types that can be designed as composite steel beams in Tekla Structural Designer are dependent on the design code.

Refer to the following table for details.

	Rolled	Plated	Westok cellular	Westok plated	FABSEC	DELTABEAM
AISC	Yes	No	No	No	No	No
Eurocode	Yes	Yes	Yes	Yes	No	No
BS	Yes	Yes	Yes	Yes	No	No
IS	N/A - Design of composite beams to IS codes is currently beyond scope					
AS	N/A - Design of composite beams to AS codes is currently beyond scope					

Plated beams - Eurocode head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to Eurocodes. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

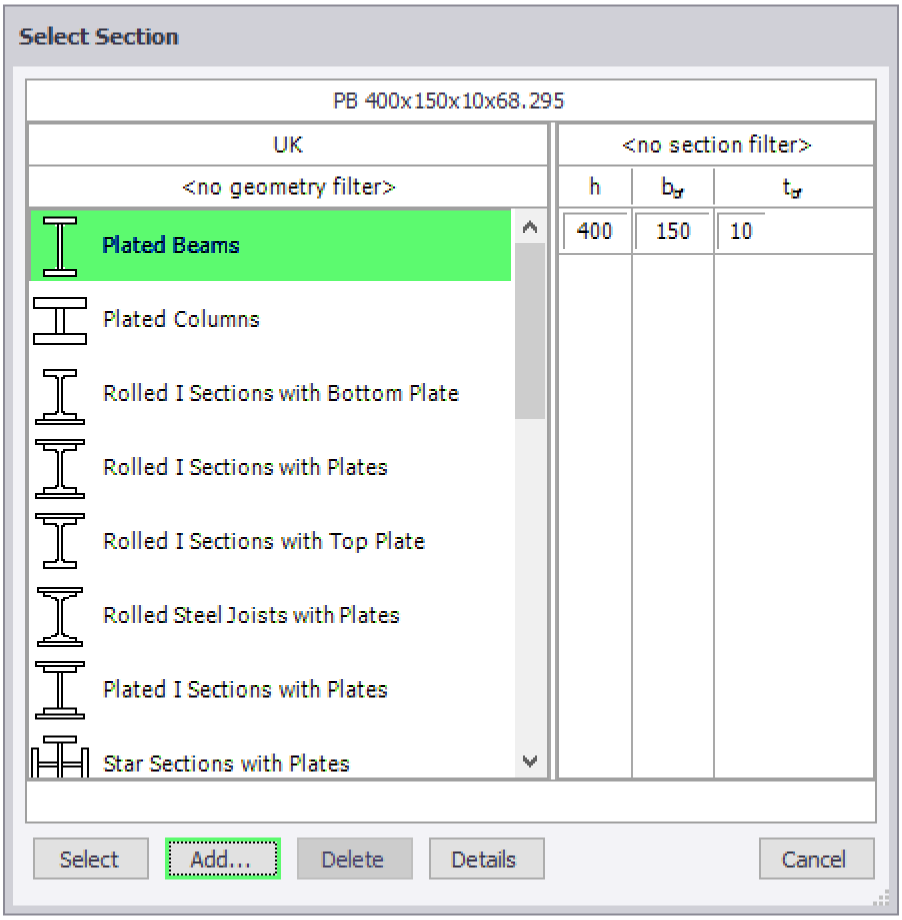


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - BS head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to BS 5950. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.



NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Composite floor construction

Deck Type, Angle and Condition

The deck type and angle used in the beam design are determined from the properties of an adjacent slab item. If there are multiple adjacent slab items with different properties, it is the users responsibility to indicate which one governs.

- When specifying the slab item properties you will find that a wide range of profiled metal decks have been included for manufacturers from many countries. PC Planks are also available, but only for the EC Head Code.

- The slab item’s rotation angle relative to the global X axis is used to set the profiled metal deck as spanning at any angle between 0° (parallel) and 90° (perpendicular) to the direction of span of the steel beam.

The beam’s “condition” is:

- restricted to internal if it has composite slabs attached along its full length on both sides,
- restricted to edge if it has no composite slabs on one side,
- defaulted to edge (but editable) if it has composite slabs on both sides but not along the full length.

Shear connector type

The shear connection between the concrete slab and the steel beam is achieved by using shear studs.

NOTE The use of channel connectors or Hilti™ connectors is currently beyond the program scope.

Head Code: Eurocode, BS

19mm diameter studs with 100 and 125 nominal height (95 and 120 as welded height) are offered. 22mm diameter studs are also offered but only for precast plank decks. Studs do not have a given capacity as their resistance is derived.

Effective width used in the design - Head Code: ACI/AISC

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

   NOTE If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.

   The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

Tekla Structural Designer will calculate the effective width of the compression flange, be for each composite beam as per Section I3.1a (360-05/-10).

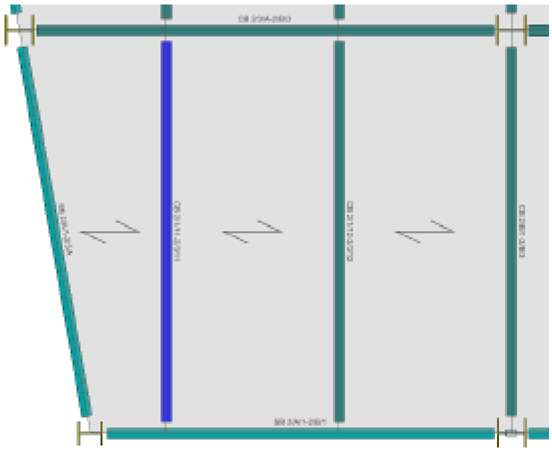
For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the center to center of supports

- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

Although the program calculates b_e, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgement may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, b_e(right) = beam span/8
- to the left of the beam, b_e(left) = one half of the shortest distance to the center line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure Design > Settings > Composite beam - Update effective width prior to design check is checked on
2. The floor construction properties will then display an Override effective width option.

a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.

b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Effective width used in the design - Head Code: Eurocode

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

> **NOTE** If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.
>
> The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

Tekla Structural Designer will calculate the effective width of the compression flange, b_{eff}, for each composite beam as per Clause 5.4.1.2 of EC4.

Unless hollowcore units are used, it is taken as the smaller of:

- Secondary beams: the spacing of the beams, or beam span/4
- Primary beams (conservatively): 80% of the spacing of beams, or beam span/4
- Edge beams: half of above values, as appropriate, plus any projection of the slab beyond the centreline of the beam.

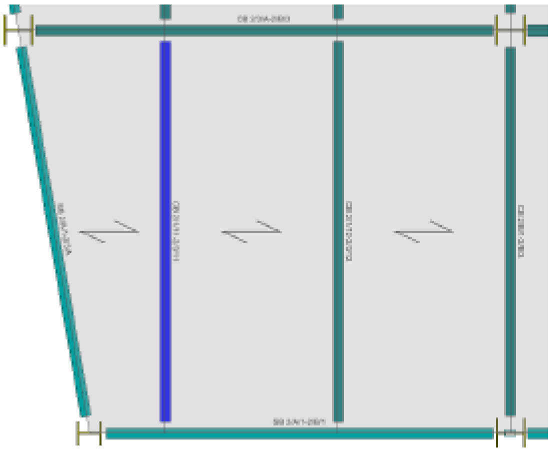
For hollowcore precast plank units only, we calculate the effective width for each side of the beam as the minimum of:

- Assumed concrete fill (500 mm) + recommended gap
- beam span/8 – span taken as the center to center of supports
- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

These effective breadths are used in both strength and serviceability calculations.

Although the program calculates beff, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgement may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, beff(right) = beam span/8
- to the left of the beam, beff(left) = one half of the shortest distance to the center line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure Design > Settings > Composite beam - Update effective width prior to design check is checked on
2. The floor construction properties will then display an Override effective width option.
    a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.

b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Effective width used in the design - Head Code: BS

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

NOTE If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.

The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

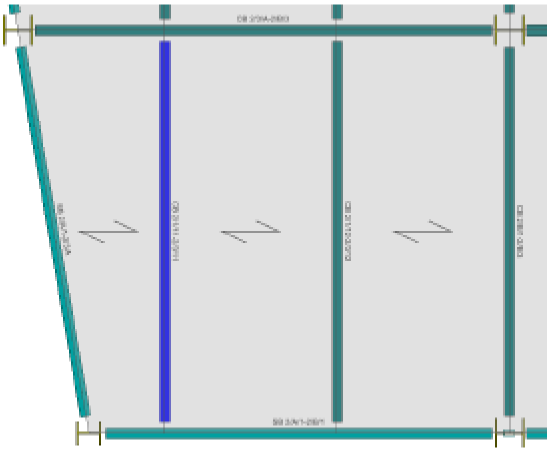
Tekla Structural Designer will calculate the effective width of the compression flange, be, for each composite beam as per section 4.6 of BS 5950 : Part 3 : Section 3.1 : 1990.

For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the centre to centre of supports
- one half of the distance to the centre line of the adjacent beam (for slabs spanning perpendicular)
- 40% of the distance to the centre line of the adjacent beam (for slabs spanning parallel)
- the distance to the edge of the slab

Although the program calculates be, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgement may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, be(right) = beam span/8
- to the left of the beam, be(left) = one half of the shortest distance to the centre line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure Design > Settings > Composite beam - Update effective width prior to design check is checked on
2. The floor construction properties will then display an Override effective width option.
    a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.
    b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Precast concrete planks (Eurocode only)

NOTE The design of composite beams with precast concrete planks is only available for Eurocodes. It is not currently supported for other Head Codes.

General limitations and assumptions

The following limitations and assumptions apply to the use of precast concrete planks:

Cross-section classification is restricted to Classes 1 & 2.

As per normal composite beams there is no requirement to check for transverse force as it is assumed there are no loads or support conditions that would necessitate this.

A balanced condition is assumed during the construction stage and the top flange of the beam is treated as laterally restrained in construction. This condition should be evaluated against the particular application, if it is not suitable then it should be cleared.

Both hollow core units and solid planks are assumed to act compositely only with perpendicular secondary beams and not primary beams parallel to the span of the PC units. Beams neither parallel nor perpendicular to the PC slab are termed angled and are also designed non-compositely.

Precast unit

Design of the precast units themselves is not carried out. It is assumed that the application and loading conditions of the particular precast unit is justified before design of the composite beam is carried out.

The ability to model slab openings is not restricted. The effect an opening has on the behaviour of the precast plank is however, not taken into account and the engineer should verify this to be safe.

P401 restricts the design of composite beams with precast concrete units to the following:

- Hollow core units with circular or circular elongated openings along their length (150mm – 260mm deep). It is assumed all hollow core units modelled will have circular or circular elongated cores. Cores with other cross-sectional geometries may need additional design and verification, this is beyond the scope of SCI P401
- Solid precast planks (75mm – 100mm deep)
- Downstand beams

Deeper units can be chosen than the sizes stated above, however design will not be carried out.

It is assumed that the concrete infill does not contribute to the overall weight of the slab.

If a solid slab is chosen, the contribution of the precast slab is ignored in resistance and stiffness calculations.

If a hollow core unit is chosen, the contribution of the concrete topping is ignored in resistance and stiffness calculations.

Steel Beam

The following applies to steel beam sections:

- Minimum flange width
    - Internal beam - 220mm for shop-welded and 235 for site welded shear connectors.
    - Edge beam – 2 * (6 * stud diameter)

  These recommendations can be reduced by decreasing the bearing - special provisions must be made after consultation with both the precast manufacturer and the steelwork providers*
- Web openings are ignored in design*
- No significant point loads are applied to the composite beam*
- Beam must not behave as a cantilever
- For solid precast units only the topping is to be included as the joints between the units may not be in good contact
- For hollow core units only the precast plank will be taken into account during design
- A warning is issued in this case and subsequent design is carried out assuming that the engineer has justified the particular condition as safe.

Bearing

A default bearing of 75mm minimum is used. This can be reduced but the engineer must consult the precast manufacturer and steelwork provider. The minimum flange width is therefore also reduced.

Concrete properties

Overall properties of the slab should be specified. It is up to the engineer to decide those that govern the overall slab (PC plank + topping). It is these properties that are used to carry out design calculations and slab self-weight.

Loading

Slab self-weight

The concrete infill in the hollow cores is not taken into account in the calculation of the overall weight of the slab.

Where either no topping or structural topping is used, both dry and wet overall self-weight is calculated from the self-weight of the precast unit plus any topping. Where a non-structural topping is used, the engineer is expected to input the overall self-weight of the slab themselves.

Significant Point Loads

Significant point loads are beyond the scope of design in SCI P401. A warning is issued if a significant point load is present on the composite beam. Subsequent design calculations are carried out assuming the point load has no effect on the composite behaviour of the beam. The engineer must carry out additional hand calculations to justify this assumption is safe.

Shear Connectors

19mm and 22mm diameter shear studs are allowed in composite design with precast planks.

Should the engineer choose to place shear connectors in pairs, no dimensional check is carried out. It is assumed that the engineer has justified their use in pairs.

Longitudinal Shear

It is assumed the shear force is divided equally between the two sides of the beam flange.

The factors that influence the longitudinal shear capacity of your composite beam are:

- Concrete strength, slab depth and slab width – you cannot change these independently for the longitudinal shear check, since they apply equally to the entire composite beam design,
- The areas of Transverse and Other reinforcement which you provide in your beam

Transverse

Transverse reinforcement is designed to ensure VEd ≤ VRd. Additional reinforcement to that detailed in design may be required for other purposes.

Refer to SCI P401 for recommended minimum bar sizes and spacing of transverse reinforcement. In the case of a solid slab, the additional mesh is ignored in transverse reinforcement calculations as only either mesh or loose bars can be chosen. Additional mesh, however, can be applied to the slab reinforcement – see “Other”.

It is possible to increase the maximum spacing of transverse reinforcement from that shown in SCI P401 Table 3.1, however it must be noted that this is being done under the engineer’s own judgement.

EN 1992-1-1, 6.2.4 is used to determine the design resistance VRd to the longitudinal shear at the potential failure surface a-a (shown in Figure 4.7 in SCI P401). Failure surface b-b however is not checked in design.

Refer to SCI P401 for recommendations on the detailing of transverse reinforcement and minimum bar length.

Other

Any “other” slab reinforcement in the topping applied to a hollow core unit is ignored in design.

Composite Moment of Inertia

When determining the moment of inertia of a composite section with a hollow core unit, the section is taken as a solid slab (i.e. hollow cores aren’t taken into account).

For stiffness calculations the concrete below the neutral axis is considered as it will contribute some stiffness. However when carrying out resistance calculations, this concrete is ignored.

Concrete slab

NOTE	While you can define concrete slabs in both normal and lightweight concrete, design using lightweight slabs is only available for the Eurocode.

Eurocode

Warnings are issued in the design if you do not comply with the following constraints:

- Normal weight concrete range C20/25 - C60/75 - See EN 1994-1-1:2004 Clause 3.1(2),
- Lightweight concrete range LC20/22 - LC60/66 - See EN 1994-1-1:2004 Clause 3.1(2),
- Minimum density for lightweight concrete 1750 kg/m3 - see EN 1994-1-1:2004 Clause 6.6.3.1(1).

Metal deck

Minimum lap distance

The position and attachment of the decking is taken into account in the longitudinal shear resistance calculations.

The applied longitudinal shear force is calculated at the center-line of the beam, and at the position of the lap (if known). If the position of the lap is not known, then the default value of 0mm should be used (that is the lap is at the center-line of the beam) as this is the worst case scenario.

Stud strength

The stud properties you can choose from are appropriate to the stud source.

All types of stud may be positioned in a range of patterns.

Stud groups - Head Code ACI/AISC

You can allow group sizes of 1, 2, 3 or 4 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 3 or 4 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 2 studs in a group.

For each group that you allow you must enter the 'Set distance emid to' - as either >= 2in or < 2in and you also specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

Stud groups - Head Code Eurocode or BS

You can allow group sizes of 1 or 2 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 2 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 1 stud in a group.

For groups with 2 studs you must specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

NOTE It is up to you to check that a particular pattern fits within the confines of the rib and beam flange since Tekla Structural Designer will draw it (and use it in design) anyway.

Optimize shear interaction

If you choose the option to optimize the shear interaction, then Tekla Structural Designer will progressively reduce the number of studs either until the minimum number of studs to resist the applied moment is found, until the minimum allowable interaction ratio is reached or until the minimum spacing requirements are reached. This results in partial shear connection.

Connector layout

When running in Auto-design mode you may not want to specify the stud layout at the start of the design process. To work in this way check Auto-layout to have the program automatically control how the stud design will proceed. When the beam is subsequently designed Auto-layout invokes an automatic calculation of the required number of studs, which is optimized to provide an efficient design.

NOTE 'Auto layout' can actually be checked regardless of whether you are auto designing the beam size or checking it. The combination of 'Check' design with 'Auto layout' of studs can be used to assist you to rationalize your designs e.g.

to force a beam to be the same size as others in the building but have Tekla Structural Designer determine the most efficient layout of studs.

You may choose to perform the initial design with Auto-layout checked and then refine the spacing with Auto-layout cleared if the spacing is not exactly as you require. This may arise if for instance the theoretical design needs to be marginally adjusted for practical reasons on site.

Auto-layout for perpendicular decks

For perpendicular decks, the Auto-layout dialog provides two options for laying out the studs:

- Uniform
- Non-uniform

Uniform

The Uniform option forces placement in ribs at the same uniform spacing along the whole length of the beam.

Whether the stud groups are placed in every rib (as shown above), alternate ribs, or every third rib etc. can be controlled by adjusting the limits you set for Minimum group spacing ( ) x rib and Maximum group spacing ( ) x rib.

The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

NOTE Example:

If you set Minimum group spacing 2 x rib and Maximum group spacing 3 x rib, then the program will only attempt to achieve a solution with studs placed in alternate ribs, or studs placed in every third rib. It will not consider a solution in which studs are placed in every rib.

Additionally, if on the Studs - Strength page, you have allowed groups of 1 stud and 2 studs; then if 1 stud per group proves to be insufficient the program will then consider 2 studs per group.

Non-uniform

If optimization has been checked (see Optimize shear interaction (page 1239)) studs are placed at suitable rib intervals (every rib, alternate ribs, every third rib etc.), in order to achieve sufficient interaction without falling below the minimum allowed by the code.

NOTE	The optimum and minimum amounts of shear connection are defaulted to 50% and 25% respectively. These can be adjusted if required.

If optimization has not been checked, studs are placed at suitable rib intervals in order to achieve 100% interaction.

Knowing the number of studs necessary to achieve the required level of interaction, it is possible that placement at a given rib interval could result in a shortfall; the program will attempt to accommodate this by working in from the ends, (as shown in the example below). If every rib is occupied and there is still a shortfall, the remainder are 'doubled-up', by working in from the ends once more.

In this example the point of maximum moment occurs one third of the way along the span, this results in an asymmetric layout. If you prefer to avoid such arrangements you can select Adjust layout to ensure symmetrical about centerline. A redesign would then result in the symmetric layout shown below.

For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

Auto-layout for parallel decks

For parallel decks, the Auto-layout again provides Uniform and Non-uniform layout options, but the way these work is slightly different.

Uniform

The Uniform option forces placement at a uniform spacing along the whole length of the beam. The spacing adopted will be within the limits you set for Minimum group spacing distance and Maximum group spacing distance. If the point of maximum moment does not occur at mid span, the resulting layout will still be symmetric.

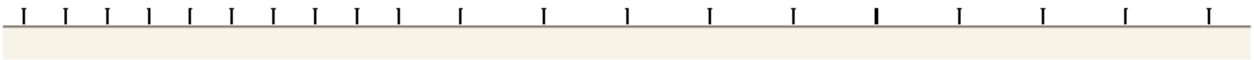
The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

Non-uniform

If optimization has been checked (see Optimize shear interaction (page 1239)) studs are placed at a suitable spacing in order to achieve sufficient interaction without falling below the minimum allowed by the code.

If optimization has not been checked, studs are placed at a suitable spacing in order to achieve 100% interaction.

If the point of maximum moment does not occur at mid span, the resulting non-uniform layout can be asymmetric as shown below.



For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

Manual Stud Layout

You may prefer to manually define/adjust the group spacing along the beam. This can be achieved by unchecking Auto layout.

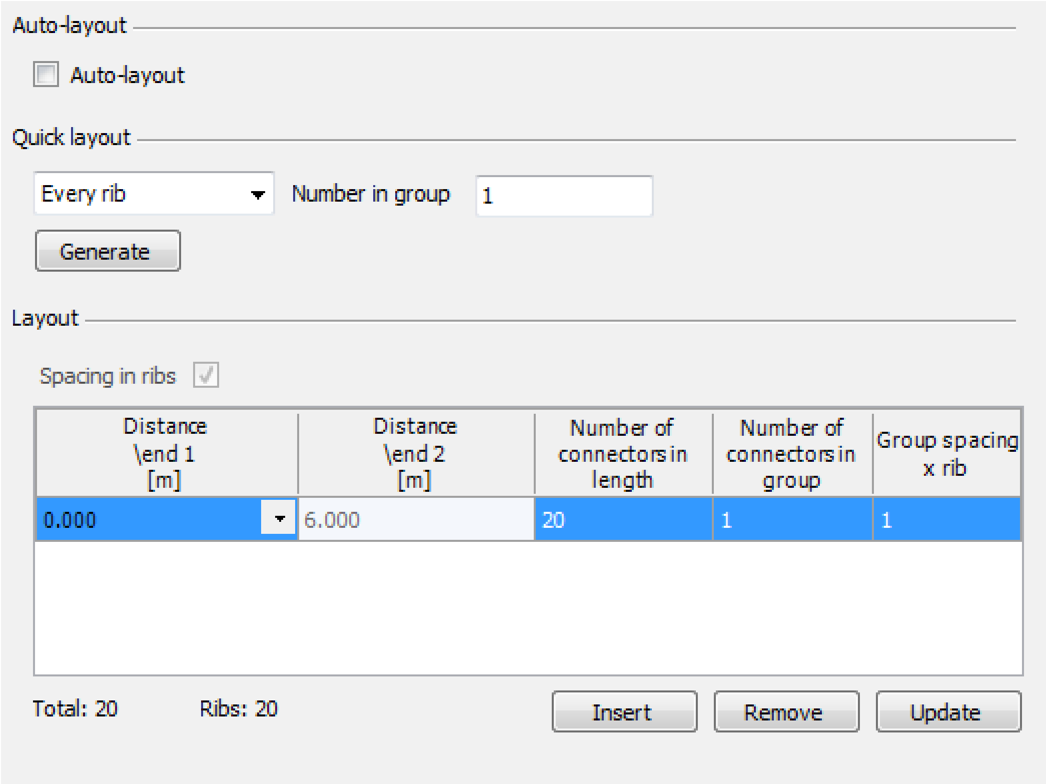
NOTE If you specify the stud spacing manually, then it is most important to note:

- the resulting design may not be the optimal design possible for the beam, or
- composite design may not be possible for the stud spacing which you have specified.

To generate groups of studs at regular intervals along the whole beam use the Quick layout facility. Alternatively, if you require to explicitly define the stud layout to be adopted for discrete lengths along the beam use the Layout table.

Manual layout for Perpendicular decks

For perpendicular decks, the dialog for manual layouts is as shown:

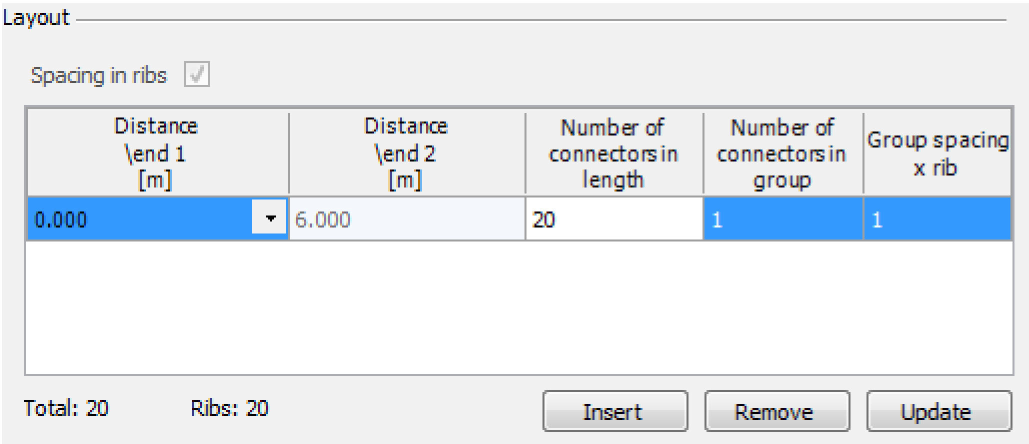


To use Quick layout, proceed in one of two ways:

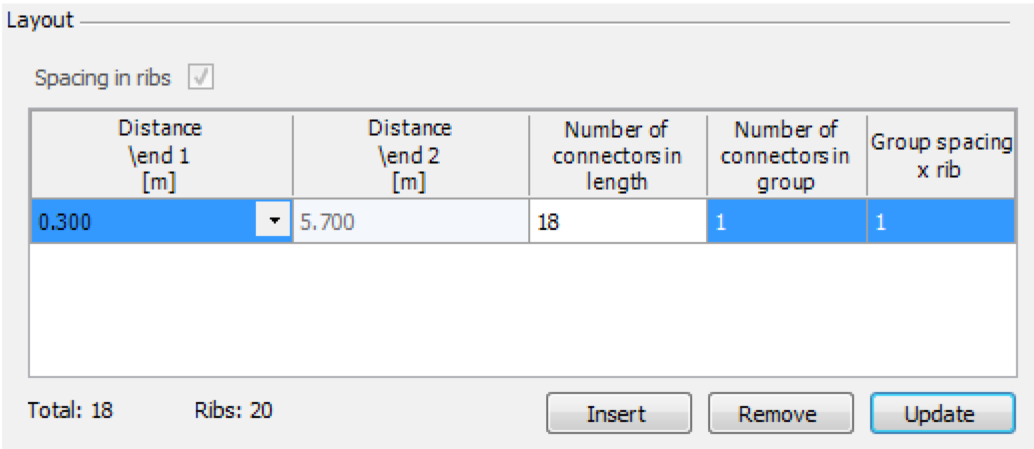
- Choose to position groups in either every rib, or alternate ribs, then specify the number of studs required in the group and click Generate.
- Alternatively: specify the total number of studs, then when you generate, if the number specified is greater than the number of ribs, one will be placed in every rib and the remainder will be 'doubled-up' in the ribs at each end starting from the supports. Similarly if the number specified is less than the number of ribs, but greater than the number of alternate ribs, one will be placed in every alternate rib and the remainder will be placed in the empty ribs. Limits of 600mm or 4 x overall slab depth, (whichever is less), are considered.

To use the Layout table:

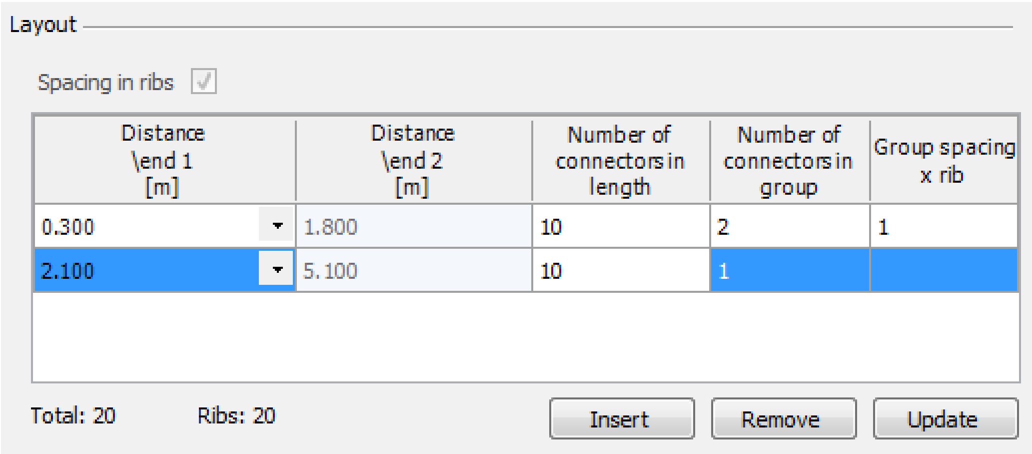
- For each segment you should define the following parameters: No. of connectors in length and No. of connectors in group; Group spacing x rib.



Your input for these parameters is used to automatically determine Distance end 2 - this latter parameter cannot be adjusted directly, hence it is dimmed.

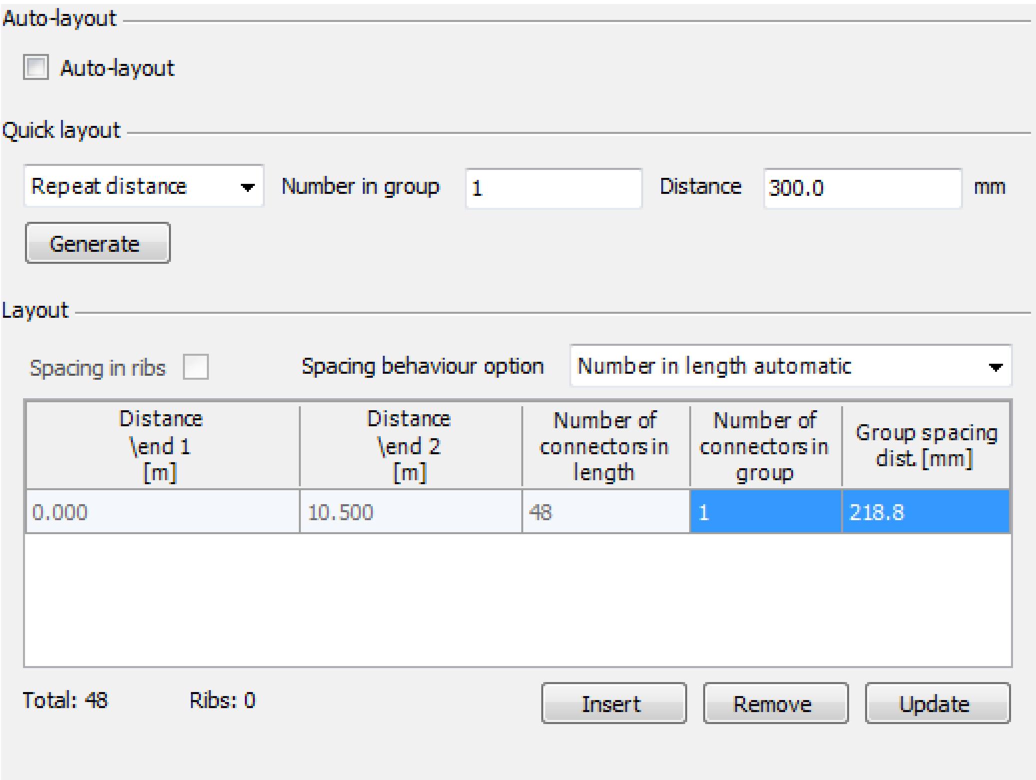


- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify a different stud layout for each segment.
- We would advise that having entered No. of studs in length, group and spacing and ignoring Distance ends 1 and 2 you click Update, this will automatically fill in the missing fields.



Manual layout for Parallel decks

For parallel decks, the dialog for manual layouts is as shown:



To use Quick layout, proceed in one of two ways:

- Choose to position groups at a set repeat distance, then specify the number of studs required in the group and click Generate.

- Alternatively: specify the total number of studs, then click Generate - the program calculates the repeat distance automatically, subject to the code limits.

To use the Layout table:

- The preferred method is to choose the option Spacing distance automatic, in which case you can adjust the No. of connectors in length and No. of connectors in group. Alternatively you could choose the option Number in length automatic and then adjust No. of connectors in group and Group spacing dist.
- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify Distance end 1 for each new segment and it's own stud layout.

Composite beam restraints

You can independently set both the top and bottom flanges of a composite beam as continuously restrained in the **Properties** window.

When the beam is initially created the decking direction is unknown until the beam is actually placed and the floor slab and direction are also created. Hence defaults are provided for each eventuality.

The defaults are:

- for perpendicular decks the deck restrains the beam top flange
- for parallel decks the deck does NOT restrain the beam top flange
- for precast decks the deck restrains the beam top flange
- for all decks the deck does NOT restrain the beam bottom flange

By setting the top flange as continuously restrained and/or the bottom flange as continuously restrained the relevant buckling checks are not performed during the design process.

When not continuously restrained, LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model. The buckling checks are based on these.

By right-clicking a member to edit its properties in the **Property dialog**, you are then able to edit the restraints. You can indicate continuously restrained sub-beams and also edit length factors.

For composite beams the buckling checks are only performed at construction stage as at composite stage they are always assumed to be fully restrained.

TIP Restraint settings for composite beams can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

Composite beam natural frequency

A natural frequency check can optionally be requested. When activated a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behaviour i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer has the option to perform a 1st order modal analysis.

Composite beam transverse reinforcement

This reinforcement is be provided specifically to resist longitudinal shear.

Since the profile metal decking can be perpendicular, parallel or at any other angle to the supporting beam the following assumptions have been made:

- if you use single bars they are always assumed to be at 90° to the span of the beam,
- if you use mesh then it is assumed to be laid so that the main bars are at 90° to the span of the beam.

The reinforcement you specify is assumed to be placed at a position in the depth of the slab where it is able to contribute to the longitudinal shear resistance

Automatic transverse shear reinforcement design

It is possible to automatically design the amount of transverse shear reinforcement for each beam. This is achieved in Tekla Structural Designer by checking the Auto-select option on the Transverse reinforcement tab of the Composite Beam Properties.

NOTE The Auto-select option for designing transverse shear reinforcement is only available when the beam is in auto-design mode.

If you are checking a beam, then you must specify the transverse shear reinforcement that you will provide, and then check out this arrangement.

The auto-selected bars can be tied into the stud group spacing by checking the Bar spacing as a multiple of stud spacing option. Alternatively, the spacing can be controlled directly by the user.

Bar spacing as a multiple of stud spacing

When the option Bar spacing as a multiple of stud spacing is checked, the Transverse Reinforcement tab provides the user with controls on the bar size and the multiples of stud spacing.

These can be used to achieve a selection of say, 12mm diameter bars at 2 times the stud spacing, with a slightly greater area than a less preferable 16mm diameter bars at 4 times the stud spacing.

Controlling the bar spacing directly

When the option Bar spacing as a multiple of stud spacing is not checked, the Transverse Reinforcement tab provides the user with direct control on the bar size and the bar spacing.

Allow non-composite design

Typically, at the outset you will know which beams are to be non-composite and which are to be composite and you will have specified the construction type accordingly. However, circumstances can arise in which a beam initially intended to be composite proves to be ineffective. Examples might be:

- very small beams,
- beams with a significant point load close to a support,
- beams where the deck is at a shallow angle to the beam, hence the stud spacing is impractical,
- beams where, for a variety of reasons, it is not possible to provide an adequate number of studs, and
- edge beams, where the advantages of composite design (e.g. reduced depth) are not so clear

Where Tekla Structural Designer is unable to find a section size which works compositely, you can ask for a non-composite design for the same loading. You will find that this facility is particularly useful when you right-click a key beam in the model in order to perform an individual member design.

To invoke non-composite design

1. Select the composite beam(s) as required.
2. In the Properties Window select Allow non-composite design

Steel column design

Click the following links to find out more:

- Steel column overview (page 1250)
- Simple columns (page 1251)

Steel column overview

Tekla Structural Designer allows you to analyse and design a structural steel column which can have moment or simple connections with incoming members, and which can have fixity applied at the base. The column can have incoming beams which may also be capable of providing restraint, and may have splices along its length at which the section size may vary. You are responsible for designing the splices appropriately.

In its simplest form a steel column can be a single pin ended member between construction levels that are designated as floors.

More typically it will be continuous past one or more floor levels, the whole forming one single entity typically from base to roof.

Steel columns that share moments with steel beams form part of a rigid moment resisting frame.

In all cases you are responsible for setting the effective lengths to be used appropriate to the provided restraint conditions. All defaults are set to 1.0L.

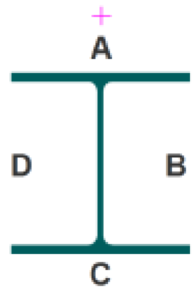
Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not considered.)

If working to either AISC or Eurocodes, the design will take into consideration additional moments resulting from the eccentricity of pinned beam end connections. If working to other head codes, while the eccentricity moments are calculated, they are not used in the design.

Steel member orientation

Tekla Structural Designer considers member orientation when displaying analysis results. Therefore, to apply the sign convention correctly, you need to know which is the end 1 and which is the end 2 of the member. For columns you also need to be able to identify the four faces: A, B, C & D.

If you select the **Direction** option for a member in **Scene Content**, Tekla Structural Designer displays a direction arrow the member which points points from end 1 to end 2 of the member. For columns the direction is always from bottom to the top and the arrow is always drawn adjacent to **Face A**. Looking down from the top of a column, Face B, C, and D then follow in the clockwise direction.



Simple columns

A steel column can be designated as a `simple column' - in which case specific design rules are required.

A simple column should not have any applied loading in its length.

Simple columns are subject to axial forces and moments due to eccentricity of beam reactions.

In order to prevent end fixity moments you would have to manually pin the ends of the column.

NOTE The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as “beyond scope” when designed.

Steel column fabrication

Fabrication types summary - all head codes

The steel column fabrication types that can currently be designed in Tekla Structural Designer are dependent on the design code, and also the construction type specified in the column properties - refer to the following table for details.

Construction: Non-composite column

	Rolled	Plated	Concrete filled	Concrete encased
AISC	Yes	No	No	No
Eurocode	Yes	No	No	No
BS	Yes	No	No	No
IS	Yes	Yes	No	No

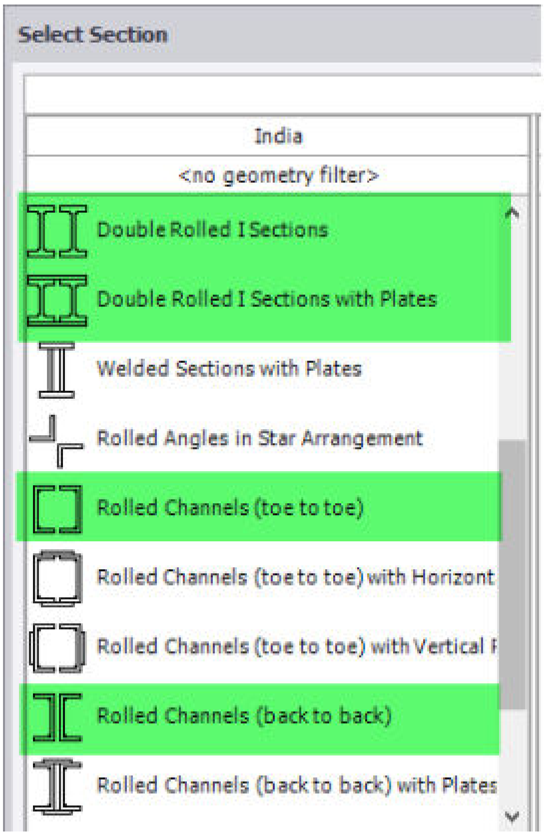
AS	Yes	Yes	No	No

Construction: Composite column

	Rolled	Plated
AISC	No	No
Eurocode	No	No
BS	No	No
IS	No	No
AS	No	No

Plated columns - Indian head code

When the Fabrication type is set to Plated, a range of steel compound sections can be designed to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections

- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created (page 1008) specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- High Shear case with minor axis moment is beyond scope.
- Design of the lacing or battening system is beyond scope.
- Only parallel flange sections can be used.

Plated columns - Australian head code

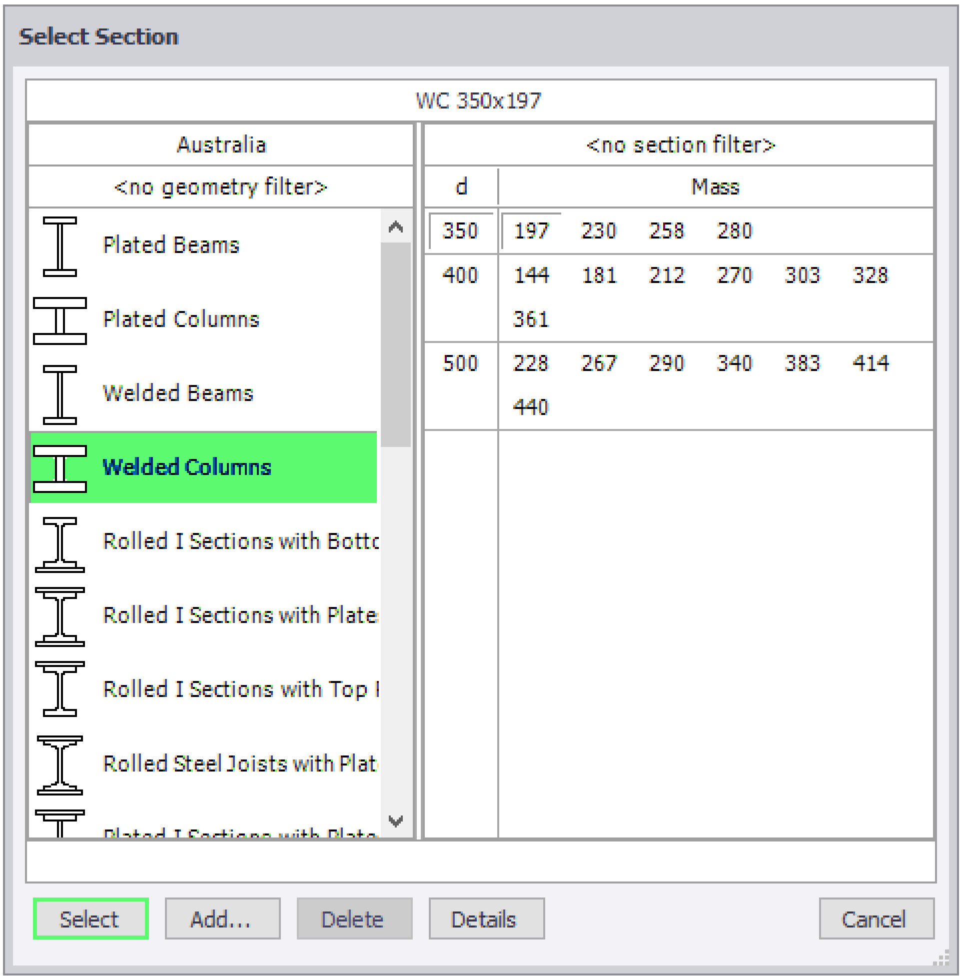
When the Fabrication type is set to Plated, only welded doubly symmetric I-section columns can be designed to the Australian code.

A pre-defined range of these is available from the **Welded Columns** page of the Select Section dialog - these can be both checked and autodesigned.

NOTE Autodesign requires a **Design section order** to be specified, the Welded Columns design section order is provided for this purpose.

While you can add to the range of welded columns in the Select Section dialog by clicking the **Add...** button, any such user-defined columns can only be analyzed, but are not designed.

Plated columns available on other pages of the Select Section dialog can be analyzed, but are not designed.



Composite, concrete filled, and concrete encased steel columns

Whilst composite columns, concrete filled hollow sections and concrete encased sections can be specified in Tekla Structural Designer they are not designed.

For composite columns and concrete filled hollow sections, the analysis uses the bare steel inertia and not the 'effective composite inertia'. This is conservative as the lower stiffness of the bare section will promote more second-order effects. On the other hand for concrete encased columns Tekla Structural Designer takes into account the encasement based on the size specified by the user.

When working to the US head code: AISC 360-16, sub-section I1.5 deals with the calculation of stiffness for concrete filled hollow sections and concrete encased sections used in the Direct Analysis Method (DAM). In accordance with C2.3, Tekla Structural Designer takes account of the standard stiffness adjustment factor of 0.8 with τb set to 1.0 given the additional notional load of 0.1% is also applied. I1.5 requires these types of member to take τb as 0.8

when considering the flexural stiffness. The user can achieve this for concrete encased columns by adjusting the Modification Factors in the Analysis Settings (page 2278). (For composite columns and concrete filled hollow sections which both use the bare steel stiffness, no adjustment is required).

Steel column restraints

Restraints to flexural and torsional buckling are determined from the incoming members that connect to the column. The buckling checks are based on these restraints.

Restraints are considered effective on a particular plane providing they are within ±45° to the local coordinate axis system.

In all cases Tekla Structural Designer sets the default unrestrained length factor between restraints to 1.0.

You have the control to set any unrestrained length to be continuously restrained over that length - when set in this way the relevant buckling check is not performed during the design process.

NOTE The Steel Column Properties window only allows you to set entire stacks as either continuously restrained or unrestrained. In order to specify restraints between the incoming members within each stack it is necessary to open the Steel Column Property dialog instead.

TIP As an alternative to using the Steel Column Property dialog, restraint settings for steel columns can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

Lateral torsional buckling (LTB):

- Members framing into either Face A or C are by default assumed to provide full LTB restraint. You therefore need to consider whether or not your particular configuration of incoming members is capable of providing this level of LTB restraint. If necessary you can edit the default restraint provision by selecting the Face A override and/or Face C override checkboxes as appropriate.

  NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 1250)

Compression/Strut buckling:

- Members framing into either Face A or C will by default provide restraint against major axis compression buckling. Members framing into either Face B or D will by default provide restraint against minor axis compression buckling. You can remove these default restraints if required by selecting the Major override and/or Minor override checkboxes.

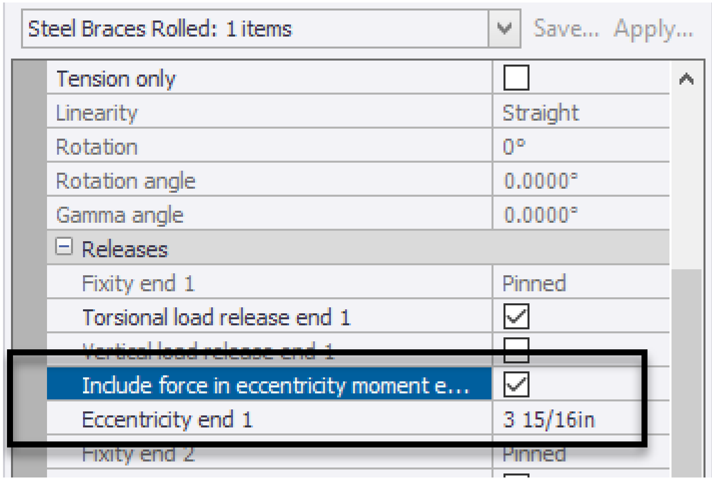
Steel column connection eccentricity moments

Overview

Nominal eccentricity moments that arise from beam end reactions are considered in Tekla Structural Designer's steel column design.

These moments do not come directly from the global analysis but instead are calculated at the ‘load analysis’ post-processing stage as follows:

- At each level the eccentricity of each connection is taken as half the depth of the supporting column, plus an additional user defined offset from the column face.
- At each level the pinned beam end reactions connecting to the column at each face are determined.
- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

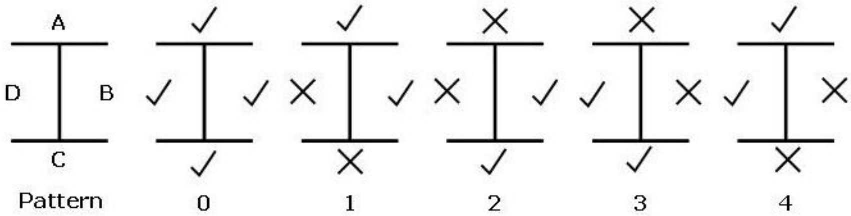
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are ‘true’ patterns switching live load ‘on’ and ‘off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design

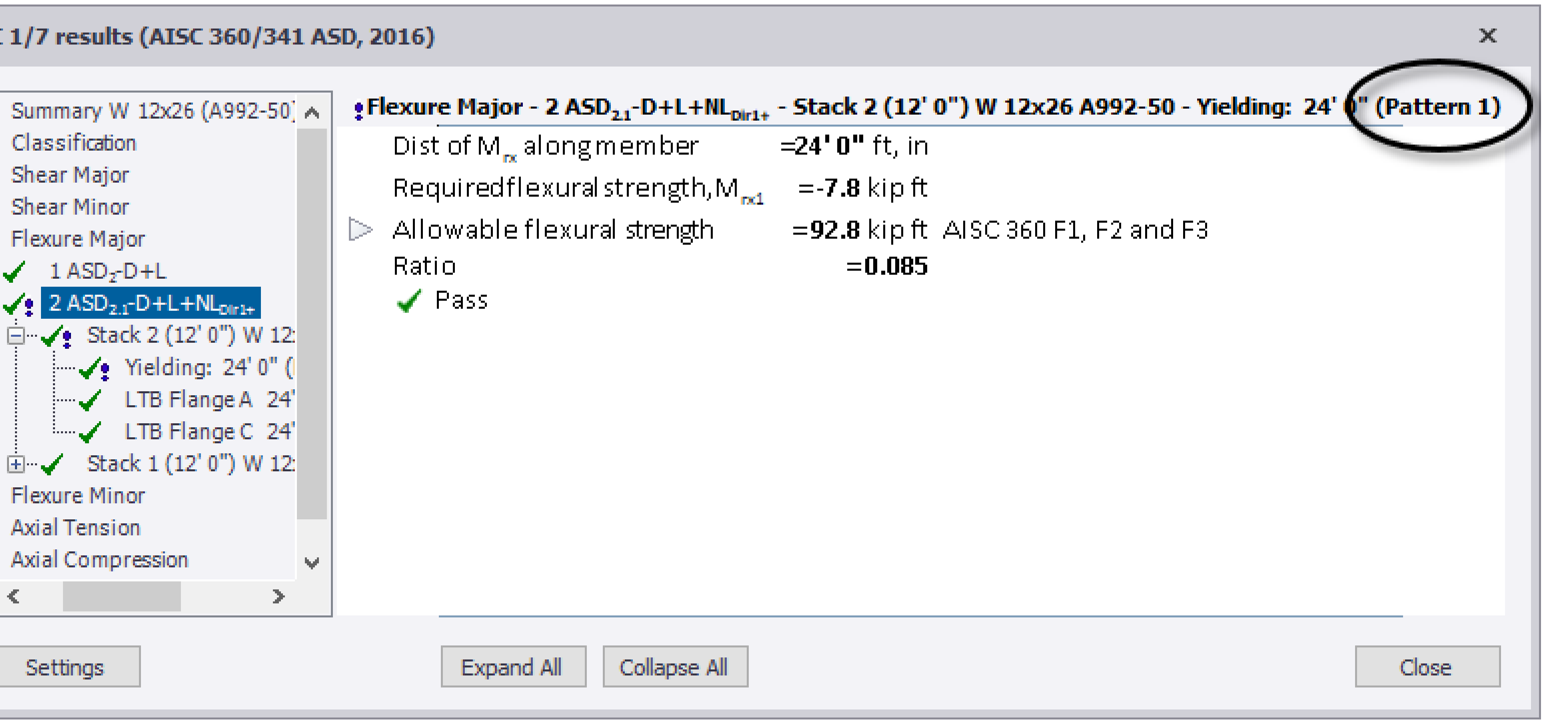
NOTE Patterned eccentricity moments are only considered in the design for the **AISC** or **Eurocode** head codes. If working to other head codes, while the patterned eccentricity moments are calculated, only the fully loaded pattern (P0) is used in the design.

In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

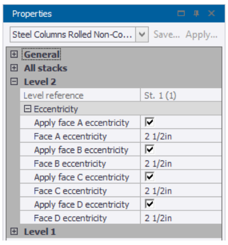
- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.
- Since it is not known whether a reduced axial force with more eccentricity moment is a worse case than full axial and a lower (or even zero, in the balanced case) eccentricity moment, the design loops through all patterns in order to consider each eventuality.
- The patterned eccentricity moments are considered in all design checks apart from 'Shear', (which is unaffected).

- To keep the design details to a manageable level, results for every pattern are not listed; the pattern which produces the governing design forces is listed in the check combination and location tree and details heading, (as shown below).



Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



As long as the option to apply eccentricity at a face is checked, the total eccentricity at that face is taken as half the dimension of the supporting column, plus the additional eccentricity from the face as specified in the **Properties** window shown above.

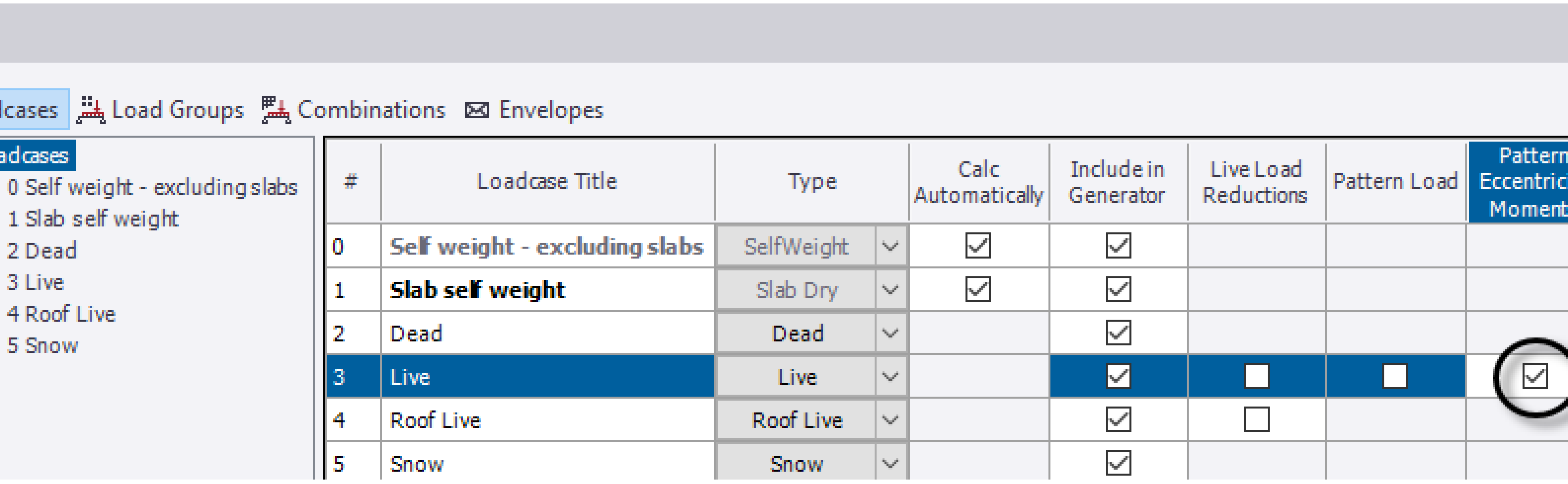
If you uncheck the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 1250)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
    a. Click **Model Settings > Loading > General**
    b. Select **Use patterning of eccentricity moments for steel columns**
    c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
    a. Select a live loadcase that you want to be patterned
    b. Select **Pattern Eccentricity Moments for Steel Columns**
    c. When patterning has been selected for each of the required loadcases, click **OK**

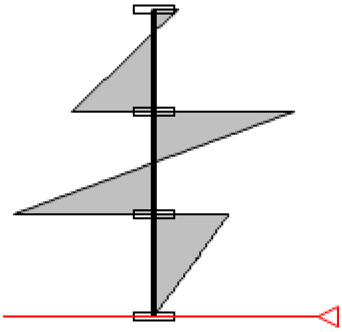


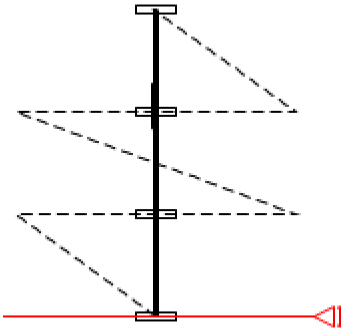
Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View (page 716).

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:



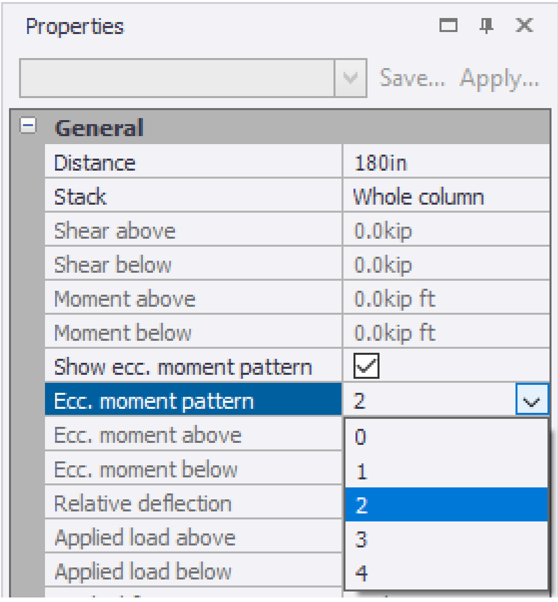


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.



Splice and splice offset

Splices are allowed at floor levels only and must be placed at changes of angle between two adjacent stacks and at changes of section size or type. A validation error will result if this is not the case. The splice can be given an offset from the floor level - the default of 500mm is considered not to be structurally significant.

You must detail the splice to resist the applied forces and moments. The detail should provide continuity of stiffness and strength. Splices given considerable offset should be designed to take account of the P-d moment (also known as moment induced by strut action) at the position as well as the forces from the analysis.

NOTE To remind you of the above requirement the following warning is issued when splice loads are exported to BIM as part of the analysis results: *"The loads given do not include any additional moment due to member buckling."* A similar warning is also issued when splice loads are included in reports. This is an explicit requirement in the British Standards (Clause 6.1.8.2 of BS 5950-1), although as the same engineering principle applies the world over, we issue it for all head codes.

Each lift (length between splices) of a general column can be of different section size and grade. Different section types within the same column are not allowed due to the particularly complex design routines that general columns require. You are responsible for guaranteeing that the splice detail ensures that the assumptions in the analysis model are achieved and that any difference in the size of section between lifts can be accommodated practically.

Steel column web openings

NOTE In the current release of the program the design or checking of columns with web openings is “Beyond Scope”

You can define rectangular or circular openings and these can be stiffened on one, or on both sides.

Web openings can be added to a column by a 'Quick-layout' process or manually.

The 'Quick-layout' process, which is activated using the check box on the Web openings dialog page, adds web openings which meet certain geometric and proximity recommendations (taken from Table 2.1 of SCI Publication P355 if the Head Code is set to Eurocode, or taken from SCI Publication P068 if the Head Code is set to BS). The openings so created are the maximum depth spaced at the minimum centers recommended for the section size.

Web openings can also be defined manually. With Quick-layout cleared, the `Add' button adds a new line to the web openings grid to allow the geometric properties of the web opening to be defined.

Steel brace design

Steel brace overview

Tekla Structural Designer allows you to analyze and design a steel member with pinned end connections for axial compression and tension.

Steel braces can be specified as rolled I-sections, C-sections, T-sections, rectangular, square and circular hollow sections, angles, double angles, and flat sections.

NOTE If working to the US head code, a brace specified as a flat section can only be analyzed, but design is beyond scope.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

Applied loading

The following points should be noted:

- Loads for the brace are derived from the building model.
- Element loads cannot be applied directly to the brace itself.

- Live and Roof Live load reductions are not applied.
- Moments due to self weight loading are ignored.

Design Forces

The design forces for strength checks are obtained from an analysis of the entire structure. Braces can be subject to axial compression or tension, but will not be subject to major and minor axis bending.

Input method for A and V Braces

A and V Braces should be modeled using special tools which can be found on the 'Steel Brace' drop list in the 'Steel' section on the 'Model' tab.

Although it is also possible to model the exact same brace arrangement using individual elements created using the simple 'Steel Brace' command, it is important to note that whilst the Notional Loads \ EHFs (Equivalent Horizontal Forces) calculated for models built using the A or V Brace tools are correct, this is not the case when the A or V braces are built up out of individual brace members. In this latter case, elements of the vertical loads that are supported by the bracing system are 'lost' and are not included in the Notional Load \ EHF calculations with the result that the calculated Notional Loads \ EHFs are not correct.

Steel brace in compression

Effective length factors are defined for each axis of buckling.

- Effective length factor y-y
- Effective length factor z-z

NOTE If designing to British Standards, see Steel brace in compression - BS 5950-1:2000 (page 1263)

Steel brace in tension

The net area of the section is required for tension checks. This can be specified either as:

- Percentage value
- Effective net area

NOTE If designing to British Standards, see Steel brace in tension - BS 5950-1:2000 (page 1264)

Steel brace in compression - BS 5950-1:2000

If either an Angle (single or double) or Channel or Tee section has been selected as the brace member then a 'Connection at each end' property box appears which allows selection of an appropriate Clause from Table 25 of the BS code to describe the connection type.

The default Clauses from Table 25 are as follows:

Single Angle	4.7.10.2a
Double Angle	4.7.10.3d
Channel	4.7.10.4b
Tee	4.7.10.5b

Notes:

1. Clauses 4.7.10.2b, 4.7.10.2c, 4.7.10.3b, 4.7.10.3c, 4.7.10.3d and 4.7.10.3e only apply to Bolted connections so in these cases 'Bolted' should be selected in the 'Connection' property box. ('Bolted' is the default connection.)
2. For Angle (single and double), Channel and Tee sections the Effective Length Factors will be auto-completed according to the connection Clause selected but these Factors can be changed if required. For Angle (single and double) sections the length Lv-v is always assumed to be L/3 and the Effective Length Factor v-v will act on this Lv-v.
3. For single Angle sections the longer leg is assumed to be the connected element unless 'Short attached' is checked on the Size page of the dialog.
4. For double unequal Angle sections, whichever leg is not the back-to-back leg is assumed to be the connected element when connection Clauses 4.7.10.3a and 4.7.10.3b are selected, and vice versa with Clauses 4.7.10.3c, 4.7.10.3d and 4.7.10.3e

Steel brace in tension - BS 5950-1:2000

Brace tension capacity is calculated according to section type as follows:

A. Hollow sections (CHS, RHS, SHS):	Gross area capacity
B. Angle, Channel, Tee sections:	Reduced effective net area capacity (Clauses 4.6.3.1 and 4.6.3.2)
C. I/H and any other sections:	Effective net area capacity (Clause 4.6.1)

Notes:

1. For section types B and C listed above, an Effective net area (Ae) should be entered either as a percentage of the gross area (Ag) or as an absolute

value. The default is to use 100% of Ag, and an absolute value cannot be used if autodesign is also selected.

2. The 'Geometry' property needs to be selected manually if autodesign is also selected, otherwise the 'Geometry' property does not appear visible.

Steel joist design

Steel joist design overview

Steel joists, (or bar joists), are a specific type of members used in the United States. They are simply supported secondary members that do not support any other members - they only support loaded areas, i.e. slab and roof loads. Steel joists are constrained to standard types specified by the US Steel Joist Institute, and standardized in terms of span, depth and load carrying capacity.

In Tekla Structural Designer steel joist design is performed in accordance with the 44th Edition of the Steel Joist Institute (SJI) specification, which uses a similar approach to that embodied in AISC 360-05/10/16 LRFD and ASD.

Standard types

Steel joists are constrained to standard types specified by the Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity. There are four standard types of steel joist available in Tekla Structural Designer.

- K series joists - parallel chord steel joists (2 variations; rod or angle webs) - depths 8" to 30" with spans up to 60ft.
    - Including 2.5 K series joist substitutes - a depth of 2.5in, intended to be used for spans up to 10ft.
- KCS series joists - K series adapted and specially designed for constant moment/shear along length (position of point loads become irrelevant).
- LH series joists - long span joists - depths 18" to 48" for clear spans up to 96ft.
- DLH series joists - deep long span joists - depths 52" to 120" for clear spans up to 240ft.

Special Joists

"SP" suffixes can be added to K, LH and DLH Series joists. Special Joists can handle 'non-uniform' loading situations. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Joist Girders

These are provided as an option to support steel joists. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Assumptions and limitations

The following assumptions and limitations currently apply to the design of steel joists in Tekla Structural Designer:

- All steel joists are considered as simply supported members.
- Steel joists cannot be released axially.
- Cantilevered joists cannot be defined.
- Joist Girders are able to support other joists and brace members, other joists are only able to support brace members.
- Joist inertia and area values are taken directly from the Steel Joist Institute tables, with the exception of Joist Girders for which you must provide the relevant data.
- For all joist types. any resulting load, other then those in the major axis, that exceeds the user specified Ignore Forces Below setting on Design Options is reported as a Warning in the results viewer along with the type and value of force detected.
- Design is currently beyond scope for SP joists and Joist Girders.
- For all steel joists, it is assumed that the top chord is sufficiently braced against lateral torsional buckling.
- There is no restriction on the minimum span for which a joist can be defined even though due to their open web nature joists can be almost impossible to fabricate for very small spans. The user should check the suitability of using such a joist in these situations.
- For steel joists which support a generic concrete slab, it is assumed that the minimum concrete slab thickness of 2 inches (50mm) is present. (SJI Steel Joist Specification 5.9.2). This is not checked by Tekla Structural Designer.
- In design, the user is expected to refer to the bridging requirements in the SJI Specification and decide the appropriate details for the relevant scenario. This is not checked by Tekla Structural Designer.
- 'Non uniform' loads are accommodated by KCS joists. E.g. parapet snow drift load, partition walls. If no KCS joists can be found then SP joists can be used for these loads but these are not designed/checked.
- Top chord extensions used as eaves and awnings are not designed.
- Camber of the joists is not shown in the graphics nor handled by Tekla Structural Designer.
- The design and specification of the joist seats (regular or sloping) is not handled by Tekla Structural Designer.
- Double joist configurations cannot be defined in Tekla Structural Designer.

- The design does not consider the minimum bearing requirements for K, KCS, LH or DLH joists – these are the responsibility of the designer/ engineer.
- Customisation of KCS joists to accommodate any applied concentrated loads is not considered byTekla Structural Designer.
- Loadings for accessories to the joist are not included in the standard load tables. An allowance for this should be included by the designer in the loading of the model.
- Steel joists are not designed for composite action and when supported by conventional composite a validation warning is issued. Composite Joists, CJ Series, cannot be defined in Tekla Structural Designer.
- Moment connections at steel joist ends are not allowed.
- Duct openings are stated in the standard Steel Joist Tables for standard panel sizes. Actual spacing and layout of the duct arrangements are beyond the scope of Tekla Structural Designer.
- Fire resistance requirements are not designed nor checked in Tekla Structural Designer, the designer must ensure suitable compliance.
- There is no design for net uplift currently although uplift can exist in a combination provided it is overcome by other gravity loads. The designer should ensure that any uplift due to Wind loads is accurate as the current version of the Wind Wizard does not determine wind forces for 'Components and Cladding'.
- Sloping joists are permitted providing the loading is normal to the joist and the span will be taken as the sloped length. However, joists with sloping top chords are not allowed.
- SJI allows grades other than 50 ksi [345 MPa] to be used but the Safe Load Table values are based on 50 ksi [345 MPa]. Tekla Structural Designer **defaults to Grade A992 – 50**, it is the user's responsibility to check the suitability of this or any other grade they wish to use. No adjustment is made for higher or lower grades and this is entirely the designer's call.

Loading

The loads on the joist are from 'one-way' load decomposition. The joist is analyzed as a pin ended beam, only loads in the plane of the web are considered.

Joist Girders should only be used to support steel joists, and should therefore have regularly spaced point loads of similar magnitude along their length. As they are not designed or checked various types of loading could be present, so the user should verify that they have been used appropriately.

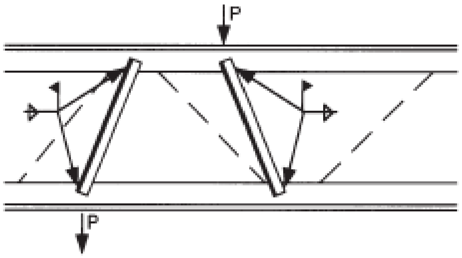
Steel Joists are essentially used to support full length uniformly distributed loads, however it is acceptable for the joist to be designed to support other load configurations. This requires the loading pattern on the joist to be assessed and classified into one of the following loading types: Uniform (or near uniform), Equivalent Uniform, or Non-Uniform.

The classification of loading type is made in accordance with the following methods:

<table>
<tr><th>Loading Type</th><th>Method</th></tr>
<tr><td>Uniform</td><td>No concentrated (point) loads are allowed.
Loads must be (member) Full UDL, UDL, VDL or Trapezoidal Load acting in the Major or Global Z directions.
Loads must be applied over the full length of the beam.
A percentage load tolerance is calculated and provided this is less than the uniform load tolerance specified in Design Settings, then the load combination is classified as ‘Uniform’.</td></tr>
<tr><td>Equivalent Uniform</td><td>Determine the position of the point of zero shear relative to the centre span point of the joist.
If the point of zero shear is located outside the maximum eccentricity of zero shear limit specified in Design Settings, then the procedure is ended and the load combination is classified as ‘Non-uniform’.
If the point of zero shear is located within the maximum eccentricity of zero shear limit then an ‘Equivalent Uniform’ load is established.
The equivalent uniform load is then used to calculate maximum shear force, bending moment and deflection. The percentage variation of these values to the actual values from the beam analysis is then calculated and compared with the equivalent load tolerance limit specified in Design Settings.</td></tr>
<tr><td>Non-Uniform</td><td>All loads not qualifying as ‘Uniform’ or ‘Equivalent Uniform’ to the above methods are considered as ‘Non-uniform’.
<hr>NOTE It is possible for an individual loadcase e.g. Live to be classified as ‘Non-uniform’ which when combined with other loads becomes ‘Equivalent uniform’. Thus, when designing for strength (comparing with the ‘black value’ in the SJI Tables) the loading is valid whereas when checking the deflection (comparing with the ‘red value’ in the SJI tables) the service design could be invalid (Fail).<hr></td></tr>
</table>

Concentrated Loads

All joists supporting concentrated loads require special treatment by the manufacturer/fabricator even if the loading configuration can be configured as ‘Uniform’ or ‘Equivalent Uniform’ since the joist design usually presumes that all concentrated loads are applied at panel points. It is common practice for “field installed members” to be located at all concentrated loads not occurring at panel points as illustrated below.



In Tekla Structural Designer concentrated loads on K, LH and DLH joists are limited to the **maximum sum of concentrated loads** limit specified in Design Settings. This value is unfactored and in the event that the sum of all **unfactored** point loads in all loadcases within a combination exceeds this value, the relevant load combination is classified as ‘Non-uniform’.

Uplift

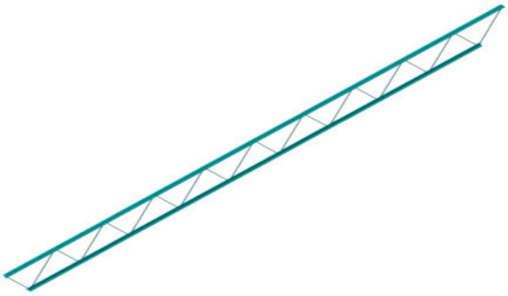
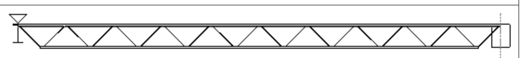
If (net) uplift is detected for a combination no design is performed for it and a warning message is displayed.

NOTE If uplift exists in a loadcase but is ‘overcome’ by positive loading in another loadcase then the design is valid and no warning is displayed.

Joist member reports

Joist design results can be viewed on the screen and incorporated into member design reports. Joists are also included in material listing reports.

The member design report for steel joists is configurable, but limited to the following chapters:

Chapter	Content
Picture	
Drawing	

<table>
<tr><th>Chapter</th><th>Content</th></tr>
<tr><td>Loading</td><td>
Loading
<table>
<tr><th>Loadcase</th><th>Source</th><th>Direction</th><th>In Proj.</th><th>Span/Stack</th><th>Type</th><th>Q₁ [klf]</th><th>Pos [ft, in]</th><th>Length [ft, in]</th></tr>
<tr><td>0 Self weight - excluding slabs</td><td>User</td><td>Global Z</td><td></td><td>1</td><td>Full UDL</td><td>0.01</td><td></td><td>26'9 3/8"</td></tr>
<tr><td>2 Dead</td><td>Decomposition 1-way</td><td>Global Z</td><td></td><td>1</td><td>UDL</td><td>0.10</td><td>0"</td><td>26'9 3/8"</td></tr>
<tr><td>4 Live</td><td>Decomposition 1-way</td><td>Global Z</td><td></td><td>1</td><td>UDL</td><td>0.08</td><td>0"</td><td>26'9 3/8"</td></tr>
</table>
Load Reductions
<table>
<tr><th>Loadcase</th><th>Imposed Load Reductions</th><th>Span</th><th>Reduction Factor</th><th>Tributary Area [ft²]</th><th>K_{LL} Factor</th></tr>
<tr><td>4 Live</td><td>No</td><td></td><td></td><td></td><td></td></tr>
</table>
</td></tr>
<tr><td>Design Summary</td><td>
<table>
<tr><th>Design Condition</th><th>#</th><th>Design Value</th><th>Design Capacity</th><th>Units</th><th>U.R.</th><th>Status</th></tr>
<tr><td>Min. Joist Depth</td><td>-</td><td>16</td><td>13 25/64</td><td>in</td><td>0.837</td><td>✓ Pass</td></tr>
<tr><td>Strength</td><td>62</td><td>0.19</td><td>0.23</td><td>klf</td><td>0.826</td><td>✓ Pass</td></tr>
<tr><td>Deflection Dead</td><td>62</td><td>0.11</td><td>-</td><td>klf</td><td>-</td><td>-</td></tr>
<tr><td>Deflection Live</td><td>62</td><td>0.08</td><td>0.14</td><td>klf</td><td>0.593</td><td>✓ Pass</td></tr>
<tr><td>Deflection Total</td><td>62</td><td>0.19</td><td>0.23</td><td>klf</td><td>0.826</td><td>✓ Pass</td></tr>
</table>
Head code: United States (ACI/AISC), design code: AISC360/341 ASD (2010)
</td></tr>
<tr><td>Design Calculations</td><td>
Min. Joist Depth
Span =26' 9 3/8" ft, in
Minimum depth =1' 1 3/8" ft, in
Joist depth =16 in
Ratio =0.837
✓ Pass
Strength
62 ASD₁-D+L - Critical
Required total load =0.19 klf
Design total load =0.23 klf
Ratio =0.826
✓ Pass
</td></tr>
</table>

Steel truss design

Steel truss design overview

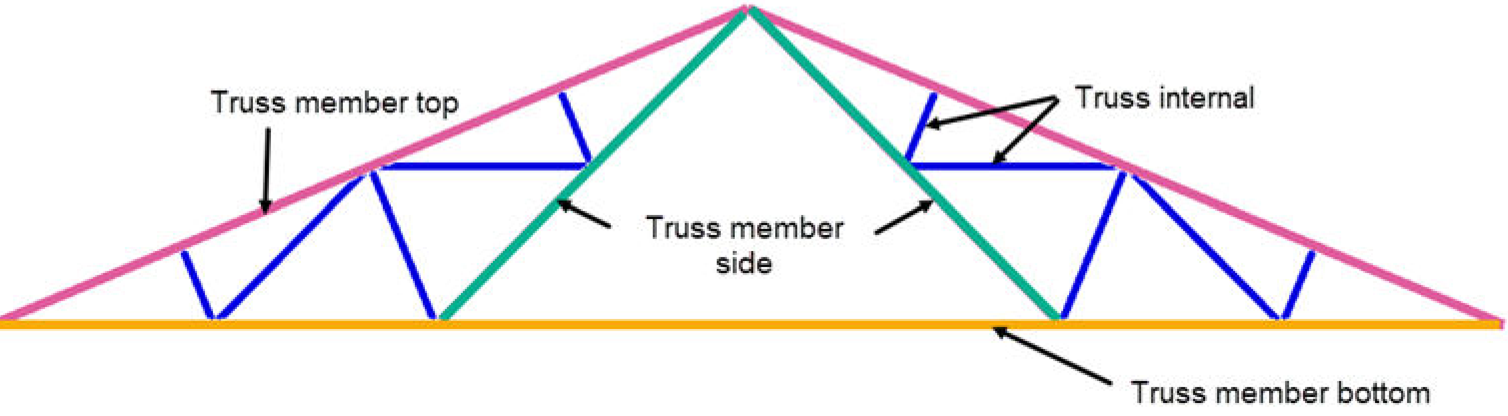
In Tekla Structural Designer although trusses can be defined in any material, design is restricted to steel and cold formed truss members only.

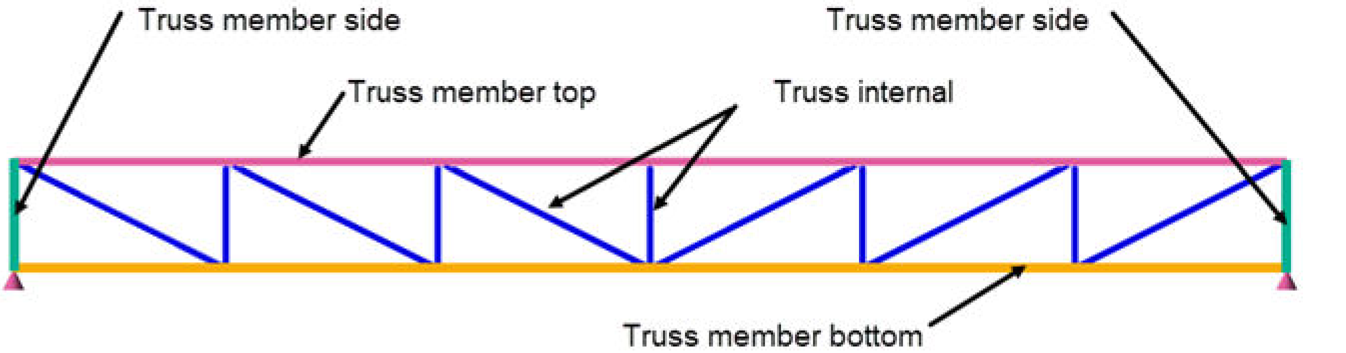
The Truss Wizard provides a wide range of standard truss shapes and configurations. This covers most practical layouts of truss. However, you can also modify a standard truss to produce one of slightly different configuration or manually ‘stick build’ your own truss from truss members if it is an unusual layout.

Trusses created by the Truss Wizard will comprise a mix of the following truss member types:

- Truss member top
- Truss member bottom
- Truss member side
- Truss internal

These are shown in two of the standard configurations below:





Solver elements are created directly between the member insertion points - they do not take into account major and minor snap points, or any offsets that might have been specified in the member properties. Consequently, all intersecting members ‘node’ at the same point – i.e. all internals meet along the set out line of the chords; this assumes that set out lines are coincident with the centroidal axes. Therefore, no in–plane eccentricities are considered in the analysis and design of the trusses.

A wide range of section types can be defined that includes all the common rolled sections. There is no restriction on the type and size of sections that can be connected within the truss. The practicality and efficiency of connections between members is your responsibility. A 'beyond scope' status is issued if a section type has been applied that cannot be checked for a given design condition.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

The design checks performed depend on the truss member types as follows:

- **Truss member top and bottom** - these are continuous members with axial force and bending principally in the plane of the truss. They are designed as beams (page 1208) for all internal forces (axial, major and minor axis bending and shear) depending upon the section type used. Where tension exists in a chord member, the tension capacity is based on the effective net area.
- **Truss internal** - these are restricted to be pin ended, and are designed to the appropriate clauses for tension and compression members. Primary bending moments due to self weight and secondary moments due to eccentricity of their connections are ignored. Effective lengths for compression and effective net area for bolted and welded connections can be taken into account via the properties of the truss members.
- **Truss member side** - depending on the truss type selected in the Truss Wizard these will either default to pin ended or fully fixed. If pin ended, the design checks are the same as for Truss internal. If fully fixed at both ends (as in the case of a Vierendeel truss), the design checks are the same as for Truss member top and bottom.

  NOTE If a (pin ended) side is intersected by another truss member then it is still designed for axial force only, but a warning is issued indicating the forces that have been ignored.

For top and bottom chords, conditions of restraint can be defined in and out-of-plane for strut buckling and, top and bottom flange for lateral torsional buckling (LTB). It is upon these that the buckling checks are based. Incoming members are identified by the program and sensible default values for whether these provide restraint or not are set up (see Assumptions and Limitations (page 1272)). Restraint cannot be added where no incoming member exists but full control of the effective length factors is provided. In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. You can also indicate chord sub-beams to be continuously restrained over their length where appropriate.

NOTE Effective length factors and continuous sub-beam restraints are edited by right clicking on individual chords and selecting Edit *chord reference* from the context menu - these can then be specified from the respective lateral and strut restraint pages of the Properties dialog. Continuous sub-beam restraints can also be edited in Show/Alter State > Restraints, as can restraint of internals to chord (defaults to in-plane only for strut and unrestrained for LTB).

Results of your truss design can be viewed on the screen and incorporated into a report. Truss members are listed as a separate type in the Material Listing report.

Assumptions and Limitations

Limitations

The following limitations apply:

- Web openings, plated sections including Fabsec beams (with or without openings) and Westok beams cannot be used as truss members
- Chord members cannot be placed vertically
- The arch member of a bowstring truss is drawn and designed as a series of facets and not as one continuous curved member
- Truss internals cannot be loaded directly and no loads from floors and roofs are decomposed to them - sides can be loaded, but forces other than axial are then ignored in the design if the pinned ends are not removed.
- Truss chords are not by default excluded from diaphragm action within a floor slab, but they can be deliberately excluded if required. See: https://teklastructuraldesigner.support.tekla.com/support-article/2816265

Assumptions - Restraints

- In both top and bottom chords the node points are assumed not to have incoming out of plane members unless you define such members in the model. Hence, at these positions only in-plane strut buckling restraint is assumed as a default. You can of course change these.
- In a top chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,
- - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out-of plane strut buckling restraint,
  - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- In a bottom chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,
  - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out of-plane strut buckling restraint,
  - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- Lateral restraints to the top or bottom flange of a chord are assumed to be capable of resisting restraint forces not less than those specified in the relevent section(s) of the design code.
- In all cases, for all member characteristics it is assumed that you will make a rational and 'correct' choice for the effective lengths between restraints *The default value for the effective length factor of 1.0 may be neither correct nor safe.*

Portal frame design

If portal frames are modelled and designed in Tekla Structural Designer they will be designed elastically for the forces determined from the 3D analysis, in the same manner as other steel beams and columns.

A more economic design can be obtained by exporting individual frames to **Tekla Portal Frame Designer**. This industry leading portal frame software performs an elastic-plastic analysis and design and undertakes member stability checks to EC3 or BS5950. Once designed, the resulting sections can then be returned to the Tekla Structural Designer model.

NOTE Tekla Portal Frame Designer is a separately purchasable product.

See also

Export to Tekla Portal Frame Designer (page 319)

13.6 Concrete member and slab design handbook

To get started with designing concrete members in Tekla Structural Designer see:

- Concrete member design workflow (page 1274)
- Concrete member autodesign (page 1280)
- Concrete member design and detailing groups (page 1281)
- Concrete member cracked or uncracked status (page 1286)
- Concrete beam design properties (page 1289)
- Concrete column design aspects (page 1300)
- Concrete wall design aspects (page 1306)
- Interactive concrete member design (page 1310)

To get started with designing concrete slabs, see:

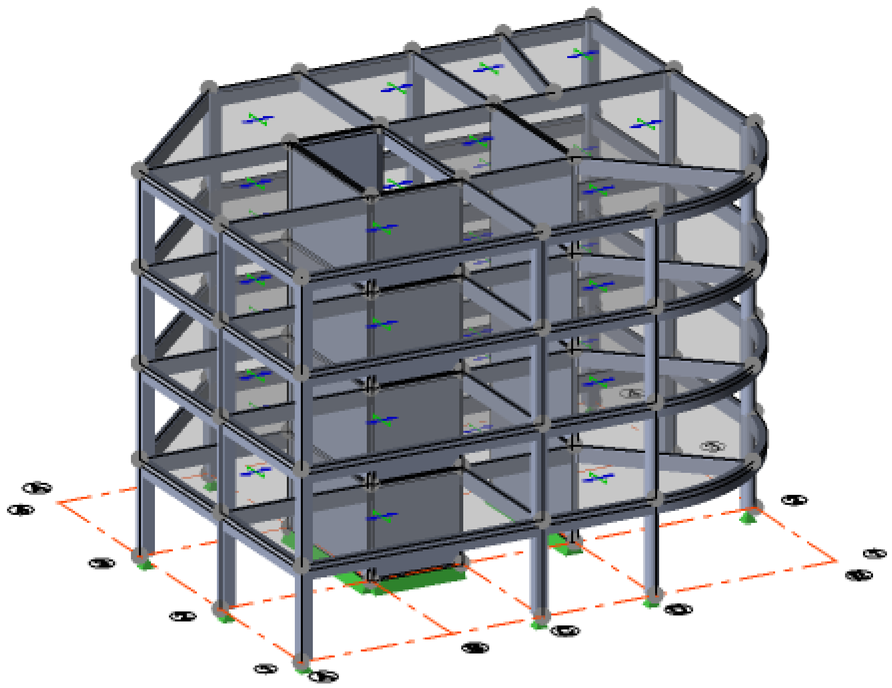
- Concrete slab design (page 1354)

For guidance to help reduce the overall design time, see

- Working with large models (page 1037)

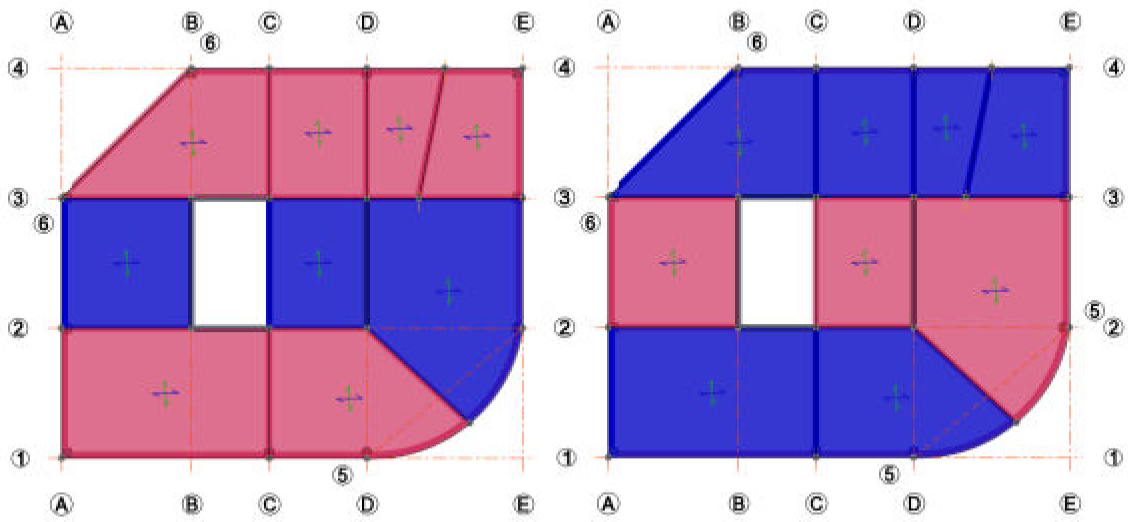
Concrete member design workflow

The following example illustrates the typical process to analyze and design all the beams, columns and walls in a concrete structure.



Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Set all beams columns and walls into autodesign mode

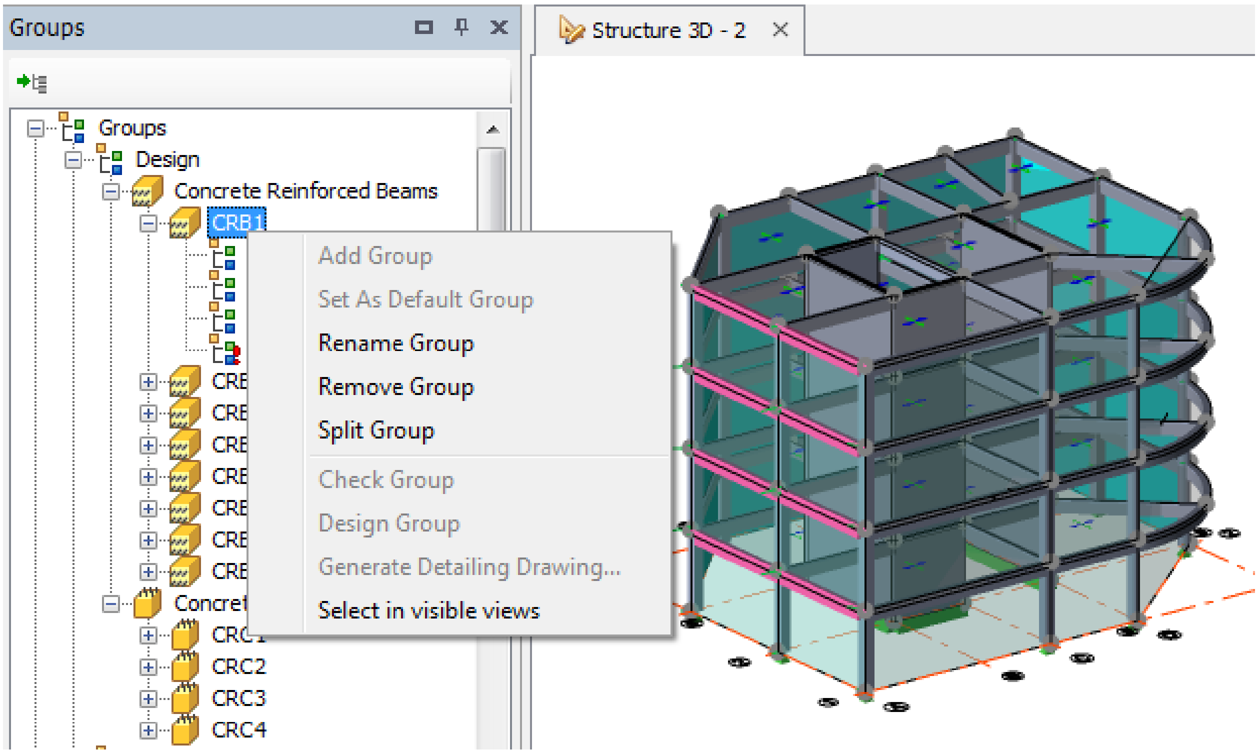
For the first pass, in order to get an efficient design at the outset, it is suggested that you set all members to "autodesign" with the option to select bars starting from Minima.

Related concept

The autodesign process for concrete members (page 1280)

Review beam and column design groups

Provided that the concrete beam and column options are checked in Design Settings > Design Groups, the design groups shown in the Groups tab of the Project Workspace will be applied in the beam and column autodesign processes.



Groups will initially have been established for members sharing the same geometry, but you should consider reviewing and amending them if required.

Review beam, column and wall design parameters and reinforcement settings

The member design parameters and reinforcement settings should be carefully considered prior to running the design.

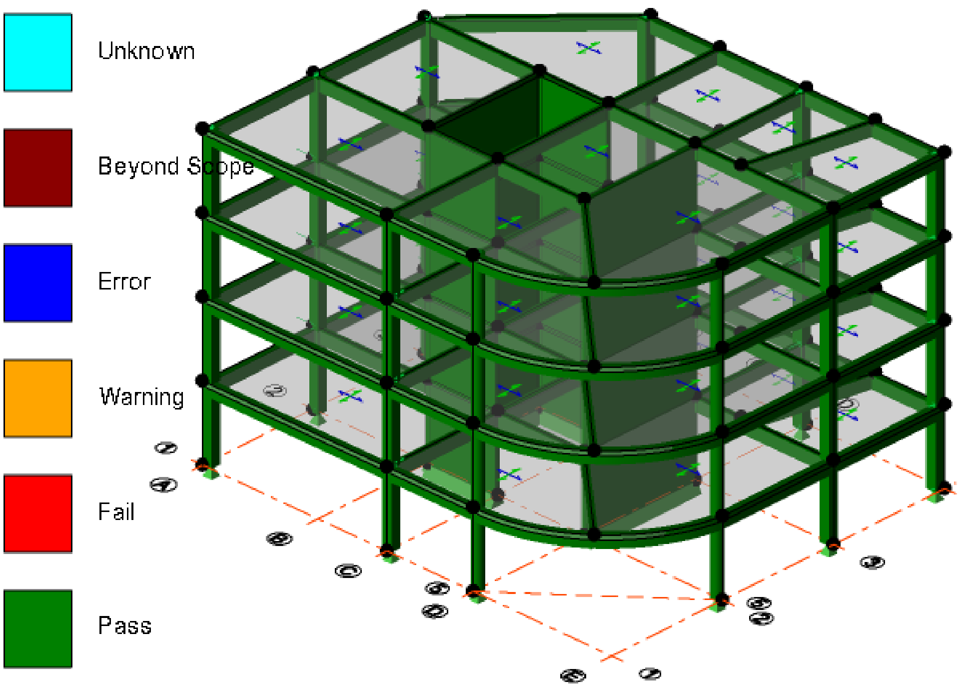
Perform the concrete design

By running Design Concrete (Static) from the Design ribbon, you effectively combine analysis and design (with the exception of slab design) into a single automated process.

Up to three separate analyses are automatically performed in order to generate the design forces required for the concrete beam, column, and wall design:

- 3D Analysis
- Grillage chasedown analysis
- FE chasedown analysis

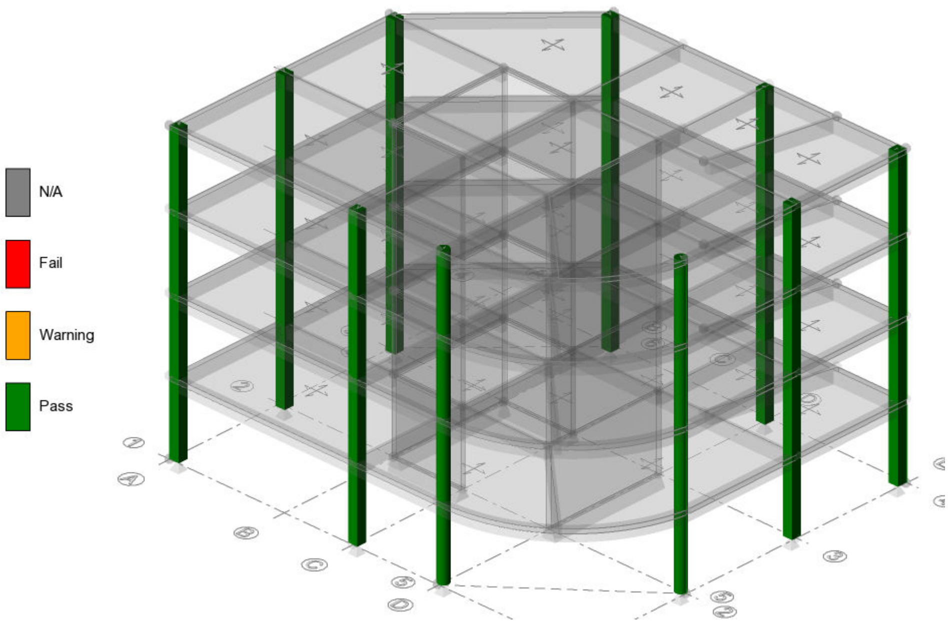
NOTE The sets of forces established from the FE chasedown analysis are considered by default for the design of each concrete member type. They can however be switched off should you decide that they are not required. The control for doing this for beams is located in Design Settings > Concrete > Beam > General Parameters. A similar control is provided for columns and walls also.



NOTE Reinforcement is designed, but member sizes are not changed during the design process.

Review stability issues

Issues relating to stability will be flagged in the Design branch of the Project Workspace Status tab. They can also be review graphically from Show/Alter State in the Review View.



Tabular results can be investigated from the Review View by clicking Tabular Data in the ribbon and then selecting the required View Type.

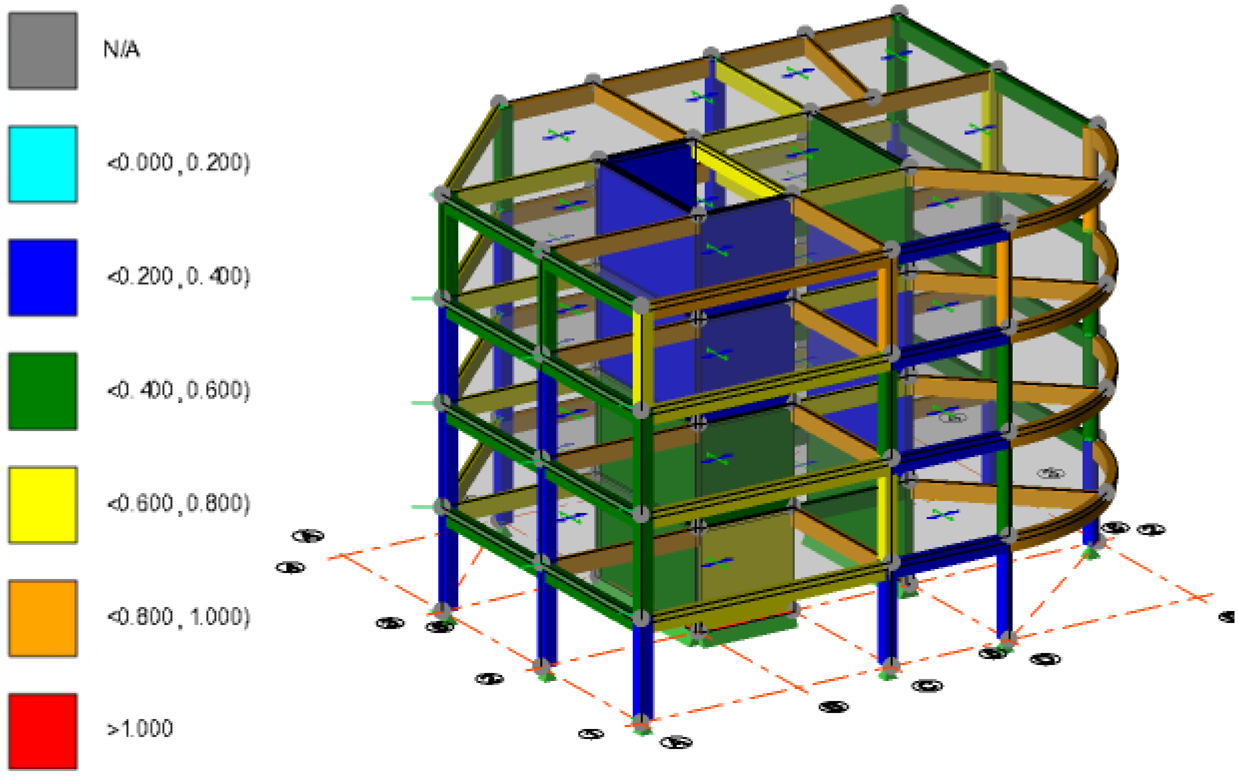
Structure 3D - 2 | Review Data | St. 1 (1) 2D

Wind Drift

Reference	Stack No.	Deflection Dir 1	Deflection Dir 2	Drift Dir 1	Drift Dir 2	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2
C1	4	2.9	-0.9	0.7	-0.2	4306.679	-17968.546	✓ Pass	✓ Pass
C1	3	2.2	-0.7	0.8	-0.2	3784.083	-13780.301	✓ Pass	✓ Pass
C1	2	1.4	-0.5	0.8	-0.2	3833.286	-12311.811	✓ Pass	✓ Pass
C1	1	0.6	-0.2	0.6	-0.2	5189.121	-12372.690	✓ Pass	✓ Pass
C2	4	2.3	-0.9	0.6	-0.2	5033.645	-17968.546	✓ Pass	✓ Pass
C2	3	1.8	-0.7	0.7	-0.2	4518.169	-13780.301	✓ Pass	✓ Pass

Review the design status and ratios

You can display the Design Status and Ratios from the Review View in order to determine if any remodelling is required.



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

If you make any changes, to see their effect simply re-run Design Concrete (Static) once more.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

Print calculations

Create a model report that includes the member design calculations that have been performed. (The default Member Design Calcs report includes these along with design calculations for other member types in the model).

Concrete member autodesign

The design mode for each member is specified in its properties.

Autodesign (concrete beam)

- When Autodesign is selected an iterative procedure is used to select longitudinal bars for each bending design region on the beam, both top and bottom. Similarly an iterative procedure is used to select links/stirrups for each shear design region on the beam.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete beams have been set to be designed using Design and detailing groups (concrete) (page 1281), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Rationalization of Reinforcement

The Auto-design process returns a set of information about the reinforcement to be provided in each design region of the beam. The number and size of the longitudinal bars in the top and bottom of the beam is given as well as the size, number and spacing of the shear links/stirrups.

This information is then "rationalized" to give an arrangement of longitudinal reinforcement that provides a solution for the beam as a whole whilst still meeting the requirements of the individual design regions.

The rationalization process is carried out separately for the longitudinal bars in the top of the beam and those in the bottom of the beam.

The arrangement of shear links/stirrups is not rationalized. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

Autodesign (concrete column)

- When Autodesign is selected an iterative procedure is used to design both the longitudinal bars and links. This applies the spacing maximisation method which attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.

- For rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, autodesign will attempt a two layer solution. For circular sections, if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete columns have been set to be designed using Design and detailing groups (concrete) (page 1281), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Autodesign (concrete wall)

- When Autodesign is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and links/ties. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

Select bars starting from

This option appears for concrete members when Autodesign is selected. It sets the autodesign start point for both longitudinal bars and links/stirrups/ties.

The options are:

- Minima (default)
- Current bars

Selecting Minima removes the current arrangement and begins with the minimum allowed bar size from the selection order.

NOTE When a member is in check mode, it can still be autodesigned "on the fly" by choosing Design Member from the right-click menu. In this case it uses the **Select bars starting from** option currently assigned to the member for autodesign. (This property is hidden when in check mode, but can be confirmed if required by switching the member back to Autodesign mode.)

Concrete member design and detailing groups

Why use concrete design and detailing groups?

Concrete beams and columns and isolated foundations are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design and detailing purposes - to standardise designs, reduce processing time, and reduce the volume of output created.

In a manual process, the Engineer might select a number of sufficiently similar members to form a "design group" to carry out a single design that is sufficient for all members in the group. Using this single set of design results, they would then create sub-groups of the members in the design group to produce a set of output details for each of these sub-groups.

In Tekla Structural Designer, concrete design groups are analogous to the manually created design groups described above. Concrete detailing groups are analogous to the sub-groups.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same number of stacks, bases must be of similar lengths in X and Y, and similar depths etc. The same rules also constrain manual group editing.

NOTE Grouped design and detailing is optional and can be deactivated if required:

From the **Design** tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

What happens in the group design process?

When the option to design a specific concrete member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the reinforcement so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the reinforcement is adequate for each and if this proves not to be the case, it is increased as necessary and the revised reinforcement is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

Concrete design group requirements

Concrete member design groups are formed according to the following rules:

Member type	Design group rules
Concrete beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties and nominal cover. • All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.
Concrete column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties and nominal cover.
Pad base	• A pad base may be in only **one** design group. • Each base in the group must have an identical cross section and depth. • Each base in the group must have identical eccentricities in X and Y. • Each base in the group must have identical material properties and nominal cover.
Pile cap	• A pile cap may be in only **one** design group. • Each pile cap in the group must have an identical cross section and depth. • Each pile cap in the group must have identical eccentricities in X and Y. • Each pile cap in the group must have identical material properties and nominal cover

Detailing group requirements

Each parent design group is sub-divided into one or more detailing groups.

Although there can be a "1 to 1" relationship between a design group and a detailing group, in practice there will often be a "1 to many" relationship as each design group is likely to require several detailing groups to allow for differences in the connected geometry.

Detailing groups are formed for the different concrete member types based on the following rules:

Member type	Design group rules
Concrete beam	• A detailing group may be associated with only **one** parent design group. • A beam element may be in only **one** detailing group. • Detailing groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • The cross section, including flange width where appropriate, span length and material properties in span • *"i"* of **all** beam elements in the group must be identical . • All beam elements in the group must have identical plan offsets. • All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans. • All beam elements in the group must have identical inclinations. • The support types and sizes, including the attached structure above and below the beam element, must be identical in all beam elements in the group however different support types and sizes in individual multi-span continuous beams are acceptable i.e. support *i* in beam element *j* must be identical to support *i* in all other beam elements in the group but supports *i* and *i+1* in beam element *j* may be different.
Concrete column	• A detailing group may be associated with only **one** parent design group, • A column may only be in **one** detailing group,

Member type	Design group rules
	• All columns in the detailing group must have an identical number of stacks, • All columns in the group must have an identical cross-section, rotation and alignment/snap levels/offsets in stack 'i'. In a multi-stack column, the cross-section may be different in each stack, i.e. the cross-section in span 'i' may be different to that in span 'j'. • Stack 'i' and stack 'i+1' must be co-linear for **all** columns, OR must be non-co-linear with an identical degree of non-co-linearity for all columns. The exact inclination must be the same for stack 'i' in all columns. • At every level each column is considered to be either "internal" or "external" (depending on if it has beams framing into it on all four sides, or not). These settings do not have to be identical for columns to be in the same group, but only if you have selected the option: Provide ties through floor depth for internal columns in Design Options > Concrete > Column > Detailing Options.
Pad base	• A detailing group may be associated with only **one** parent design group. • A base may be in only **one** detailing group. • The attached column cross-section above the base must be identical for all bases in the group however different support types are acceptable.
Pile cap	• A detailing group may be associated with only **one** parent design group. • A pile cap may be in only **one** detailing group. • The attached column cross-section above the base must be identical for all pile caps in the group however different support types are acceptable.

Group management

Automatic Grouping

Concrete beams and columns are grouped automatically.

In **Model Settings** > **Grouping** the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

Detailing groups cannot be edited manually.

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Model Editing and Group Validity Checks

When new beam elements are created when a "split" or "join" command is run the resulting beam elements are automatically placed in existing design and detailing groups [or new groups created].

Concrete member cracked or uncracked status

Assuming concrete sections are cracked has a direct affect on analysis - smaller Modification Factors (page 2290) are applied to cracked sections, (typically a cracked member is assumed to have half the stiffness of an uncracked member), causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.

The *Assume cracked* setting in the member properties is used to specify the cracked/uncracked status of individual concrete beam spans, column stacks and wall panels. While it can be set directly in the member properties, it can also be set or toggled graphically (page 875) in a Show/Alter State Review View. For meshed walls a Review Wall Stress (page 876) feature is provided to speed up this process.

Wall cracked properties

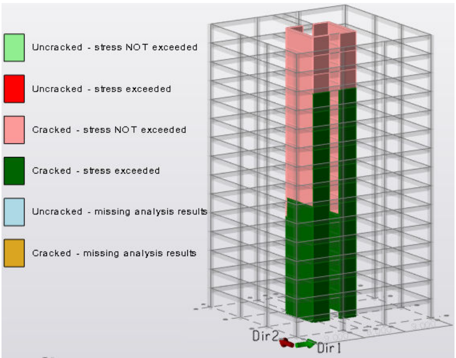
For engineers investigating the sway characteristics of taller buildings the cracked/uncracked status of walls is a complex topic of critical importance.

A couple of challenges faced by engineers are:

1. If using general stiffness adjustments - what are the realistic adjustment values for cracked and uncracked panels?

2. How can the cracked/uncracked status of wall panels be easily determined?

Tekla Structural Designer provides a **Review Wall Stress (page 876)** feature in Show/Alter State to significantly assist with the latter task for meshed panels. This graphically displays the current cracked/uncracked status of each panel and indicates in which panels the stress threshold has and has not been exceeded. It then allows you to update the cracked status to be compatible with the threshold.



Use of the Review Wall Stress feature requires a degree of engineering judgement - you will need to make your own choices with regard to:

- strength or service level stress?
- basing the stress threshold on max instead of in-plane stress contours?
- whether the panel cracks for any level of stress, or could a higher threshold be considered?

You will also need to be familiar with the limitations and assumptions (page 876) that apply.

Workflow for reviewing wall cracked properties

To determine an appropriate cracked status for wall panels, you could adjust the following basic workflow to suit your needs:

1. Analyze the model to determine some initial stresses.
2. Open a Review View and click Show/Alter State.
3. In the Show/Alter State **Properties** window:
   - Set the *Attribute* to **Assume cracked**
   - Set the *[M]ode* to **Review Wall Stress (page 876)**
   - Set the *Result type* to **Strength** or **Service** as you deem appropriate.

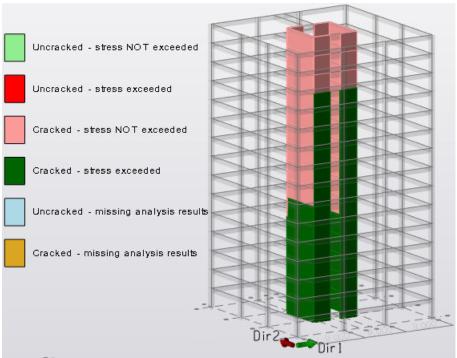
- Set the *Stress type* to **In-plane tension - Y** or **Max tension - Y** as you deem appropriate.

NOTE If using **In-plane tension - Y** contours:

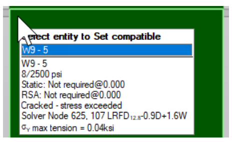
- these give an average stress through the thickness of the wall and therefore exclude local stress concentrations that exist as a result of out of plane bending
    - (engineers often choose to ignore out of plane bending when considering the overall cracked status of panels)

- Enter the *Stress threshold* that you want to work to.

In the below example, the most likely likely scenario where everything was initially set as cracked is shown; in lower panels this is a correct setting, in the higher ones they could be changed to uncracked.



4. By hovering the cursor over an individual panel a tooltip is displayed. This reports the maximum stress value from all nodes in the panel and the case/combination in which it occurs. This value is used to determine if the stress threshold has been exceeded.

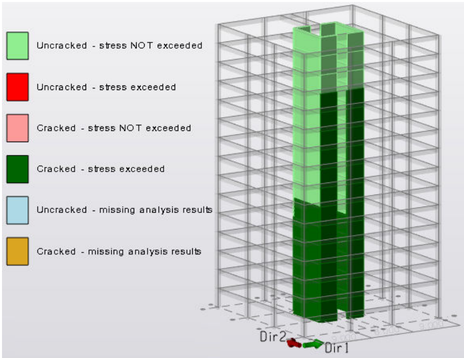


NOTE To investigate reported stresses in more detail you might choose to open a Results View to Display 2D results (page 677) of tension stress contours for specific envelopes or combinations.

---

5. At this point you could choose to manually set or toggle the cracked/ uncracked status (page 875) of the individual panels, however **Auto update** provides a quicker way to make all panels compatible.
6. To make all panels automatically compatible:

- Click [Auto update] to expose the Auto update button.
- Click the Auto update button



Each panel is updated as follows:

- uncracked panels in which the stress threshold has been not exceeded are unaffected,
- uncracked panels in which the stress threshold has been exceeded are set to cracked,
- cracked panels in which the stress threshold has been exceeded are unaffected,
- cracked panels in which the stress threshold has not been exceeded are set to uncracked.

7. Reanalyze the model to take into account the revised cracked/uncracked properties.
8. Review the results once more and make further adjustments if required.

See also

Modify assumed cracked settings (page 875)

Concrete beam design aspects

Concrete type

While you can apply both normal and lightweight (LW) concrete in the beam properties, beam design using lightweight concrete is only available for Eurocodes.

When using other Head Codes beams can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

Deflection control (ACI/AISC)

Tekla Structural Designer implements both a simplified, and also a more rigorous method for controlling deflection.

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

When the rigorous method is applied, immediate short-term deflections, and also long-term deflections resulting from creep and shrinkage of flexural members are considered.

Structure supporting sensitive finishes

Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting Structure supporting sensitive finishes under the Design Control heading in the beam properties.

The deflection method applied to the beam depends on this setting as follows:

- beams not required to support sensitive finishes adopt the simplified method.
- beams required to support sensitive finishes adopt the rigorous method.

The simplified method

For those beams that have not being chosen to support sensitive finishes, deflection is controlled by the simplified method of limiting the span to depth ratio.

This check can be satisfied by providing a total beam depth which exceeds h_{min}, (where h_{min} depends on the clear span and beam end conditions).

If this check is not satisfied the beam is then rechecked by the rigorous method.

The rigorous method

For beams that do not meet minimum thickness requirements determined by the simplified method, or that support sensitive finishes, deflections should be calculated by rigorous method.

The parameters listed below directly affect the rigorous calculation method and hence require consideration:

1. Does the beam support sensitive finishes?
    - If the Structure supporting sensitive finishes beam property (located under the Design Control heading) is cleared the simplified method may suffice in which case rigorous calculations are not required.
2. Can the beam flanges be taken into account?
    - Selecting Consider flanges (located under the Design Control heading) can assist in reducing the calculated deflections.
3. Does the beam contain compression steel?
    - In the current release of the program it has been assumed that the beam always contains compression steel for the rigorous method deflection calculations.
4. 4. Are the deflection limits appropriate?
    - These limits are editable in the beam properties, the default ratios being span/360 for live load, and span/480 for total load affecting sensitive finishes.
    - The default span\over limits tend to produce conservative results.
    - You are also given the flexibility to specify absolute deflection limits as an alternative, or in addition to the above span\over limits.
5. Is the long term deflection period correctly specified?
    - The default value is 5 years, but this can be edited via Design Options > Beam > General Parameters
6. Is the time at which brittle finishes are introduced correctly specified?
    - The default value is 1 month, but this can be edited via Design Options > Beam > General Parameters
7. Are the percentages of dead load applied prior to brittle finishes and long term live load correctly specified?
    - The deflection check takes account of these percentages, (defaulting to 50% of dead load applied prior to brittle finishes and 33% of long term live load). You are able to adjust each of these by selecting the loadcase name from the left hand side of the Loadcases dialog and then adjusting the value.

Deflection control (AS 3600)

Tekla Structural Designer controls deflections either by limiting span to depth ratios, or by applying the simplified method. The choice of method being set via Design Options > Concrete > Beam > General Parameters.

The simplified method

In the simplified deflection calculation procedure actual short-term deflection is calculated using the mean value of modulus of elasticity of concrete at appropriate age (Ecj) and an effective second moment of area of member (I_{ef}).

When the simplified method is applied, the shrinkage parameters that are required are specified on the Design Options > Concrete > Beam > General Parameters page.

Limiting span to depth ratios method

The deflection of reinforced concrete beams is not directly calculated and the serviceability of the beam is measured by comparing the calculated limiting effective span/effective depth ratio (L_{ef}/d)

Structure supporting sensitive finishes

Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting **Calculate deflection after installation of finishes** under the Design Limits heading in the beam properties.

When this option has been selected an additional span\over or absolute limit can be specified and checked against in the beam properties.

• beams not required to support sensitive finishes adopt the simplified method.

• beams required to support sensitive finishes adopt the rigorous method.

Parameters affecting deflection

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. 1. Increase reinforcement if deflection check fails
    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results
    - If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.
2. Consider flanges
    - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Deflection control (Eurocode BS and IS)

Tekla Structural Designer controls deflection by comparing the calculated limiting span/effective depth ratio L/d to the maximum allowable value $(L/d)_{max}$

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. Increase reinforcement if deflection check fails

    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results

    If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.

2. Structure supporting sensitive finishes (Eurocode only)

    - This beam property (located under the Design Control heading, and on by default) is used to control the value of the f2 parameter used in the deflection check. When unchecked f2 will be taken as 1.0.

3. Consider flanges

    - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Ignore lateral instability (Eurocode)

This option (located under the Design Control heading) allows you to consider or ignore lateral instability for slender spans to EC2 clause 5.9(1) (off by default). When the option is checked on the slender span check is excluded from design.

Consider flanges

Flanged beam properties can be specified under the Design Control heading in the beam properties, by selecting Consider Flanges.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

The characteristic behaviour of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Related concept

Flanged concrete beams (page 1297)

Design parameters (Eurocode only)

Located under the Design parameters heading in the beam properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Nominal cover

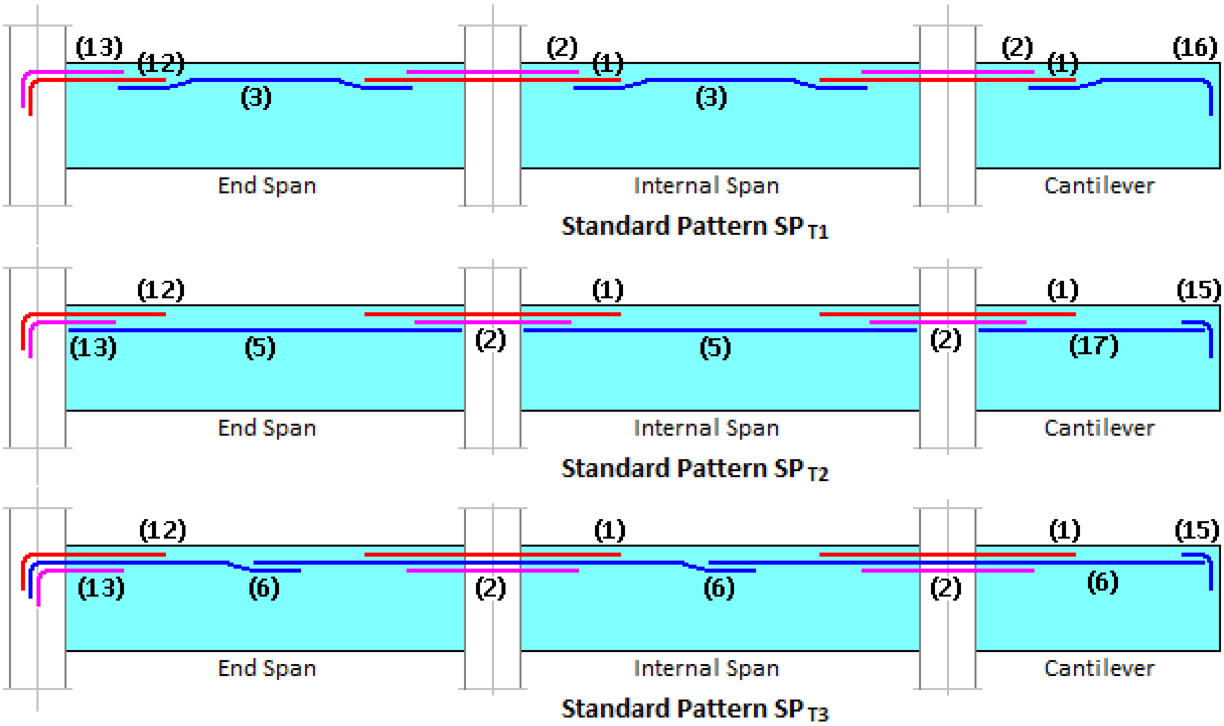
In the beam properties, the nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement - longitudinal bar patterns

Under the reinforcement heading in the beam properties, there are three Standard Patterns available for top reinforcement, SP_{T1}, SP_{T2} and SP_{T3} and two

Standard Patterns available for bottom reinforcement, SP_{B1} & SP_{B2} as illustrated in the figures below.

Standard Patterns of Top Reinforcement



The bars used in the Standard Top Patterns are:

(1) Straight bar extending to approximately 25% or 33% of each span (end points of this bar are determined by the design region settings)

(2) Straight bar extending to approximately 10% of each span (end points of this bar are determined by the design region settings) - if required by the design

(3) Double cranked bar lapped with bar (1)

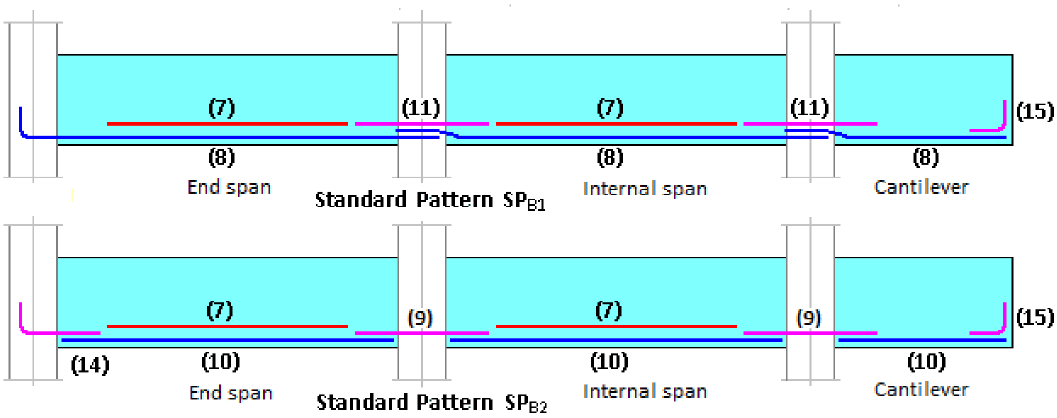
(5) Straight bar running approximately from face to face of beam supports

(6) Single cranked bar running from center span to center span with the option to merge bars if they are the same size and number to extend the bar over several spans

(12) Bob bar

(13) Bob bar

Standard Patterns of Bottom Reinforcement



The bars used in the Standard Bottom Patterns are:

(4) Bar with a bob at each end

(7) Straight bar with a length approximately 70% of span - if required by the design

(8) Single cranked bar extending over several spans or over one span only and lapped within a support - with bob if it continues over an end span.

(9) Straight bar

(10) Straight bar running approximately from face to face of beam supports

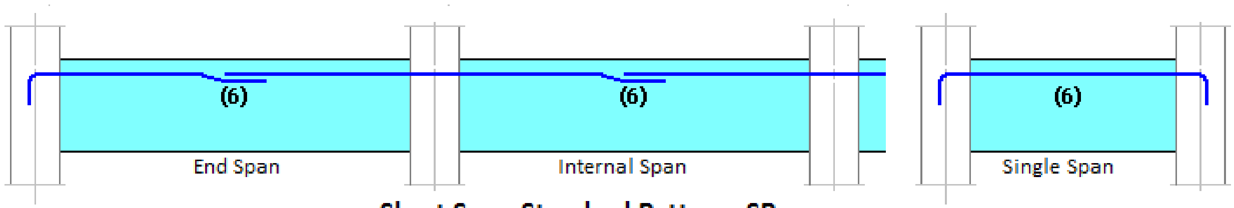
(11) Straight bar

(14) Bob bar

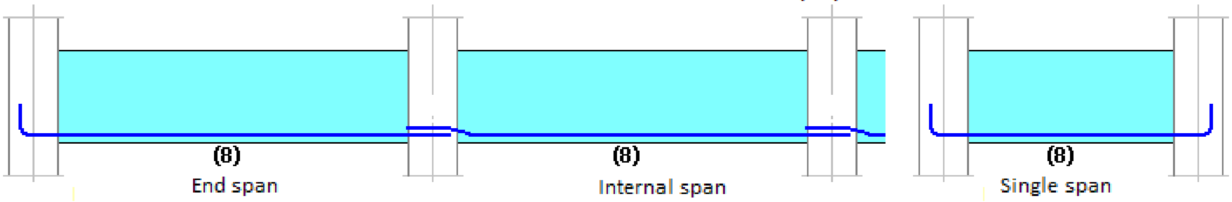
Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in **Design Options > Beam > Reinforcement Settings** and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams



Short Span Standard Patterns SP $_{T1,\ T2,\ \&\ T3}$



Short Span Standard Patterns SP $_{B1\ \&\ B2}$

Flanged concrete beams

Consider flanges

Flanged beam properties can be specified under the Design Control heading in beam properties, by selecting Consider Flanges.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

The characteristic behaviour of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Validation of slabs for use in the flange effective width calculations

If a slab is present (and provided that a user defined flange has not been specified), the program automatically validates the slab as a potential candidate for being a beam flange using a number of criteria, the main ones being;

• the slab can be on one or both sides of the beam but

• it must extend for a distance ≥ the slab depth from the vertical face of the beam and

• it must extend for the full span length of the beam

• the slab must be a reinforced concrete slab

• if there are slabs on both sides of the beam, they may be of different depths and these depths may vary along the length of the beam

The effective width of any **valid** slab on each side of the beam, $b_{eff,i}$, is calculated and the results that are appropriate at the mid-span length point are displayed along with the flange depth, under **Design control** in the Beam Property dialog.

NOTE When automatically calculated, the flange width and depth are only displayed in the Beam Property dialog and not in the Beam Properties window, (because the width and depth could vary if multiple beams were to be selected).

Include flanges in analysis

Selecting this option allows the flanged beam section properties to be considered in the analysis, stiffening the beam and reducing the deflection.

Consider as isolated (ACI/AISC)

ACI 318 clause 8.12.4 states:

"Isolated beams, in which the T-shape is used to provide a flange for additional compression area shall have a flange thickness not less than one-half the width of web and an effective flange width no more than four times the width of web."

When the **Consider flanges** check box is selected, an **Isolated Beam** check box is displayed to control whether or not the above code limit is applied. When the check is performed, if the flange geometry does not meet the above requirements the flanges are ignored.

NOTE Our understanding is that while this limit usually applies to precast beams, it is not usually applied to in-situ construction. Therefore, by the default the Isolated Beam check box is cleared, which means that the above check is not performed.

Effective Width of flanges (ACI/AISC)

For ACI 318-08 and ACI 318-11

For "T" shaped flanged beams the effective flange width, b_{eff}, is given by :

b_{eff} = MIN(L/4, 16*h_f + b_w, b_1 + b_2 + b_w) - O_{wi}

For beams with slab one side only, the effective flange width, b_{eff} is given by :

b_{eff} = MIN(L/12+ b_w, 6*h_f + b_w, b_i + b_w) - O_{wi}

where

L = span length

IF construction is continuous:

= distance of center-to-center of supports

ELSE

= MAX(clear span + h, distance between centers of supports)

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centerline of any supporting steel beam

O_{wi} = the user specified allowance for an opening

For ACI 318-14

All limits on the flange width apply to the overhangs on each side of the beam. It is also clarified in this version that the clear span should be used in these calculations.

$b_{eff,i} = MIN(l_n/8, 8*h_f , b_i) - O_{wi}$

Effective Width of flanges (Eurocode)

The effective width of the compression flange is based on L_0, the distance between points of zero bending moment.

For flanged beams the following values of L_0 are to be used;

For a simply supported beam $L_0 = L$

For a continuous beam, the value of L_0 may be obtained using the following simplified rules;

End span of a continuous beam with a pinned end support $L_0 = 0.85*L$

End span of a continuous beam with a fixed end support $L_0 = 0.70*L$

Internal span of a continuous beam $L_0 = 0.70*L$

where

L = the clear length of the span under consideration

The effective flange width, b_{eff}, is given by;

$b_{eff} = b_w + \sum b_{eff,i}$

where

$b_{eff,i}$ = the effective width of the flange on side i of the beam

$= MIN[0.2*L_0, b_i,(0.2*b_i + 0.1*L_0)] - O_{w\ i}$

where

L_0 = the distance between points of zero moment as defined above

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centreline of any supporting steel beam

b_w = the width of the beam

O_{wi} = the user specified allowance for an opening

NOTE If the slab thickness varies on each side of the beam, the thinner value is used in calculating the beam properties.

The above calculation for b_{eff} is also used for "Γ" beams with a slab on only one side although in this case, b_1 or b_2 as appropriate is = 0.

Concrete column design aspects

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Apply rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the Rigid zones not applied option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the Apply rigid zones option that is provided in the column properties under the Design control heading.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the positions of releases in analysis model are affected, and member start and end points for design are also adjusted.

The Apply rigid zones setting is located under the Design control heading in the column properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the column properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Confinement reinforcement

Located under the Confinement reinforcement heading in the column properties, the **Provide support regions** setting determines the way each stack is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.

- Cleared - the same confinement reinforcement is designed for the whole stack.

Slenderness

Located under the Slenderness heading in the column properties, the significant parameter within the slenderness criteria is a choice of how the column is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) which is the clear dimension between the restraining beams at the bottom of the stack and the restraining beams at the top of the stack. The unsupported length may be different in each direction.

Effective concrete beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the column. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the stack if it is within the depth of the stack section from the end of the stack, and if its centre is nearer to this end of the stack than the far end. Therefore, at a node at a stack join, if the top of the beam is below the node by a dimension greater than the depth of the stack below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the stack above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a stack, Tekla Structural Designer considers whether there is a flat slab restraining the stack at that end. The Use slab for calculation... upper/ lower, major/minor options, (which are located under the Stiffness heading in the column properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the stack if it is within half the slab depth from the end of the stack, and if its centre is nearer to this end of the stack than the far end.

If, at an end of the stack, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the stack beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next stack (and so on). If there is no stack beyond this restraint (i.e. this is the end of the column), the unsupported length ends at the end of the column.

Sway/Drift Checks

By default all stacks of all columns are taken into account in the sway/drift, wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the column properties, these parameters provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.

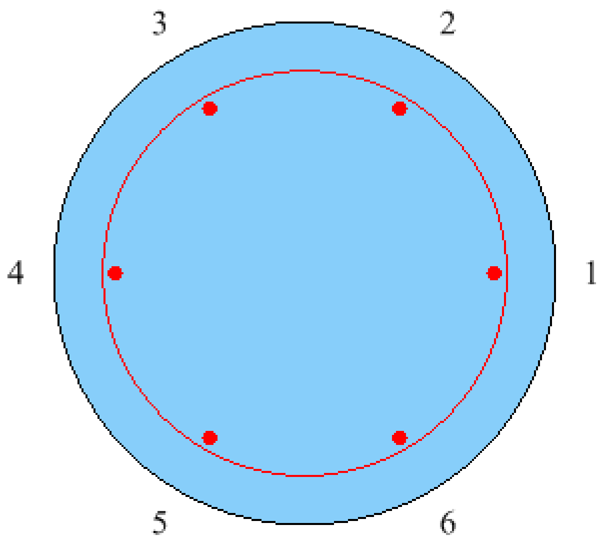
Nominal cover

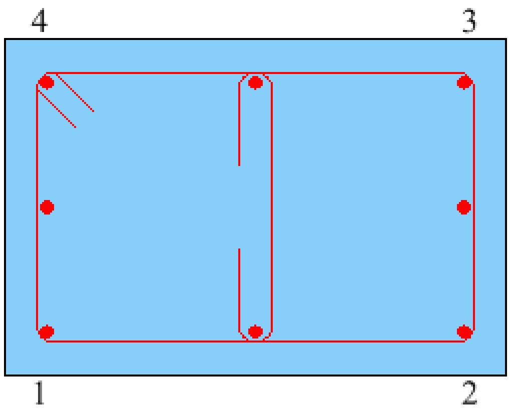
In the column properties, the nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

The type and grade of vertical and confinement reinforcement to be considered for the design are specified under the Reinforcement heading in the column properties.

Where possible bars are arranged in a single layer:

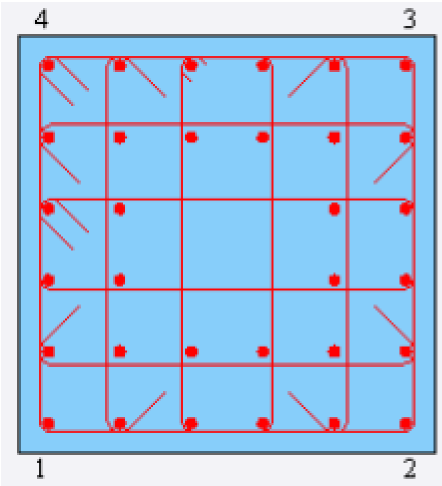


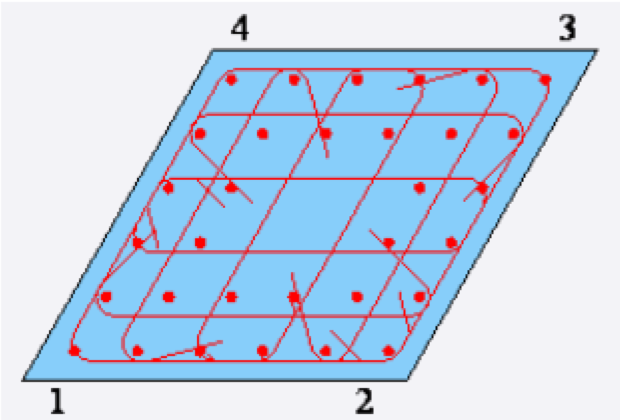


Using a 2nd layer of reinforcement

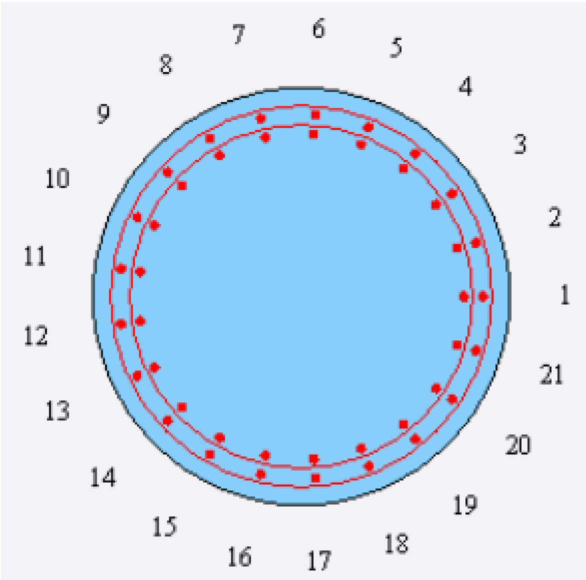
- In Autodesign: for rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, design will attempt a two layer solution.
    - Rectangular and parallelogram sections: a second layer is added in line with the first bar along the adjacent length.

NOTE At least 2 intermediate bars are needed in each direction for a 2nd layer to be added.





- Circular sections: if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).



- A link is added along the second layer for all 3 section shapes.
- In Interactive Design: you can choose to toggle the second layer on or off in the Interactive Design dialog.
- When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

Concrete wall design aspects

Concrete type

While you can apply both normal and lightweight concrete in the wall properties, wall design using lightweight concrete is currently beyond scope.

End 1 and End 2 extensions

Wall extensions (End 1/End 2) can be applied in the wall properties in order to remove physical overlaps with adjoining walls and columns without compromising the integrity of the underlying analysis model.

Negative extensions can be created automatically where appropriate. Extensions can also be defined manually if required, in which case they can be input with either positive or negative values:

- A positive extension extends the wall length beyond its insertion point.
- A negative extension trims the wall back from the insertion point.

Although the length of the wall used in the analysis model (L_{wall}) is unchanged, the wall length that is used in the design, quantity reporting and drawings changes to $L_{wall,d}$

Reinforcement layers

Either one or two layers of reinforcement can be specified in the wall properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the wall properties, the following parameters relating to shrinkage and creep can be specified for individual walls.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Sway/Drift Checks

By default all panels of all walls are taken into account in the sway/drift , wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the wall properties, these parameters provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All panels' to exclude the entire wall, or you can exclude a particular panel by clearing the check box located under that panel only.

Confinement reinforcement

Located under the Confinement reinforcement heading in the wall properties, the **Provide support regions** setting determines the way each panel is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.
- Cleared - the same confinement reinforcement is designed for the whole panel.

Slenderness

Located under the Slenderness heading in the wall properties, the significant parameter within the slenderness criteria is a choice of how the wall is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) which is the clear dimension between the restraining beams at the bottom of the panel and the restraining beams at the top of the panel. The unsupported length may be different in each direction.

Effective Concrete Beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the wall. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the panel if it is within the depth of the panel section from the end of the stack, and if its centre is nearer to this end of the panel than the far end. Therefore, at a node at a panel join, if the top of the beam is below the node by a dimension greater than the depth of the panel below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the panel above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a panel, Tekla Structural Designer considers whether there is a flat slab restraining the panel at that end. The Use slab for calculation... upper/ lower, major/minor options, (which are located under the Stiffness heading in the wall properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the panel if it is within half the slab depth from the end of the panel, and if its centre is nearer to this end of the panel than the far end.

If, at an end of the panel, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the panel beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next panel (and so on). If there is no panel beyond this restraint (i.e. this is the end of the wall), the unsupported length ends at the end of the wall.

Nominal cover

Nominal concrete cover is specified in the wall properties.

For walls, it is measured as follows:

- For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.
- For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/confinement transverse reinforcement that may be present.

Reinforcement

Under the Reinforcement heading in the wall properties, the Reinforcement layers, Form and Include end zones properties can be combined as required in order to obtain a range of reinforcement patterns, e.g:

- Single layer, using mesh reinforcement
- Two layers, using mesh reinforcement
- Single layer, using loose bars

- Two layers, using loose bars
- End zones, with a single layer of mesh in the mid zone
- End zones, with two layers of mesh in the mid zone
- End zones, with a single layer of loose bars in the mid zone
- End zones, with two layers of loose bars in the mid zone

Interactive concrete member design

The combined analysis and design processes, **Design Concrete (Static)**, **Design All (Static)** etc. are complemented by the program's interactive member design facilities. These allow you to interact with the concrete member designs to override the results arising from the auto-design process.

The following interactive member designs are provided:

- Interactive concrete beam design (page 1310)
- Interactive concrete column design (page 1315)
- Interactive concrete wall design (page 1335)

Generally you are advised to perform interactive member designs only after the Design All process has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.

The Interactive design dialogs display a limited selection of the relevant critical design results including bar details and allow you to make changes to the number, size and spacing (for links/ties only) of the selected bars.

After making changes, you are able to see the effect on the displayed results - you then have the option of canceling or accepting the changes.

Interactive concrete beam design

Opening the Interactive Beam Design Dialog

The Interactive Beam Design dialog (page 1311) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Beam Design dialog is displayed.
3. Interactively adjust the reinforcement as required until the design is satisfactory.

Changing the bar pattern used in the Interactive Beam Design Dialog

The interactive dialog can be opened from any of the 2D or 3D Views as follows:

1. If the Interactive Beam Design Dialog is open, click Cancel to close it.
2. If necessary, re-select the beam to be designed.
3. Change the Top and Bottom longitudinal bar pattern in the Properties Window as required.
4. Hover the cursor over the beam until its outline is highlighted, then right-click.
5. From the context menu select Interactive Design...

The Interactive Beam Design Dialog opens and reinforcement is automatically re-selected for the beam based on the new bar pattern.

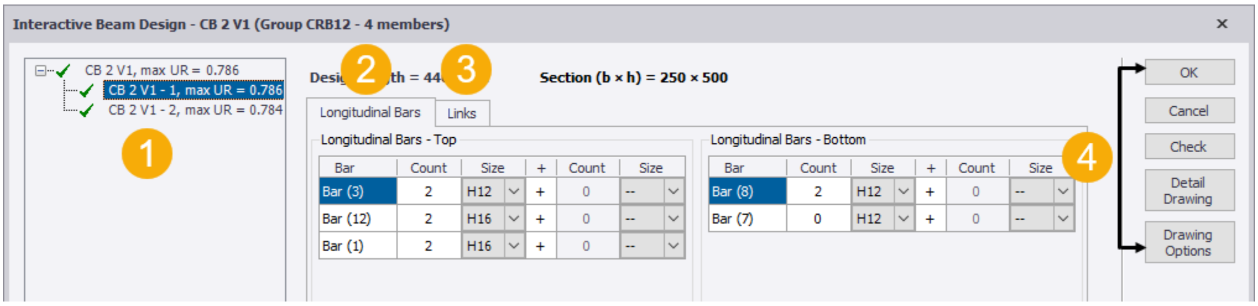
Interactive Beam Design dialog

The **Interactive Beam Design dialog** shows the current reinforcement and check results for each beam span in the selected beam line. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

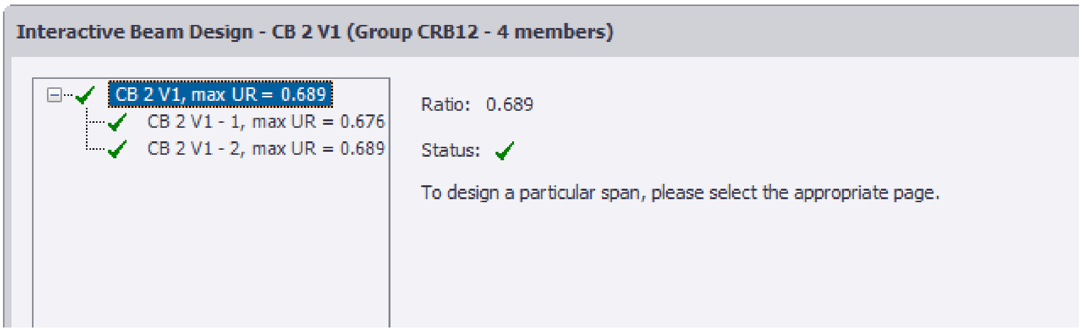
To display the dialog:

1. Right click on an existing concrete beam.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Beam/span summary pane

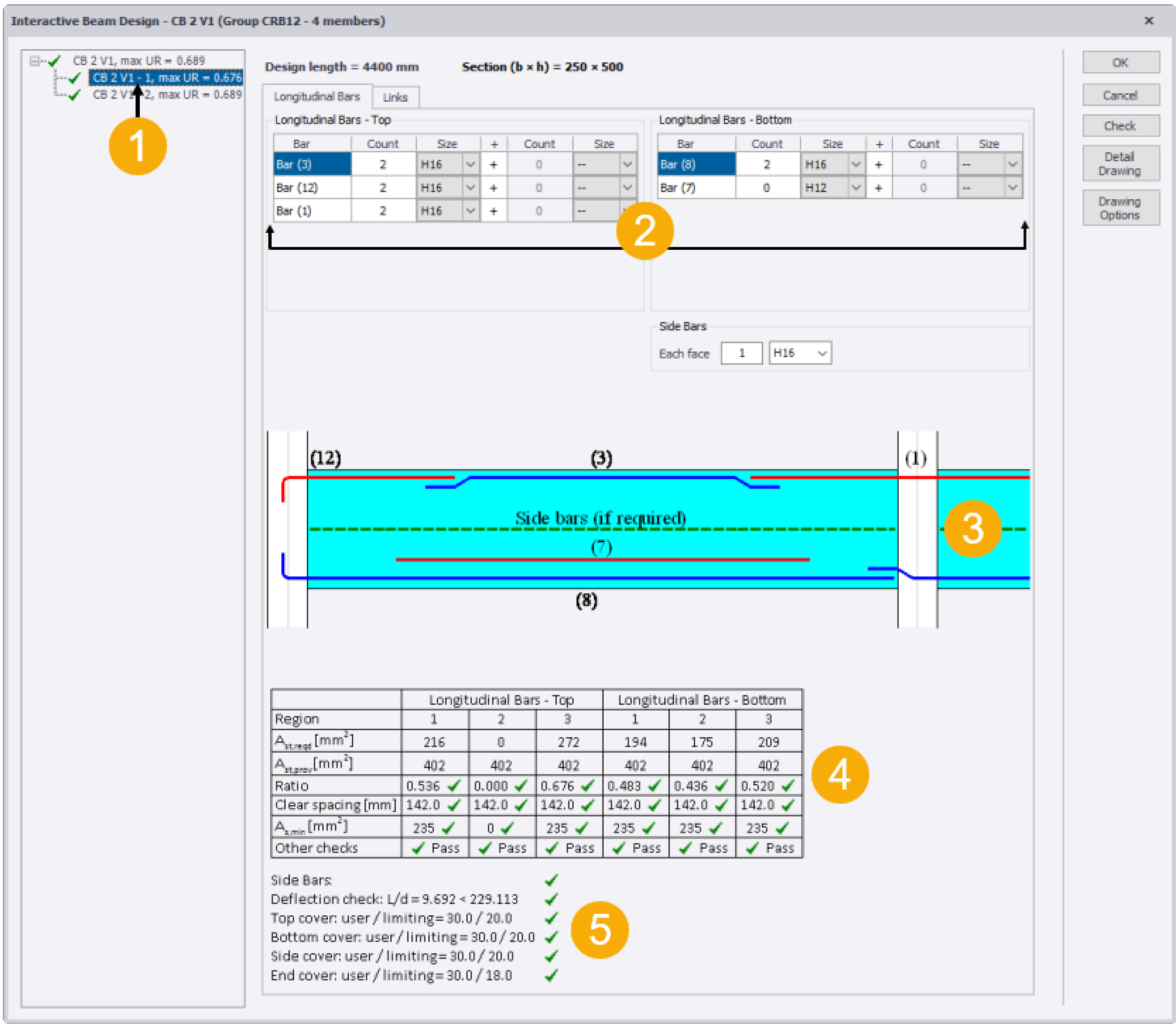


The top row in this pane shows the beam line summary, consisting of the overall utilization ratio and design status.

Subsequent rows show the design status of each span and associated utilization ratio.

- To design a particular span, click on the corresponding row for that span in the summary pane.

2. Longitudinal Bars tab

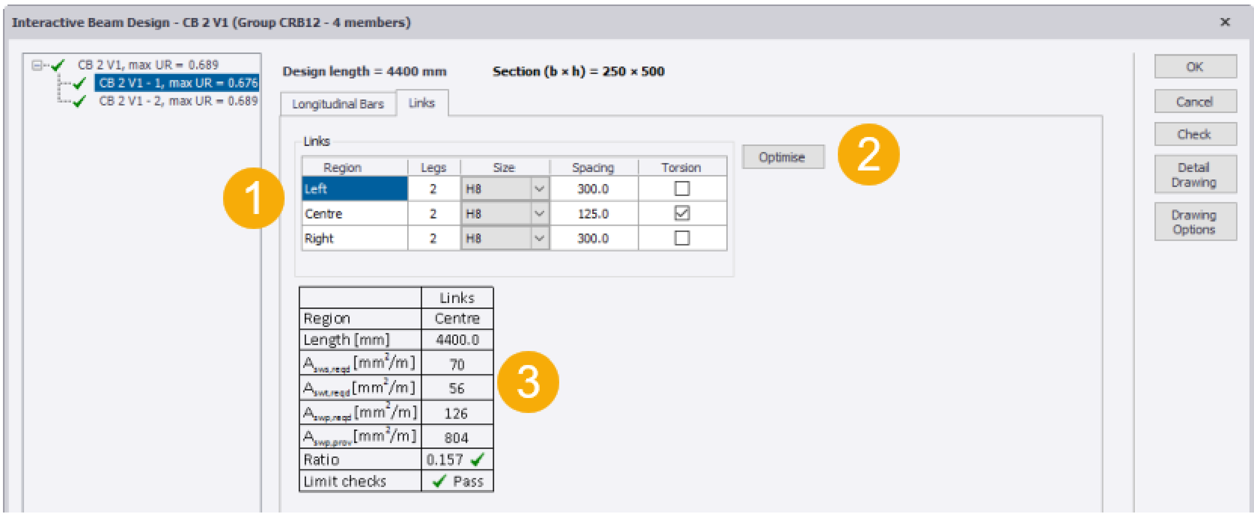


1. **Span Summary**: Displays the design status of the selected span and the associated utilization ratio.
2. **Bar Selection Tables**: Used for editing the longitudinal bars into the beam.
   - Each row in the table is labeled with a specific “bar number” (taken from the standard patterns applied to the beam in the Properties Window); these represent bar locations within the beam.
   - Two different bar sizes can be defined in each row, the only restriction being that the second bar must always be smaller than the first.
   - The number of bars of each size is defined using the **Count** field.

- When bars are joined to the adjacent span, changing those bars within this span has the effect of changing those bars in the adjacent span, as they are effectively the same bar. (This is only done when the spans are "matching" in terms of their alignment and dimensions.)

3. **Bar Pattern Layout**: a schematic diagram representing the top and bottom patterns assigned to the beam.
4. **Design Summary Table**: Displays critical results for each of the design regions from all combinations:
    - Area of reinforcement required, $A_{s,reqd}$
    - Area of reinforcement provided, $A_{s,prov}$
    - Reinforcement area utilization ratio
    - Smallest clear spacing between bars
    - Minimum required reinforcement area, $A_{s,min}$
5. **Additional checks**: Side bar, deflection and cover check results are displayed below the design summary table.

3. Links/Stirrups tab



1. **Link/Stirrup Selection Table**: Specifies the number of link/stirrup legs, size and spacing in each of the regions.
2. **Optimise Button** : This calculates the optimum length of the central region given the reinforcement that you have selected. The button is not be visible when the beam is in a design group with other beams, and is also not visible when the span is a cantilever.
3. **Link/Stirrup Design Summary Table**: Displays the most critical result from all combinations:

- Region length
- Link/Stirrup area over spacing required for shear, $A_{sw,reqd/s}$
- Link/Stirrup area over spacing required for torsion, $A_{swt,reqd/s}$
- Link/Stirrup area provided, $A_{sw,prov}$
- Link/Stirrup utilization ratio

4. **Buttons**: (See separate section below.)

Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog

See also

Interactive concrete member design (page 1310)

Interactive concrete column design

Opening the Interactive Column Design Dialog

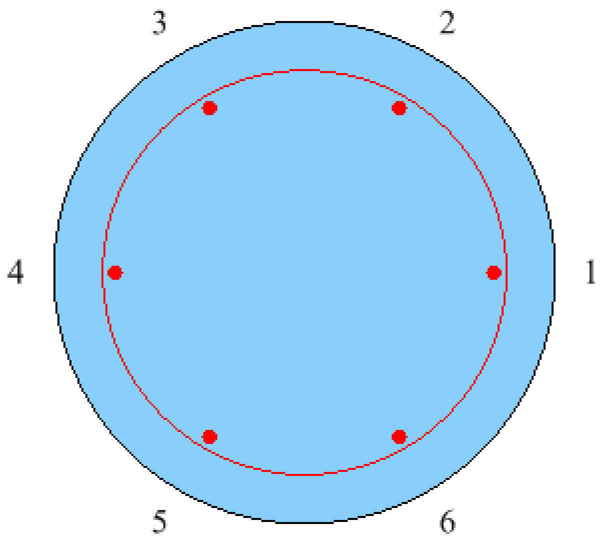
The Interactive Column Design dialog (page 1327) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Column Design dialog is displayed.
3. Click on an individual stack in the Column/stack summary pane (page 1328).
4. Interactively adjust the reinforcement as required for the chosen stack until the design is satisfactory.

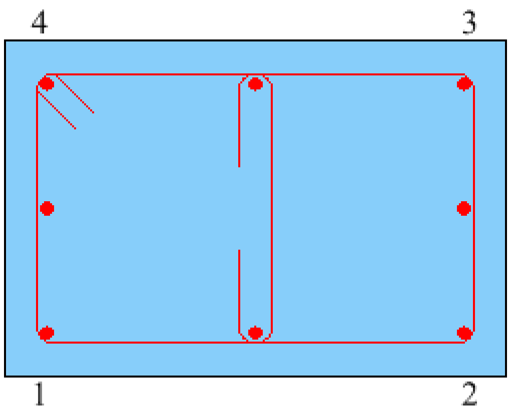
Arranging bars in the Interactive Column Design Dialog

The way in which bars are arranged depends on the column shape.

In circular columns, bars are arranged simply by modifying the bar size and count fields.



In rectangular columns bars are arranged as described below.



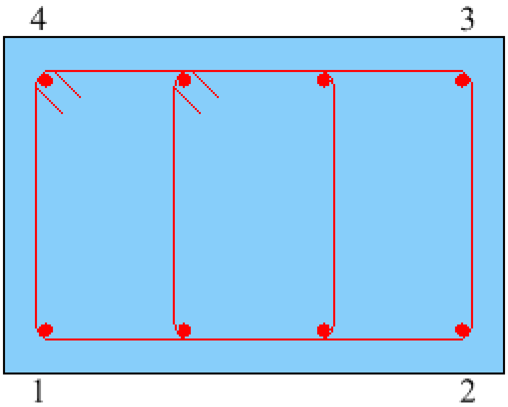
Principal bars exist at fixed locations; they are labeled with numbers in the cross-section, as shown above. You can only change the principal bar sizes, not their locations.

Intermediate bars are the unnumbered bars in the cross-section. You can change both their size and number. They are defined in the bar location table by reference to the principal bars between which they lie.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	1	249.0	✓	3-4	1	249.0	✓
2-3	1	149.0	✓	4-5	1	149.0	✓

A count of "1" for each intermediate length in the bar location table indicates that a single intermediate bar is positioned between each of the principal bars.

If the count is increased to "2" for Int. length 1-2, but reduced to "0" for Int. length 2-3, the following arrangement is achieved. (Two intermediate bars are positioned between principal bars 1 and 2, but there are now no intermediate bars between principal bars 2 and 3.)



Note that Int. lengths 3-4 and 4-5 are adjusted automatically in the table to match.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	2	166.0	✓	3-4	2	166.0	✓
2-3	0	298.0	✓	4-5	0	298.0	✓

Link / Tie arrangements in rectangular and parallelogram sections have the following basic options:

- Single links/ties,
- Double links/ties,
- Triple links/ties,
- Cross links/ties.

Tie bars are used with these arrangements. Link/Tie arrangements in other section shapes use standard link/tie positions with additional tie bars where required.

Using a 2nd layer of bars

It is sometimes not possible to find a solution using a single-layer of reinforcement - (a problem more common in hi-rise structures with large columns in lower stacks).

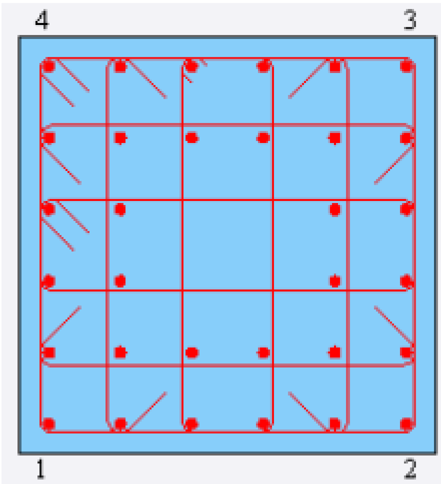
Selecting the **Use 2nd layer** option in the interactive design provides you with more flexibility to find a solution in these cases.

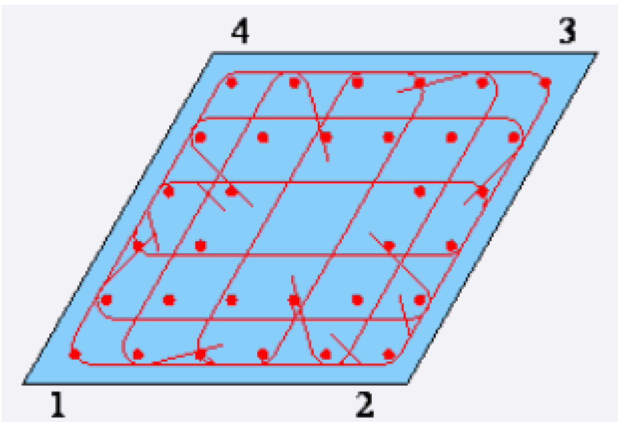
A 2nd layer can be applied to rectangular/parallelogram and circular sections.

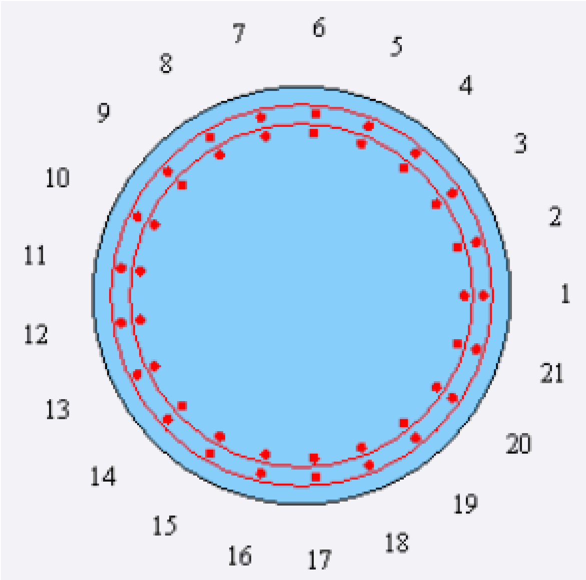
Rectangular/parallelogram: a second layer in line with the first bar along the adjacent length

- At least 2 intermediate bars are needed in each direction

A link is added along the second layer for all 3 section shapes.







When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

Column interaction diagrams (US customary units)

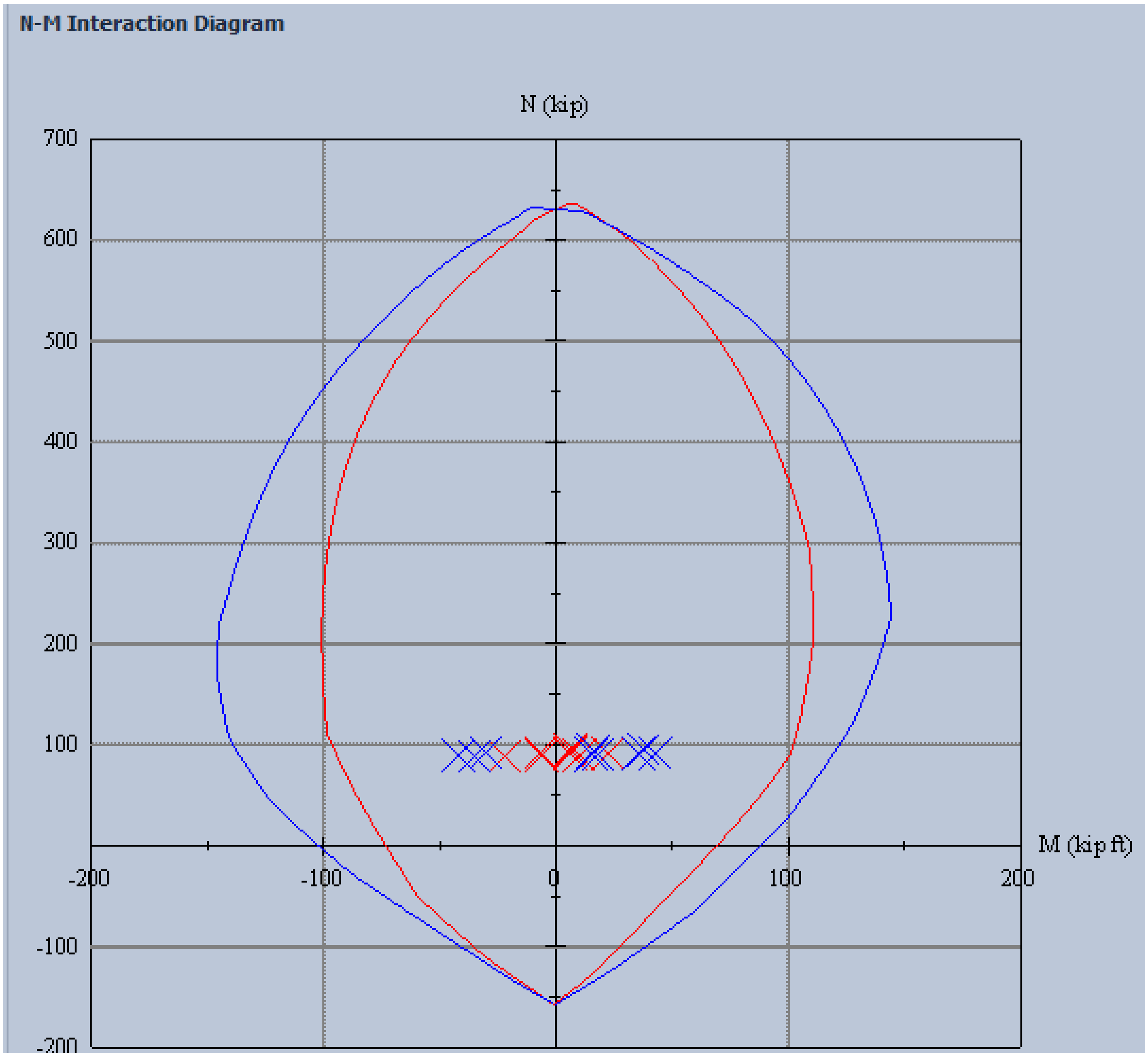
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

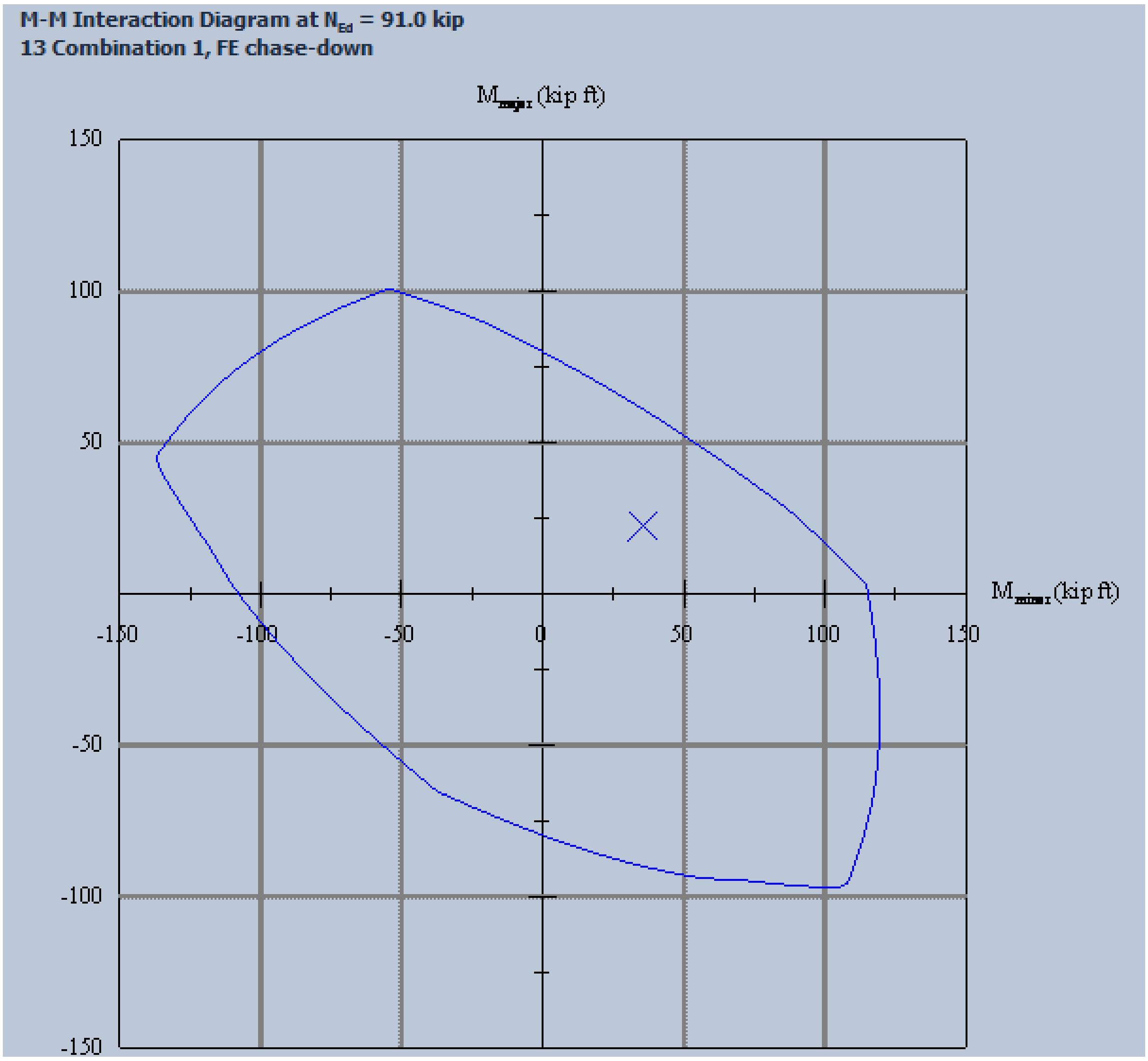


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

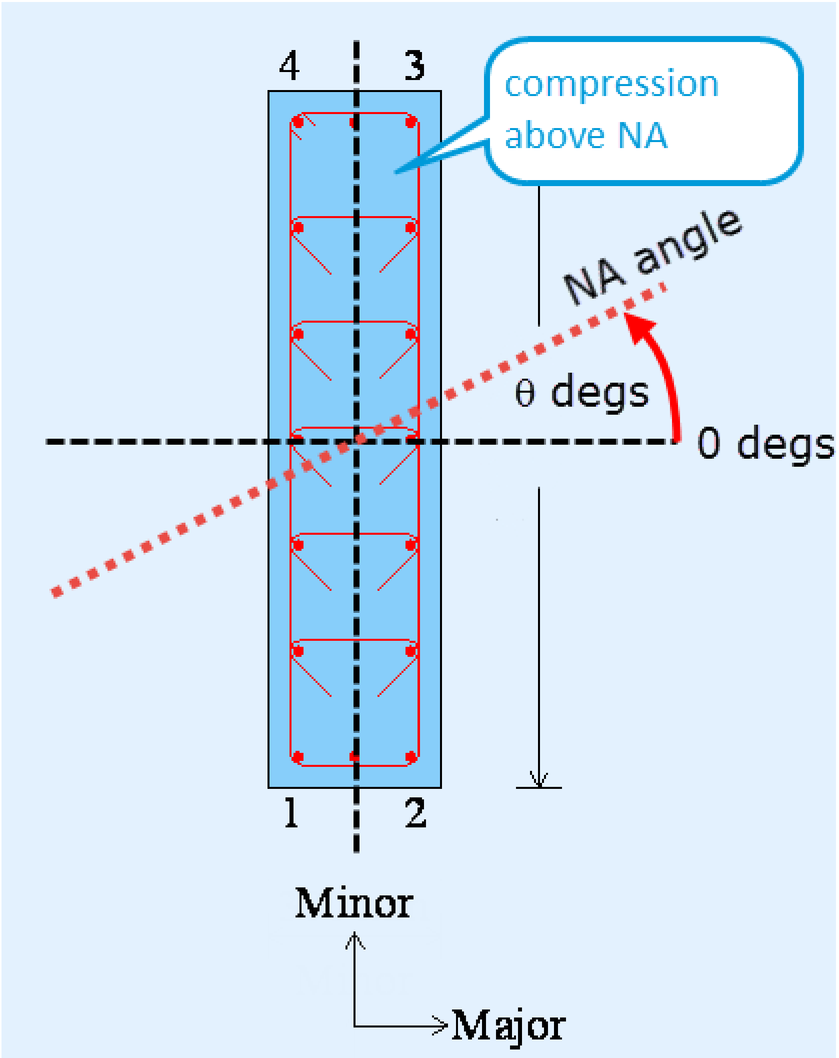
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla

Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Column interaction diagrams (metric units)

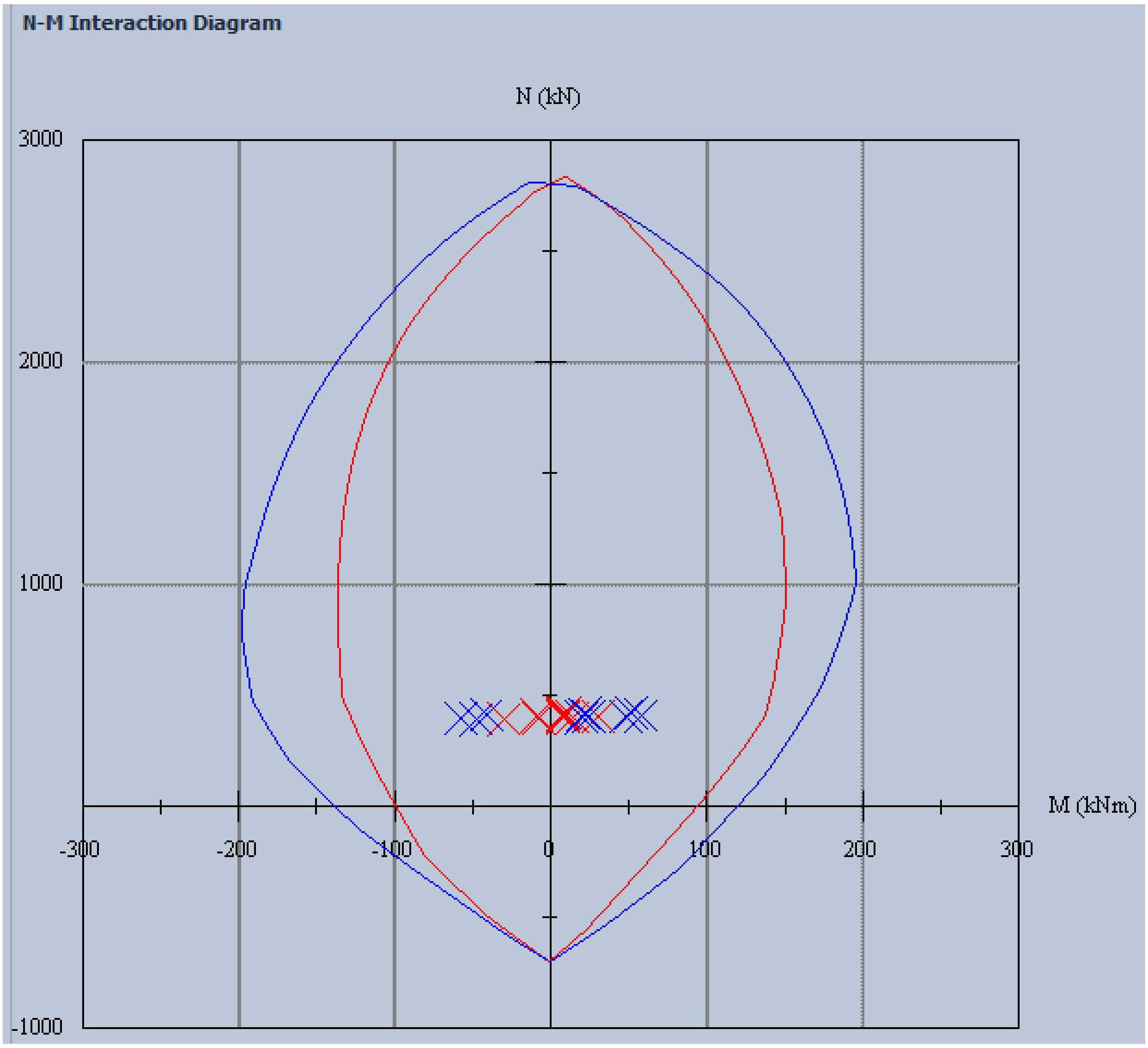
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

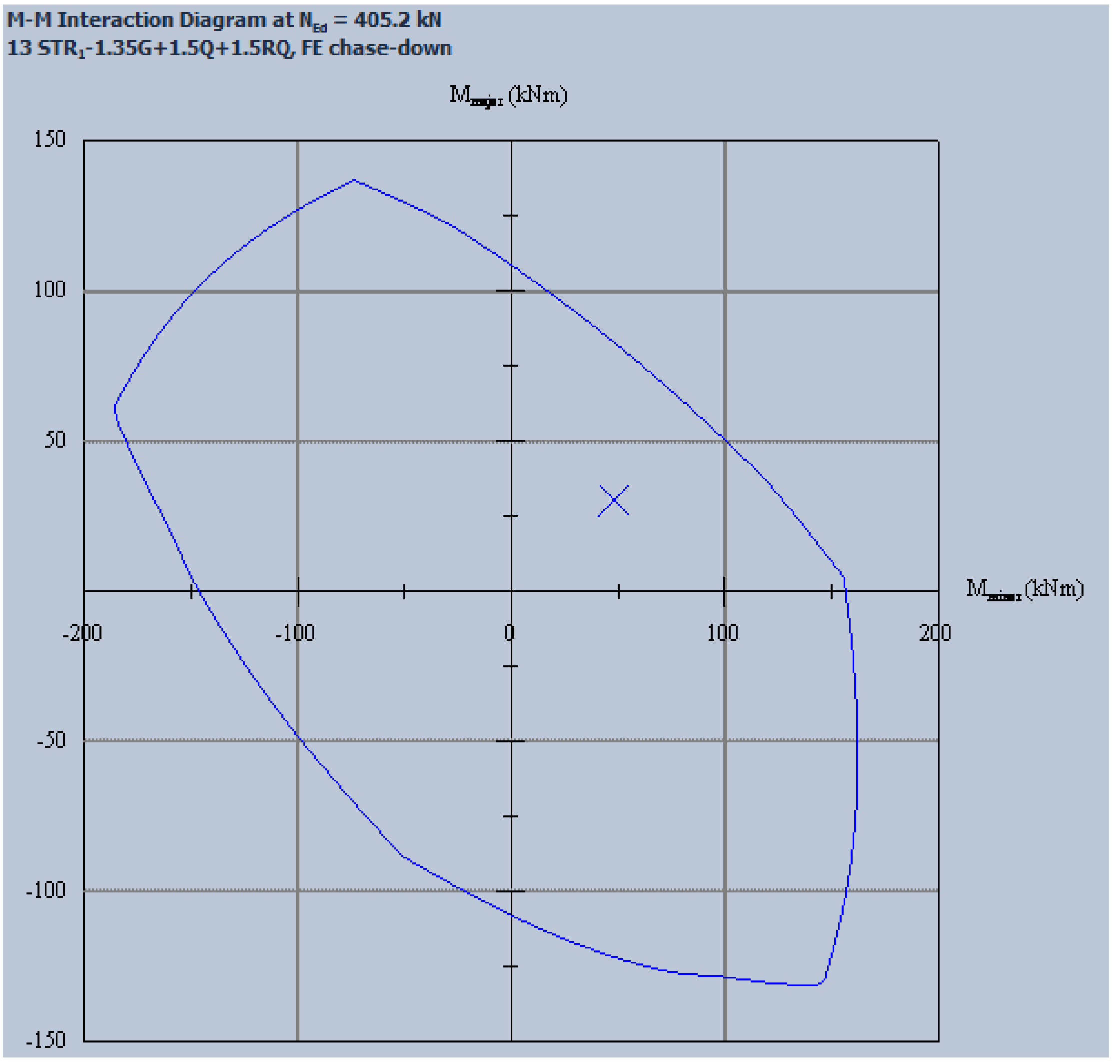


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

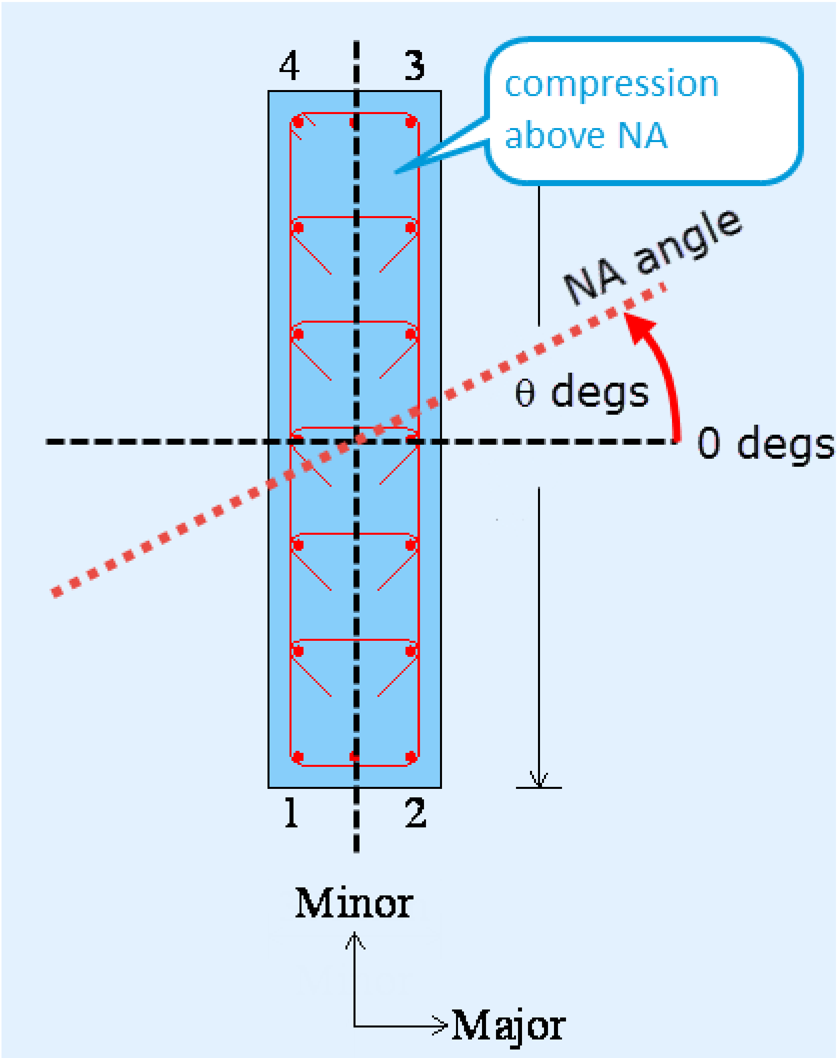
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla

Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional column design cases for user defined forces

Additional design cases can be specified in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected stack on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Column Dialog, select Additional Design Cases (page 1333) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all column stacks where appropriate.

The additional loading cases are now always checked whenever the regular combinations are checked.

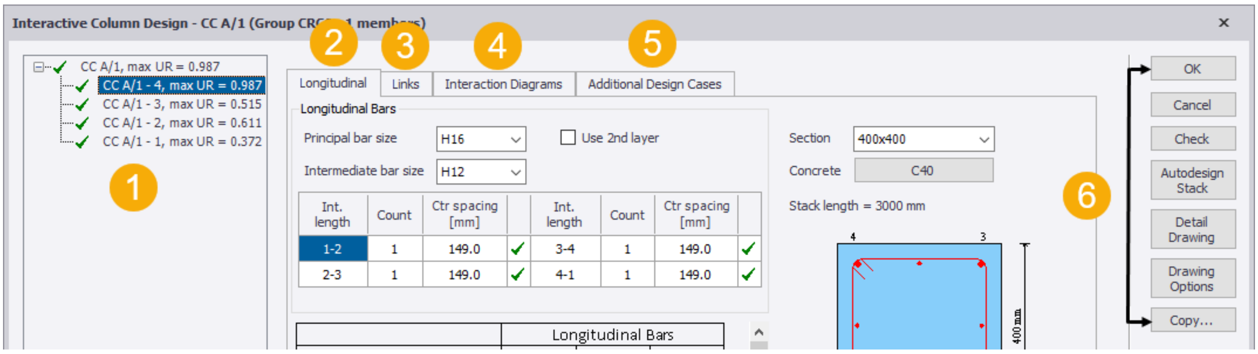
Interactive Column Design dialog

The **Interactive Column Design dialog** shows the current reinforcement and check results for each stack in the selected column. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

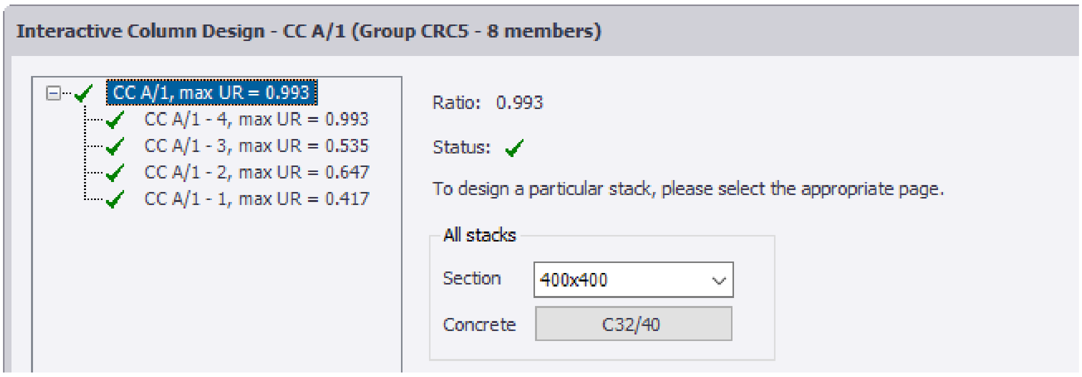
To display the dialog:

1. Right click on an existing concrete column.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Column/stack summary pane



The top row in this pane shows the column summary, consisting of the overall utilization ratio and design status.

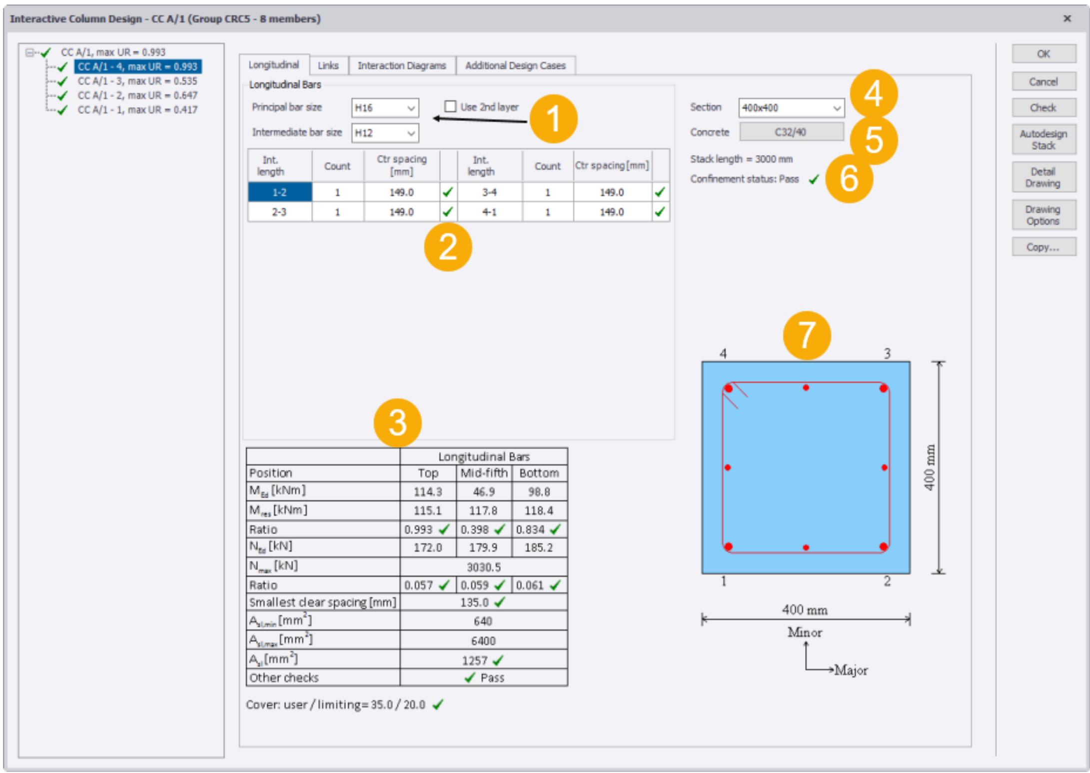
- With this row selected you can edit the section size and grade for all stacks simultaneously.

Subsequent rows show the design status of each stack and associated utilization ratio.

- To design a particular stack, click on the corresponding row in the summary pane.

2. Longitudinal tab

All straight-edged cross sections have “Principal” bars located at shear tie corners. Between these, evenly spaced identical “Intermediate” bars can be located. Circular sections have 6 or more evenly spaced bars around the edge of the section.



1. **Longitudinal Bars**:
    - **Principal bar size**: Used to change the size of **all** principal bars.
        - **Use 2nd layer**: Check the box to allow a second bar layer if required.
            - **Spacing**: For circular sections the user can control the layer spacing.
    - **Intermediate bar size**: (Not displayed for circular columns) Used to change the size of **all** intermediate bars.
2. **Bar Location Table**: Used for adding intermediate bars into the cross-section:
    - Int. length - identifies the edge along which the bars are positioned
    - Count - for changing the number of intermediate bars along the length

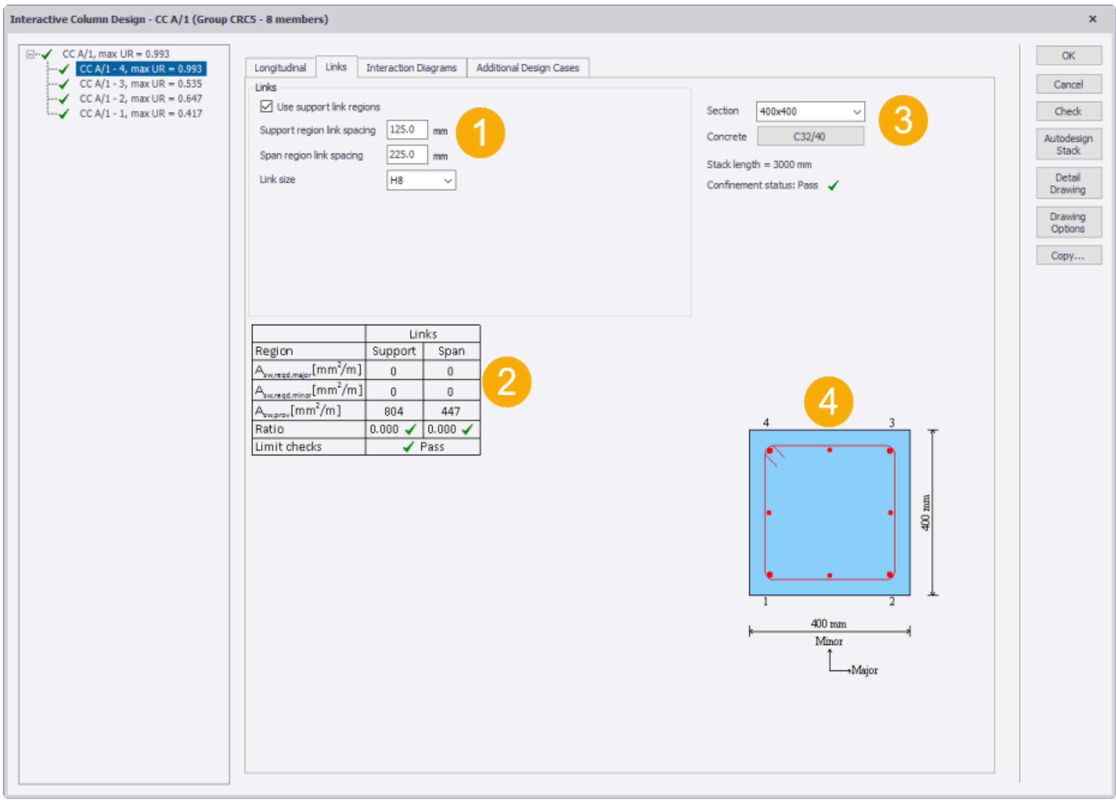
- Ctr spacing - the centerline spacing for the current number of bars along the length
- Status - indicates when the maximum bar spacing limit has been exceeded. (When the minimum bar spacing limit is exceeded this is displayed elsewhere in the Design Summary Table).

3. **Design Summary Table**: Displays the most critical result from all combinations:
    - Design and Resistance Moments and Moment Ratios
    - Axial Force, Axial Resistance and Axial Ratios
    - Smallest clear bar spacing
    - Minimum area of steel
    - Area of steel provided
4. **Section Droplist**: Used for changing the section size for the current stack.

    NOTE If the droplist is used to change the section shape an autodesign is performed.

5. **Concrete Droplist**: Used for changing the concrete grade for the current stack.
6. **Confinement status** : This status is determined based on the requirements for bars being tied.
7. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Tie locations
    - Section dimensions
    - Principal bar labels

3. Links/Ties tab



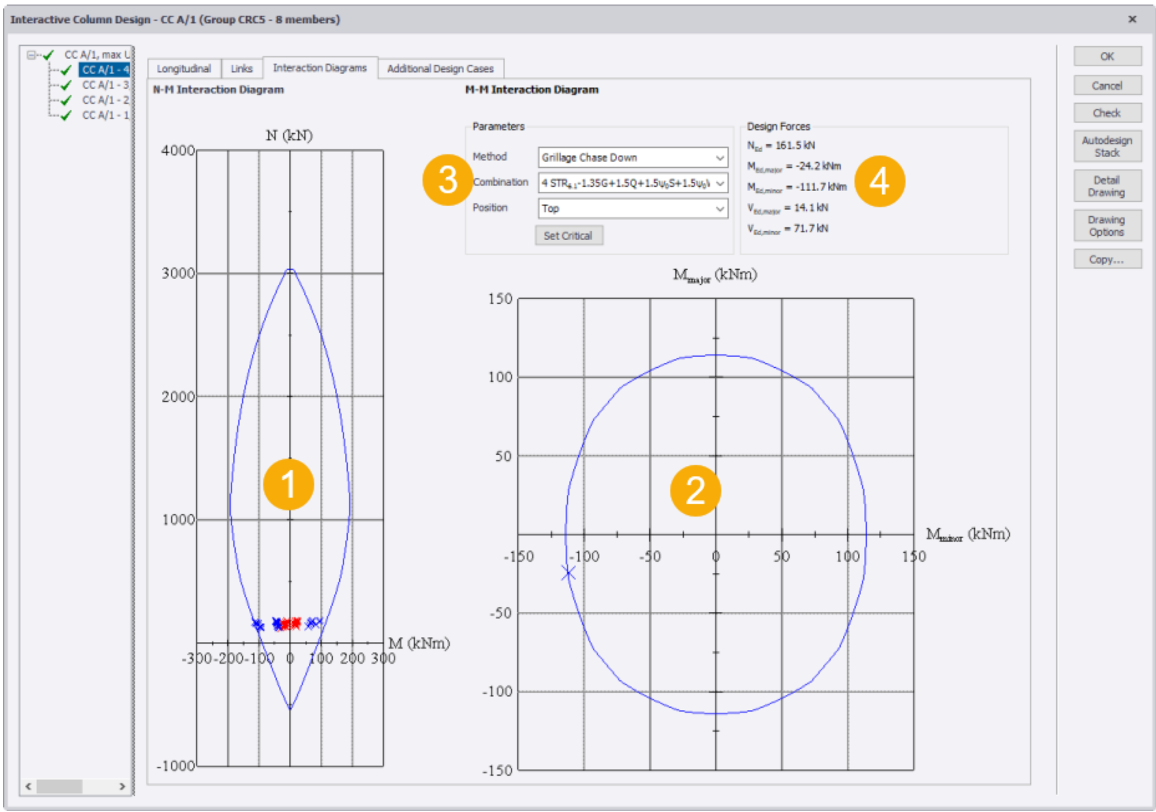
1. **Links**:
    - **Use support region links/ties** : Select to design support regions for the links/ties.
    - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
    - **Link/Tie size**: Used to change the size of link/tiebars (all must have the same size).
2. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
    - Region length
    - Link/Stirrup area over spacing required, major
    - Link/Stirrup area over spacing required, minor
    - Link/Tie area over spacing provided
    - Link/Tie utilization ratio

3. **Section Droplist**: Used for changing the section size for the current stack.

NOTE If the droplist is used to change the section shape an autodesign is performed.

4. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions
   - Principal bar labels

4. Interaction Diagram tab

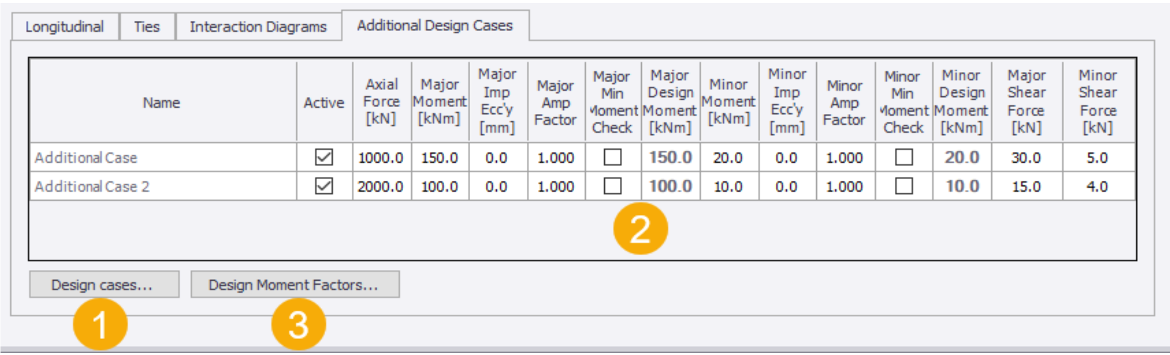


1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
   - The curves for bending about the major axis are shown in red
   - The curves for bending about the minor axis are shown in blue

2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.
   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.
4. **Design Forces**: The design forces applicable for the selected parameters.

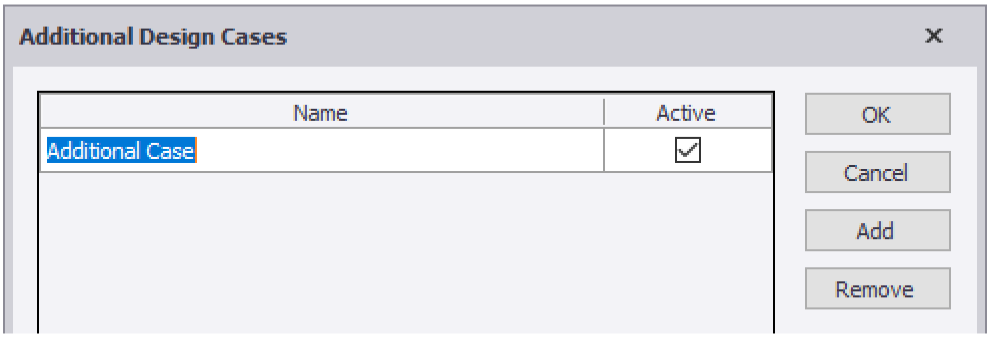
5. Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.



Name	Active	Axial Force [kN]	Major Moment [kNm]	Major Imp Ecc'y [mm]	Major Amp Factor	Major Min Moment Check	Major Design Moment [kNm]	Minor Moment [kNm]	Minor Imp Ecc'y [mm]	Minor Amp Factor	Minor Min Moment Check	Minor Design Moment [kNm]	Major Shear Force [kN]	Minor Shear Force [kN]
Additional Case	☑	1000.0	150.0	0.0	1.000	☐	150.0	20.0	0.0	1.000	☐	20.0	30.0	5.0
Additional Case 2	☑	2000.0	100.0	0.0	1.000	☐	100.0	10.0	0.0	1.000	☐	10.0	15.0	4.0

1. **Design cases...**button: Click to open a dialog in which to add any additional design cases.



The design case names defined here are available to all columns and walls in the model.

The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given stack you must:

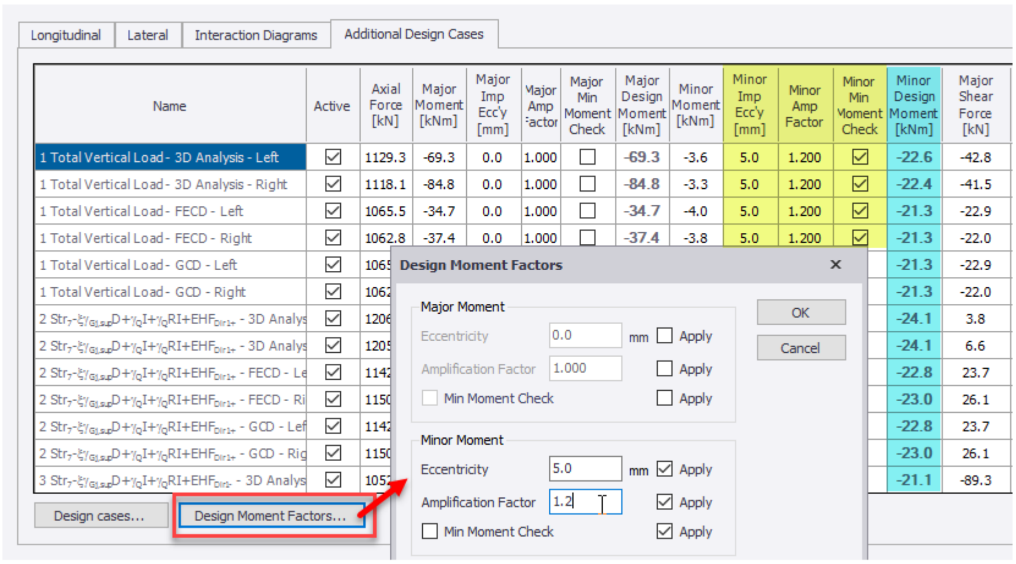
   - Select the required stack in the stack summary
   - Click the Active box for the design case
   - Enter the design forces

3. **Design Moment Factors...** button: Offers three potential “Design Moment” adjustments for each direction:

   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
   - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
   - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension “h” in the direction considered).

   When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

   The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with “Active” option checked on) as shown below:



6. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Autodesign Stack	Performs an autodesign reselecting bars for the current stack
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog
Copy	Opens a new dialog enabling the selective coping of section/concrete grade/reinforcement from the selected stack to other stacks in the same column.

See also

Interactive concrete member design (page 1310)

Interactive concrete wall design

Opening the Interactive Wall Design Dialog

The Interactive Wall Design dialog (page 1345) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the wall you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Wall Design dialog is displayed.
3. Click on an individual panel in the wall/panel summary pane (page 1345).
4. Interactively adjust the reinforcement as required until the panel design is satisfactory.

Wall interaction diagrams (US customary units)

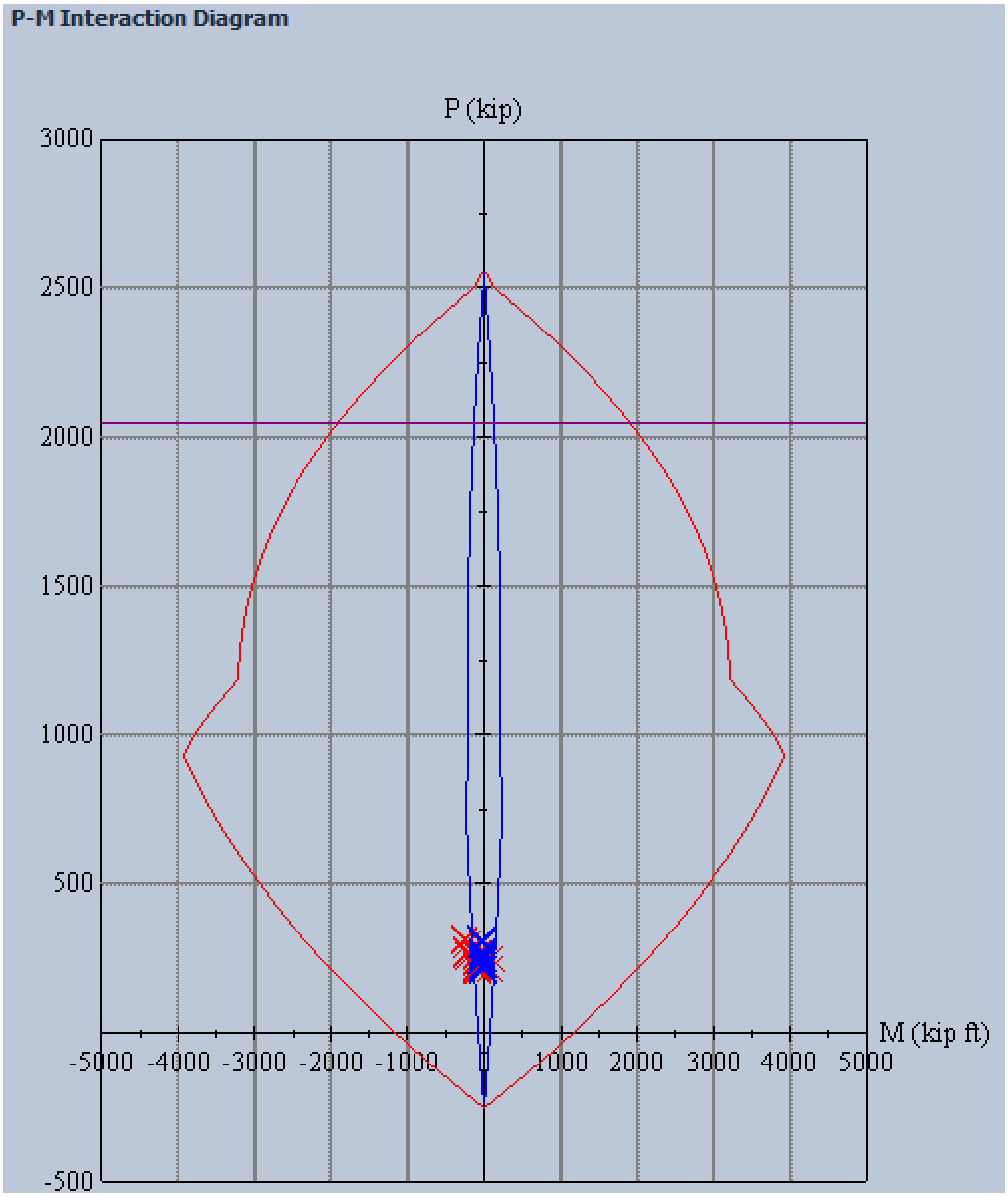
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

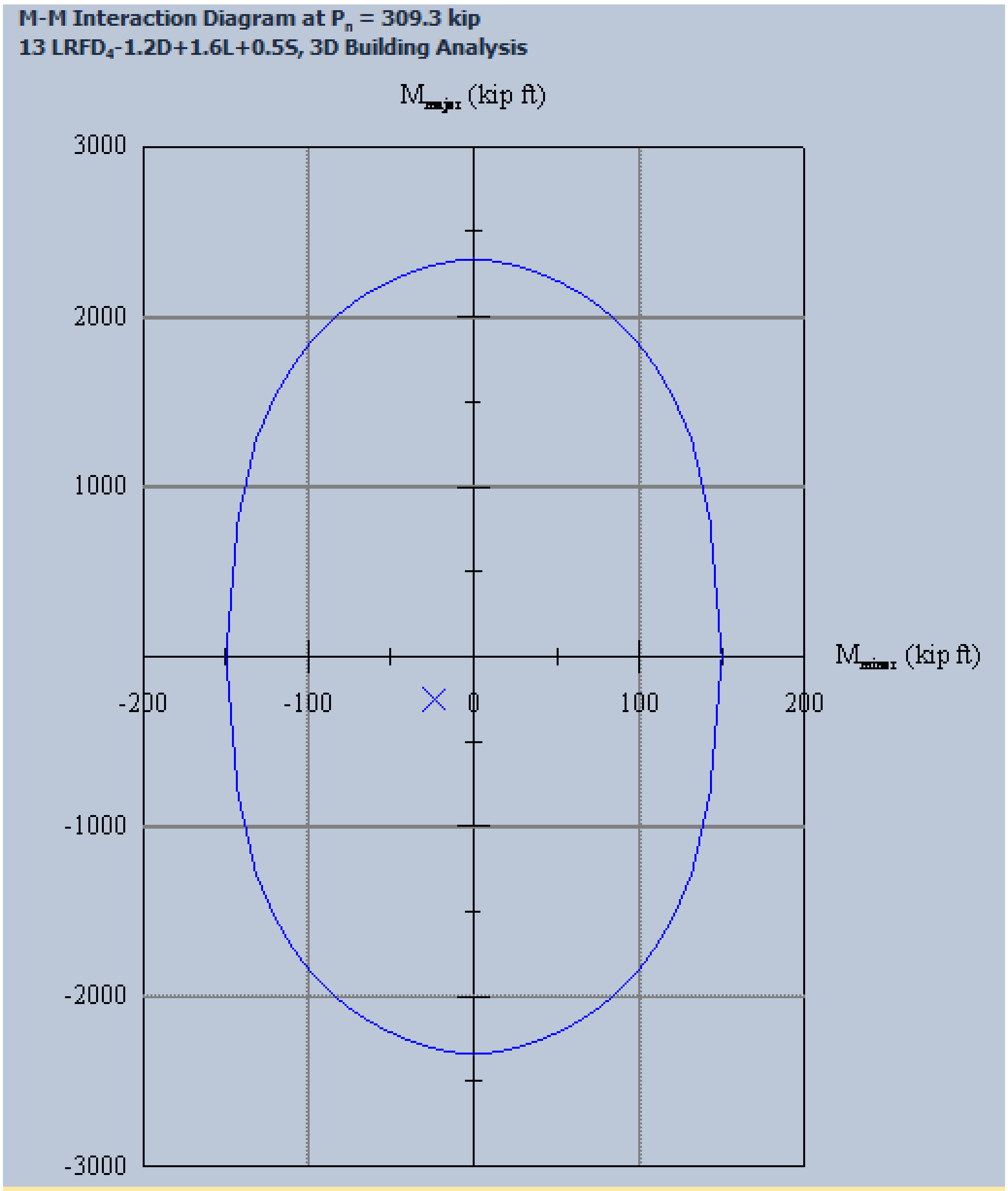
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



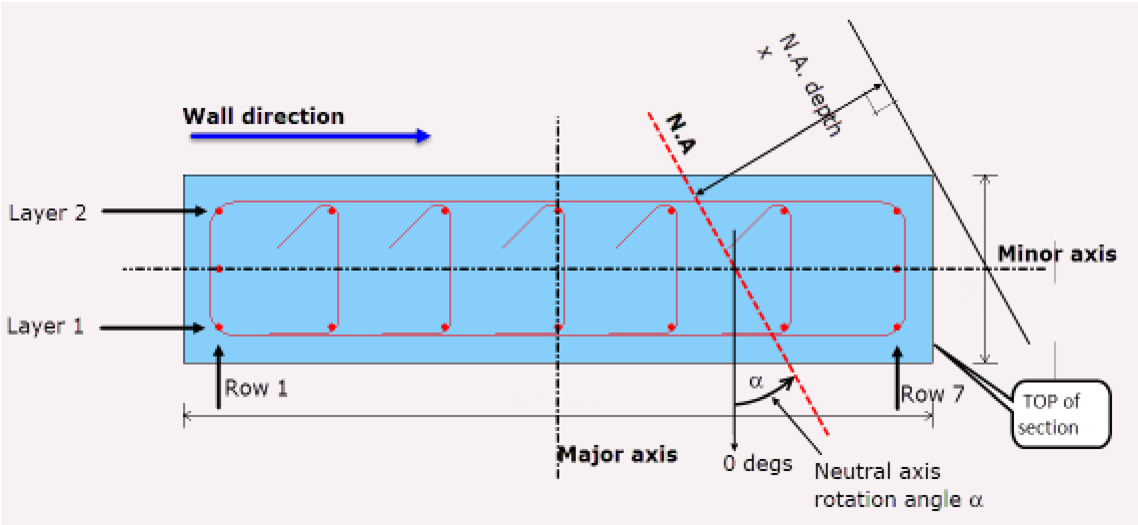
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Wall interaction diagrams (metric units)

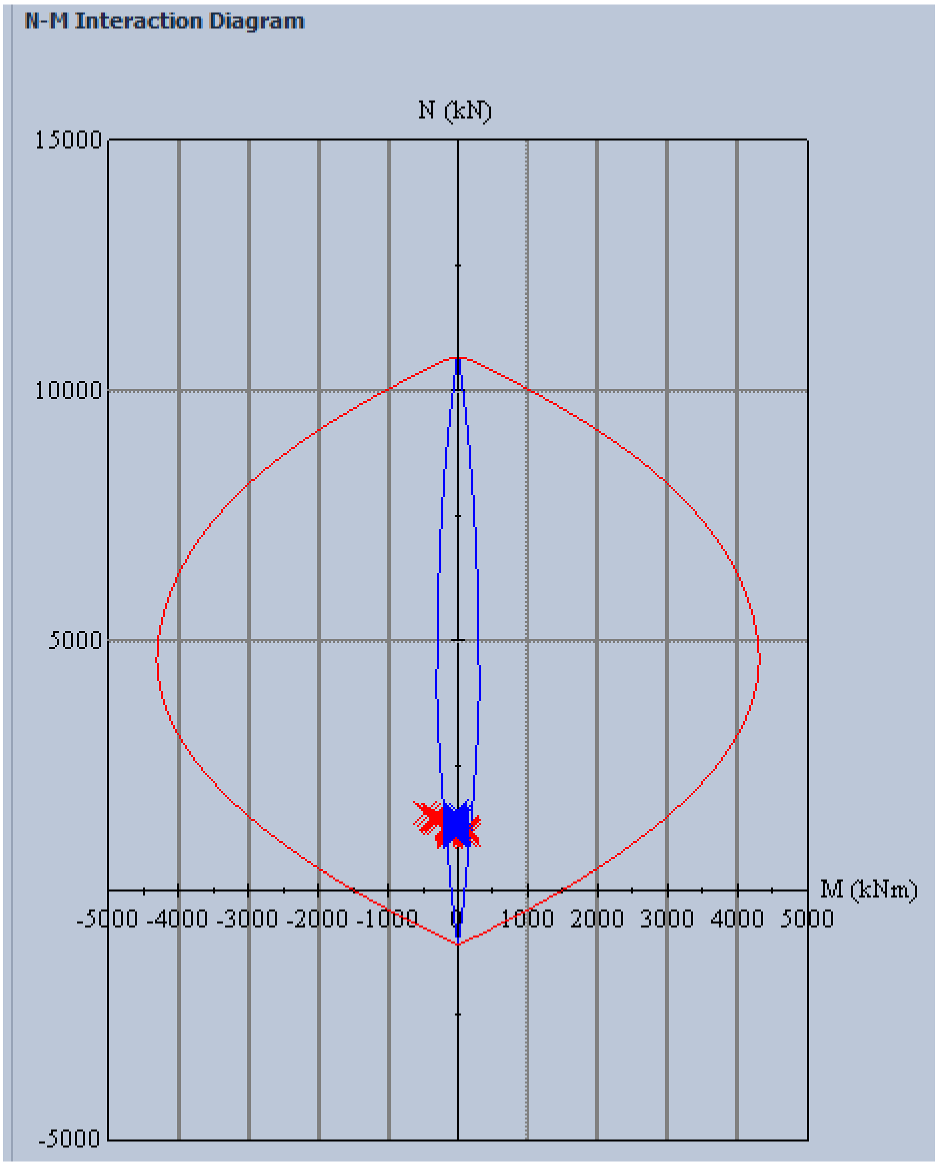
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

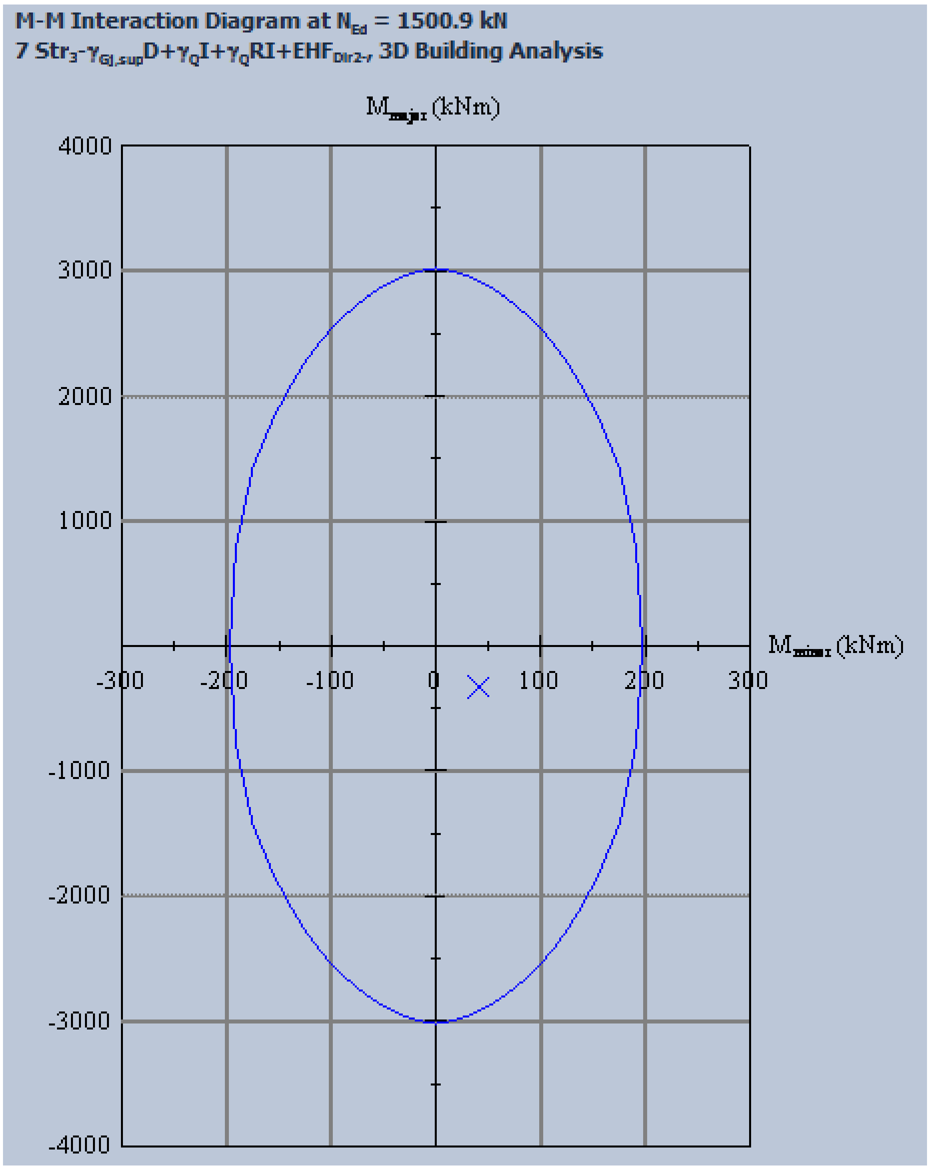
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



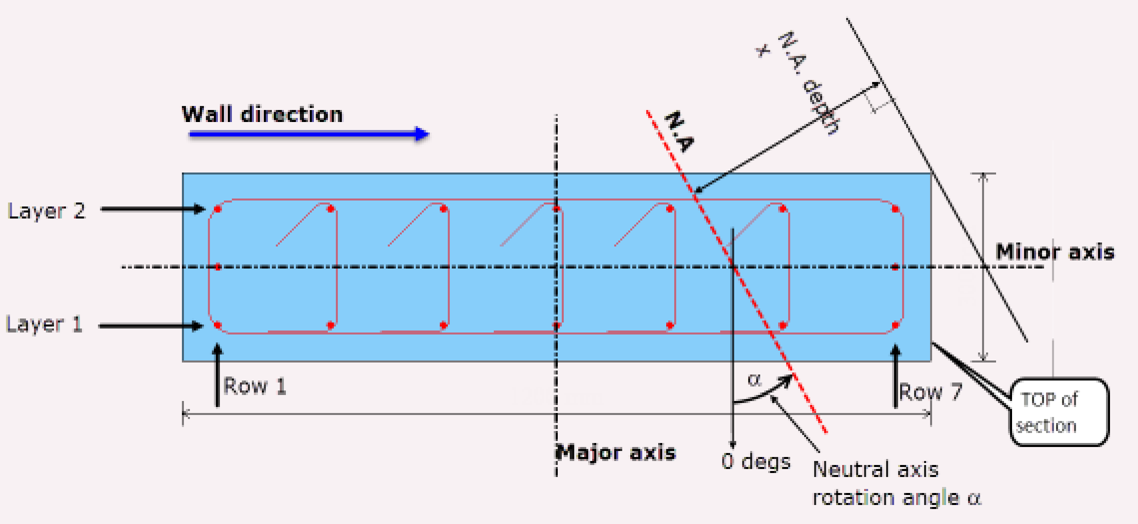
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional wall design cases for user defined forces

Additional design cases can be specified typically in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected panel on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Wall Dialog, select Additional Design Cases (page 1352) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all wall panels where appropriate.

The additional loading cases are always checked whenever the regular combinations are checked.

NOTE Additional design cases can also be set up directly from **Result Lines** in order to facilitate local section design around openings - see the following topic.

Using result lines for local section design around openings

Interactive design of local sections around/between openings is possible using **Result Lines**.

The design is performed either as a column or wall section (as specified by the user), using those design forces determined along the lengths cut by the result lines. Engineering judgement is therefore required when positioning the lines to ensure suitable design forces are obtained.

For further details of the process, see: Manage, display and design result lines (page 688)

Related video

Interactive design using Result lines

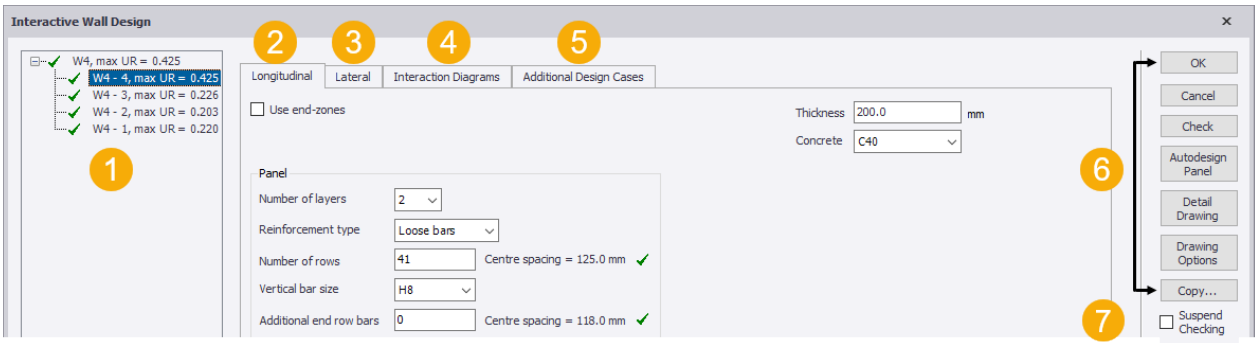
Interactive Wall Design dialog

The **Interactive Wall Design dialog** shows the current reinforcement and check results for each panel in the selected wall. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

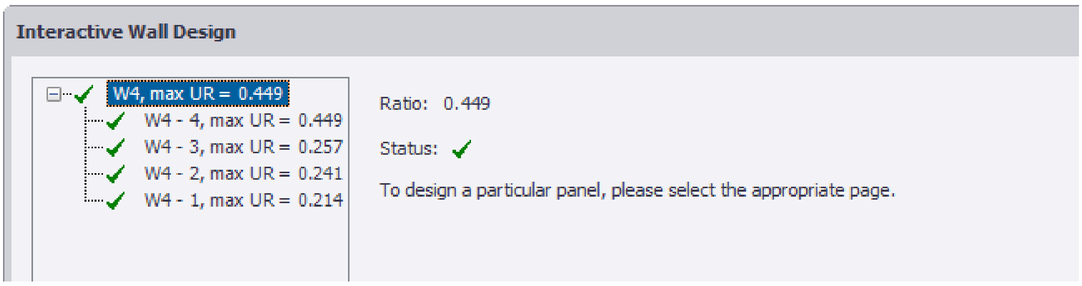
To display the dialog:

1. Right click on an existing concrete beam.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Wall/panel summary pane



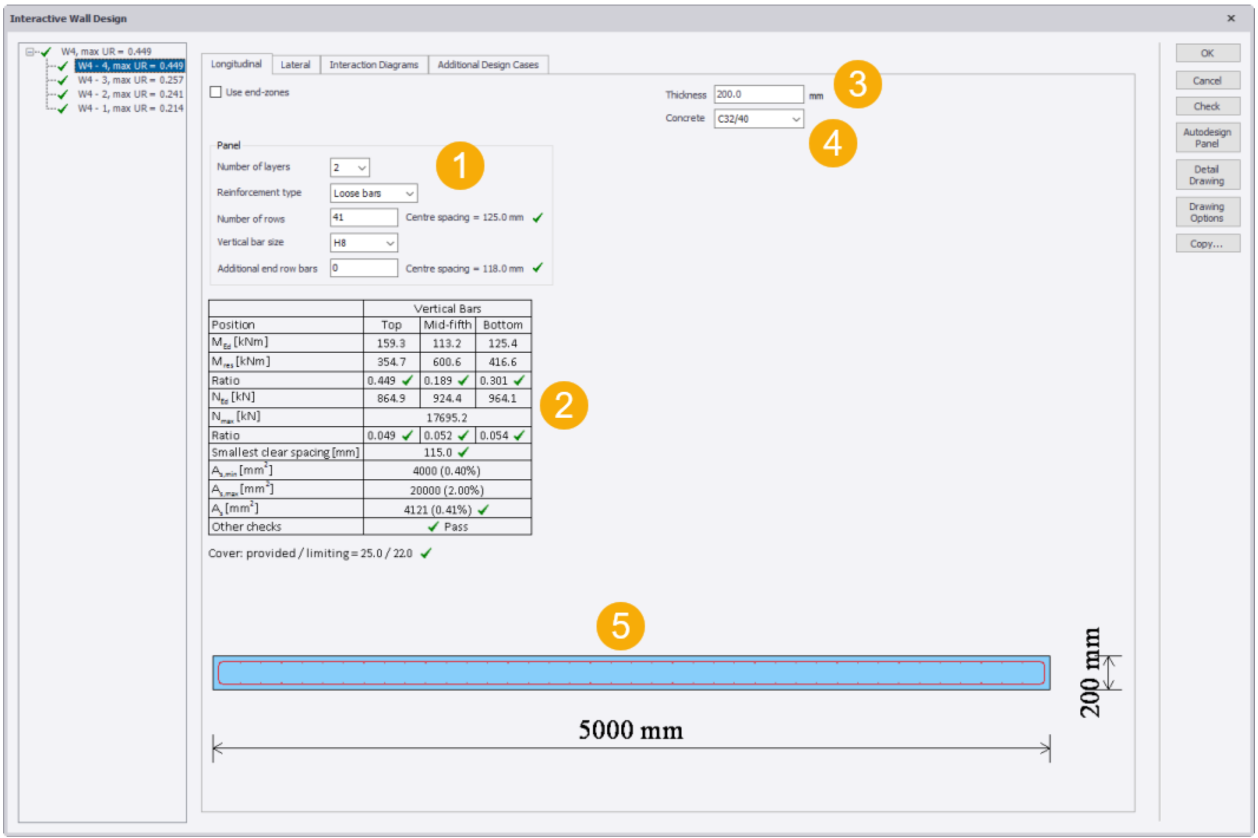
The top row in this pane shows the Wall summary, consisting of the overall utilization ratio and design status.

- With this row selected you can edit the thickness and grade for all stacks simultaneously.

Subsequent rows show the design status of each panel and associated utilization ratio.

- To design a particular panel, click on the corresponding row for the panel in the summary pane.

2. Longitudinal tab (no end-zones)

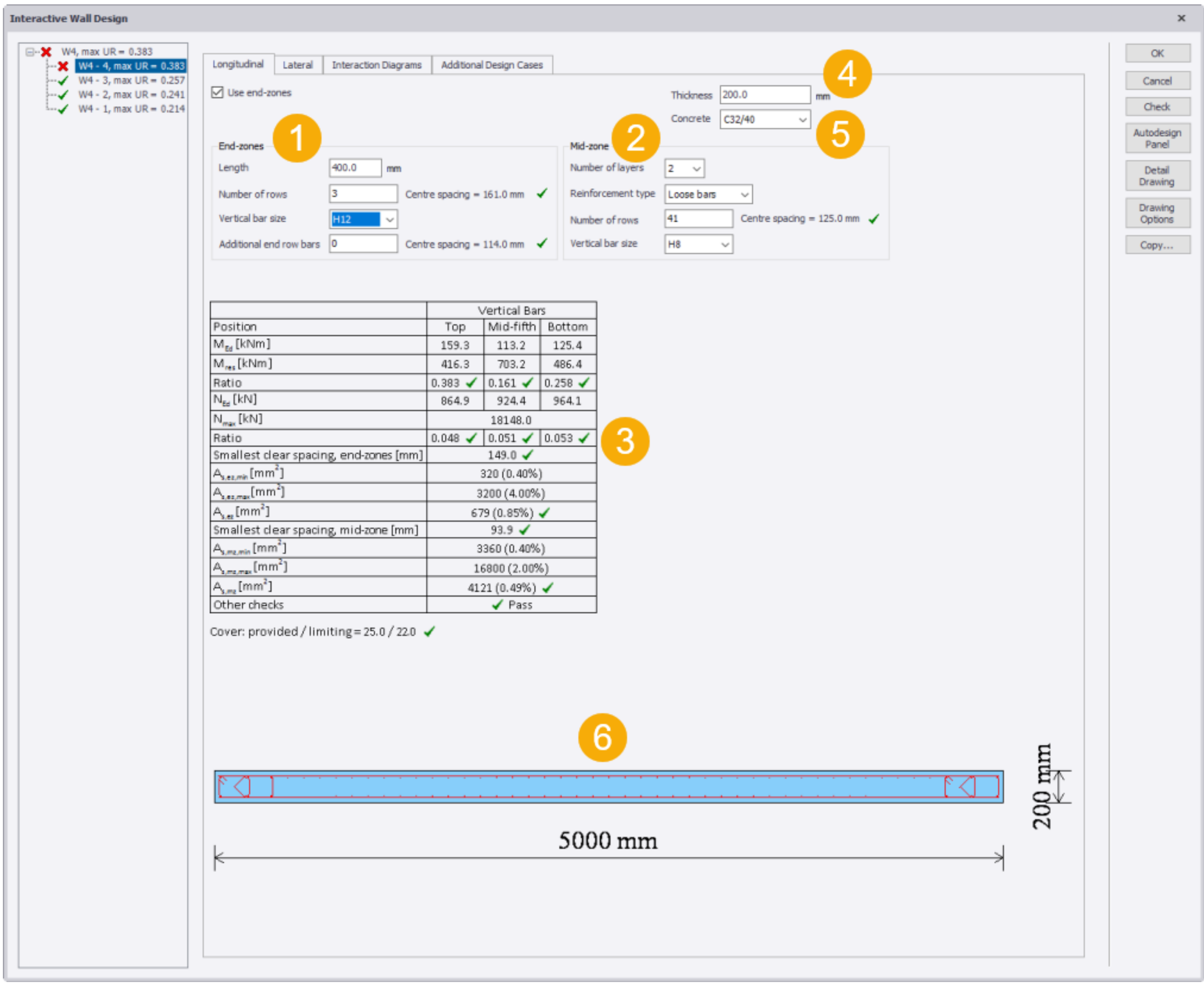


Use end-zones : With this box unchecked end-zones are not used.

1. **Panel**: Used for adding either one or two layers of bars in the panel.
   - Number of layers - (1, or 2)
   - If Reinforcement type = loose bars
     - Number of rows - the number of bars in each layer
     - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
     - Vertical bar size - specifies the size to be checked
     - Additional end row bars - specifies the number of additional bars in the end row
     - Centre spacing (end rows) - the spacing between layers (measured centre to centre)
   - If Reinforcement type = mesh
     - Mesh size - specifies the mesh size to be checked

    - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
  - Additional end row bars - specifies the number of additional bars in the end row
  - Centre spacing (end rows) - the spacing between layers (measured centre to centre)
2. **Design Summary Table**: Displays the most critical result from all combinations:
   - Design and Resistance Moments and Moment Ratios
   - Axial Force, Axial Resistance and Axial Ratios
   - Smallest clear bar spacing
   - Minimum area of steel
   - Area of steel provided
3. **Thickness**: Used for changing the thickness of the current panel.
4. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
5. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions

2. Longitudinal tab (with end-zones)



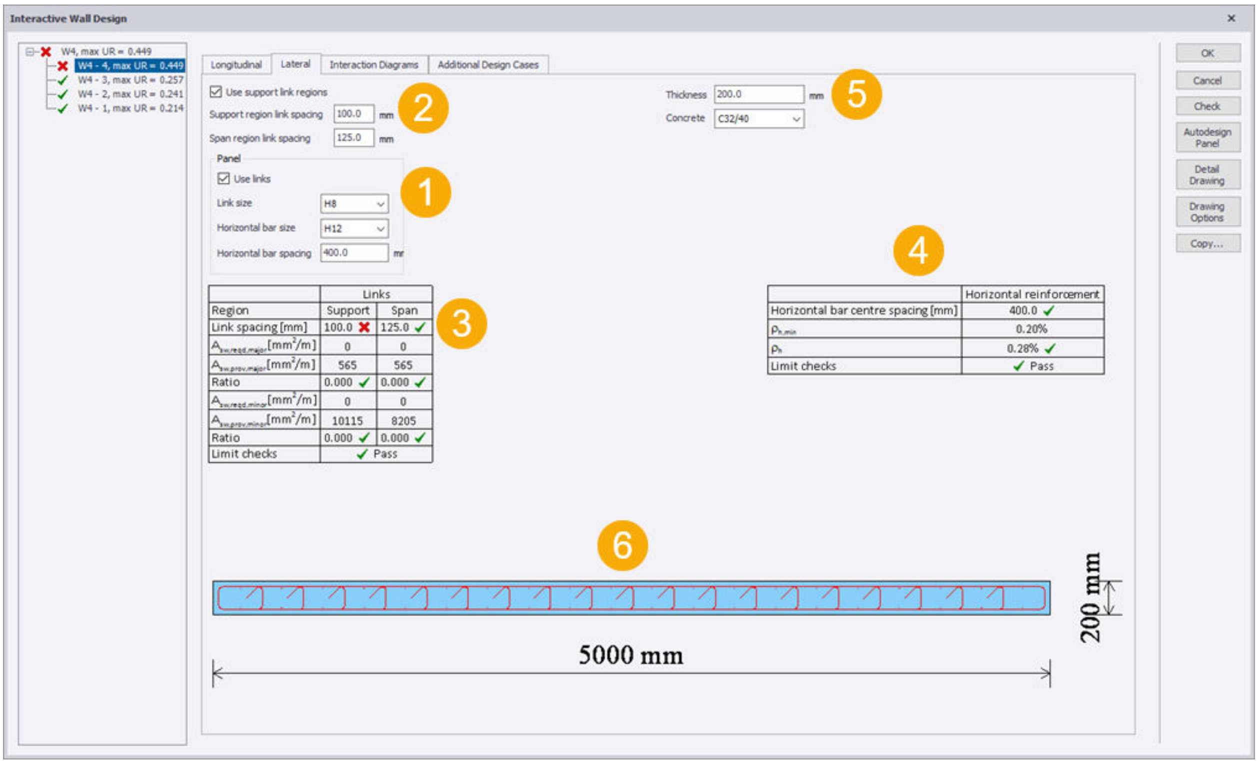
	Vertical Bars		
Position	Top	Mid-fifth	Bottom
M_{Ed} [kNm]	159.3	113.2	125.4
M_{res} [kNm]	416.3	703.2	486.4
Ratio	0.383 ✓	0.161 ✓	0.258 ✓
N_{Ed} [kN]	864.9	924.4	964.1
N_{max} [kN]	18148.0		
Ratio	0.048 ✓	0.051 ✓	0.053 ✓
Smallest clear spacing, end-zones [mm]	149.0 ✓		
$A_{s,ez,min}$ [mm^2]	320 (0.40%)		
$A_{s,ez,max}$ [mm^2]	3200 (4.00%)		
$A_{s,ez}$ [mm^2]	679 (0.85%) ✓		
Smallest clear spacing, mid-zone [mm]	93.9 ✓		
$A_{s,mz,min}$ [mm^2]	3360 (0.40%)		
$A_{s,mz,max}$ [mm^2]	16800 (2.00%)		
$A_{s,mz}$ [mm^2]	4121 (0.49%) ✓		
Other checks	✓ Pass		

Use end-zones : With this box checked end-zones are used.

1. **End-zones**: Used for adding bars in the end-zones.
   - Length - length of each end-zone
   - Number of rows - the number of loose bars in each end-zone
   - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
   - Vertical bar size - specifies the size to be checked
   - Additional end row bars - specifies the number of additional bars in the end row
   - Centre spacing (end rows) - the spacing between layers in the end-zone (measured centre to centre)

2. **Mid-zone**: Used for adding either one or two layers of bars in the panel between the end-zones.
   - Number of layers - (1, or 2)
   - If Reinforcement type = loose bars
     - Number of rows - the number of bars in each layer
     - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
     - Vertical bar size - specifies the size to be checked
   - If Reinforcement type = mesh
     - Mesh size - specifies the mesh size to be checked
     - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
3. **Design Summary Table**: Displays the most critical result from all combinations:
   - Design and Resistance Moments and Moment Ratios
   - Axial Force, Axial Resistance and Axial Ratios
   - Smallest clear bar spacing
   - Minimum area of steel
   - Area of steel provided
4. **Thickness**: Used for changing the thickness of the current panel.
5. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
6. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions

3. Lateral tab



1. **Panel**:
   - **Use links/ties** : Select to specify links/ties.
     - **Link/Tie size**: Used to change the size of link/ties (all must have the same size).
   - **Horizontal bar size**: Used to specify the size of horizontal bars.
   - **Horizontal bar spacing**: Used to specify the vertical spacing of horizontal bars.
2. **Use support region links/ties**: Select to design support regions for the links/ties.

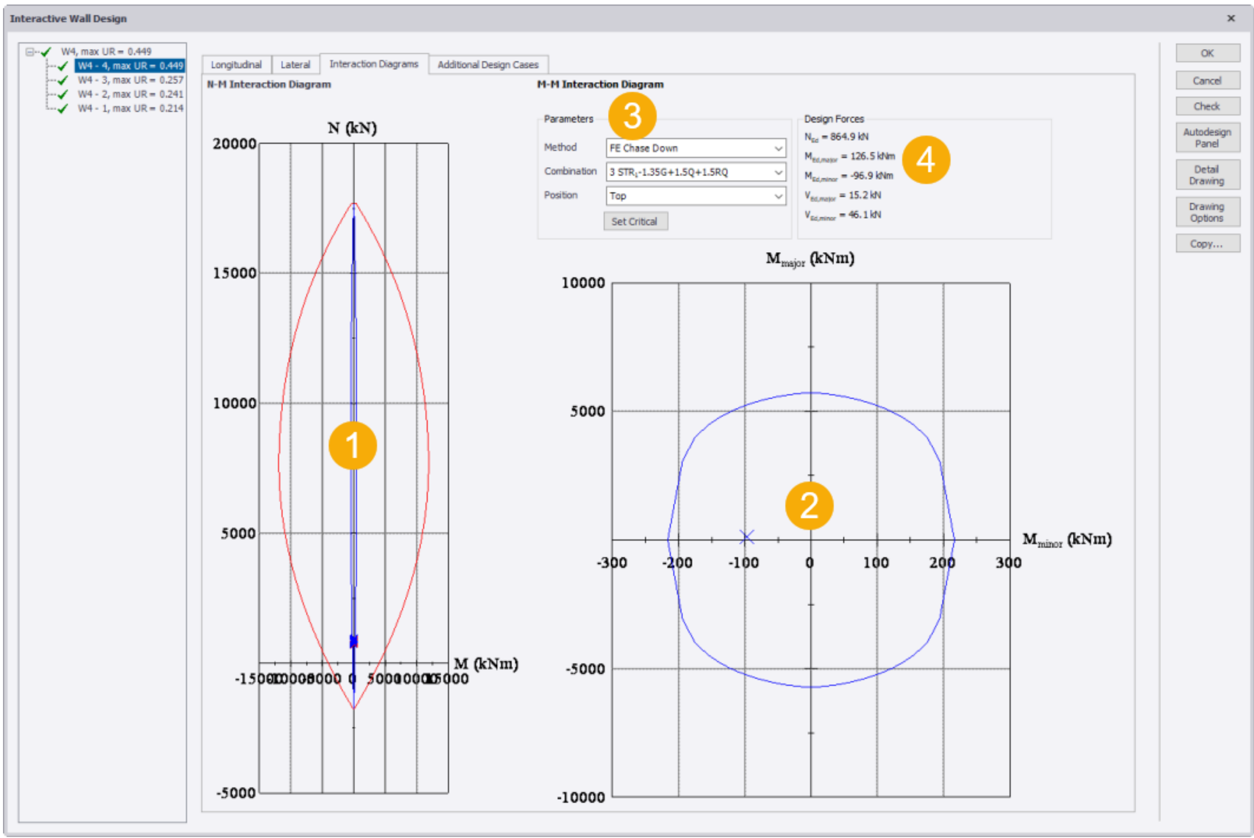
   NOTE Only displayed provided Use links/ties box is also checked.

   - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
3. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
   - Link/Tie area over spacing required, major
   - Link/Tie area over spacing provided, major
   - Link/Tie utilization ratio, majo

- Link/Tie area over spacing required, mino
- Link/Tie area over spacing required, mino
- Link/Tie utilization ratio, minor

4. **Horizontal Reinforcement Summary Table**: The table displays the horizontal bar spacing and reinforcement ratios.
5. **Thickness**: Used for changing the thickness of the current panel.
6. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions

4. Interaction Diagrams tab

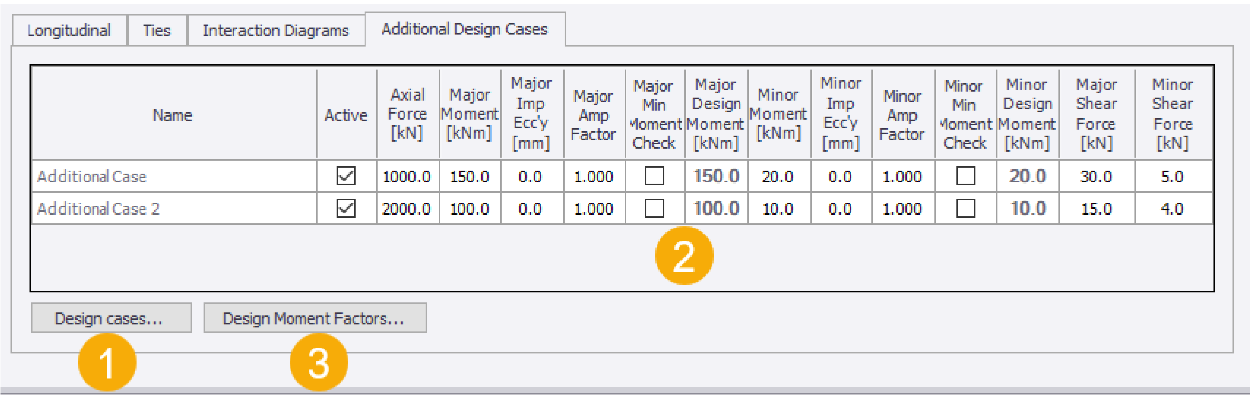


1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
   - The curves for bending about the major axis are shown in red
   - The curves for bending about the minor axis are shown in blue

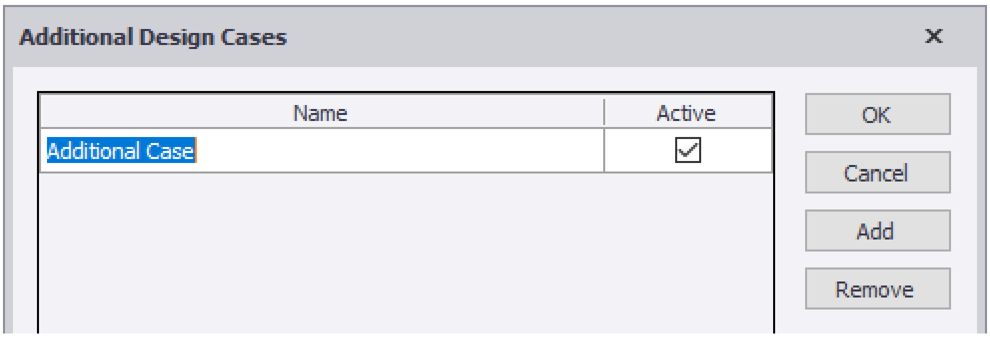
2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.
   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.
4. **Design Forces**: The design forces applicable for the selected parameters.

5. Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.



1. **Design cases...**button: Click to open a dialog in which to add any additional design cases.



The design case names defined here are available to all columns and walls in the model.

The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given panel you must:

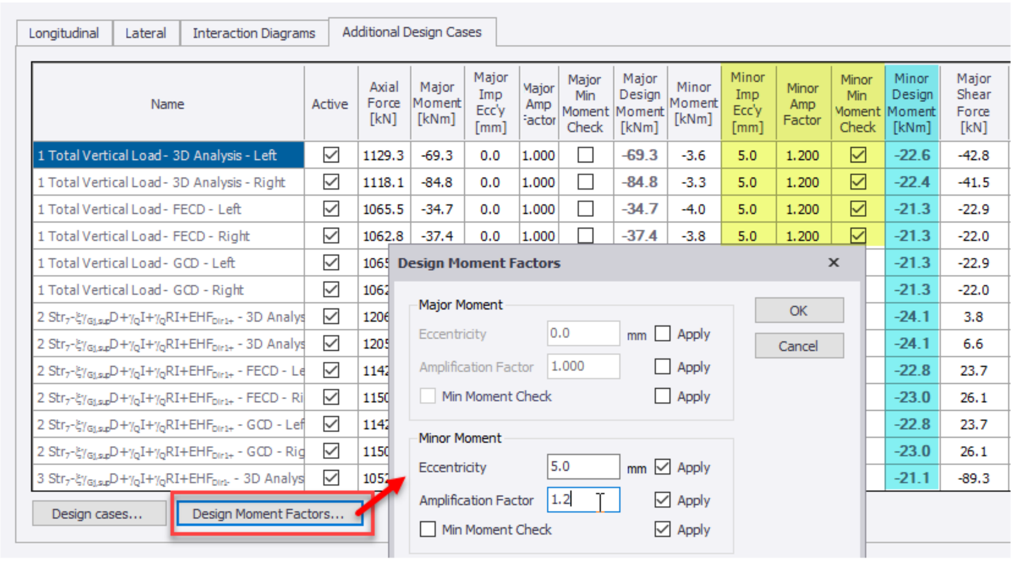
   - Select the required panel in the panel summary
   - Click the Active box for the design case
   - Enter the design forces

3. **Design Moment Factors...** button: Offers three potential "Design Moment" adjustments for each direction:

   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
   - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
   - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension "h" in the direction considered).

   When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

   The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with "Active" option checked on) as shown below:



6. Buttons

Button/option	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Autodesign Panel	Performs an autodesign reselecting bars for the current panel
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog
Copy	Opens a new dialog enabling the selective coping of wall thickness/ concrete grade/reinforcement from the selected panel to other panels in the same wall.

7. Suspend Checking option

By default this option is unselected and checks are made after every edit. For large structures with many combinations interactive checking takes time which can be frustrating if you want to make several changes.

When this option is selected, checking is suspended. (The option also changes to red to highlight that this is the case).

Checks are only carried out and results updated when the option is unselected once more.

See also

Interactive concrete member design (page 1310)

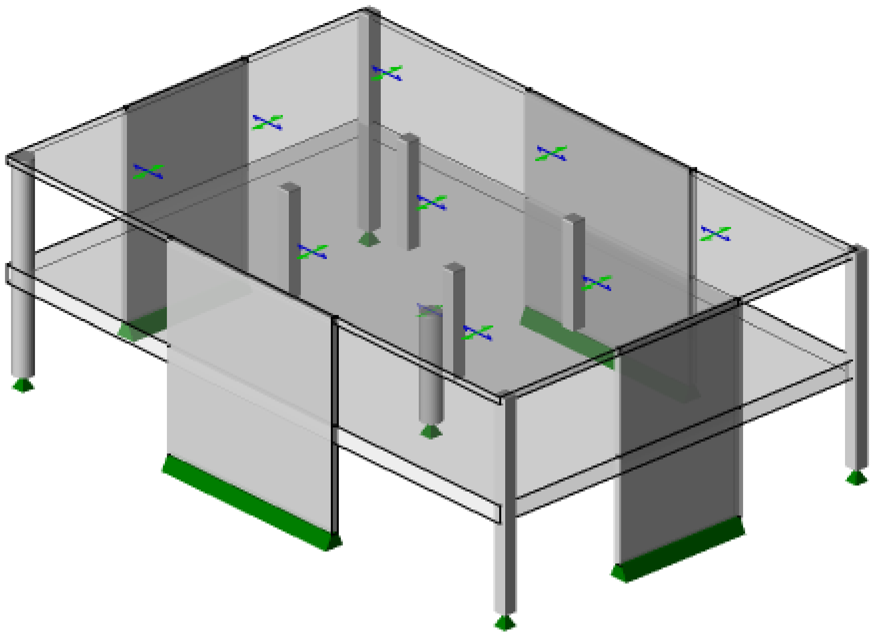
Concrete slab design

To get familiar with the concrete slab design processes. click the following links:

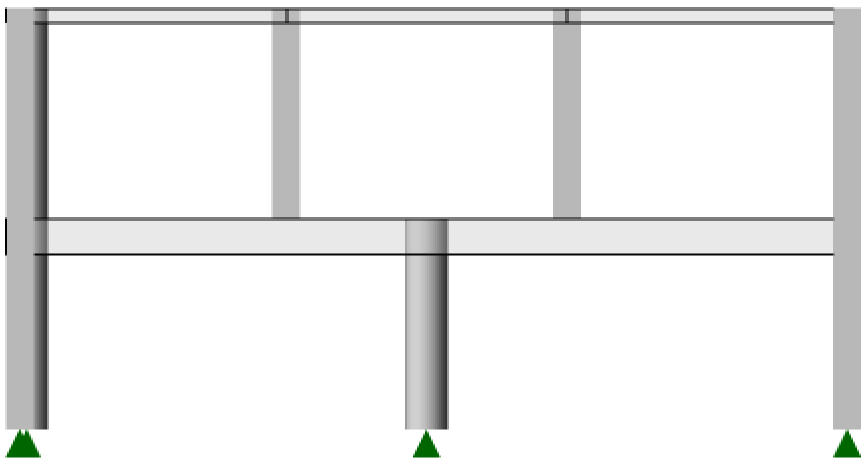
- Flat slab design workflow (page 1355)
- Slab on beams design workflow (page 1367)
- Concrete slab design aspects (page 1373)

Flat slab design workflow

A simple flat slab model as shown below is used in order to demonstrate the techniques involved in the slab design process.

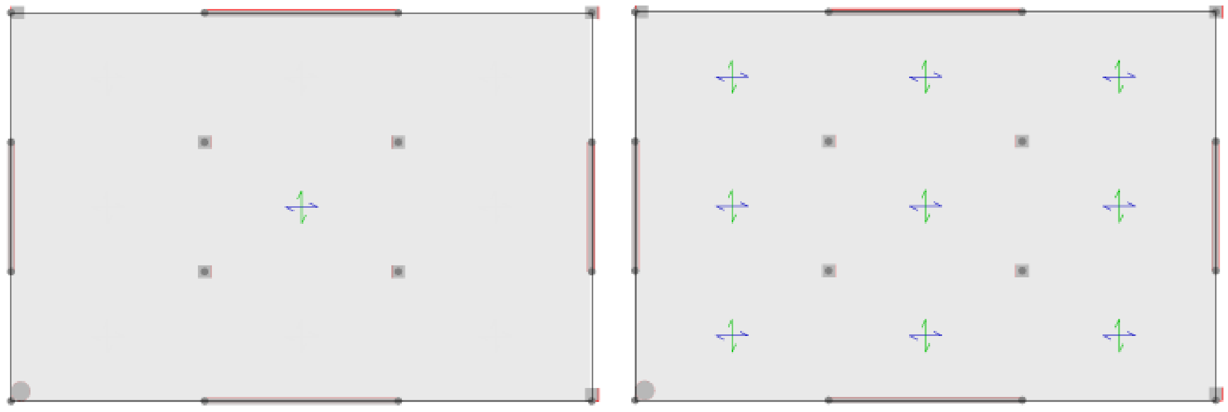


Note that there is a transfer level at the first floor:

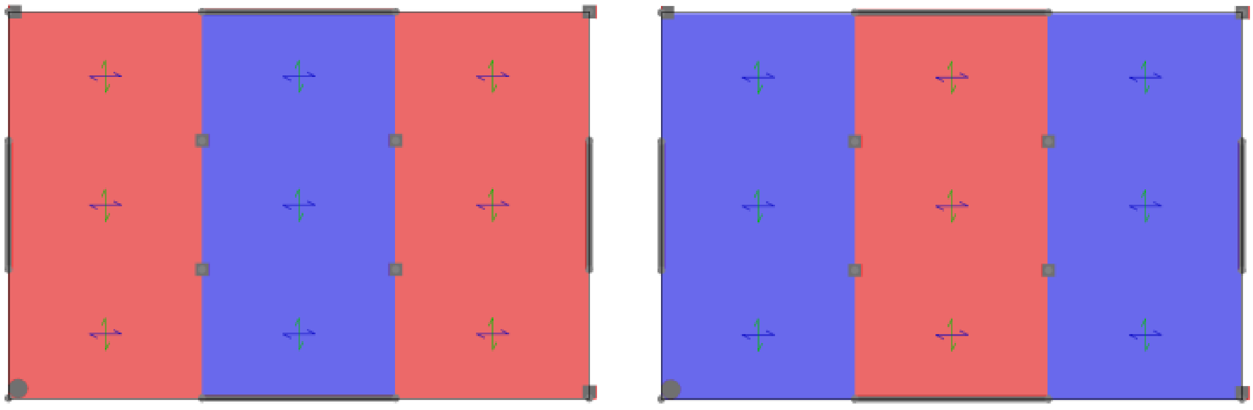


Set up pattern loading

If necessary you should consider manually splitting and joining slab panels to facilitate management of the pattern loading process.



By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Establish slab design moments

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Analyze All**, **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

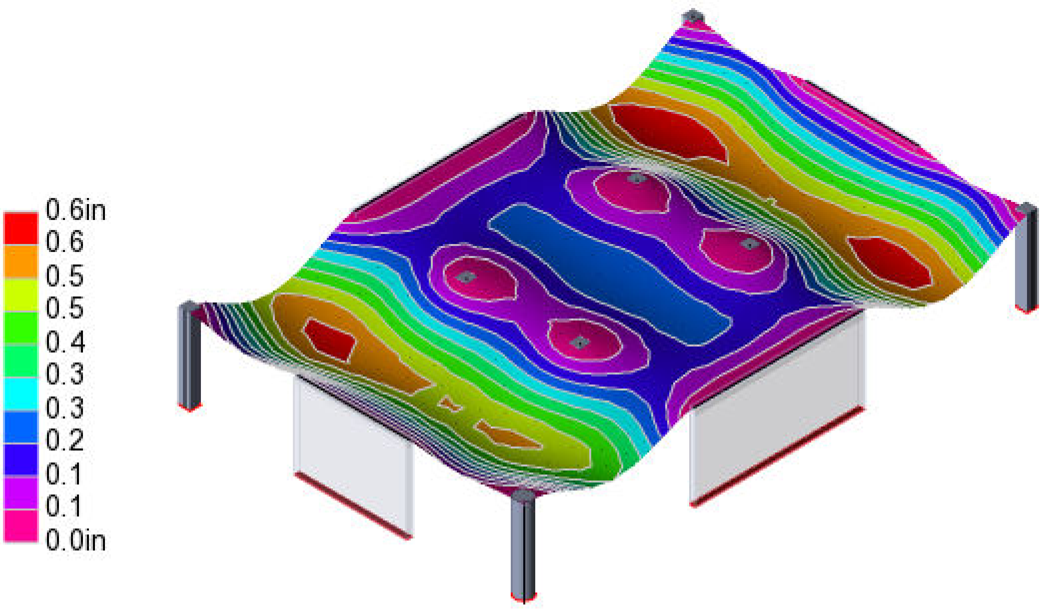
NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Consider simple (linear) deflection (US customary units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.

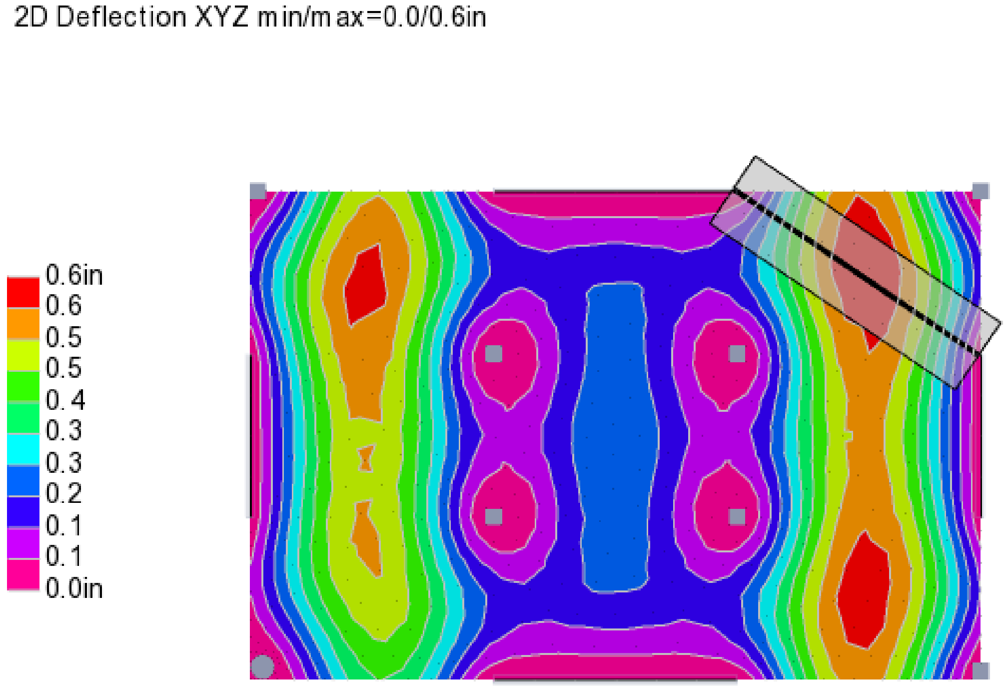
2D Deflection XYZ min/max=0.0/0.6in



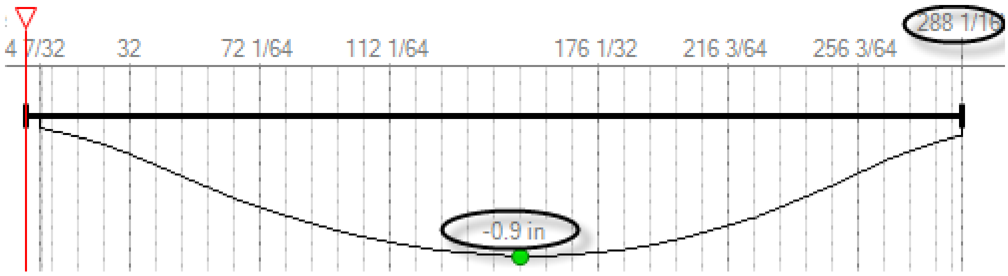
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

If the Head Code you are working to prevents you from running a **Rigorous Deflection Check** you can still use a "Deemed-to-Satisfy" method to assess deflections utilizing slab strips as outlined below:

1. With the level view displayed in 2D, click Create Strip and create a strip between supports.



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



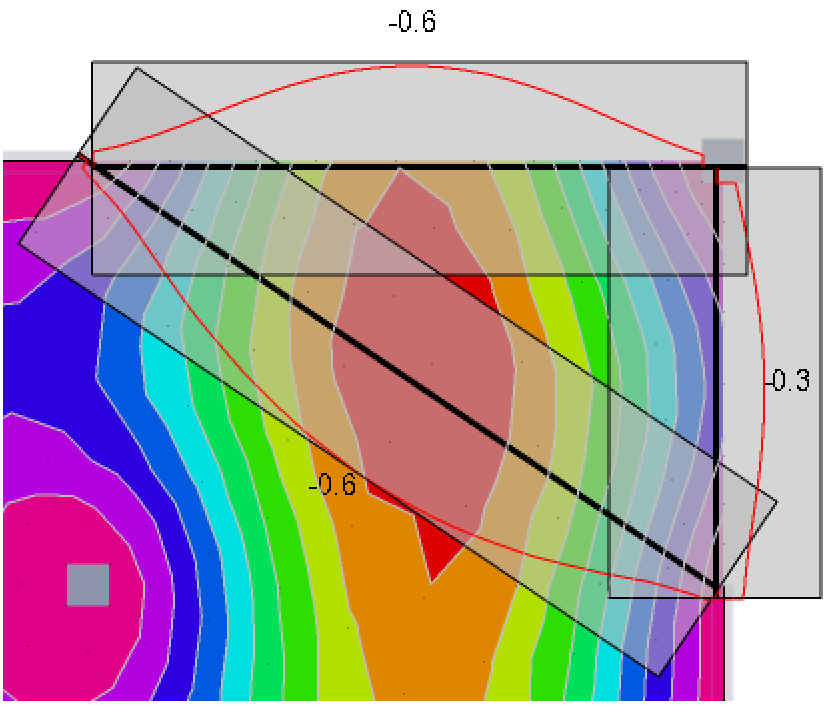
In this example there is a requirement to limit total deflection to span/240

Taking the span as the length of the slab strip: 288in

Deflection limit: 288/240 = 1.2 in

Actual deflection: 0.9 in - the check passes

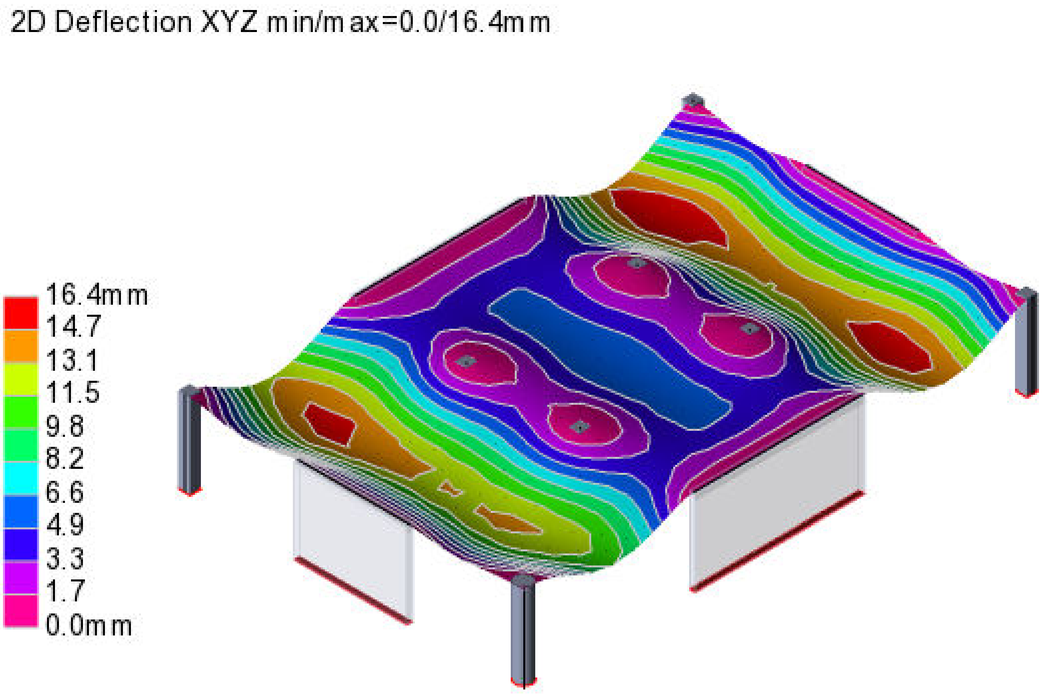
The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Consider simple (linear) deflection (metric units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

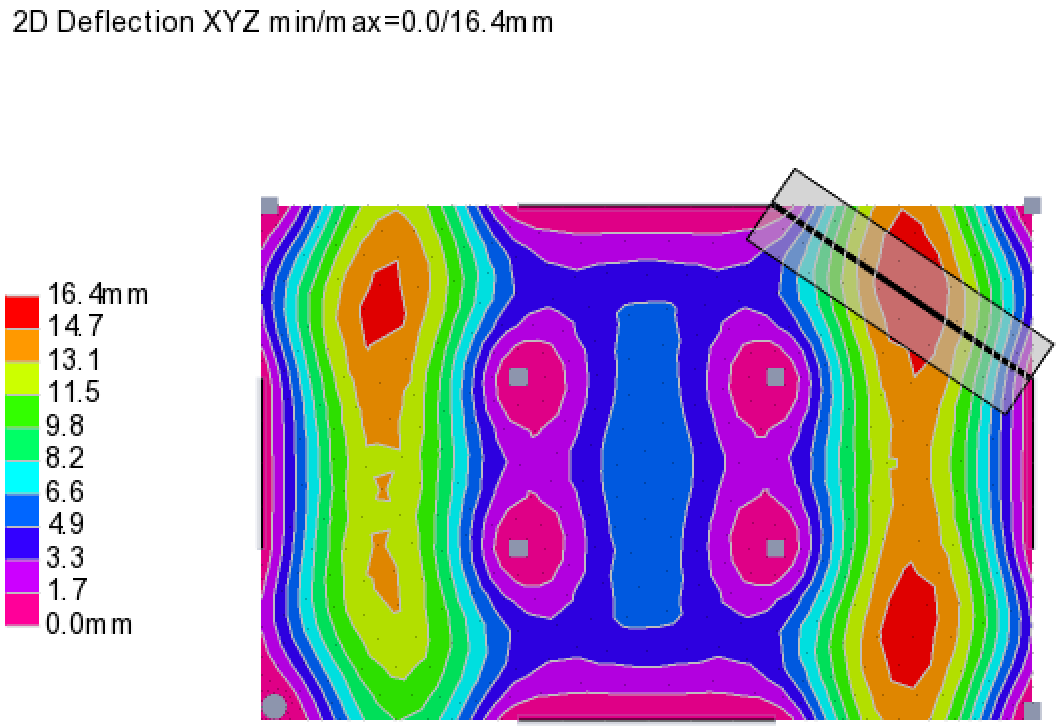
NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.



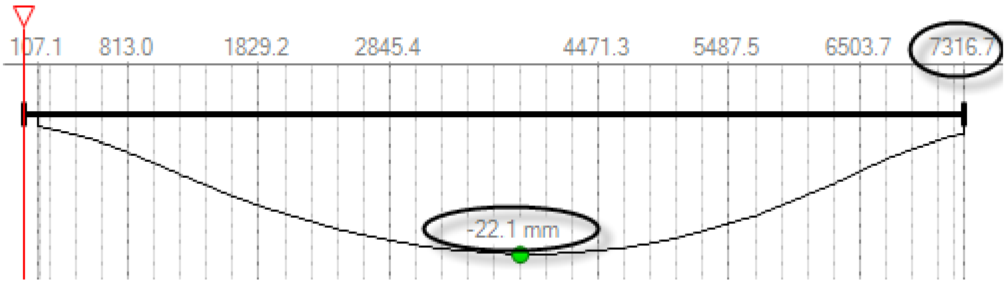
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

If the Head Code you are working to prevents you from running a **Rigorous Deflection Check** you can still use a "Deemed-to-Satisfy" method to assess deflections utilizing slab strips as outlined below:

1. With the level view displayed in 2D, click Create Strip and create a strip between supports.



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



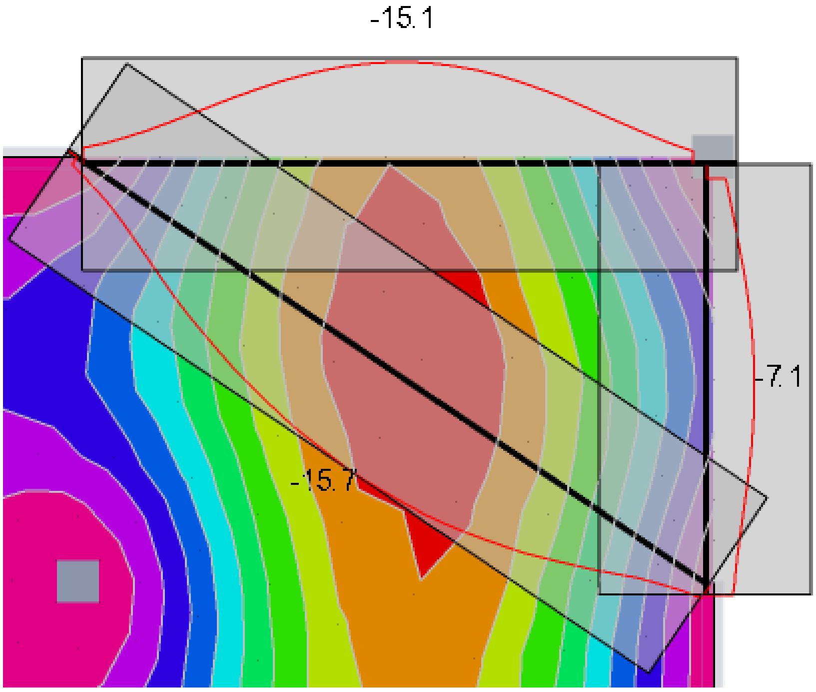
In this example there is a requirement to limit total deflection to span/250

Taking the span as the length of the slab strip: 7317mm

Deflection limit: 7317/250 = 29mm

Actual deflection: 22mm - the check passes

The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Select a Level, or sub-model

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

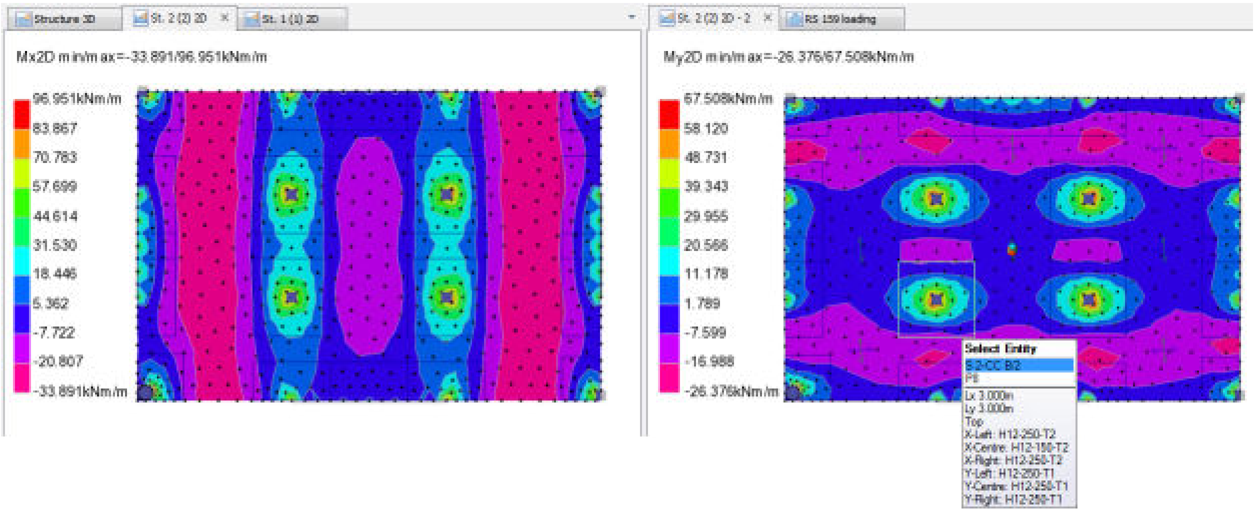
NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mx moments on in one view on the left and My moments in a second view on the right, as below:



By doing this, it is possible to see how patches extend over the peaks.

Typically, at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

In a "slab on beam" situation, you may want to add beam and wall patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

Design panels

NOTE Panel design is dependent on the areas of patches (patch areas which are excluded from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the Design ribbon run Design Slabs in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Slabs or Check Slabs. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

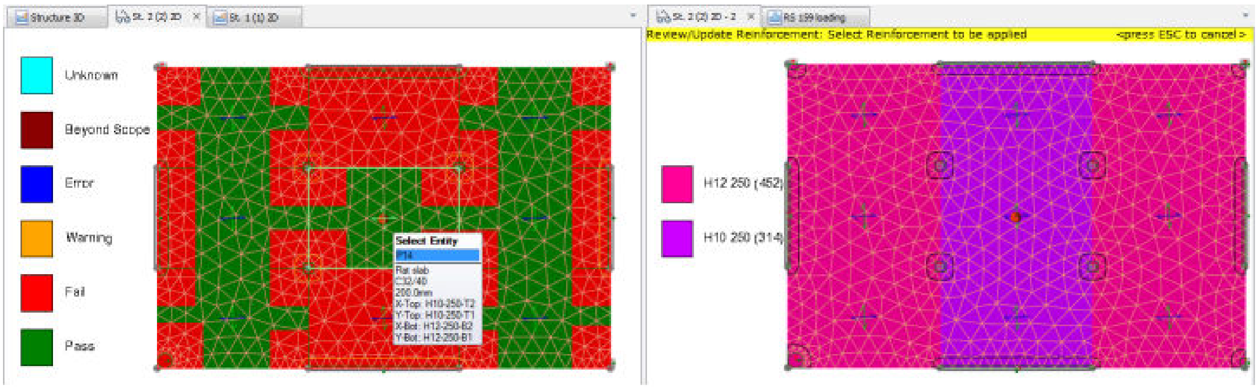
NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Slabs will re-design slabs (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Slabs will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

Once again it is suggested that you use split Review Views to examine the results as indicated below.



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

The view on the left shows Slab Design Status, the view on the right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/ patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a flat slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Design ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

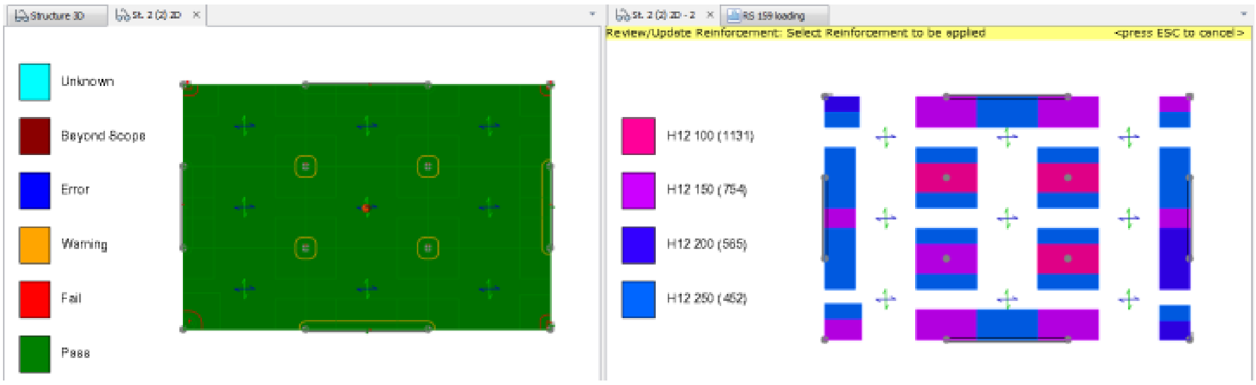
- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Column patches - Is the size reasonable - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the slab reinforcement to be matched (if the slab has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.



Add and run punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added over the entire floor, or structure by windowing it. You can then select any check and review the properties assigned to it. Internal/edge/corner locations are automatically determined (with a user override if you require). Once added click Design Punching Shear and the checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the slab

NOTE In order to see the punching check status in the Review View you might need to first switch off Slab Items and Slab Patches in Scene Content.

Rigorous deflection check

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis. For futher details, see: Calculate slab deflections (page 931)

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

Print calculations

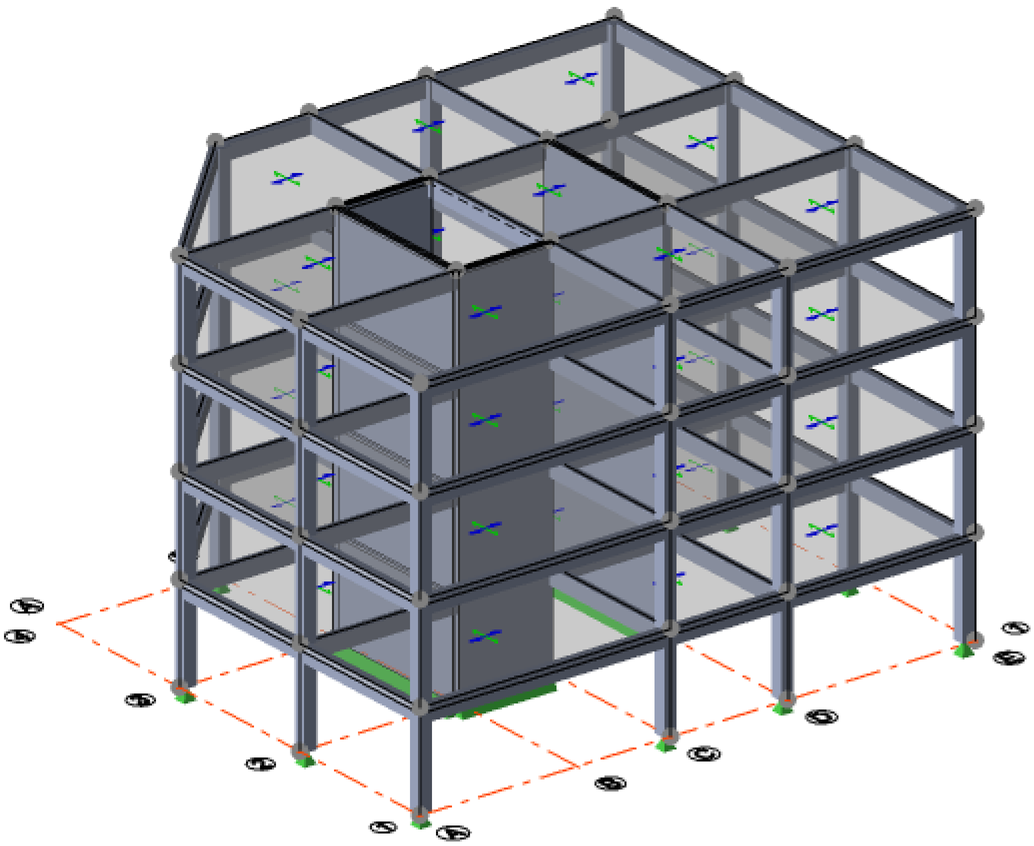
Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

See also

Apply user defined utilization ratios (page 786)

Slab on beams design workflow

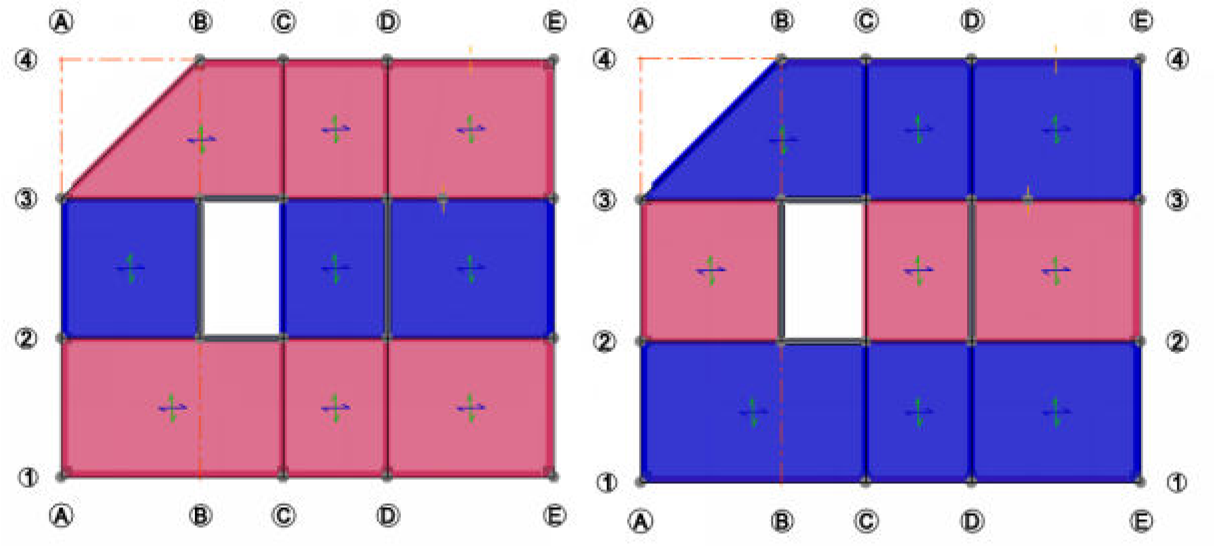
A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.



Note that not all the slab panels are rectangular.

Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Establish slab design moments

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Analyze All**, **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Select a Level

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

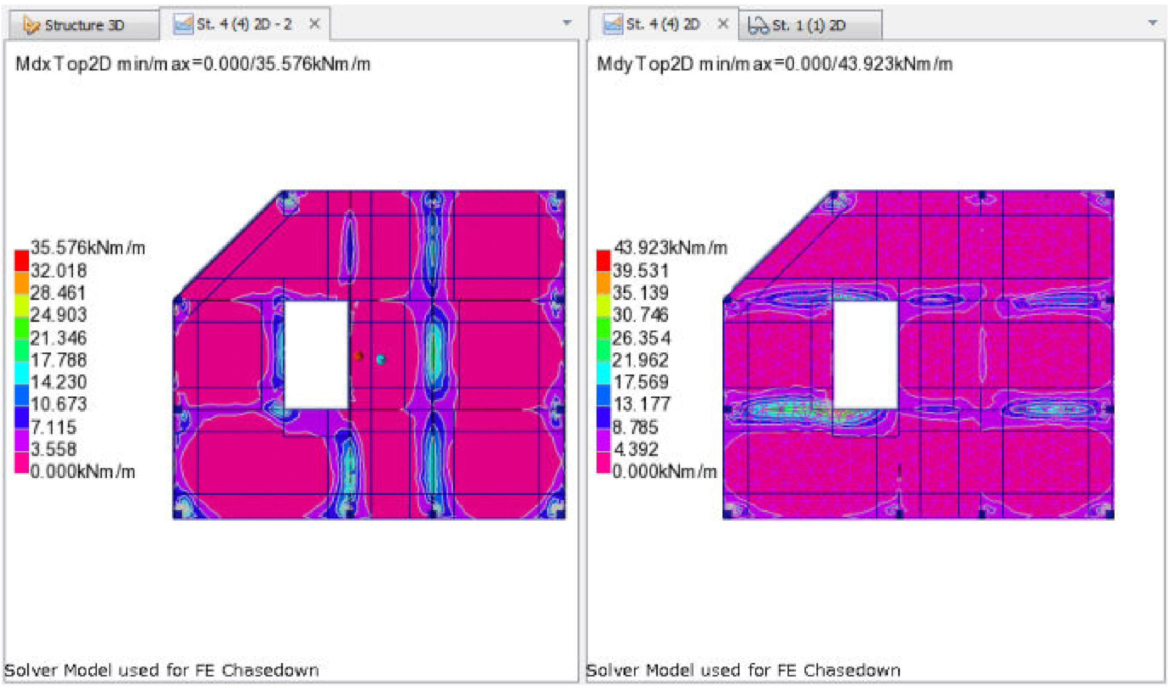
Add beam and wall top patches

You may optionally want to add beam and wall "top surface" patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating beam patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mdx top moments on in one view on the left and Mdy top moments in a second view on the right, as below



By doing this, it is possible to see how patches extend over the moment contours.

It is suggested that at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

Design panels

NOTE Slab on beams panel design takes account of any beam or wall patches (by excluding the patch areas from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the Design ribbon run Design Slabs in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Slabs or Check Slabs. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

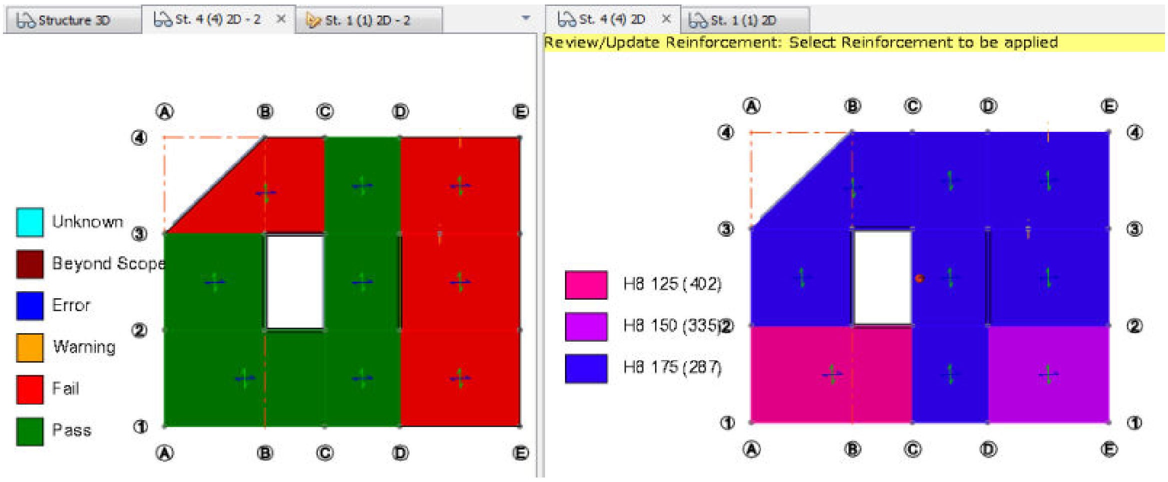
NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Slabs will re-design slabs (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Slabs will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

It is suggested that you use split Review Views to examine the results as indicated below.



The view on left shows Slab Design Status, (with slab patches turned off in Scene Content to assist clarity), the view on right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.
- If the span-effective depth check is failing, review the panel properties to confirm that edge categories are set correctly, (and if the panel shape is irregular consider whether it is being suitably idealized as a rectangular panel). See: Slab on beam idealized panels (page 1378)

Design beam and wall patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design any beam or wall patches that you may have defined.

To do this, either:

- From the Design ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

See also

Apply user defined utilization ratios (page 786)

Concrete slab design aspects

Concrete type

While you can apply both normal and lightweight (LW) concrete in the slab properties, slab design using lightweight concrete is only available for Eurocodes.

When using other Head Codes slabs can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

How the load decomposition method affects slab design

Only slabs that have been specified with two-way load decomposition are designed.

One-way load decomposition in Tekla Structural Designer is a simple procedure that does not determine slab design forces. When a slab's decomposition is set as one-way it is assumed that it is some form of precast slab (presumably designed by safe load tables).

- A flat slab panel always uses two-way load decomposition.
- A slab on beams panel can either be specified to use one-way or two-way decomposition - however if it is specified as one-way it cannot then be designed in Tekla Structural Designer.

It should be noted that any in-situ slab is capable of two-way decomposition:

- When a slab is set as two-way it will only effectively span in 2 directions if its proportions and support conditions mean that there will be a two-way effect.
- For example - If a slab that has a span of 6 units in one direction and 50 units in the other is set to two-way decomposition, then although it is two-way the FE analysis will still inherently take the load one-way.

Two-way spanning slab panel design moments

When a slab panel is specified with two-way decomposition, a general FE based approach is used to determine the design moments. If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis results are also considered:

- The worst design moment (per unit width) is found in each direction of the slab - if the design moment is zero in one of the directions then the analysis has shown that the slab is effectively spanning one-way and the supplied reinforcement in this secondary direction will be selected to suit the minimum requirements of secondary reinforcement.
- Note that this FE based approach inherently caters for point loads, line loads, openings, etc and for the possibility of variable adjacent span lengths in a continuous "1-way" slab (and of course it can still be applied to the simple case of a "1-way" slab with a uniform UDL applied and uniform span lengths).

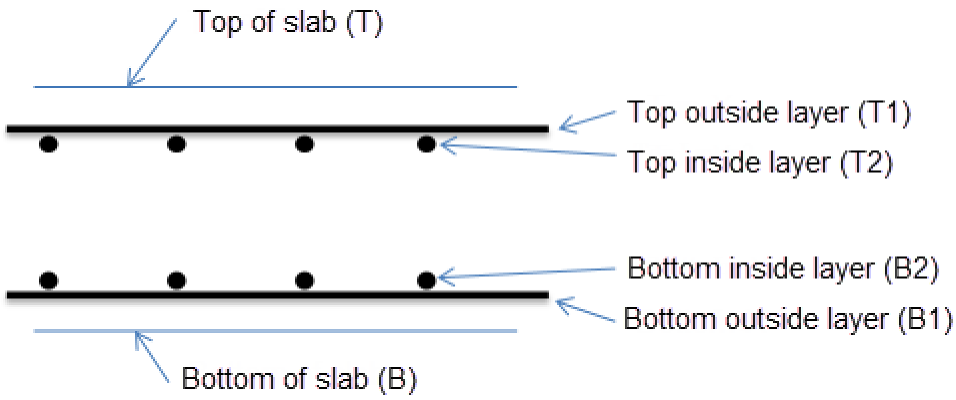
Alignment of slab panel reinforcement

X direction reinforcement is aligned to the panel span, which is determined by the slab's **rotation angle** property. Y direction reinforcement is always perpendicular to the X direction reinforcement.

For further details, see: Rotation angle for panels (page 1380)

Slab panel reinforcement

In a slab you have two surfaces: Top (T) and Bottom (B).



In each surface you have two layers of steel in orthogonal directions - X direction steel and Y direction steel. Layer 1 is the outside layer - the one closest to the surface. Layer 2 is the inside layer. Which direction is the outer layer is controlled by this slab panel setting: Outside layer in X direction.

Any of the four layers can be set to "none" if required.

Slab patch reinforcement

Additional rectangular reinforcement patches can be applied to slab panels:

- column patch - at column stack heads
- beam patch - along beams
- wall patch - along walls
- panel patch - at the panel centre
    - typically positioned centrally - but not restricted to this location and also not restricted to existing purely within one panel
    - can also be positioned under loads

These patches are either in the top or the bottom of the slab and may or may not have reinforcement defined in them. If no reinforcement is defined then the background reinforcement is used. If patch reinforcement is defined then it will either be used on its own, or if you select the "Combine with Panel Reinforcement" option, the sum of the background + patch reinforcement will be used. Note that this option is only selectable when the "Align to Panel Reinforcement" option is also selected since combining in this way would only be valid provided the reinforcement is reasonably aligned. Choosing the reinforcement to be combined also forces the "Cover as Panel" option to be selected as the program assumes the reinforcement to be in a single layer. If the reinforcement is not combined you can specify the cover to the patch reinforcement by turning off the "Cover as Panel" option.

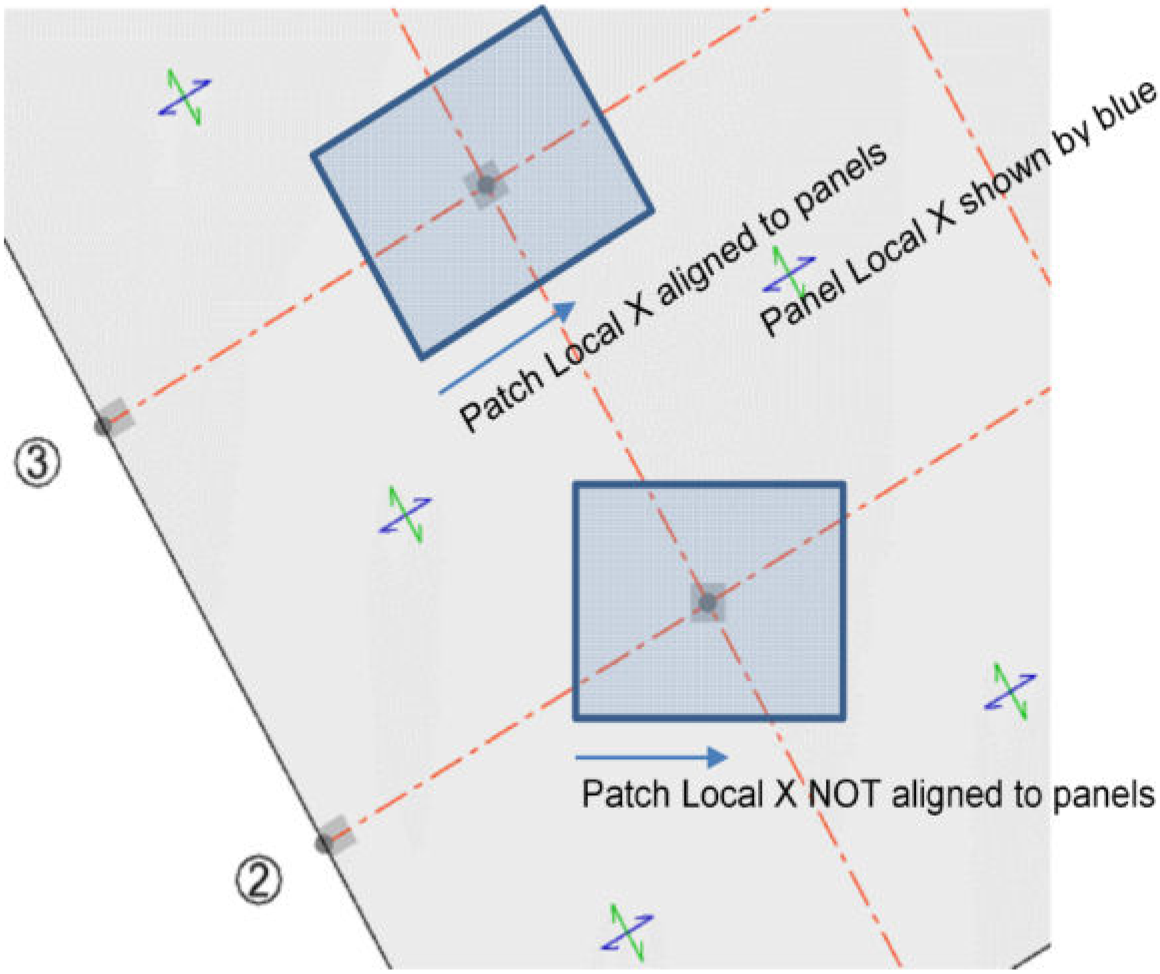
Note that patches may overlap on the plan view, and there is no restriction on this, even patches relating to the same layer of reinforcement are allowed to overlap. This situation is handled conservatively during design by simply ignoring the overlap.

Each patch manages reinforcement and the reinforcement design using a number of slab design strips. Some key points to bear in mind considering patch design are:

- A patch only manages data in the top or the bottom layers of a slab, not both.
- There can be up to 3 design strips in each direction of a patch
- There is no requirement to have design strips in both directions - there can be one design strip in one direction and none in the other.
- Within any strip there might be patch reinforcement to consider but note that:
    - The underlying panel reinforcement can be none
    - The added patch reinforcement can be none.
- If there is patch reinforcement to consider this can be considered instead of, or in addition to, relevant slab reinforcement

Slab patch strip design

For the strip designs within each patch it is necessary to establish which bar layer is to be designed and work out if and how the patch reinforcement combines with the panel reinforcement.



As shown above there are 2 distinct options:

1. Patch aligned to panel
2. Patch NOT aligned to panel:

In both cases the reinforcement that is determined for use in the design checks is some combination of the panel and patch reinforcement. Expanding upon this the cases considered are:

Patch aligned to panel:

1. Patch reinforcement type = NONE then the panel reinforcement is used for all layers
2. Combine with Panel Reinforcement = No Then the patch reinforcement is used in the patch surface and the panel reinforcement is used in the opposite surface.

3. Combine with Panel Reinforcement = YES Then the patch reinforcement is combined with panel reinforcement and used in the patch surface and the panel reinforcement is used in the opposite surface.

Patch NOT aligned to panel:

Patch reinforcement is used in the surface to which the patch applies. Reinforcement in the other surface is taken from the panel using the most aligned possibility.

Patches to both surfaces

As stated above, patch reinforcement is only ever used in the surface to which a patch applies, reinforcement in the other surface is taken from the panel.

In rare situations you may have separate patches on both surfaces; in which case you would want the patch reinforcement from the top patch to be considered on the top surface and patch reinforcement from the bottom patch to be considered on the bottom surface.

In this specific situation, for both patches the other surface doesn't necessarily need to be designed to only consider reinforcement in the panel; you can avoid this by selecting the patch property **Consider patch surface moments only**.

Span effective depth checks for irregular shaped panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

For further details, see: Slab on beam idealized panels (page 1378)

Slab on beam idealized panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

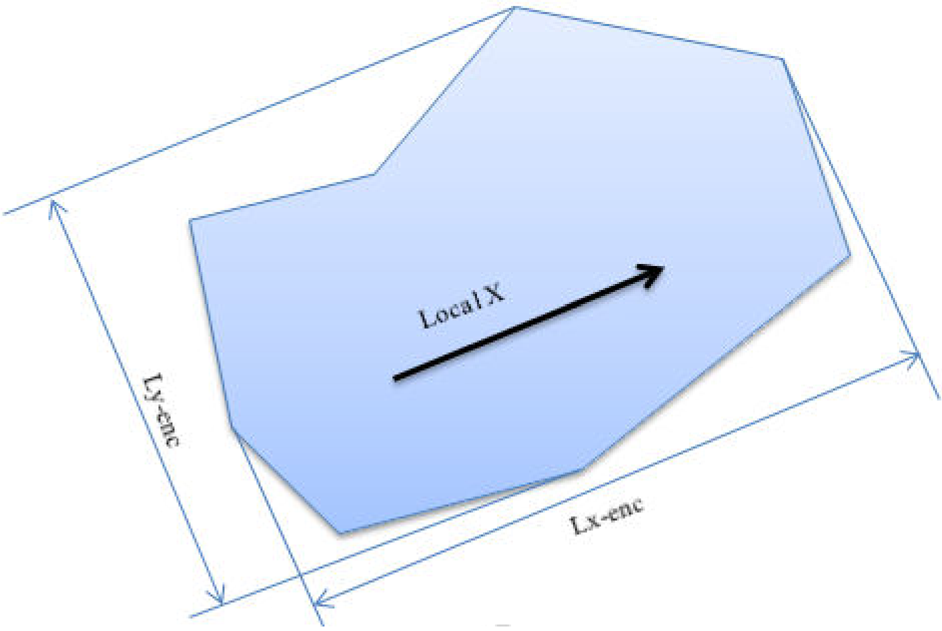
To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel rotation angle):

Lx-enc = maximum overall length of the panel measured parallel to local X

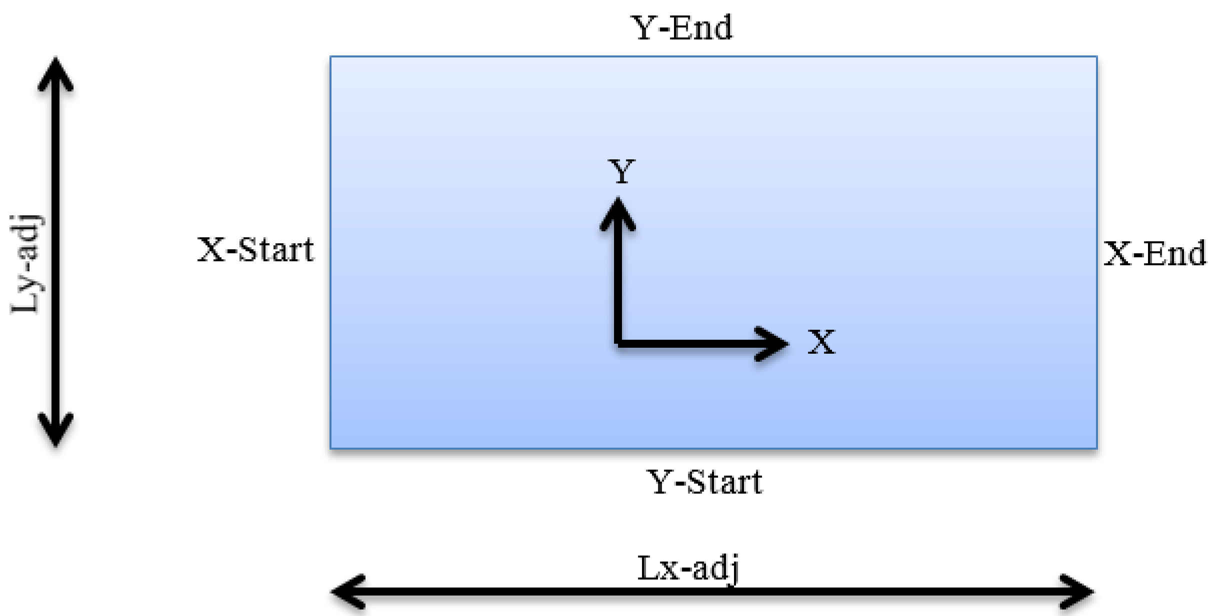
Ly-enc = maximum overall length of the panel measured perpendicular to local X

A user specified adjustment ratio is then applied to these lengths to determine the adjusted lengths. Conservatively the adjustment ratio defaults to 1.0 in both directions.

In situations where the panel does not have 4 sides, (such as the one shown below), some engineering judgement might be required when deciding on appropriate values of the adjustment ratios in each direction.



The resulting idealized panel with dimensions in X and Y is illustrated below:.



Edge Category

For the span-effective depth check, the edge categories in each direction have to be manually assigned to the idealized slab panel. The three edge categories being:

- Unsupported
- Continuous Support
- Dis-continuous Support (default)

Rotation angle for panels

The rotation angle in the panel properties is used to control the orientation of the panel span. For reinforced slabs the X direction reinforcement is aligned to the span.

Panels defined by a level

When the rotation angle = 0

- Span direction aligns with global X

When the rotation angle = 90

- Span direction aligns with global Y

Panels defined by a frame

When the rotation angle = 0

- Span direction is horizontal

When the rotation angle = 90

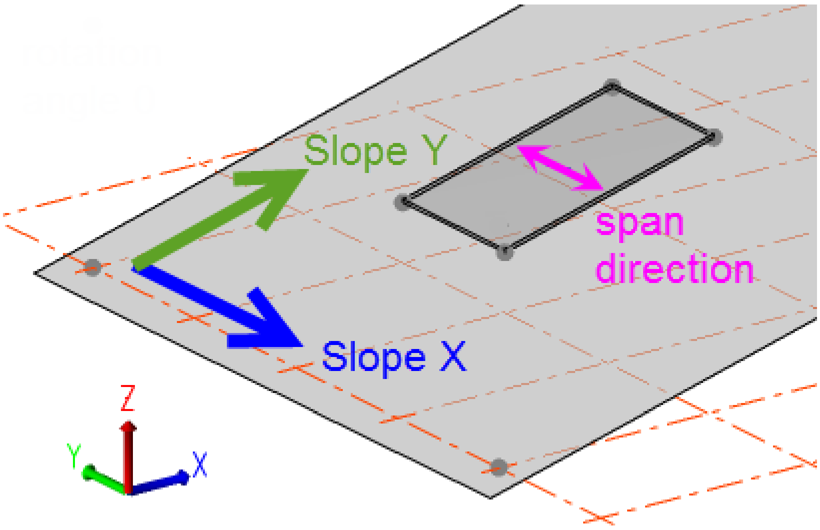
- Span direction is vertical

Panels defined by a sloped plane

NOTE In a sloped plane positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

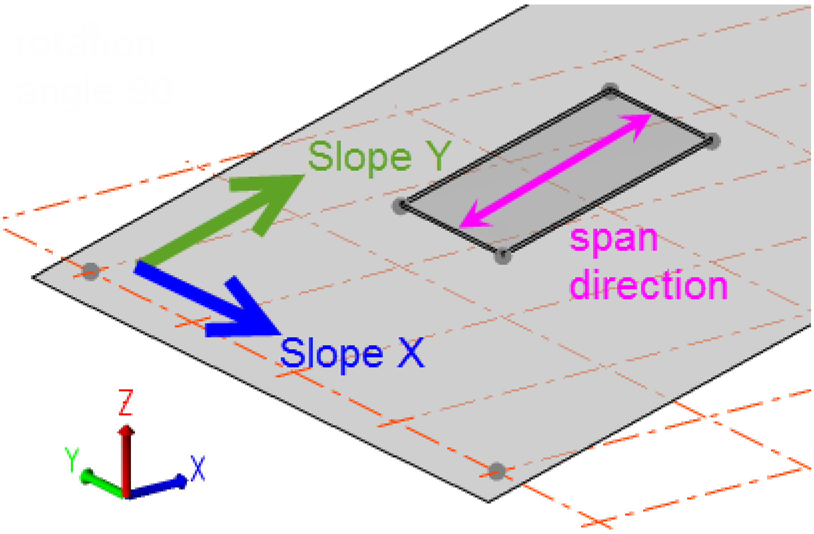
When the rotation angle = 0

- Span direction aligns with X direction of the slope (i.e. perpendicular to the slope, as shown below)



When the rotation angle = 90

- Span direction aligns with Y direction of slope (i.e. parallel to the slope, as shown below)



13.7 Slab deflection handbook

This handbook provides an overview of how iterative cracked section analysis is applied in Tekla Structural Designer for the purpose of obtaining a better estimate of slab deflections.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

Related video

Rigorous Slab Deflection (Eurocode)

Rigorous Slab Deflection Design (ACI)

Slab deflection methods

Concrete is considered a durable and economic material for floors systems. However, reinforced concrete slabs deflect. The magnitude of the deflection is more complicated for concrete as deflection increases with time. It’s long term behavior is characterized by cracking caused by flexure, shrinkage and creep. If this is not taken into consideration by allowing adequate tolerances to glass facades and internal partitions for example then problems can arise.

Tekla Structural Designer provides two alternative methods for checking deflections. Either:

- Deemed-to-satisfy checks, or
- A rigorous theoretical deflection estimation using iterative cracked section analysis

Deemed-to-Satisfy Checks

A couple of deemed-to-satisfy checks are presented here.

1. The use of a limiting span-to-depth ratio (L/d) method. This method is assumed to 'be adequate for avoiding deflection problems in normal circumstances'. It can only be considered as a rough deflection estimate and is not intended to predict how much a member will actually deflect. Total deflection is expected to be < span / 250 (Eurocode) or < span / 250 (US) and deflection affecting sensitive finishes is expected to be < span / 500 (Eurocode) or < span / 480 (US)
2. A linear analysis with adjusted analysis properties.

It is important to appreciate that these deemed-to-satisfy methods do not predict actual deflections even though the linear analysis method provides a total deflection that can be checked against the span / 250 (Eurocode) or < span / 250 (US) limit mentioned above.

When normal deflection limits do not apply, for example, due to stricter usage limits, glazed cladding systems or where a faster pace of construction is applied then the 'deemed-to-satisfy' checks are no longer applicable - the alternative is a rigorous deflection estimation which is the primary topic for this guide.

See also

- Deemed to satisfy slab deflection checks example (Eurocode) (page 1427)
- Deemed to satisfy slab deflection checks example (ACI) (page 1463)

Rigorous theoretical deflection estimation

The rigorous theoretical deflection assessment takes into account cracking, creep and shrinkage over time.

In the UK, rigorous deflection estimation is taken to mean deflection estimation in accordance with the Concrete Society Technical Report 58.

The principle of assessing deflections rigorously involves assessing the curvatures induced by both load and shrinkage, adding them together and then the total curvature is translated into a deflection.

The Technical Report discusses the importance of construction events. Total deflection at the end of every event comprises:

- An instantaneous deflection which is influenced by the extent of cracking
- An additional accumulated creep deflection
- An additional accumulated shrinkage deflection.

Once these totals are known, differential deflections between any two events can be calculated.

The Technical Report gives detailed guidance on some very complex looking calculations - it all seems very "rigorous". However, we must not lose sight that the material - Concrete is a very variable material. Furthermore, how accurately can we really predict input parameters such as event loads and timings? The report advises that deflection accuracy can only be considered an estimate in the range +15 to -30%.

In the US, the basic approach described in ACI 318 has a similar approach to cracking, interpolating between the fully cracked and the uncracked states, (although it doesn't recognize the reduction in tension stiffening). For creep and shrinkage, in ACI 318 there is a single multiplier for the deflection calculated from the cracked flexural rigidity. There are additionally two ACI Committees 435 and 209 which go into more detail about creep and shrinkage calculations. The US user can therefore either adopt the basic ACI 318 approach, or, take on board the ACI Committee 435 & 209 guidance. In special situations the TR58 approach could even be considered.

Expectations - It is incorrect to think that rigorous methods will provide greater economy. i.e. by allowing the engineer to reduce slab thickness or the quantity of reinforcement. The end result is greatly influenced by the various input parameters which each can impact on the deflection.

See also

- Rigorous slab deflection workflow (page 1384)
- Rigorous slab deflection analysis example (Eurocode) (page 1430)
- Rigorous slab deflection analysis examples (ACI) (page 1465)

Rigorous slab deflection workflow

You perform rigorous slab deflection analysis in Tekla Structural Designer from the **Slab Deflection** toolbar.

The suggested workflow is as follows:

1. Define the slab loading event sequences (page 1393).
2. Run a slab deflection analysis (page 936) and review deflections for the final load event.

   You can analyze the current or selected level (sub-model) or choose to analyze every slab in the model. Obviously your choice has an effect on the time necessary to undertake the iterative slab deflection analysis.
3. Define the slab deflection checks via the **Slab Deflection Check Catalogue** and then place Check lines (page 1421) as required.
4. Graphically review the Slab deflection status and utilization (page 1424).
5. Make adjustments as necessary until the status passes. For more information, see Slab deflection optimization (page 942).

NOTE A reasonable level of slab reinforcement should be established prior to running the Slab Deflection Analysis as this will have a significant impact on the deflection calculations.

Factors that affect rigorous slab deflection estimates

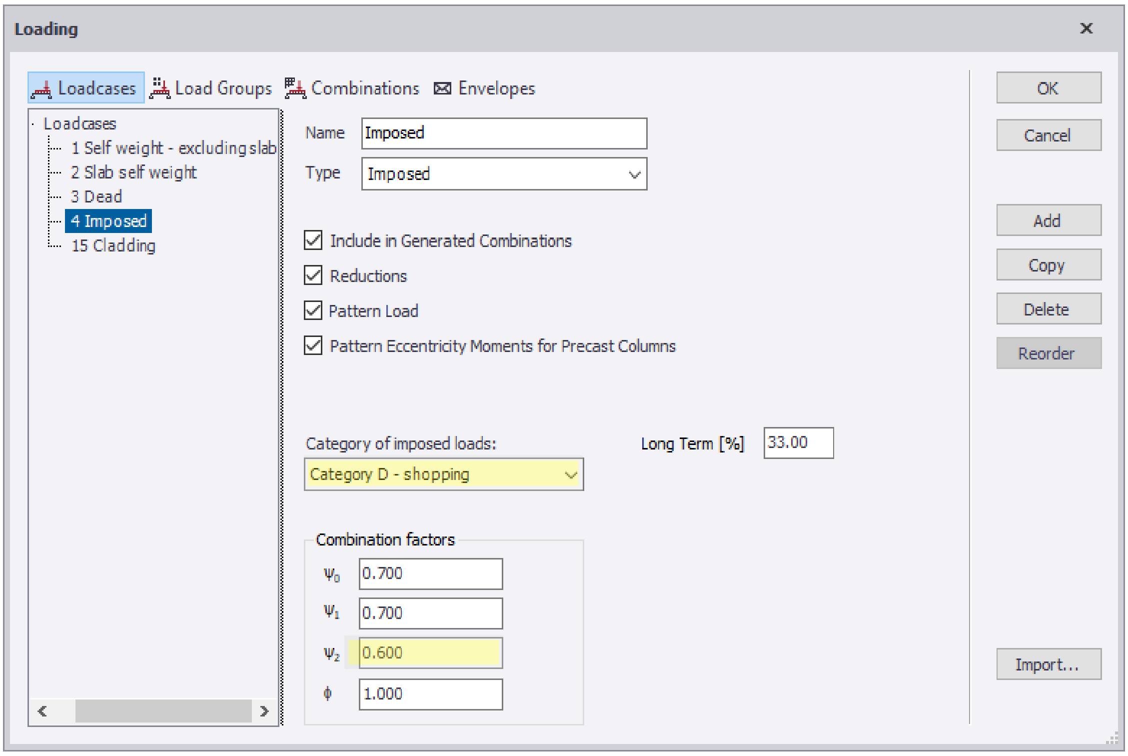
Many input parameters have an impact on the rigorous deflection estimates. These include:

- Level of Restraint
- Concrete properties
- Stiffness adjustments
- Allowance for shrinkage effects
- Event sequence parameters
- and even the assumed analysis properties of connected columns and walls

Some of the key factors are described below, these vary depend on the head code being worked to.

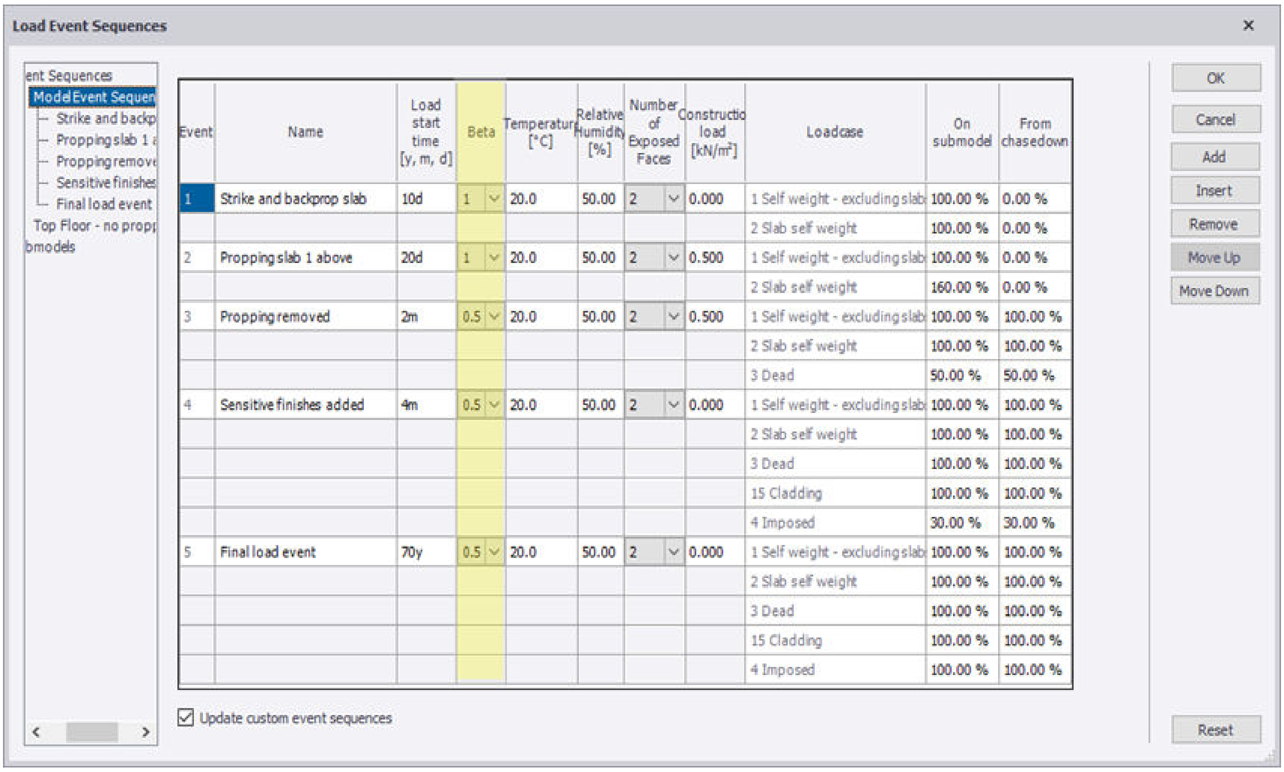
Quasi-permanent load factors (EC2)

An accurate prediction of deflection requires a realistic assessment of the loading. To the Eurocodes long term loads are termed quasi-permanent and a ψ2 factor is applied to the imposed load which varies based on the use of the structure. These are dealt with when defining the imposed load case.



Beta coefficient (EC2)

This property is set in the Event Sequence dialog.



Beta relates to the duration of the load and tension stiffening effects.

NOTE Tension stiffening occurs when the concrete is not fully cracked - because there is still concrete in the tension zone that transfers some tensile forces, the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness).

Due to phenomena such as increased cracking or local bond failure, tension stiffening effects reduce over time - Beta is used to account for these, (see EC2 Clause 7.4.3 and TR58).

- For loads with a short duration, and for cyclic loads, Beta should be set to 1.0.
- For loads with a long duration, Beta should be set to 0.5.

NOTE In Tekla Structural Designer, Beta defaults to 1.0 where the start event time is <= 30 days and 0.5 if >30 days, but may be changed for any event.

Since these phenomena are irreversible, it is not recommended that Events have a value of Beta greater than that set for any previous Event and a warning will be issued if you do this.

However, there are circumstances where you may wish to have a value of Beta=0.5 in the earlier events, and Beta=1.0 in a later Event. For example, TR58 suggests that Beta=1.0 be used for the variable part of imposed load (if you wish to consider that at all).

In this case the analysis will permit you to enter these values. However, because the reduction in tension stiffening is carried forward and is irreversible in the analysis, caution is advised.

To explain this issue more fully, the impact of choosing Beta=1.0 at a later event, for different extents of cracking, is explored below.

Consider 3 possible configurations of a model, where:

1. There are many elements that are uncracked at the end of earlier events. For a later event where the duration is short (which implies you should set Beta=1), setting Beta=0.5 could lead to an overestimation of incremental cracking, and subsequently overestimation of deflection.
2. There are many elements that are cracked in an earlier event, for which you have chosen Beta=0.5. For a later event where the duration is short (which implies you should set Beta=1), if cracking increases in that Event, setting Beta=1.0 could lead to an underestimate of incremental cracking, and subsequently an underestimate of deflection for this later event.
3. The majority of cracking occurs in an earlier event, for which Beta=0.5, if cracking does not increase in a later Event, the value of Beta in that later Event will have minimal impact on deflection. To ascertain whether the cracking increases, you may run separate analyses, with values of Beta=0.5 & 1.0, and compare the results.

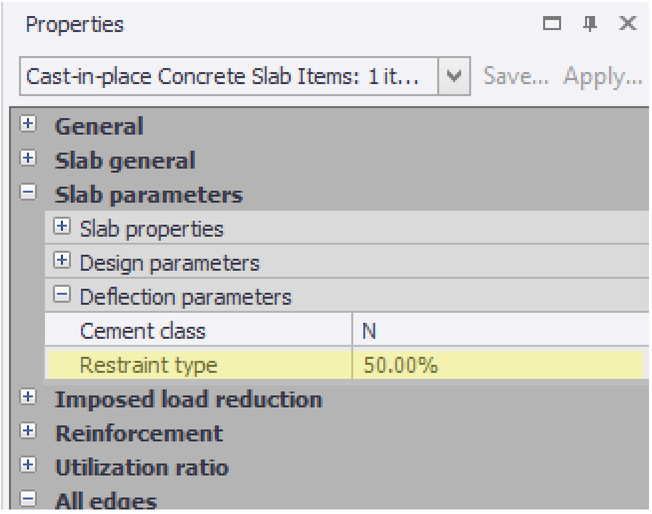
In summary, it is prudent to consider whether cracking is likely to increase in later events for which you wish to set Beta=1.0.

However, it is likely that you will observe minimal difference between the total deflection estimates, and that other assumed values will be of much greater significance.

It is straightforward to run the analysis with different values of Beta to determine whether this is the case.

Restraint type (EC2)

This property is located under the Deflection parameters heading in the slab item Properties Window.



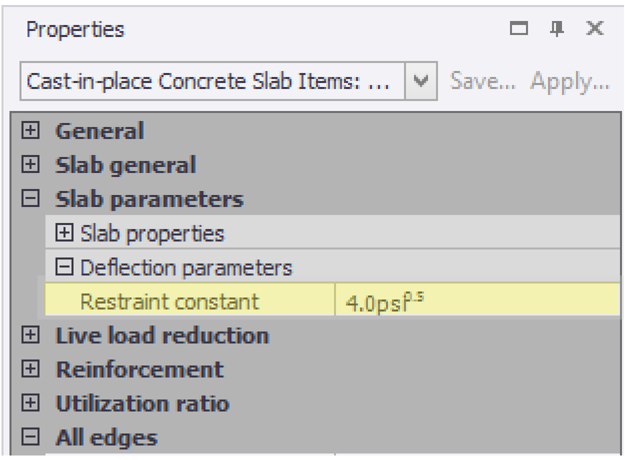
NOTE The Restraint type is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

The value to adopt is a matter of judgment - for guidance on this refer to EC2 cl 7.4.3(4) and TR58

Changing the value will affect tensile strength (f_{ct}) and hence cracking moment.

Restraint constant (ACI)

This property is located under the Deflection parameters heading in the slab item Properties Window.



NOTE The Restraint constant is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

For US customary units, restraint constant values from ACI 435 are:

- For situations with significant restraint - 4.0
- For insignificant restraint - 7.5

The default is conservatively set to 4.0.

The allowable input range is between 1 and 10 and is a user specified value by the engineer.

The restraint constant is used to determine the Modulus of Rupture, f_r = “restraint constant” x λ x √(f_c’)

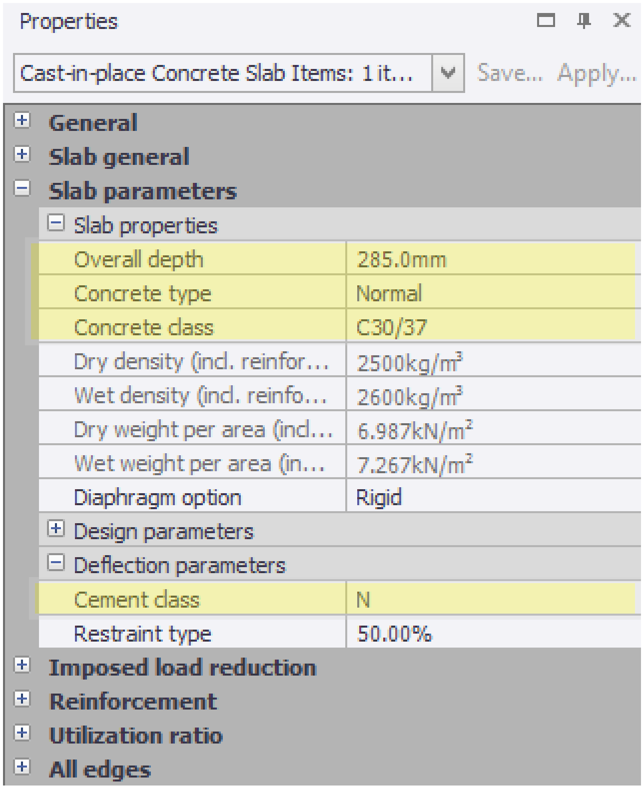
Where:

f_c’ = specified compressive strength (psi)

λ = 1.0 (for normal weight concrete)

Concrete Properties (Eurocode)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent. The Cement class can also affect deflection.



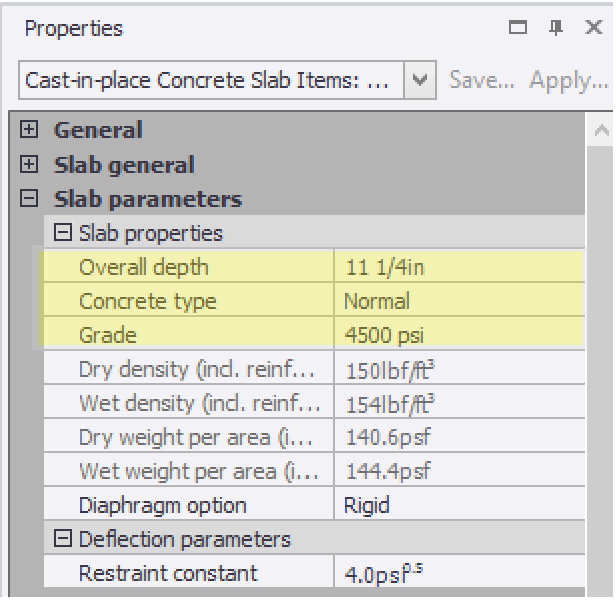
The elastic deformation of concrete largely depends on its composition (especially the aggregates). For C35/45 concrete, Eurocode 1992-1-1:2004 Table 3.1 provides a modulus of elasticity, E of 34GPa (34000 N/mm^2) as a mid range value using quartzite aggregate. For different aggregates this can range between -30% and +20%.

What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Concrete Properties (ACI)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent.



What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Stiffness Adjustments

The relative stiffness of the interconnecting elements will play a role in the force distribution and hence the deflection results.

For existing models the values can be reviewed and amended by clicking **Settings > Modification Factors** on the Slab Deflection ribbon. These are user defined values with assumed defaults.

As mentioned in the above **Concrete Properties (Eurocode)** and **Concrete Properties (ACI)** sections, an alternative to assessing the impact of a different grade could be to alter the Modification Factors specified for flat slab E and G values. i.e. Concrete material property of slab = 34000 N/mm2. What impact would using a value of 32000 N/mm^2 make? Flat slab stiffness adjustment = 32000/34000 = 0.9412

Shrinkage

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

Shrinkage is taken into consideration by making an overall adjustment to the total deflection. This approach is in line with many other software products.

The adjustment is specified in the **Slab Deflection Settings** dialog.

We recommend an allowance with an upper limit of 30%. The default is set as 0.25 (25%).

Event sequences

An Event sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

By default a model event sequence is initially applied to all sub models in the structure; if required custom event sequences can also be defined to override the model event sequence for specific sub models.

NOTE Event sequences are NOT structure event sequences, i.e. they do not describe all the events starting from the first day of the overall construction.

Click the links below to find out more:

- Construction stage events (page 1393)
- A typical model event sequence (page 1394)
- Custom event sequences (page 1402)
- Understanding event sequence deflections (page 1405)

See also

Work with event sequences (page 932)

Construction stage events

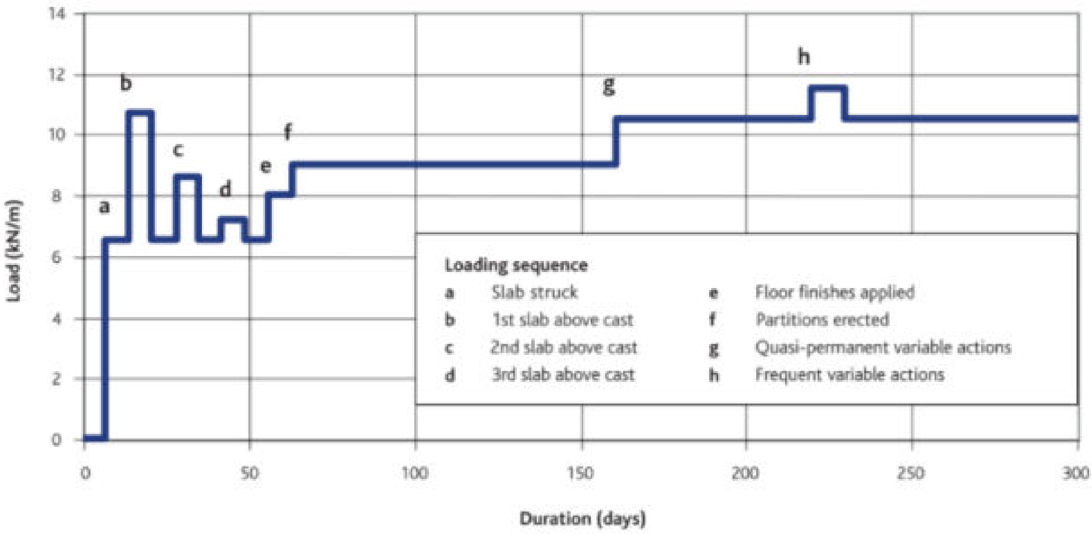
An Event Sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

Deflection is very dependant on the extent of cracking. Once the slab has cracked then it is assumed to remain cracked. The tensile stress of concrete varies with time so careful consideration of load events are required so predicted stresses can be compared with allowable at the appropriate age. Propping loads from slabs being cast above are acknowledged to be significant early loading events which will cause cracking.

In addition to the Final load event, construction stages that could be considered are:

1. Striking
2. Casting the floor above

NOTE Propping loads from casting slabs above have an impact on already struck slabs - dependant upon the extent of back-propping adopted.

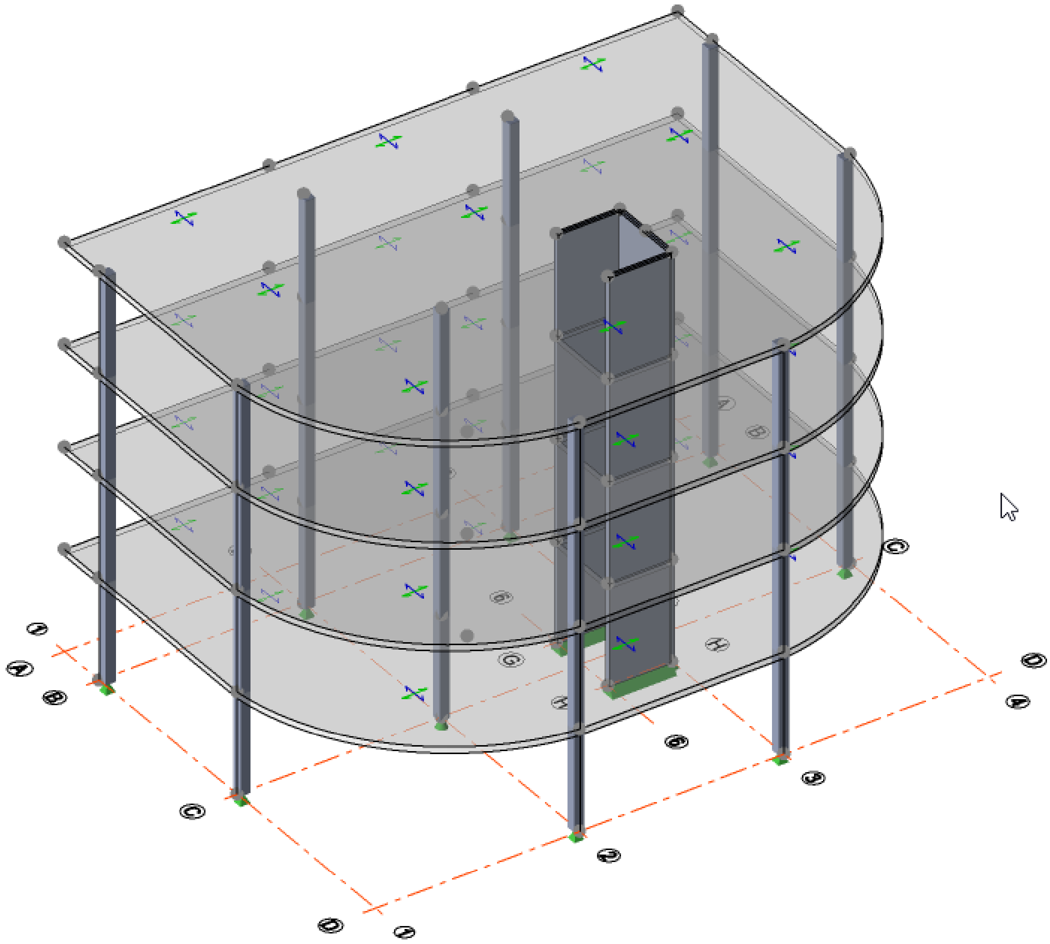


Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

3. Adding Partitions
4. Finishes

A typical model event sequence

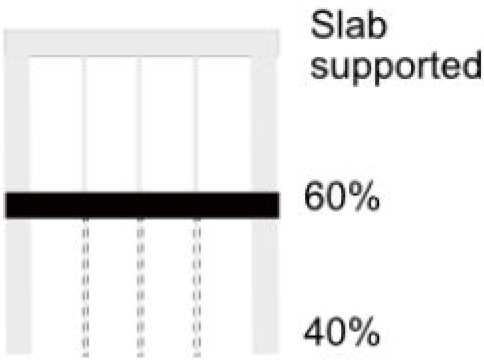
Let us consider a multi-storey building where the slab layout is the same on each level.



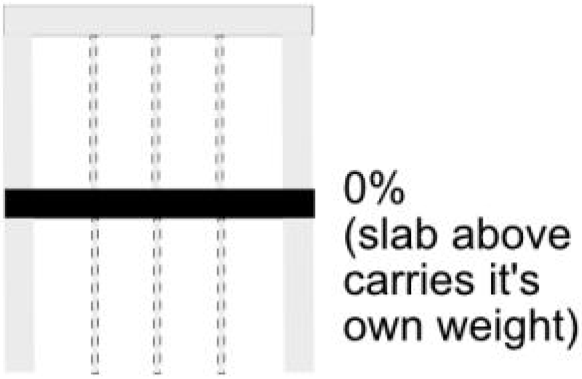
If we think about the slab model event sequence that occurs for the slab at level 1 it could be something like this:

1. Strike and backprop slab (slab carries it's own weight)

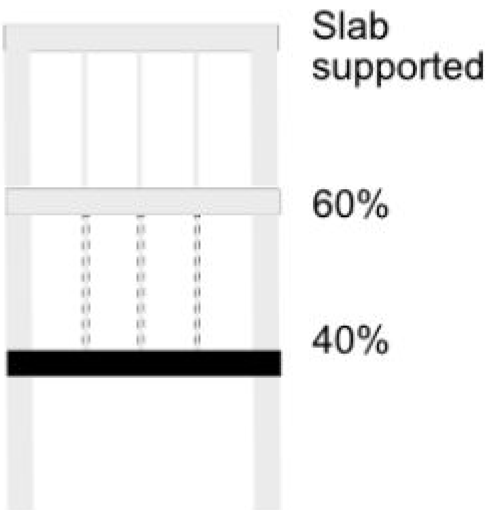
2. Cast Slab above (slab carries a proportion of the weight of slab above - the proportion is dependent on the number of levels of backpropping and there is also some debate about the efficiency of load sharing between the supporting levels. In this example we will assume 2 levels of propping and that the level directly below supports 60% and level 2 below supports 40%.)



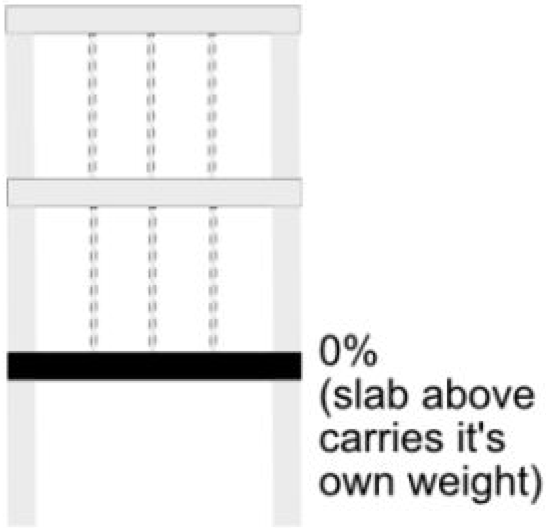
3. Strike and backprop slab above (slab above now carries its own weight)



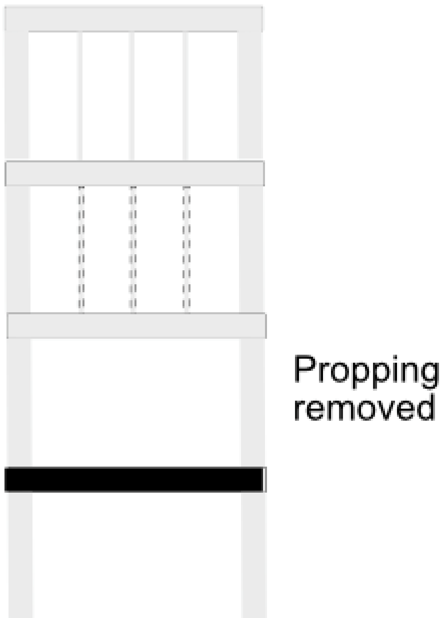
4. Cast slab 2 above (slab carries a proportion of the weight of the slab 2 above)



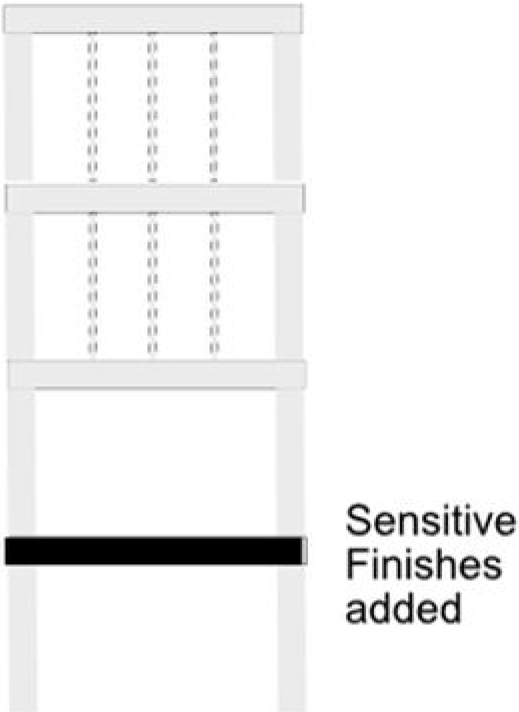
5. Strike and backprop slab 2 above (slab 2 above now carries its own weight)



6. If propping extends through 3 levels then there can be events for casting and striking the slab 3 above. In this example, the propping is removed and used two levels above.



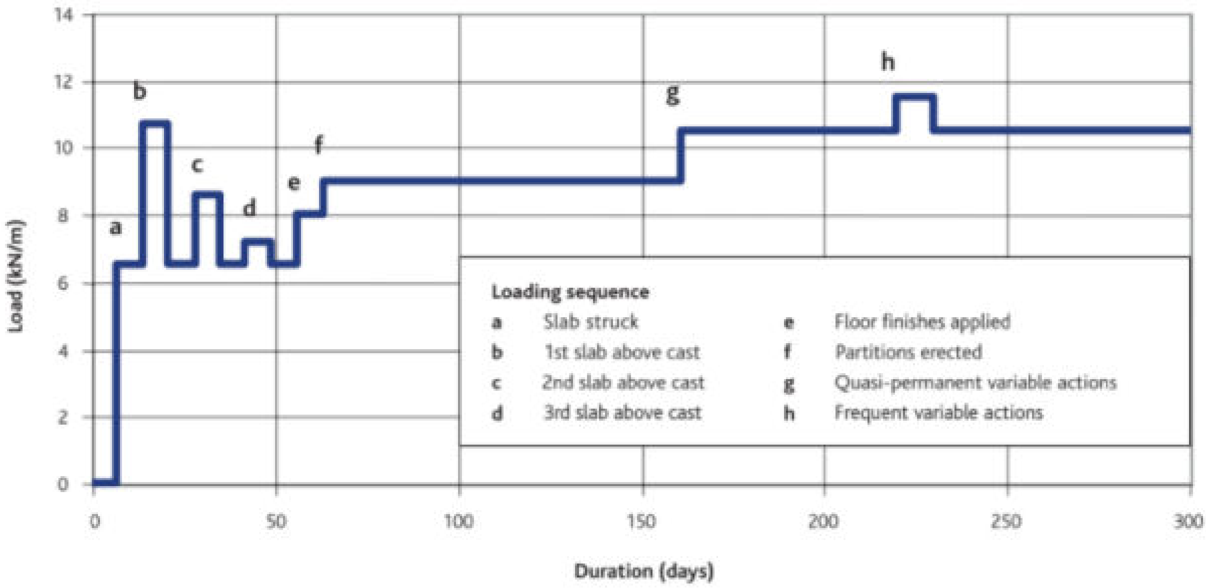
7. Additional load from finishes (in particular from sensitive finishes)



8. Start of occupation.
9. Final Load Event

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

The view shown below is a graphical representation of the above but in this case is recognising 3 levels of backpropping



Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

This sort of event sequence can be described within the Event Sequences dialog as shown below.

NOTE Some of the event parameters vary between headcodes, so both the ACI and Eurocode versions of the dialog are shown.

Model Event Sequence ACI (US Imperial units)

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Constructio load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	160.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Model Event Sequence Eurocode (metric)

:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	160.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Note the increased slab self weight when casting slab 1 above and slab 2 above where an additional 60% and 40% of the slab weight is being supported respectively. This is defined in the On submodel % column.

A slab model event loading sequence for any other internal slab i.e. level 2 would be identical to that described above.

If we now consider the uppermost slab - the roof.

- Is the slab event sequence for the roof any different?
- Can we use the slab model event sequence above for the roof?

The event sequences for casting any slabs above are not required, since there is no need to make an allowance for additional propping loads. This means that a different event sequence is necessary to deal with the roof. In Tekla Structural Designer differences in event sequences are dealt with using a Custom Event Sequence.

Technically, we could deal with the propping events in one of two ways.

1. Delete the event
2. Keep the event, but adjust the included slab self weight to allow only for the roof load as the previous event. i.e the event has no change in load to that of the previous load event.

In Tekla Structural Designer, we must deal with the change using method 2 above. This is due to the way Tekla Structural Designer deals with Custom event sequences (page 1402).

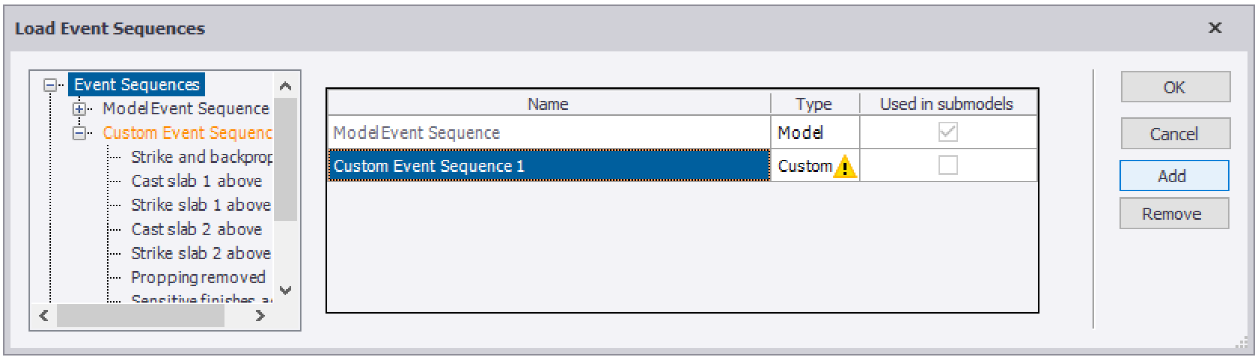
See also

Custom event sequences (page 1402)

Custom event sequences

Custom event sequences are required to deal with different slab loading sequences, such as for the roof slab and special cases like transfer slabs.

A custom event sequence can be created via the Add button on the Event Sequences page (highlighted below).



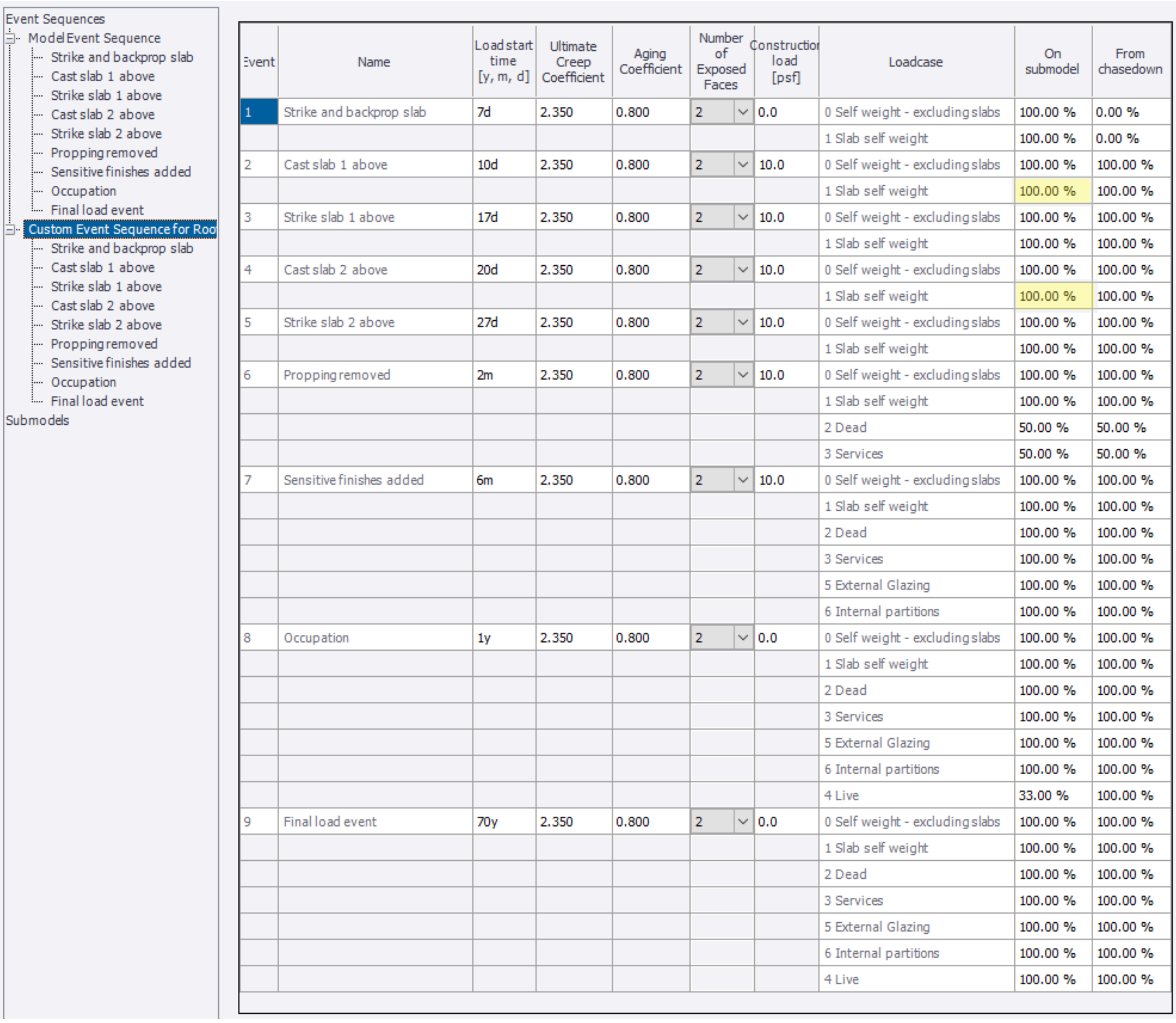
Once it has been added and given a name you can then edit the Custom event sequence by selecting it in the list.

Editing a custom event sequence

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

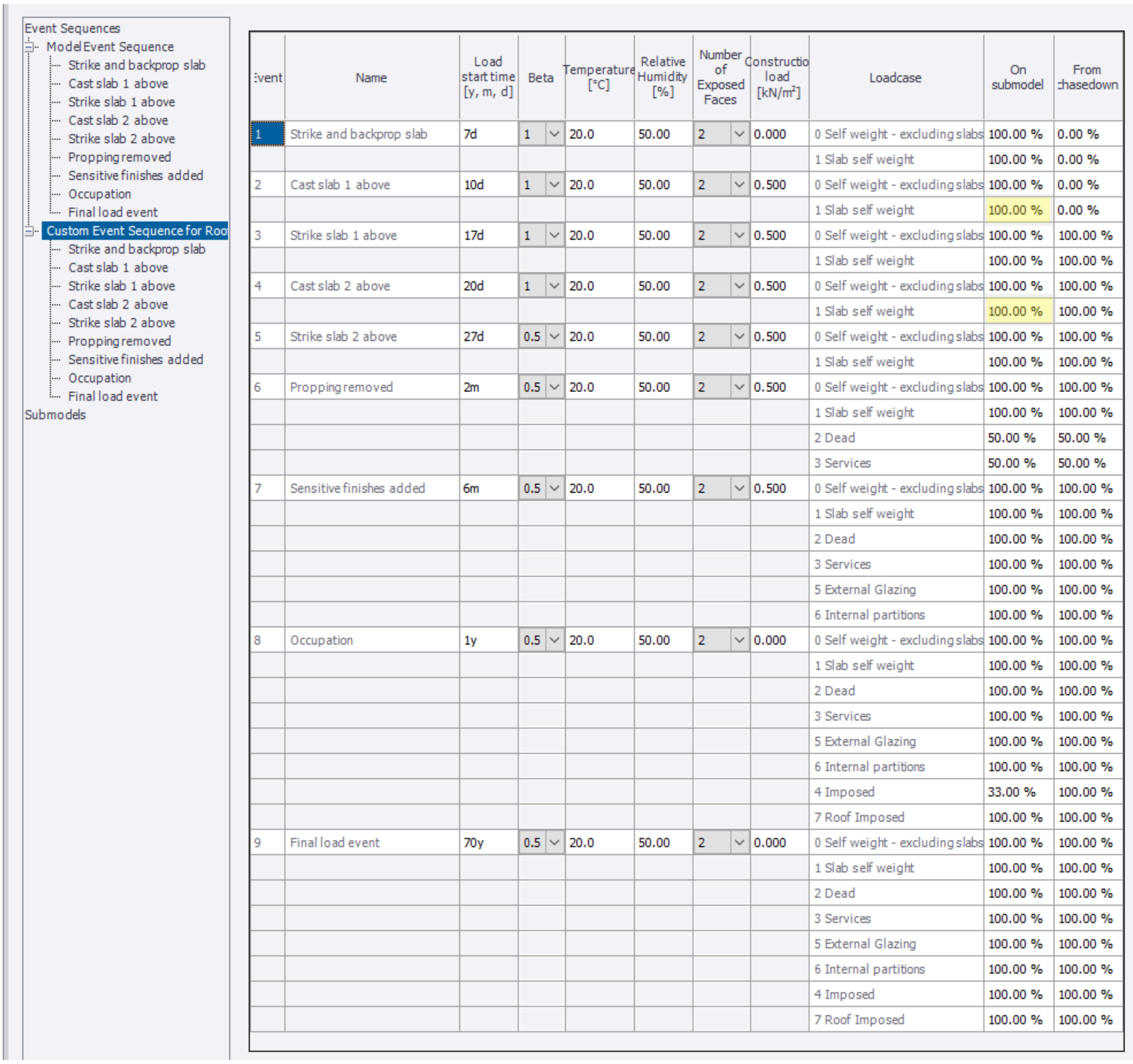
In the screenshot below, to create a custom event sequence for the roof we have edited the slab self weight load in the Cast slab 1 above and Cast slab 2 above events to 100% i.e. it is only supporting the roof slab self weight and not any extra propping load from any slabs above.

Custom Event Sequence ACI (US Imperial units)



Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Custom Event Sequence Eurocode (metric)



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %

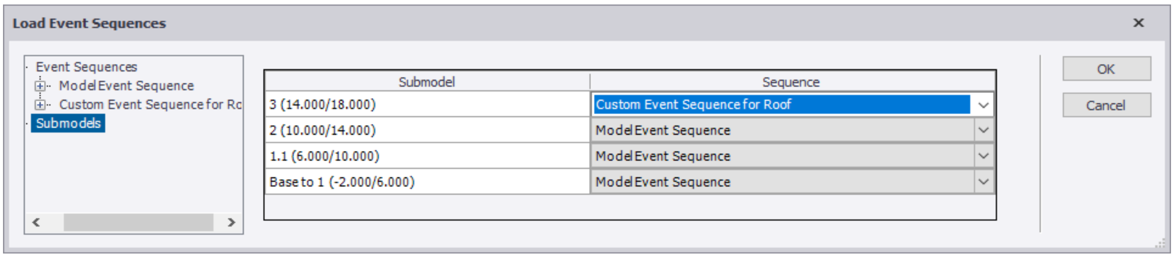
It should be noted that it is not possible to Add, Insert, Remove or change the order of Custom event sequences. It is also not possible to alter any item that is greyed out within the table. This includes events and loadcases. You can, however, edit the Load start time, Number of Exposed faces, Construction loads and the % of load to apply in the combinations (and if working to ACI: Ultimate Creep and Aging Coefficients; or if working to Eurocodes: Beta, Temperature, Relative Humidity). This could mean that careful consideration of the slab model event sequence is necessary to ensure any necessary events are included.

Assigning a custom event sequence to a submodel

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

You can assign different event sequences to different submodels (slabs) using the Submodels page of the Load Event Sequence.

In the screenshot below, the slab model event sequence is assigned to all slabs, except the roof, which has its own custom event sequence defined.



Hence, when you run a rigorous slab deflection estimate on a selected slab, it runs the Load Event Sequence specified on the Submodel page above.

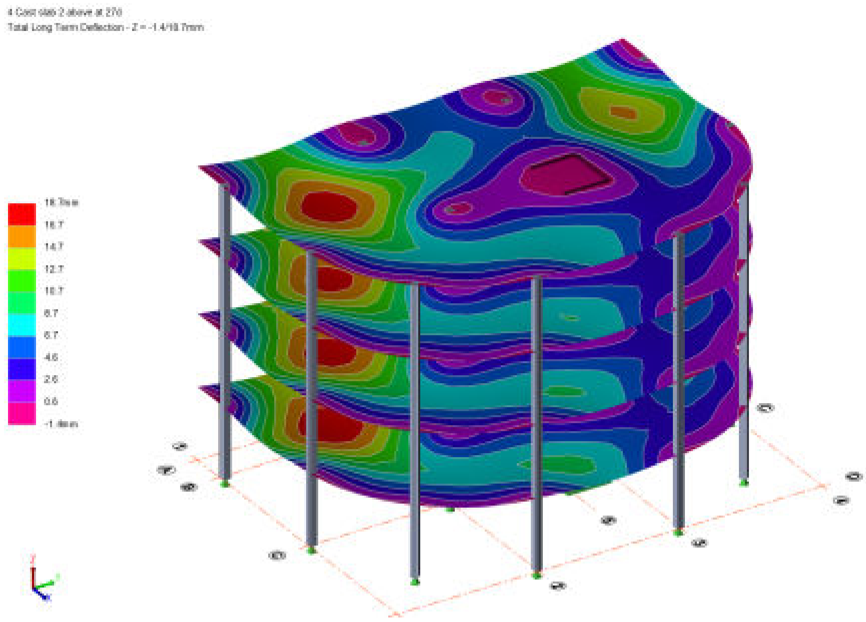
Understanding event sequence deflections

The model described in A typical model event sequence (page 1394) consists 3 identical levels and then a roof level with different loading applied. To cater for this a Custom Event Sequence has been applied to the roof submodel.

When you run **Slab Deflection > Analyse All** for this model, 4 sub-models are dealt with one after the other:

- Each one considers 9 events
- Each one does an iterative cracked section analysis at every one of these events
    - assume say, average of 20 iterations in each cracked section analysis
    - That's 180 analyses for each sub-model - over 700 analyses for the whole structure.
- There is an additional instantaneous analysis for each event.

After analysis the deflections can be reviewed for individual events. The view below shows the deflections for Event 4 “Cast slab 2 above”.



Event 4 ends when the slab is 27 days old. The view above is the deflection estimate when each of the slabs are this age. These things don't happen at the same time but it's a very convenient way to display things.

Initially, this has the potential to be confusing, however it should be easy to understand provided you remember that:

- Event Sequences are “Slab Event Sequences”. They describe the events a slab goes through where Day 0 = the day the slab is cast.
- Event Sequences are NOT structure event sequences. They do not describe the all the events starting on day 1 of the overall construction.

Slab deflection analysis sequence

The same basic process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

Having run a Slab Deflection Analysis the following analysis results are then available to review for either the chosen level (sub-model) or the entire structure dependent upon your chosen analysis

- Deflections (page 1407) - Three deflection types are available: Total, Differential, and Instantaneous.
- Extent of Cracking (page 1411) - You can also review contours to display the extent of cracking at any load event.
- Relative Stiffness (page 1413) - You can also review the relative stiffness in a particular result direction for any specified event.
- Effective Reinforcement (page 1415) - You can also review the area of effective reinforcement for a chosen result direction for each FE element.

Total, differential, and instantaneous deflection types

Once the slab deflection analysis has been run, three deflection types are available for review:

- **Total** deflection at the end of any event.
- **Differential** deflection between any two events (Start of Event and End of Event).
- **Instantaneous** deflection (not actually needed for TR 58).
  - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
  - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

Understanding differential deflections

When looking at differential deflections it is important to appreciate that an event is not a single point in time, it is a time period which has different deflections at the start and the end - so it is logical that there can be a differential deflection for a single event.

To clarify the selections requested in the ribbon:

- **Start Event** defines the **start** of the start event
- **Event** defines the **end** of the event

Therefore, if you specify the same event as the start and the end event, you will still see a differential deflection.

To clarify further, consider the scenario below:

- Event 1 - Construction loading

- Event 2 - Application of Sensitive Finishes
- Event 3 - Application of Additional Finishes
- Event 4 - Occupation (all loading with live load set at long term factor)
- Event 5 - Final loading event (as above with live load set at 100%)

To determine the maximum differential deflection relevant to the application of sensitive finishes you would define this as the deflection between Events 2 and 5 (Start of Event 2 to End of Event 5). If you opted to pick Events 1 and 5 you would get an un-conservative deflection reported since the effect of construction loading is also considered.

Slab deflection calculations in depth

Interrogating slab deflection calculations

Tekla Structural Designer's Slab Deflection Analysis is not a "black box" - the calculated results are all exposed for interrogation as required.

Several different Results views are available: Deflection, Extent of Cracking, Relative Stiffness and Effective Reinforcement. There is also a Composite modulus report available for each slab.

To help pull all the results together, the items noted above are tied together in the following way:

1. Every element is part of a slab item. Each slab item can have different effective concrete properties.

   The best way to understand this is to look at the summary of information from the Composite Modulus report for each event and consider things like:

   • Adjusted event times due to temperature and cement class

   • Adjusted creep properties due to the number of exposed faces

   • Incremental loading factors

   • The Composite Modulus Calculation

2. Every shell element can have different Effective Reinforcement:

   • This is determined automatically

   • The Effective Reinforcement results view allows you to confirm the values used.

3. Cracking and Stiffness Calculations for each event:

• This is dependent upon the effective concrete properties, effective reinforcement, and the forces that develop

• So this calculation is unique for each direction of each shell for each event.

• The stiffness of cracked sections is dependent on the degree of cracking - so the procedure is iterative (force -> stiffness -> new force, etc)

• At the converged conclusion of this you can see (and check):

- the extent of cracking via the Extent of Cracking results view

- the stiffnesses determined via the Relative Stiffness results view

Composite creep

Design codes typically provide a way of calculating an effective creep modulus at time, t for a constant load/stress applied at time t_0. Typically this is presented as $\phi(t, t_0)$.

An effective Young's Modulus is then calculated as $E_{c, eff} (t, t_0) = E_{c,28} / [1 + \phi(t, t_0)]$

However, codes typically do not give guidance on how to deal with a loading history where loads vary over the time period being considered.

Technical Report 58 introduces guidance on this topic and proposes a method by which the loading history can be taken into account. (Reference Section 8.4.1, equation 8.37 and also the example on page 36):

$$\left(\frac{\Sigma w}{E_{comp}}\right)n = \frac{w_1}{E_{eff,1}} + \frac{w_2}{E_{eff,2}} + \ldots\ldots + \frac{w_i}{E_{eff,i}} + \ldots\ldots + \frac{w_n}{E_{eff,n}}$$

Where:

n = event under consideration

w_i = incremental load in event i (= load in event i - load in event (i-1)) (note that this will be a negative value when load is removed)

$E_{eff,i} = E_{c, eff} (t_{end,n}, t_i)$ (i.e. covers period from start of event i to end of event n)

The above is logical when you consider a single member subjected to a constant loading arrangement that is increased or decreased at each event. However, when you consider an entire slab with many panels receiving different loading increments in different events it does not seem reasonable to consider all the panels together. Two examples of this are:

1. Why should the addition of cladding loads affect internal panels to the same extent as edge panels?
2. In a transfer slab why should panels that don't support columns be affected to the same extent as those which do?

The aim of TR58 is clear, in that loading on a span/panel is taken as an indication of stress. What this fails to consider is that loading on another span/panel can also induce stress (although in most situations this will be a secondary effect). It is also clear that you do not actually need "loads", you just need "relative loads" or some other measure of the relative work done in each event. With this in mind a more general approach has been developed where the relative "work done" by each panel is determined by considering the strain energy in each event:

1. Calculate total strain energy 'Q_0' for each Slab Item for a unit load case

   • 'Q_0' = sum of 2D Element strain energies

2. For each Load Event 'i'

   • Calculate incremental strain energy 'Q_i' 'Q_i' = sum of incremental 2D Element strain energies

   • Calculate equivalent incremental load factor 'λ_i

   ' $\lambda_i = Q_i / Q_0$

3. E_{comp} can be established from the equation below where "incremental work done" replaces "incremental load" in the TR58 equation.

$$\left(\frac{\Sigma\lambda}{E_{comp}}\right)n = \frac{\lambda_1}{E_{eff.1}} + \frac{\lambda_2}{E_{eff.2}} + \ldots\ldots + \frac{\lambda_i}{E_{eff.i}} + \ldots\ldots + \frac{\lambda_n}{E_{eff.n}}$$

There is an array of intermediate values which lie behind the calculation of the composite modulus, E_{comp} for each slab item, for each event. The composite modulus calculation is provided as an excel spreadsheet report.

You can either generate a report for a chosen slab, selected slabs or all slabs, dependant upon your selection method.

- To obtain slab modulus reports for all slab items, right click anywhere in a scene view and choose Export Eff. Modulus report to Excel > For all slab items
- To obtain slab modulus reports for selected slab items, select the slabs in the structure view regime, right click anywhere in the scene view and choose Export Eff. Modulus report to Excel > For selected slab items
- To obtain slab modulus reports for a chosen slab items, right click a slab panel in the structure view regime and choose Export Eff. Modulus report to Excel > For current slab items

A typical composite modulus report for a slab panel is shown below.

Composite Modulus Calculation

Event	Start [d]	Adjusted Start Temp. and Cement [d]	Incremental load factor, λ	To end of Event 1			To end of Event 2			To end of Event 3			To end of Event 4			To end of Event 5		
				φ	$E_{c,eff}$ [N/mm^2]	$\lambda / E_{c,eff}$ [mm^2/N]	φ	$E_{c,eff}$ [N/mm^2]	$\lambda / E_{c,eff}$ [mm^2/N]	φ	$E_{c,eff}$ [N/mm^2]	$\lambda / E_{c,eff}$ [mm^2/N]	φ	$E_{c,eff}$ [N/mm^2]	$\lambda / E_{c,eff}$ [mm^2/N]	φ	$E_{c,eff}$ [N/mm^2]	$\lambda / E_{c,eff}$ [mm^2/N]
1 Strike and backprop slab	10	10	6.983	0.772	19558	0.00036	1.235	15503	0.00045	1.526	13716	0.00051	2.714	9330	0.00075	2.714	9330	0.00075
2 Propping slab 1 above	20	20	4.690	-	-	-	1.019	17164	0.00027	1.307	15020	0.00031	2.381	10249	0.00046	2.381	10249	0.00046
3 Propping removed	61	61	-3.443	-	-	-	-	-	-	0.921	18036	-0.00019	1.926	11843	-0.00029	1.926	11843	-0.00029
4 Sensitive finishes added	122	121	2.734	-	-	-	-	-	-	-	-	-	1.686	12902	0.00021	1.686	12902	0.00021
5 Final load event	25550	25502	3.499	-	-	-	-	-	-	-	-	-	-	-	-	0.000	34650	0.00010
Total of $\lambda / E_{c,eff}$ [mm^2/N]						0.00036			0.00072			0.00063			0.00113			0.00123
E_c to end of Event [N/mm^2]						19558			16130			13054			9727			11776

It should be noted that Tekla Structural Designer takes into account the cement class when determining the temperature adjusted age of loading so minor variations will occur.

The effective modulus is used in determining the properties for each load event.

Extent of cracking

The effective modulus, and the effective reinforcement are used to determine the cracked or uncracked state of each shell for each event, for each direction.

Eurocode 2 provides an expression that predicts the behavior between the cracked and uncracked states. This expression uses a Distribution factor, ζ that apportions the behavior between a fully-cracked state (±1.0) and an uncracked state (0) for interpolating the stiffness when a state of partial cracking exists.

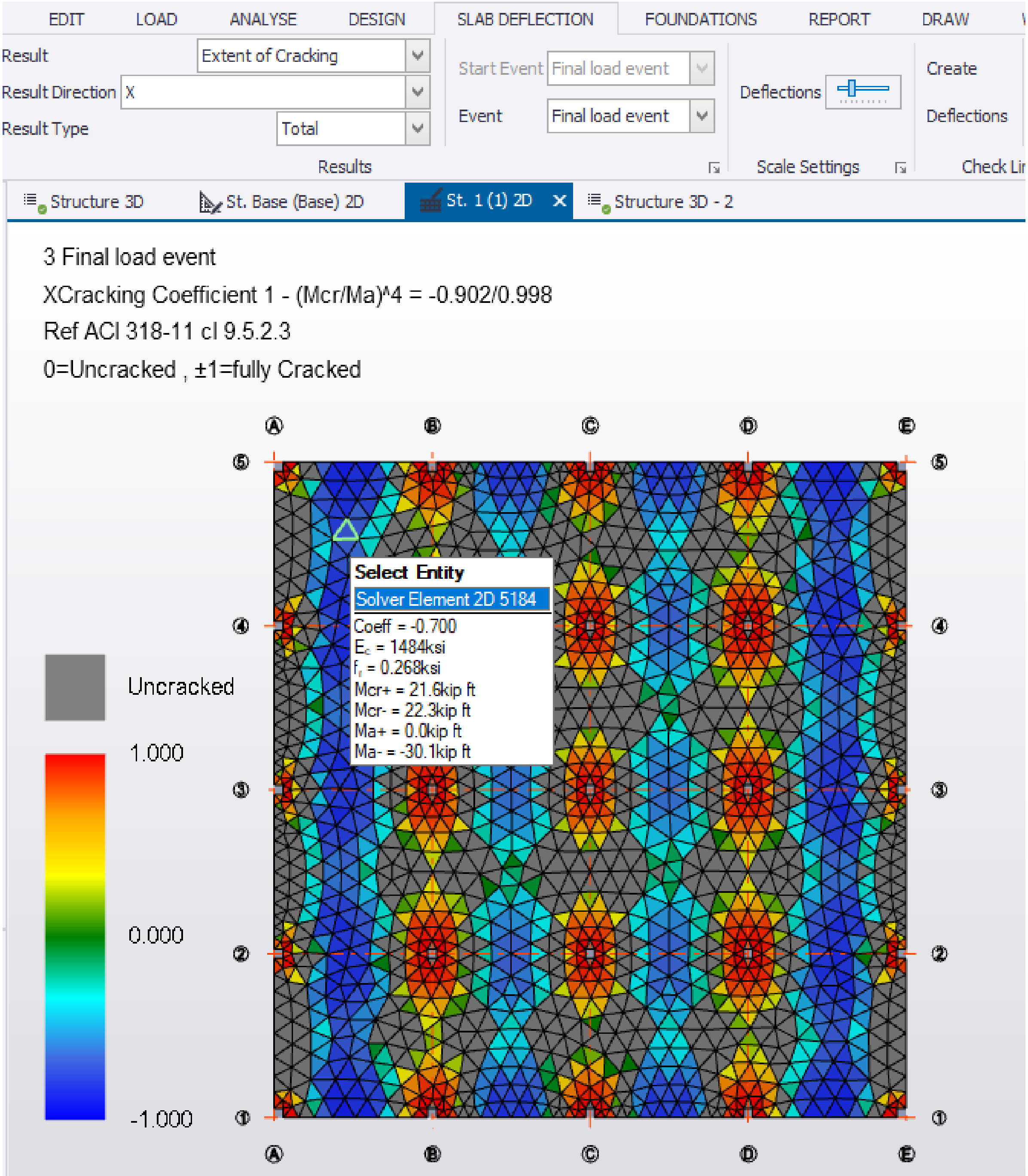
$\zeta = 1 - ß\,(M_{cr} / M)^2$

Where:

- ß is a user defined value specified in the Event Sequences and is either: 1.0 for single short-term loading 0.5 for sustained loads or many cycles of repeated loading
- M_{cr} is the hogging (positive) or sagging (negative) cracking moment.
- $M_{a(+|-)}$ is the relevant Wood-Armer moment in the direction for which the display is shown (X or Y). This is calculated from M_x, M_y & M_{xy} in the usual way, when determining the extent of cracking for a shell element for each iteration for each Event.

If you view Extent of Cracking results for a chosen result direction and cycle through the events you will see each of the FE elements shaded to indicate the extent of cracking.

At the first sign of cracking, $M_{cr} > M_a$. If you hover over an FE element the tooltip provides some intermediate calculation results to verify this distribution factor.

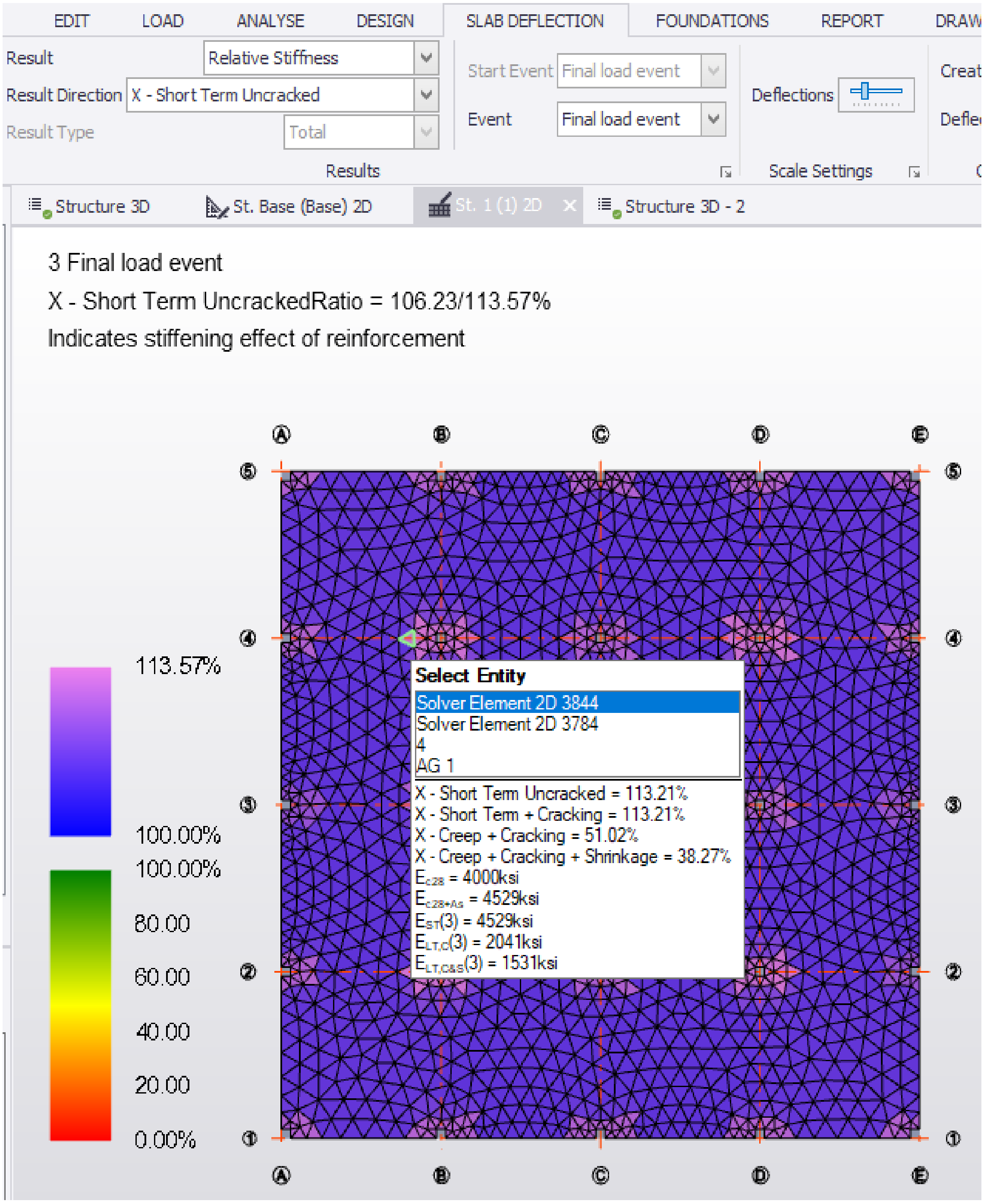


The tooltip shows the applied moment for the Event in question, rather than the worst moment of all Events up to and including the one being looked at -

this allows engineers to deduce that a greater level of cracking was caused by an earlier Event.

Relative stiffness

The tooltip provides detailed information on the relative stiffness calculations.



The tooltip lists:

- E_{c28} : E_{db}*1.05*stiffness adjustment factor. (Where E_{db} = the short term modulus from the concrete materials database). The stiffness adjustment factor used is determined from the Slab Deflection ribbon > Settings >, Modification Factors page. Provided for information - not directly used in any analysis.

- Ec_{28+As} : the short term modulus including for reinforcement. Provided for information - not directly used in any analysis.
- $E_{ST}(i)$: the short term modulus used in the instantaneous analysis (i.e. includes area of reinforcement and cracking if cracking has occurred) for the selected event.
- $E_{LT,C}(i)$: the modulus used in the final iteration of long term deflection estimation (i.e. includes area of reinforcement and cracking if cracking has occurred, and effective creep) for the selected event.
- $E_{LT,C\&S}(i)$: $E_{LT,C}(i)$ with further adjustment to allow for effect of shrinkage (= $E_{LT,C}$ / multiplier) for the selected event. The shrinkage multiplier to determine the shrinkage contribution is determined for the chosen event based upon the ratio of the maximum panel Z deflection (including shrinkage) / total Z deflection (excluding shrinkage). This provides an indication of the overall effective stiffness adjustment. Provided for information - not directly used in any analysis.

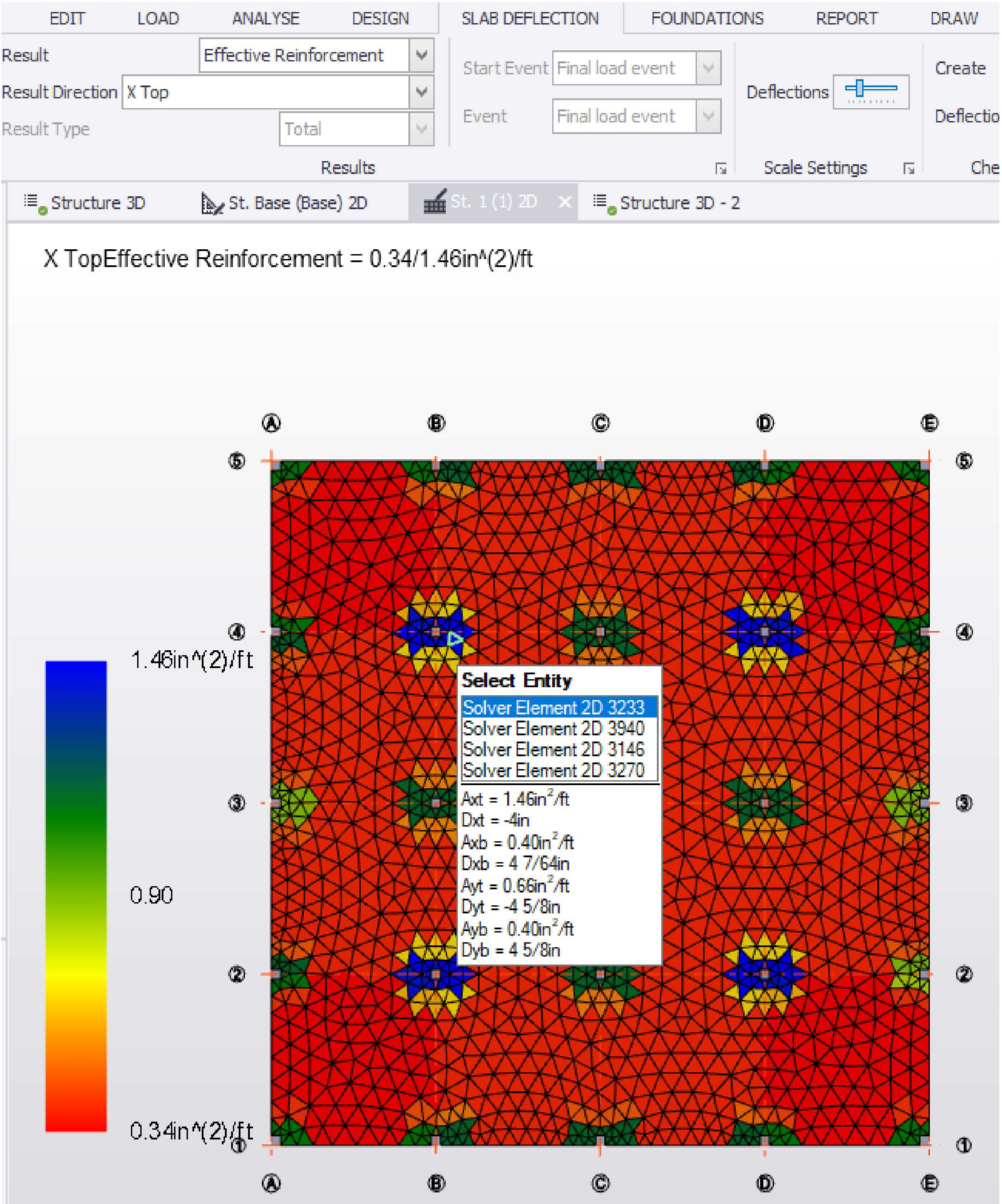
Based on the modulus, E defined above, a number of ratios are provided in the tooltip for the chosen result direction.

- Short Term Uncracked = E_{c28+As} / E_{c28}
- Short Term + Cracking = E_{ST} / E_{c28}
- Creep + Cracking = $E_{LT,C}$ / E_{c28}
- Creep + Cracking + Shrinkage = $E_{LT,C\&S}$ / E_{c28}

Effective reinforcement

Effective reinforcement for each shell element is also required for the determination of the shell's effective properties at the end of each load event.

The effective reinforcement used in the property calculations are reported to you in the 4 layers for every shell element. The information is provided as a color coded shell display from minimum to maximum reinforcement area.



If you hover over a shell the tooltip provides the following information:

- A_{xt} : X Top Effective Reinforcement.

- D_{xt} : Distance from the section centroid to the center of X top reinforcement.
- A_{xb} : X Bottom Effective Reinforcement.
- D_{xb} : Distance from the section centroid to the center of X bottom reinforcement.
- A_{yt} : Y Top Effective Reinforcement.
- D_{yt} : Distance from the section centroid to the center of Y top reinforcement.
- A_{yb} : Y Bottom Effective Reinforcement.
- D_{yb} : Distance from the section centroid to the center of Y bottom reinforcement.

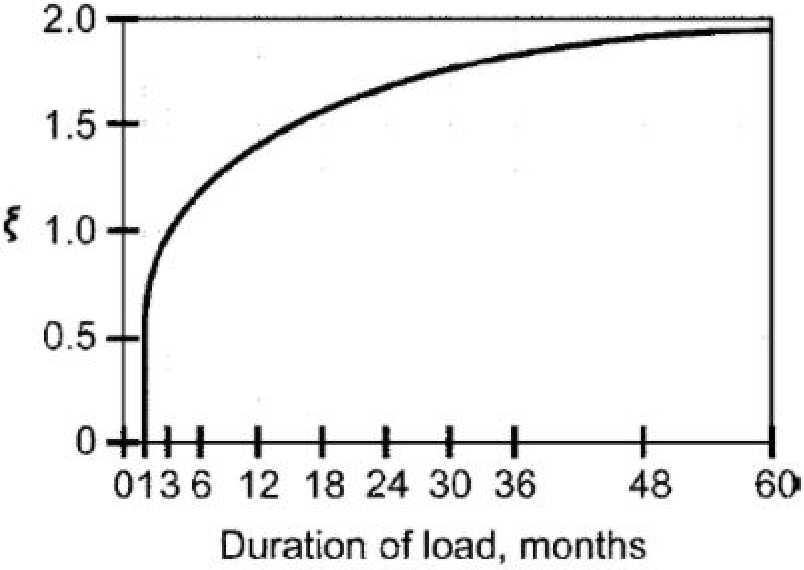
Shrinkage allowance

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

- Eurocode 2 provides a method to estimate shrinkage strains and curvatures based on exposed surface area, member size, relative humidity and reinforcement quantity and position.
- Asymmetry of reinforcement leads to curvature which leads to deflection. It is estimated this effect can contribute up to 30% to the long-term deflection.
- Technical Report 58 provides a theoretical method of estimating the additional shrinkage deflection effect in the analysis

At this time the TR58 method has not been implemented within Tekla Structural Designer. Shrinkage is taken into consideration using a multiplier, by making an overall adjustment to the total deflection (excluding shrinkage) in line with simpler adjustment proposals of the ACI code. This approach is in line with many other software products.

The simplest ACI approach makes an allowance for all long term effects (creep and shrinkage) by using an adjustment factor. This is based on the graph below and also provides some specific values.



Time	Multiplier for long-term deflections
5 years or more	2.0
12 months	1.4
6 months	1.2
3 months	1.0

Note that we said shrinkage effects and not creep and shrinkage. Creep is dealt with rigorously in Tekla Structural Designer so we need to ascertain the proportional effect of shrinkage only. ACI 435 provides some indication of the separate contribution of creep and shrinkage.

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ 4 $\sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

Comparing the different sources the ratio of shrinkage is as follows:

- Sbarounis 1.2 / 5 = 24%
- Branson 1 / 4 = 25%
- Graham and Scanlon 1 / 3.5 = 28%

  (ignore higher modulus of rupture because reduced values are considered automatically in the cracked section analysis).

The above provides a shrinkage ratio of between 24% and 28%. Hence we recommend a value of between 20%-30% is used. A 25% default is provided via the Slab deflection ribbon > Options dialog and the Creep and Shrinkage page.

The total deflection due to shrinkage effect is determined based on an identified “Total Shrinkage Event” towards the end of the event sequence. The event sequence with the latest load start time is used for calculating the shrinkage adjustment. If multiple events exist with the latest load start time then the first one is considered.

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

Using this “total shrinkage effect” we can then assign a proportion of the total shrinkage to each event.

With reference to earlier versions of ACI 435 (1966) on which the values in the graph above are based, additional values for 1 month and 3 years can be obtained. This follows the case where As’ = 0 (because compression steel is allowed for differently in the code). By comparison with the graph above, we can see closely matched values.

Duration of loading	**Factor F**		
	As' = 0	**As' = 0.5As**	**As' = As**
1 month	0.58	0.42	0.27
3 months	0.95	0.77	0.55
6 months	1.17	0.95	0.69
1 year	1.42	1.08	0.78
3 years	1.78	1.18	0.81
5 years	1.95	1.21	0.82

The values we have adopted for considering shrinkage effects are as tabulated below. The final column provides the proportion of the total shrinkage at a given time.

Time	Long Term Effects Factor	Proportion of Total Shrinkage
0	0	0.00
1 month	0.6	0.30
3 months	1	0.50
6 months	1.2	0.60
1 year	1.4	0.70
3 years	1.8	0.90
5 years and above	2	1.00

From the above, for any event, the end of event time is used to calculate a "Proportion of total shrinkage" using linear interpolation between the values discussed in the table above.

Deflection calculations on the Z deformation are then adjusted to account for shrinkage effects.

As an example, let's assume the following event sequence.

- Event 1 = 7 days, Event 2 = 10 days, Event 3 = 17 days, Event 4 = 20 days, Event 5 = 27 days, Event 6 = 2 months, Event 7 = 6 months, Event 8 = 1 year and Event 9 = 70 years

Assuming a shrinkage factor of 25% (user defined input value), a basic multiplier can be determined = 1/(1-25%) = 1.333

Event 9 Final event at 70 years analysis deflection (excluding shrinkage) = 32.4mm

Therefore, the Total deflection (including shrinkage) = 32.4 x 1.333 = 43.2 mm

Total Deflection from shrinkage alone is 43.2 - 32.4 = 10.8 mm

We can now apportion this deflection due to shrinkage, to each event based upon the event time and a proportion value.

i.e.

At 0 days proportion of total shrinkage is 0, At 1 month proportion is 0.3. Therefore using linear interpolation between these values;

- Event 1 (7 days) Shrinkage multiplier = 0.3 * 7/30 = 0.07
- Event 2 (10 days) Shrinkage multiplier = 0.3 * 10/30 = 0.1
- Event 3 (17 days) Shrinkage multiplier = 0.3 * 17/30 = 0.17
- Event 4 (20 days) Shrinkage multiplier = 0.3 * 20/30 = 0.2
- Event 5 (27 days) Shrinkage multiplier = 0.3 * 27/30 = 0.27

At 1 month proportion of total shrinkage is 0.3, At 3 month proportion is 0.5. Therefore using linear interpolation

- Event 6 (2 months) Shrinkage multiplier = 0.4

At 6 month proportion of total shrinkage is 0.6

- Event 7 (6 months) Shrinkage Multiplier = 0.6

At 1 year proportion of total shrinkage is 0.7

- Event 8 (1 year) Shrinkage Multiplier = 0.7

At 70 years proportion of total shrinkage is 1.0

- Event 9 (70 years) Shrinkage Multiplier = 1.0

The Shrinkage deflection that occurs at each event is then the total shrinkage 10.8 mm x the shrinkage multiplier calculated above.

- Event 1 (7 days) Shrinkage = 0.07 x 10.8 = 0.76 mm
- Event 2 (10 days) Shrinkage = 0.1 x 10.8 = 1.08 mm
- Event 3 (17 days) Shrinkage = 0.17 x 10.8 = 1.84 mm
- Event 4 (20 days) Shrinkage = 0.2 x 10.8 = 2.16 mm
- Event 5 (27 days) Shrinkage = 0.27 x 10.8 = 2.92 mm
- Event 6 (2 months) Shrinkage = 0.4 x 10.8 = 4.32 mm
- Event 7 (6 months) Shrinkage = 0.6 x 10.8 = 6.48 mm
- Event 8 (1 year) Shrinkage = 0.7 x 10.8 = 7.56 mm
- Event 9 (70 years) Shrinkage = 1.0 x 10.8 = 10.8 mm

For each event, the total deflection (including shrinkage) reported in the Slab deflection view regime and the tooltips is the event analysis deflection (excluding shrinkage) + the proportion calculated above using the shrinkage multiplier.

Check lines

Check lines - a unique feature inTekla Structural Designer - provide a practical way to automate deflection checking and reduce the possibility of errors.

Each check line defines a line along which deflection checks are required. Multiple check lines can be created and each line can have several different deflection checks performed (against either a total, instantaneous, or differential deflection limit).

Whilst the check lines have to initially be positioned using engineering judgment, once they are in place they provide an instantaneous means to evaluate revised deflections following changes to the model parameters and re-analysis.

Assessing slab deflections without check lines would be quite an arduous task, since deflection limits are of the form (span / fixed value) and the spans can vary on an irregular slab, hence the permissible limit would also vary. Furthermore, you may wish to check total or differential deflections at or between different load events. Would the critical location be obvious?

Setting up the checks in advance (via the slab deflection check catalogue)

The deflection checks to be performed for the check lines are set up in the Slab Deflection Check Catalogue (page 2425) before the check lines are positioned.

Each check in the catalogue is specified as a Total, Differential or Instantaneous deflection limit that is checked for a specified event, (or in the case of a Differential check between the start of an event and the end of an event).

Related task Create the deflection checks to be applied to check lines (page 935)

Application of check lines

The application of check lines is an iterative activity. They should be created at all locations where you deem deflection checks to be required.

Check lines can only be created in a 2D view. If the Create command is greyed out ensure you switch to a 2D view.

When you click Create, the Properties Window automatically includes those slab deflection checks from the Slab Deflection Check Catalogue where "Use in new Check Lines" was checked. Note that a maximum of six checks can be assigned to a single check line; if you require more than six checks, then multiple check lines can be applied at the same location.

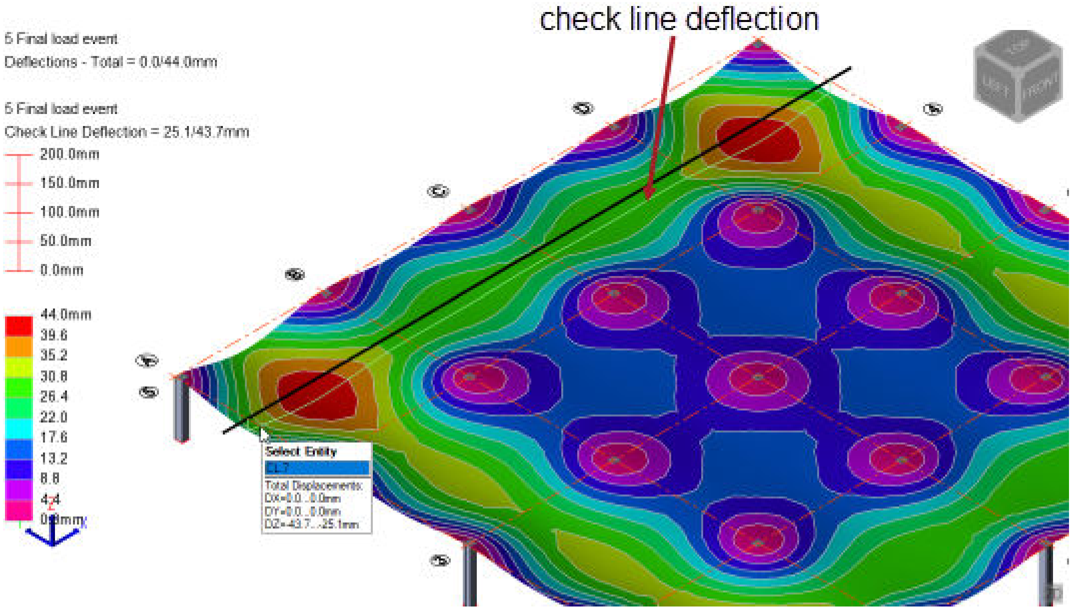
You can add further checks from the catalogue or add new checks directly from the Properties Window.

You can also edit the "Use in New Check Lines" option in the catalogue prior to running the Create command to change the default checks automatically assigned to each new check line.

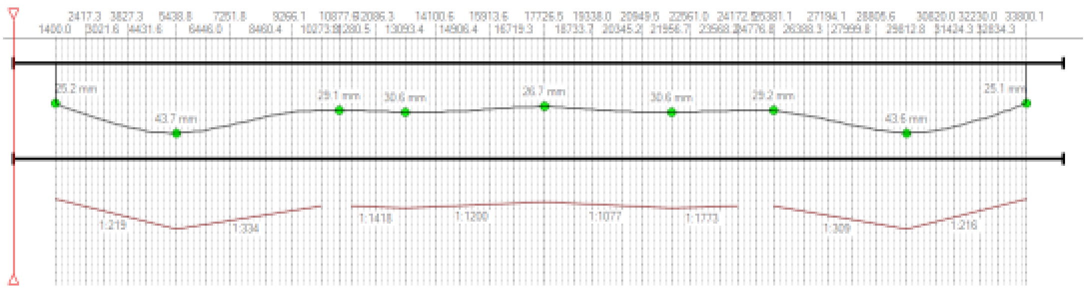
Related task Create a check line (page 935)

Displaying check line results

The Check Line deflection can be overlayed on the slab by using the Deflections command in the ribbon, it is perhaps easier to visualize if you switch the view to a 3D view of the slab using the 2D/3D toggle button in the bottom right of the window.



You can also right click on a check line and open the deflections check view. This displays a cut line through the slab, showing the deflected shape with the maximum and minimum deflection values. Beneath this it also draws the average slopes between maximum and minimum points and reports the average slope ratios.



The above deflection check view is controlled from the Loading Analysis ribbon which has droplists to enable you choose the Result Type (Total, Instantaneous, or Differential) and the Event(s).

The average slope ratios (for the appropriate result type/event) are checked against the deflection limits that have requested for the check line.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Related task Display check line results (page 939)

Check line reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

You can also generate a Slab deflection check line report for the entire structure, selected level, planes or sub structures via the Model Report command on the Report ribbon.

Related task View slab deflection reports (page 942)

Slab deflection status and utilization

The status and the utilization can be graphically displayed for both Check lines and slabs.

Check lines Status and Utilization

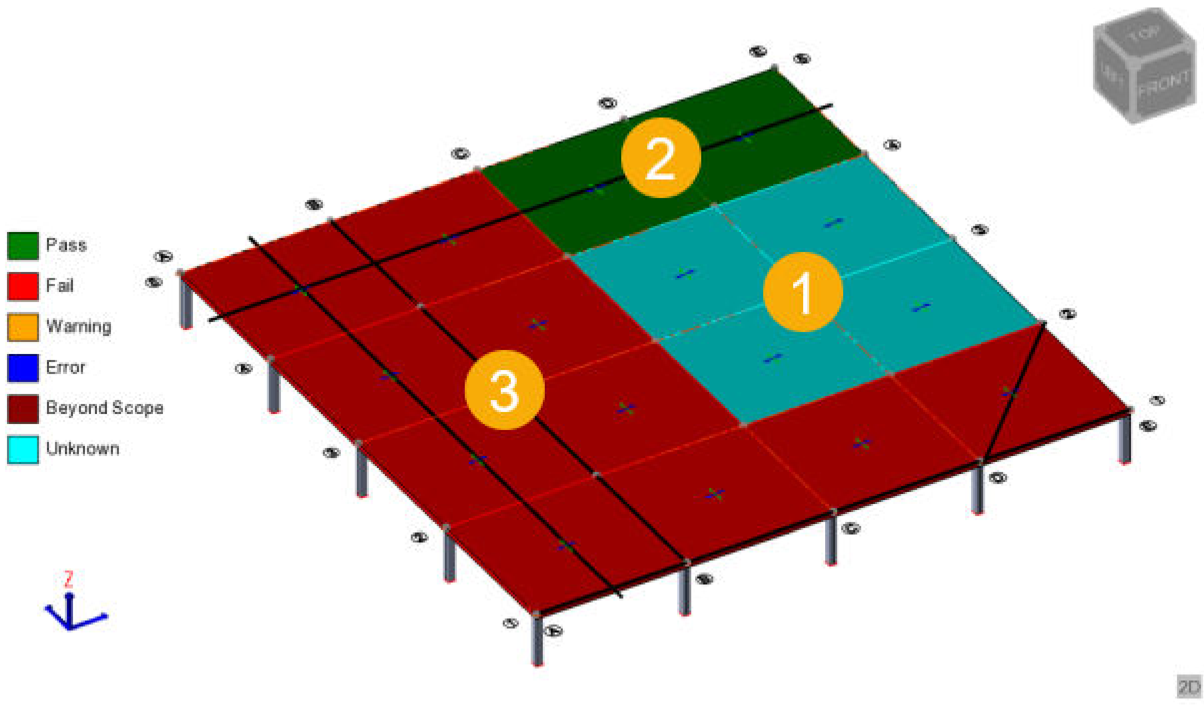
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical utilization ratio

Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization.

Related task: Display check line status and utilization (page 939)

Slab Deflection Status and Utilization

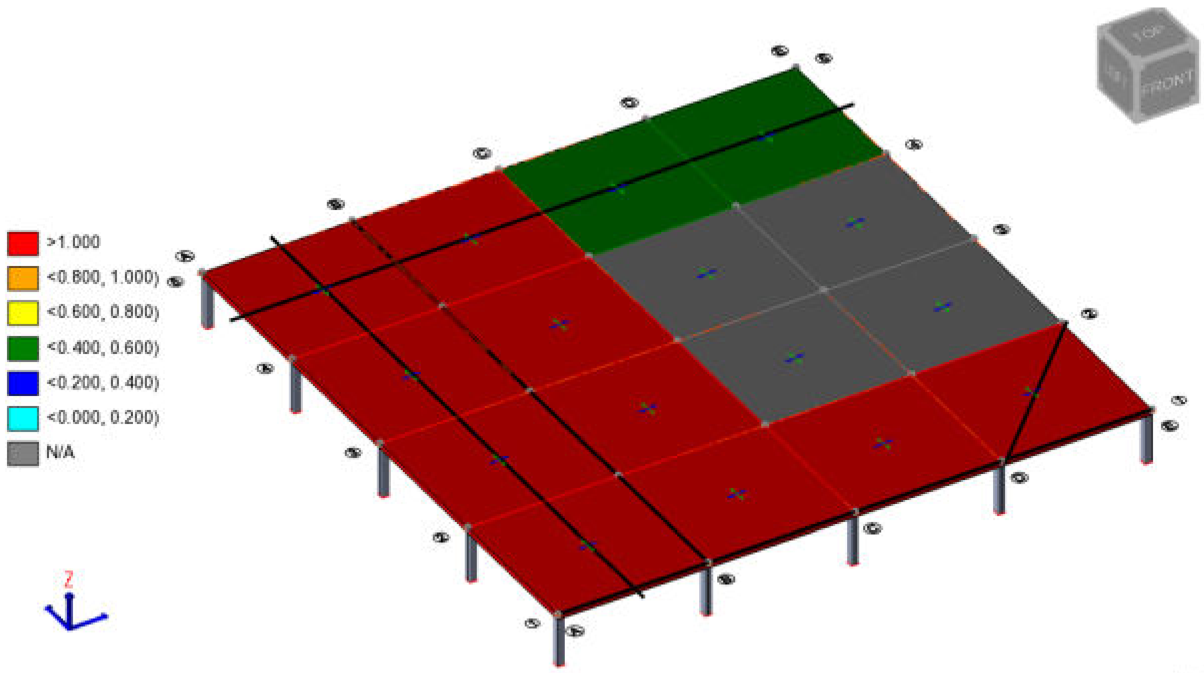
Slab deflection status is determined for each slab item as the worst status from all associated check lines detected within the slab item or touching the slab item boundary.



In the above status view:

1. No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
2. One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
3. A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Slab deflection utilization is similarly determined for each slab item as the worst utilization from all associated check lines.

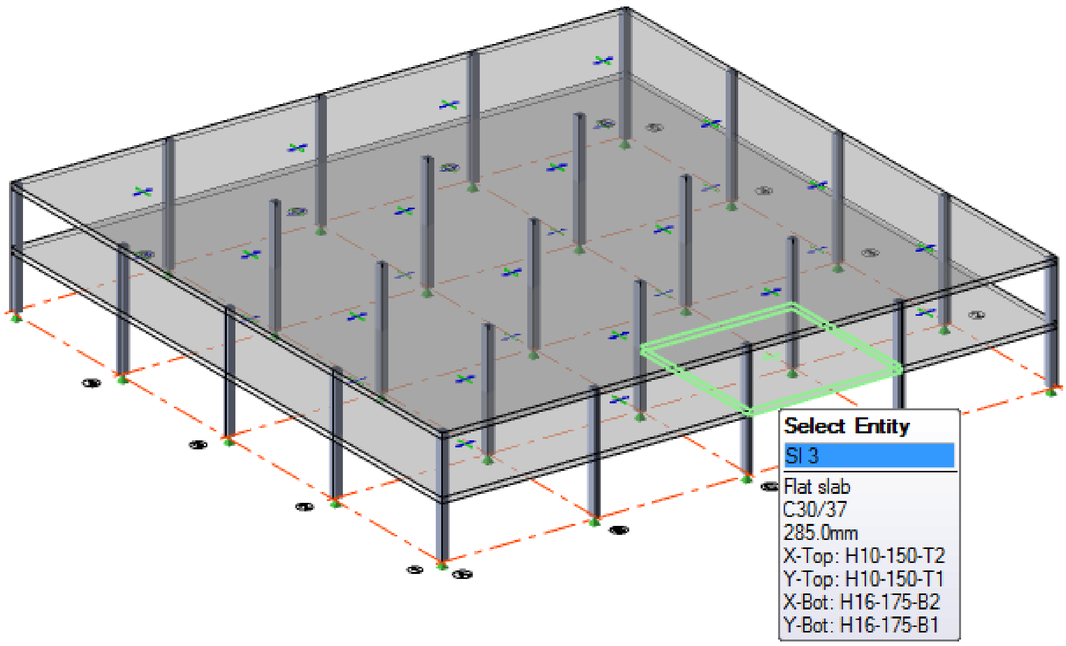


Related task: Display slab deflection status and utilization (page 940)

Slab deflection example (Eurocode)

In the following exercise slab deflections will be checked for the tutorial model shown below.

It is a simple multi-bay flat slab structure on an 8m square grid of columns. The slabs have been sized based on deemed-to-satisfy rules taken from "Economic Concrete Frame Elements to Eurocode 2 - The Concrete Society".



Geometry:

- 8m grid
- 285 thick C30/37 slab
- 400 square columns

Loading:

- Finishes - 1.5kN/m^2
- Imposed - 5.0kN/m^2
- Perimeter Cladding - 10.0kN/m

Deemed to satisfy checks (page 1427) provide one method of checking, however the main focus of the exercise will be to investigate rigorous slab deflection checks (page 1430).

Deemed to satisfy slab deflection checks example (Eurocode)

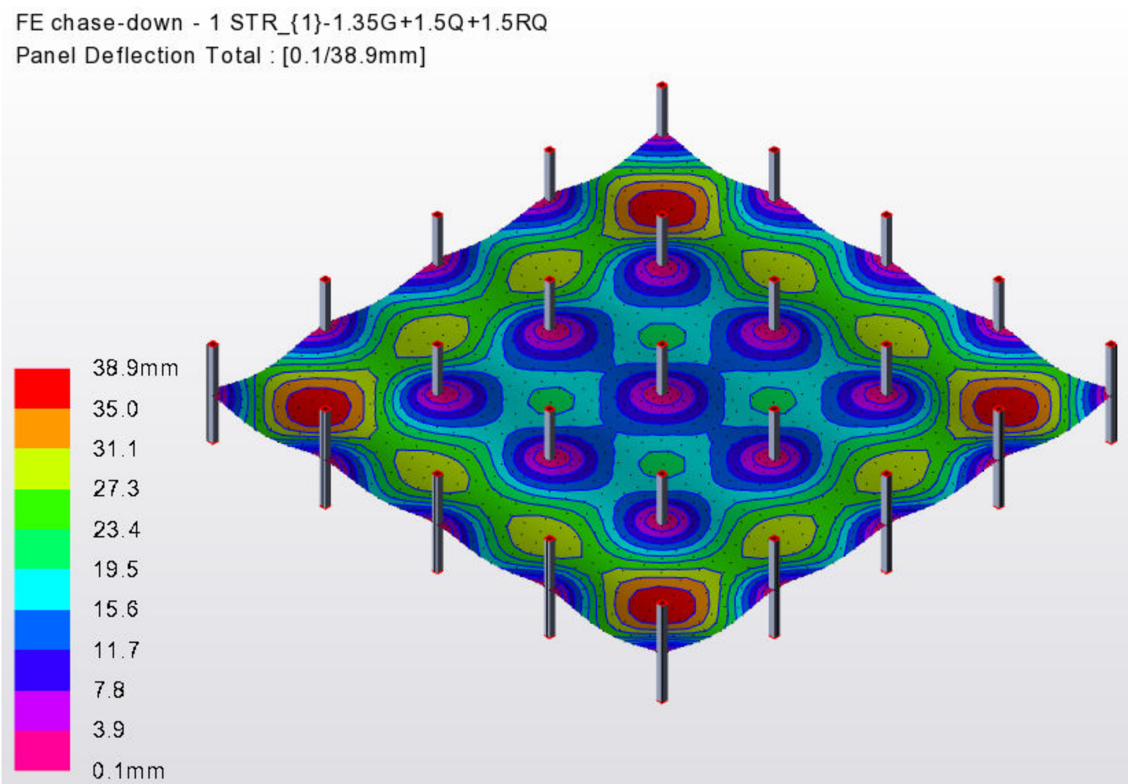
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results

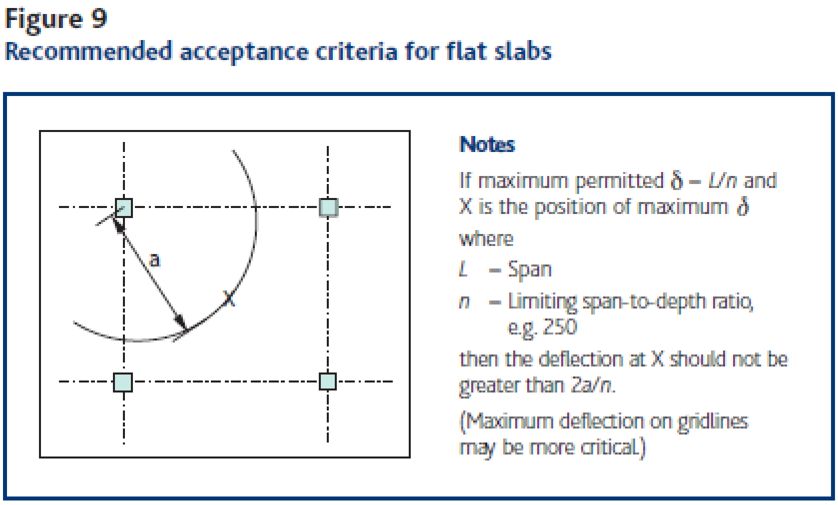


Identify critical check locations

We can see that the maximum reported deflection is 38.9mm, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs some engineering judgment might be required when assessing which deflections and span lengths require checking.

Guidance exists in How to Design Concrete Structures using Eurocode 2, The Concrete Centre, Figure 9



In our example, taking the diagonal dimension across the columns, the deemed-to-satisfy span / 250 rule provides a deflection of $[\sqrt{(8000^2+8000^2)}] / 250 = 45.3$mm.

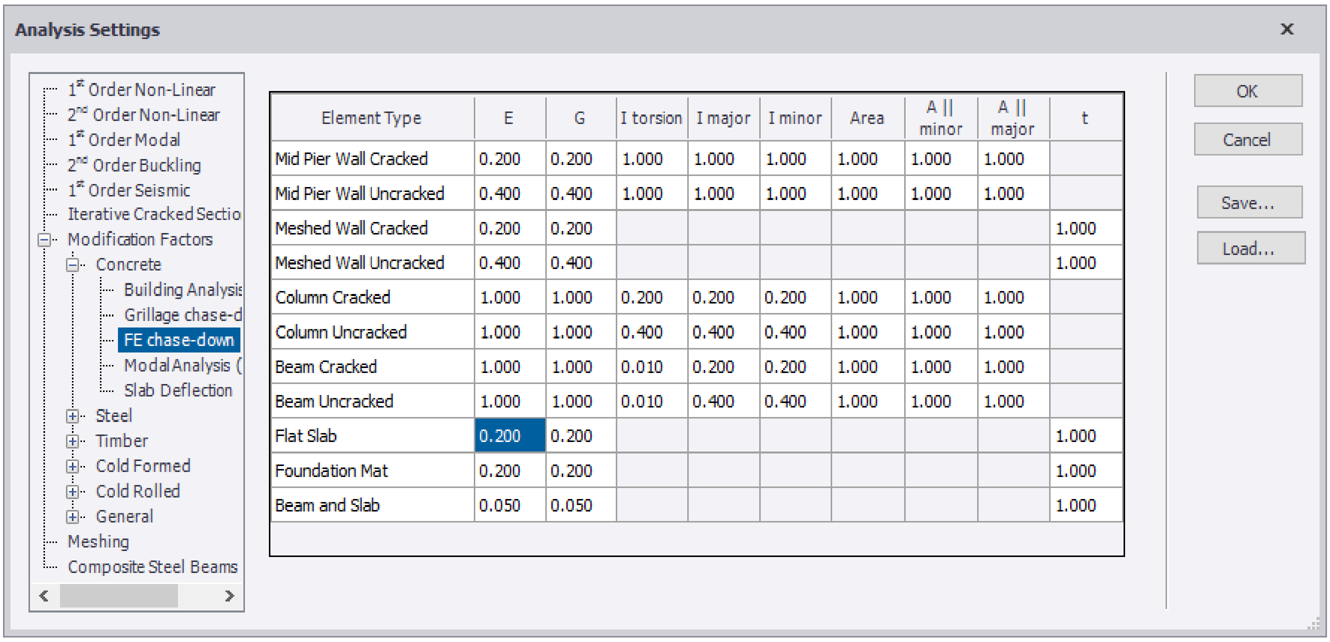
This compares favorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 250.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.2.



Rigorous slab deflection analysis example (Eurocode)

There are many input parameters that will have an impact on the rigorous deflection estimates and would therefore need to be considered. For details see: Factors that affect rigorous slab deflection estimates (page 1385)

For this exercise, initially it will be assumed that the default settings have already been reviewed and set as required.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**

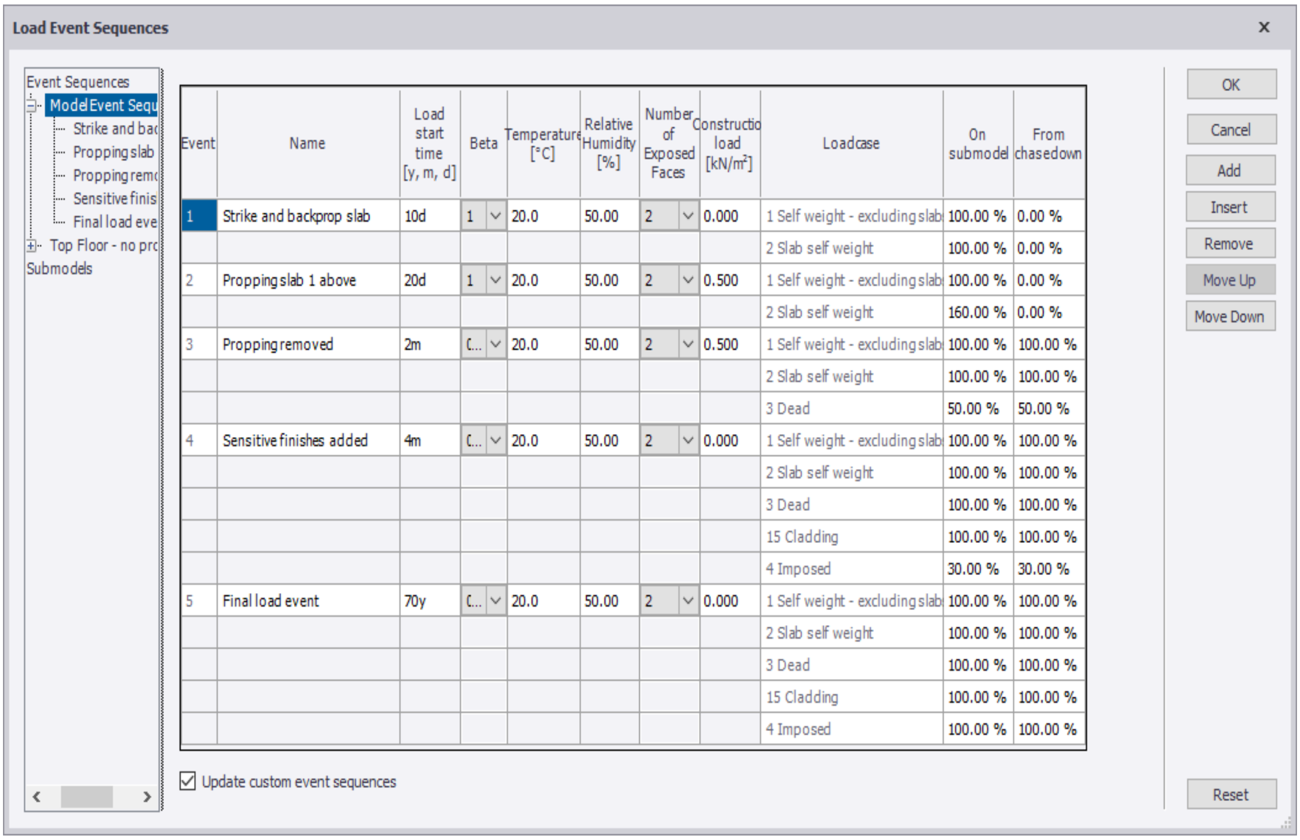
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Model Event Sequence

To review the model event sequence:

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

   This has already been setup for this example as shown below:



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	C...	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	C...	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	C...	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Deflections to the end of each of the above event periods will be calculated by the analysis.

3. Click **OK** to close the dialog.

Perform Iterative Slab Deflection Analysis

To establish some initial results (with all parameters left as default values) you could just click **Analyse All** from the Slab Deflection toolbar, however, in a real model rather than analysing all levels at once, it can be more efficient to work on just the current level, or a selected level. Obviously considering just a single level reduces the time necessary to undertake the iterative slab deflection analysis.

1. Open a 2D plan view of the level **Typical floor**
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.

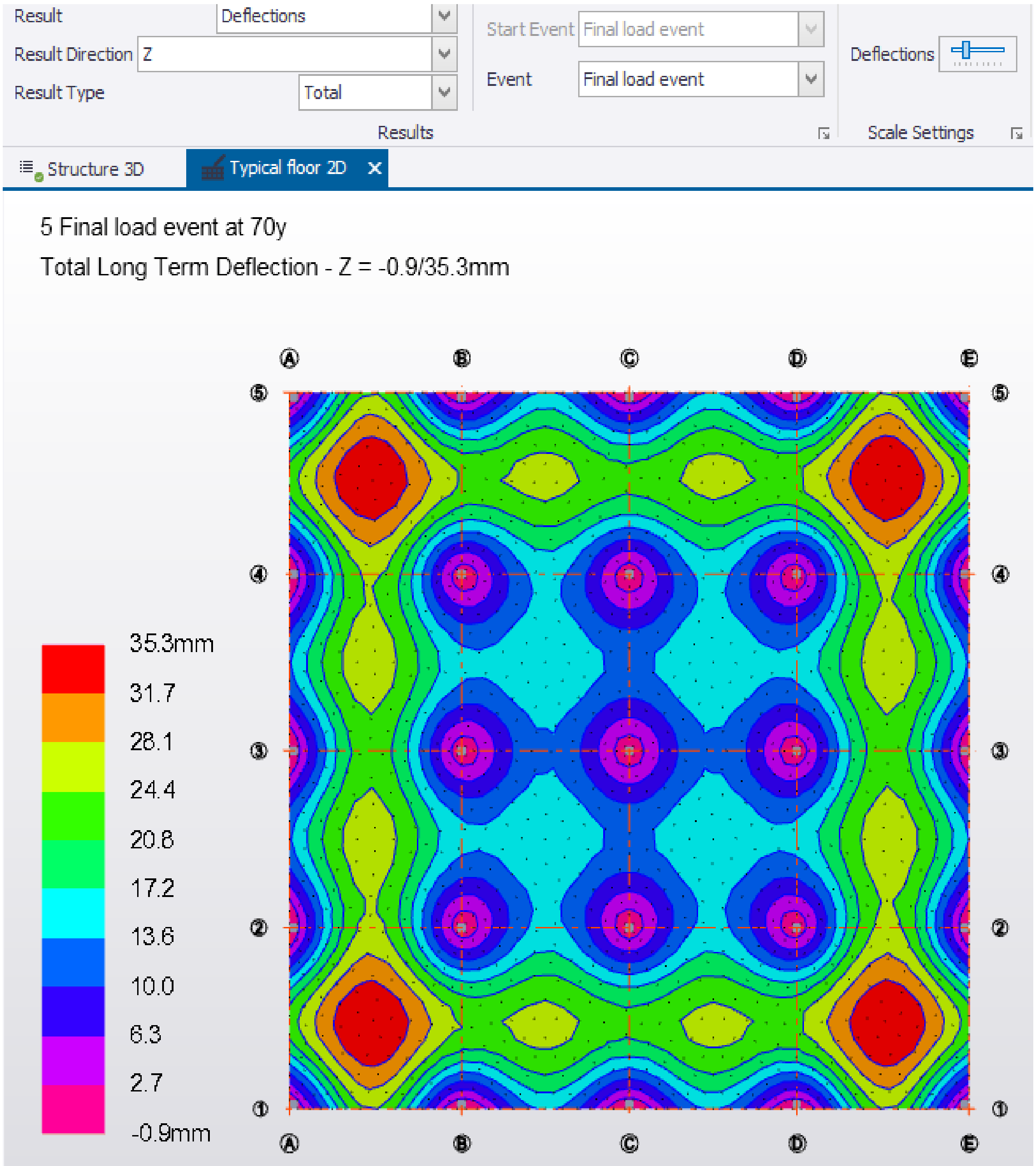
Review Deflections for Events

Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon.

You are able to review:

- Total deflection at the end of any event.
- Differential deflection between any two events (Start of Event and End of Event).
- Instantaneous deflection (not actually needed for TR 58).

The image below shows total deflection contours for the final load event.



NOTE To see the image as shown, you may need to adjust some of the selections in Scene Content, for example **Slab Patches** should be de-selected.

As a comparison with the simple approach earlier (39.7mm), the Total deflection at the final load event for the chosen location is 35.2mm.

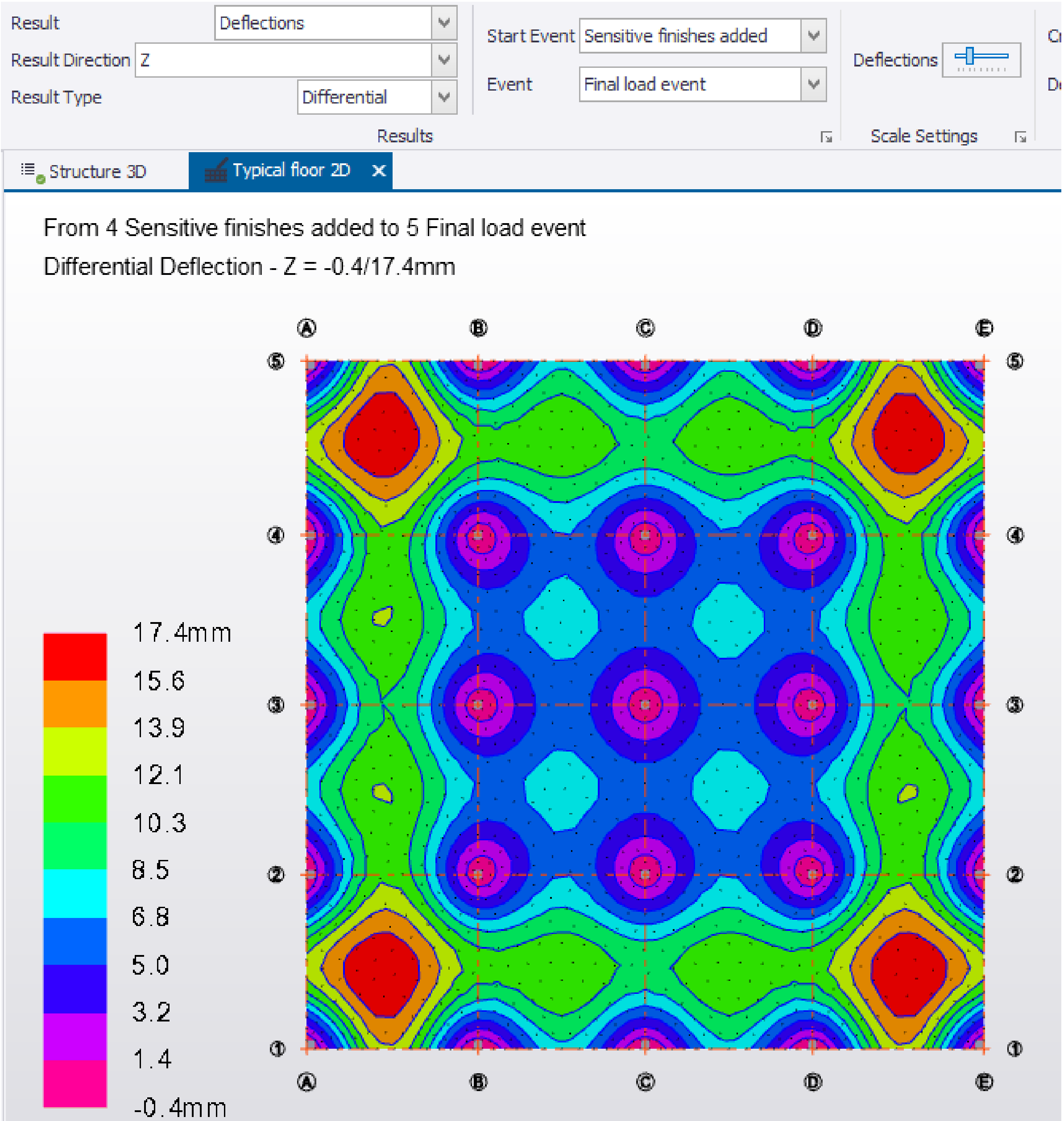
NOTE In the above contour plot the deflections are not exactly symmetrical - this is because reinforcement is in the outer layer in the Y direction making the slab stiffer in that direction.

Total deflections to the end of each of the event periods in the Event Sequence are available and could also be displayed as required.

In the Event Sequence there is an event for “Sensitive Finishes added” - we shall now show differential deflection between this and the final event.

1. From the **Slab Deflection** toolbar, change the Result Type to **Differential**
2. Select the Start Event as **Sensitive Finishes added**
3. Select the Event as **Final load event**

   The 2D view now displays the differential deflections between “Sensitive Finishes added” and the final event.


Result
Deflections
Result Direction
Z
Result Type
Differential
Results
Start Event
Sensitive finishes added
Event
Final load event
Deflections
Scale Settings
Structure 3D
Typical floor 2D
From 4 Sensitive finishes added to 5 Final load event
Differential Deflection - Z = -0.4/17.4mm
17.4mm
15.6
13.9
12.1
10.3
8.5
6.8
5.0
3.2
1.4
-0.4mm
A
B
C
D
E
1
2
3
4
5

Review Other Results

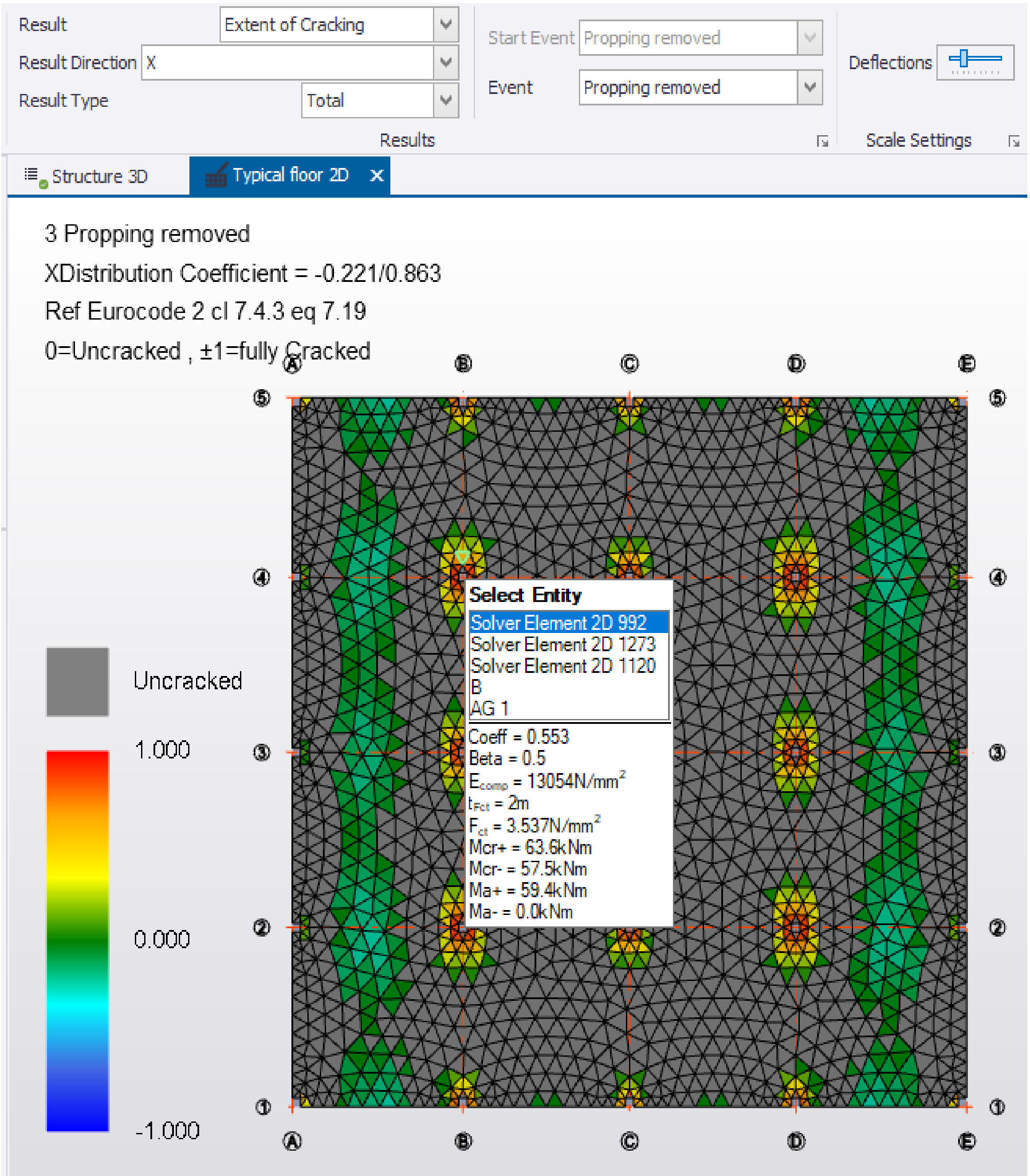
In addition to the deflections you can display the extent of cracking at any load event.

You can also review the relative stiffness in a particular result direction for any specified event.

You can also review the area of effective reinforcement for a chosen result direction for each FE element.

1. Select the Result as **Extent of Cracking**, and then:
   a. Select the Result Direction as **X**,
   b. Select the Result Type as **Total**,
   c. Select the Event as **Propping removed**,

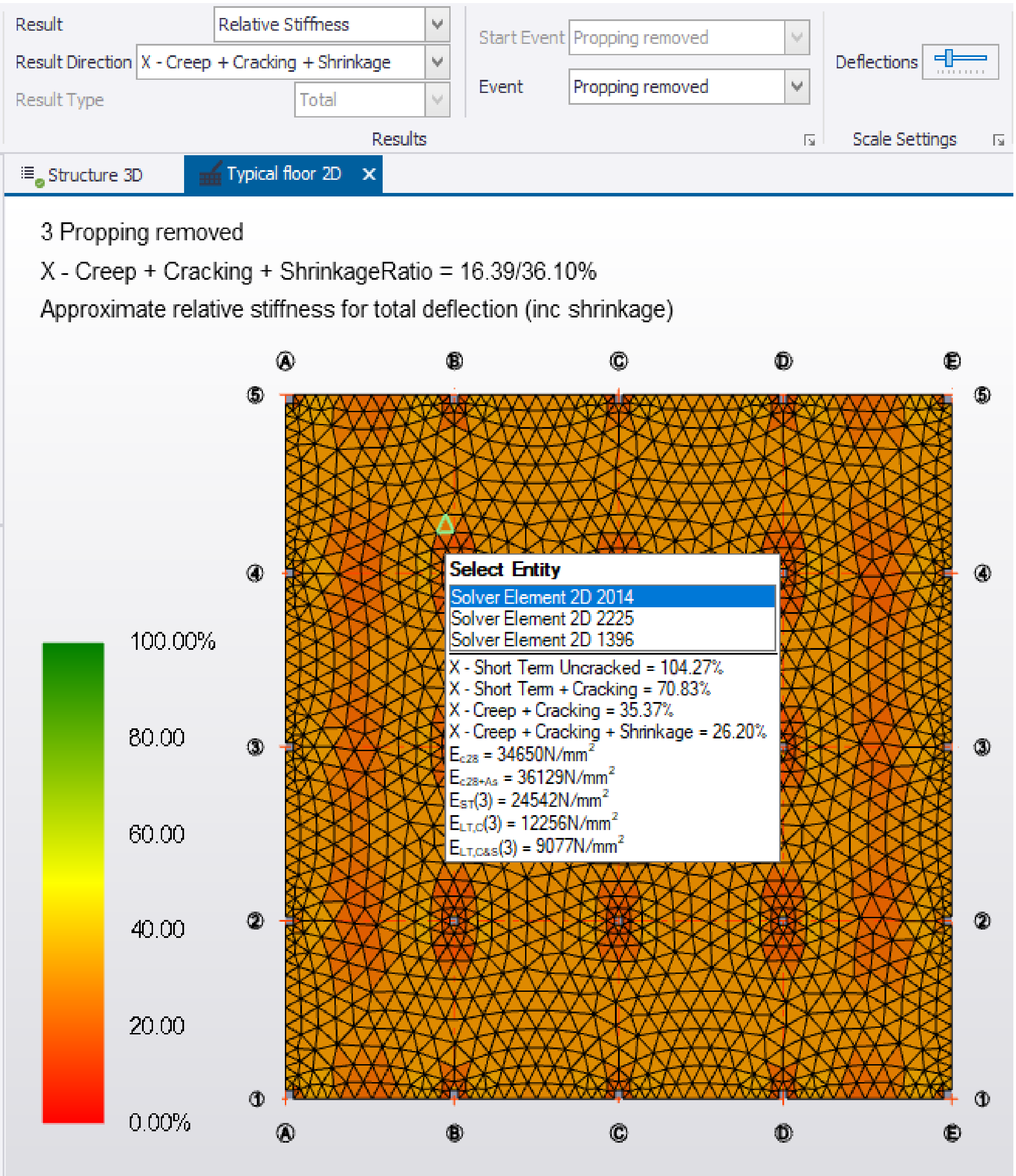
   The extent of cracking for the **Propping removed** load event is displayed.



2. Select the Result as **Relative Stiffness**, and then:

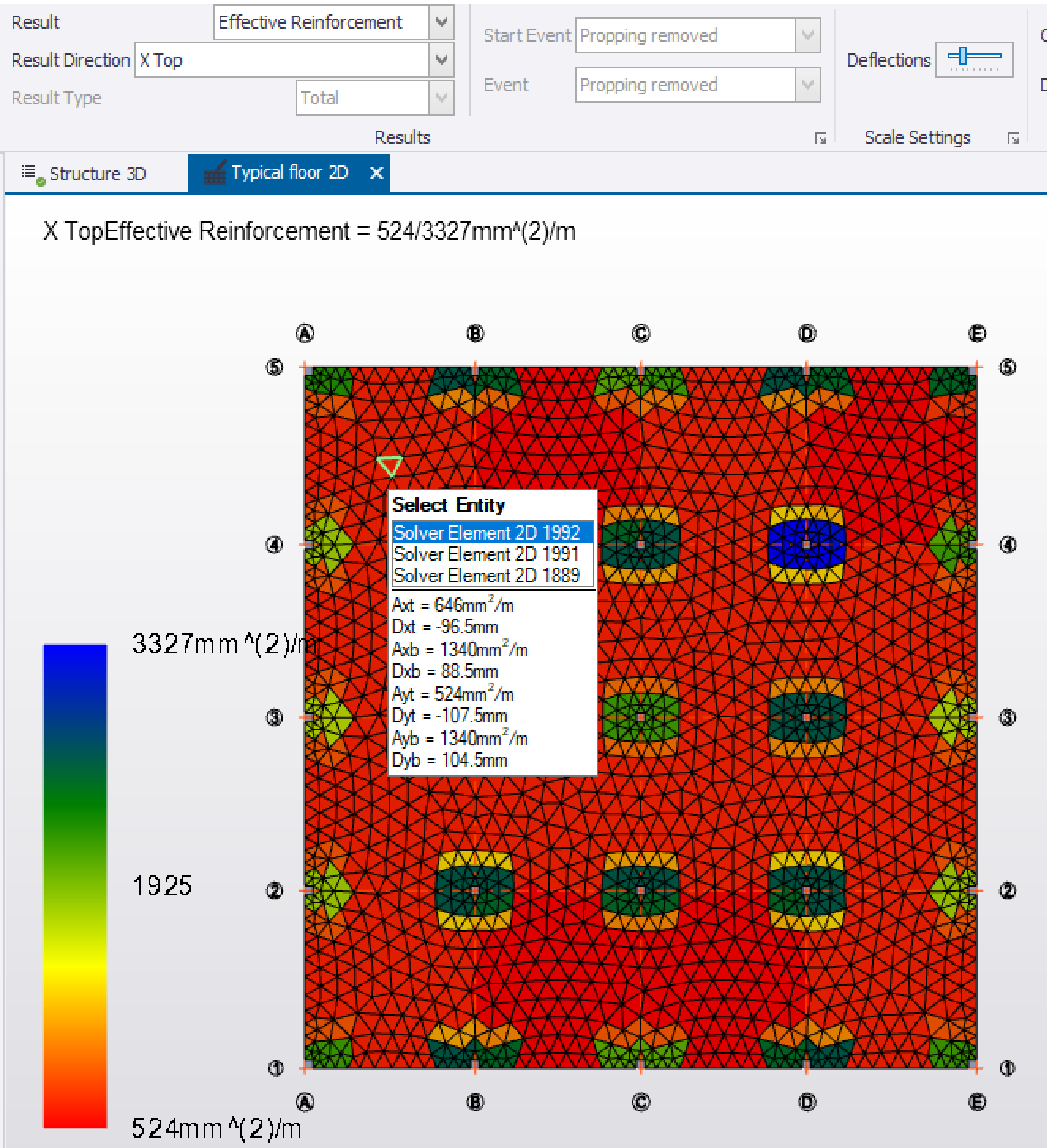
   a. Select the Result Direction as **X - Creep + Cracking + Shrinkage**,

   The relative stiffness for the **Propping removed** load event is displayed.



3. Select the Result as **Effective Reinforcement**, and then:
   a. Select the Result Direction as **X Top**,

   The effective reinforcement for the **X Top** direction is displayed.



NOTE When you hover over any FE element in the slab deflection view regime - values are provided within the tooltip.

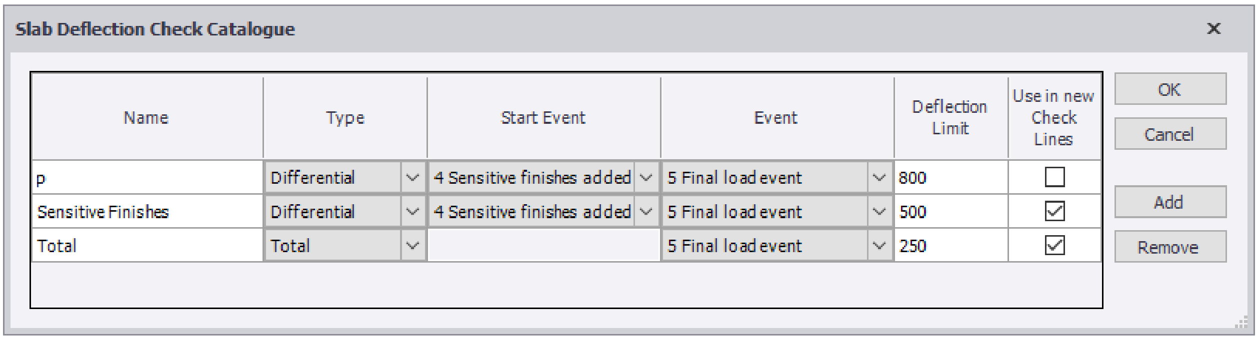
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the **Slab Deflection** toolbar, click **Deflection Checks**



   Whilst three checks have been defined above, only two of these have been set to be used in new Check Lines:

   - **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/500
   - **Total** will check the total deflections to the final load event against a deflection limit of 1/250

2. Click **OK** to close the dialog.

Place Check Lines

NOTE	Check lines can only be created in a 2D view.

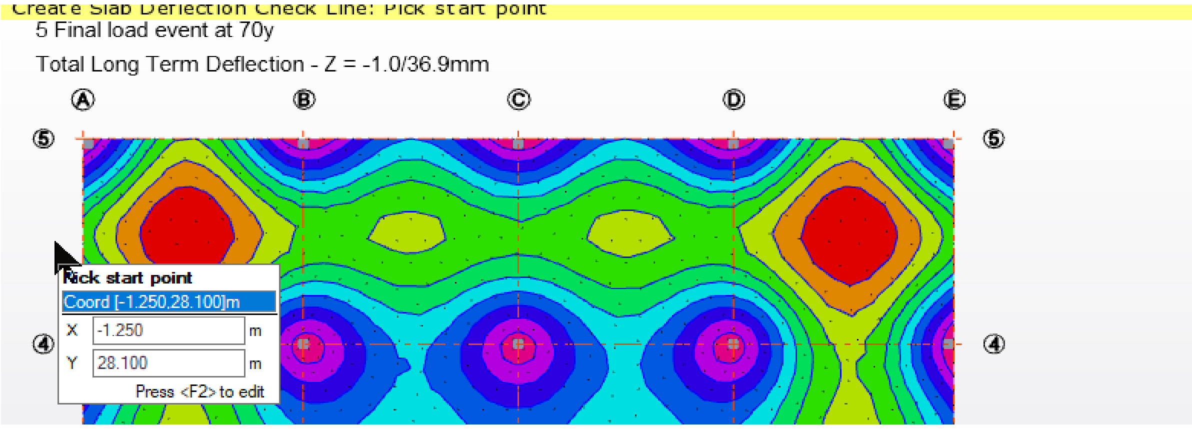
We will define several check lines in this example:

1. Ensure you are in the **Typical floor** 2D plan view and if necessary change:
   a. the Result back to **Deflections**,
   b. the Result Type to **Total**,
   c. the Event to **Final load event**
2. Click **Create**

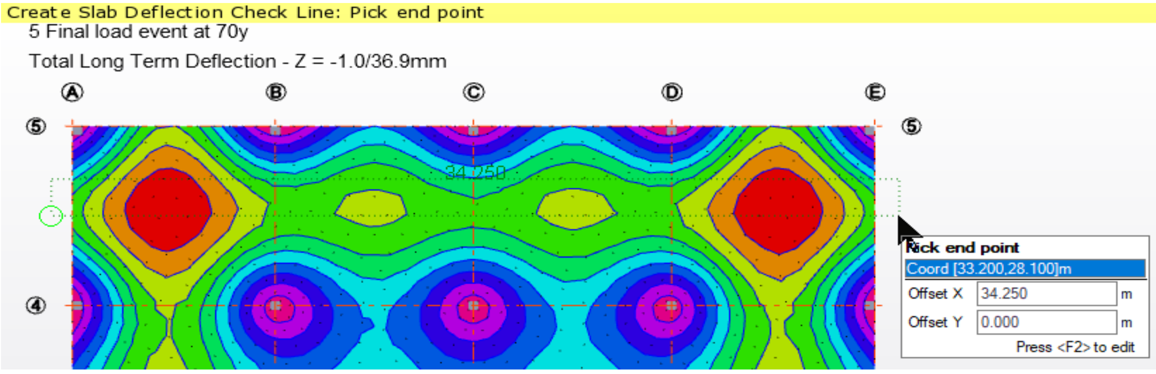
NOTE	When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which “Use in new Check Lines” was checked.

3. To place the check line at approximately mid-span between grid lines 4-5 from grid line A to E:

    a. Pick the start point to the left of grid line A as shown:

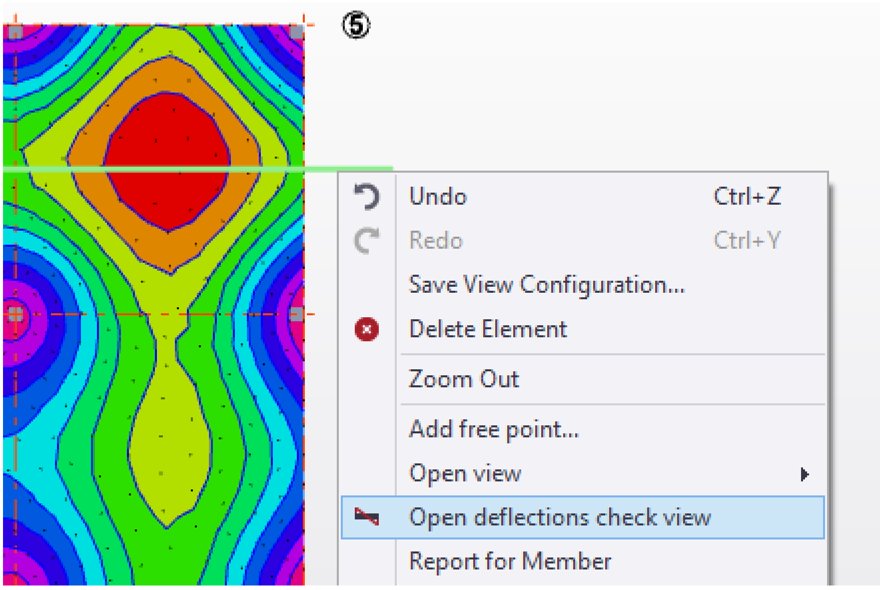


    b. Pick the end point to the right of grid line E as shown:

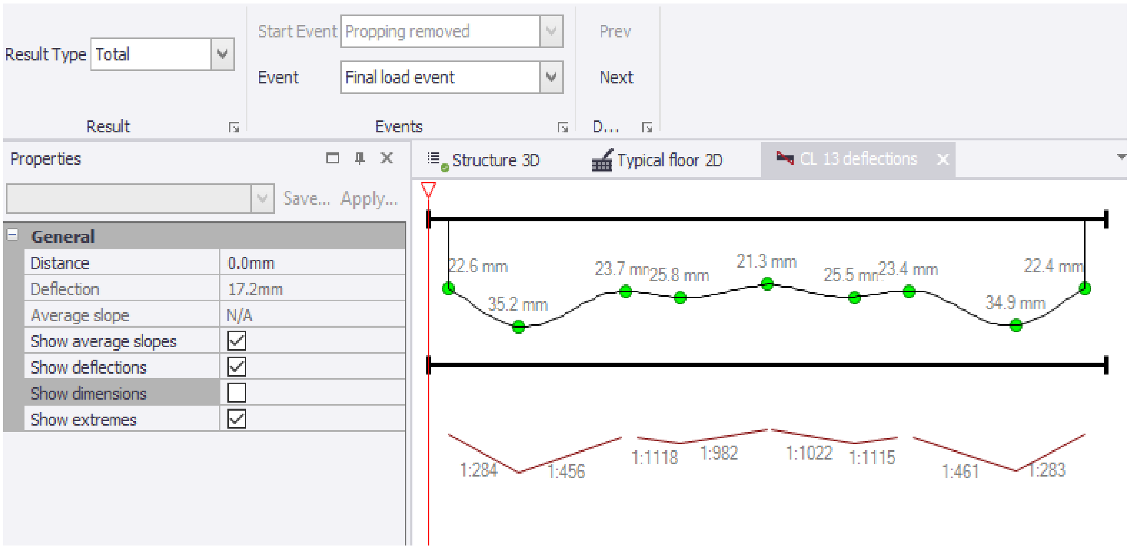


    c. Press **Esc** to end the command.

4. Right click on the check line and choose **Open deflections check view** from the context menu.



The deflection results along the length of the check line are displayed.

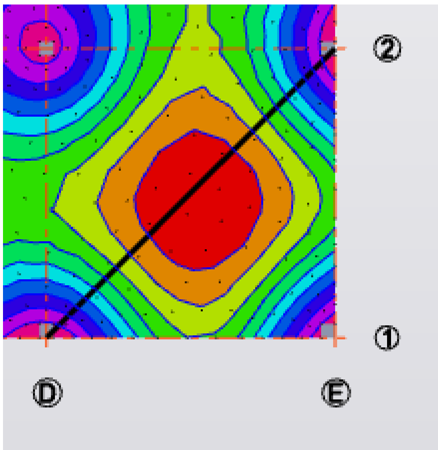


The **Slab Deflection** toolbar allows you to specify the total (as shown above), or differential or instantaneous results for the selected events.

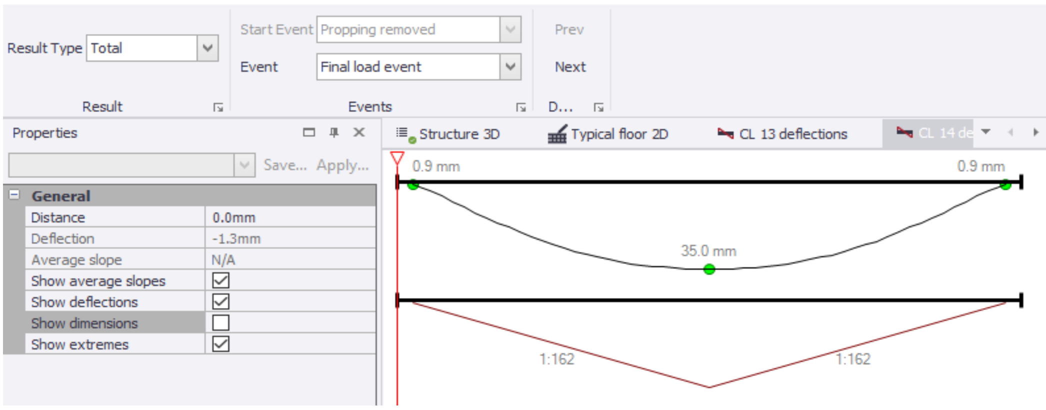
Tekla Structural Designer then draws average slopes between maximum and minimum points.

If we return to the original deemed-to-satisfy check - this was performed diagonally between columns in bottom right corner panel of the slab - we will now revisit this using a check line.

5. Create a check line running diagonally between columns in bottom right corner panel (from D/1 to E/2) where the peak deflection occurs.



6. Right click to open the deflections check view for the new check line.



A total deflection limit of Span / 250 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 125. In the view above the average slope between these points is 1 / 155, i.e. less than the limit.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Generate Check Line Reports

A tabulated report is available for each check line which itemises each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **Typical floor 2D** view, right click on the diagonal check line and select **Report for Member**

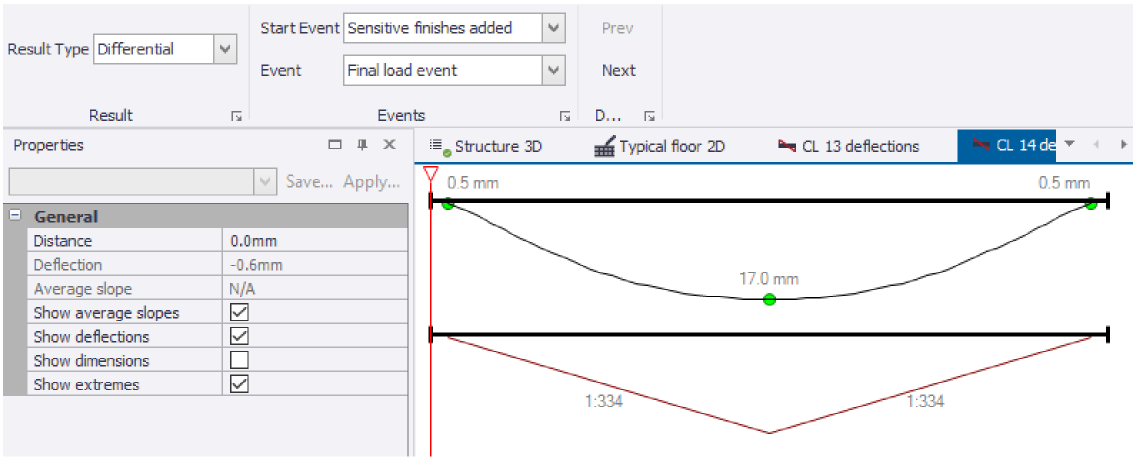
Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	16.5	5515.4	1 : 334	✓ Pass	0.749
Total	250	1 : 125	34.1	5515.4	1 : 162	✓ Pass	0.772

As previously noted, the slope above is reported as 1:155 which is not less than the allowable slope limit of 1:125 and hence a Pass.

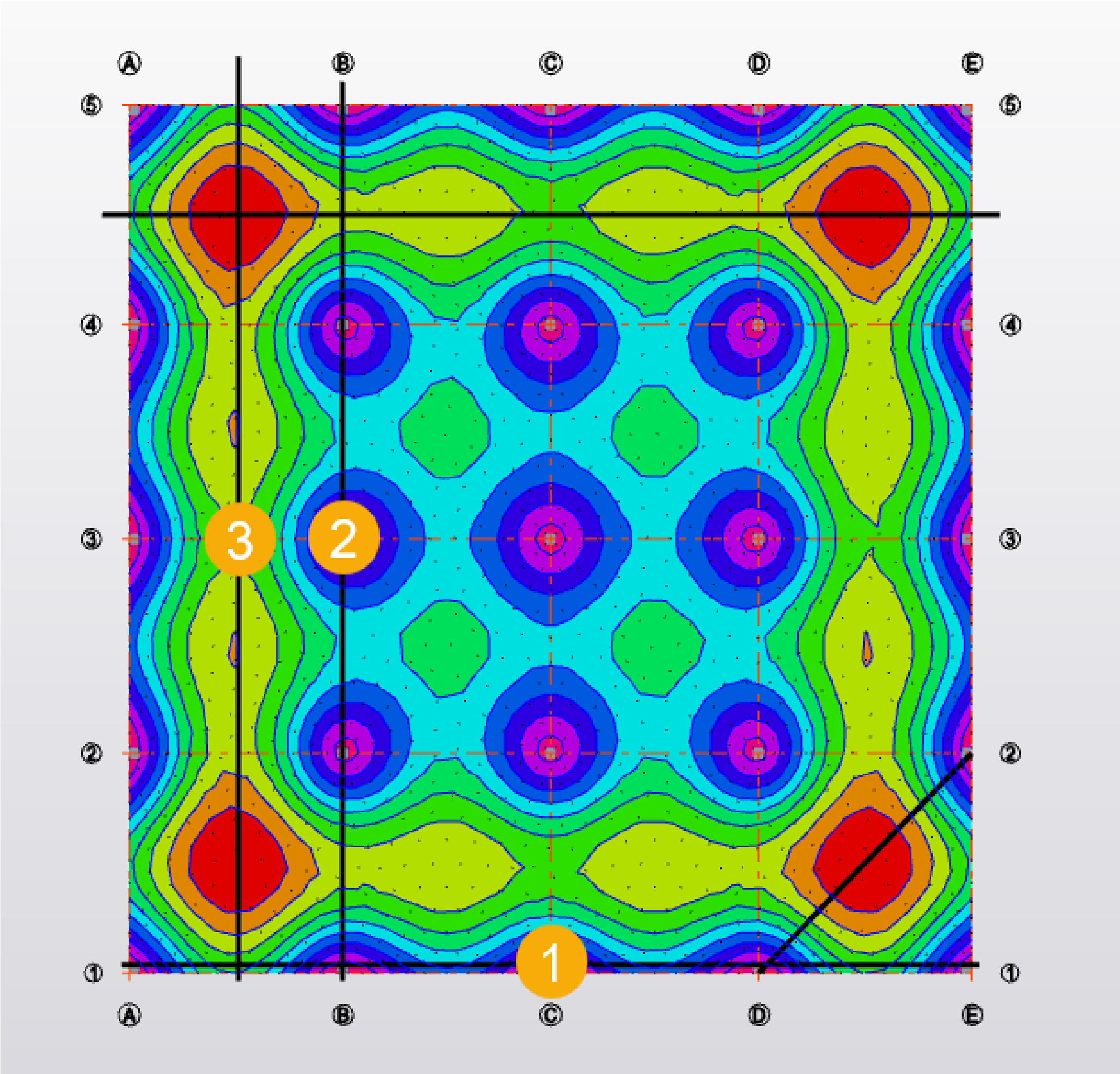
If you click within the Load Analysis Deflection view, the ribbon changes to allow you to display deflection results and slopes for the Result Type - Total or Instantaneous for a chosen event, or Differential between chosen events.

2. Switch to the Load Analysis Deflection view for the diagonal check line.
3. From the **Slab Deflection** toolbar, change the Result type to **Differential** and check deflection and slopes between the**Sensitive finishes added** and **Final load event**.



We can add as many check lines to the model as we consider appropriate.

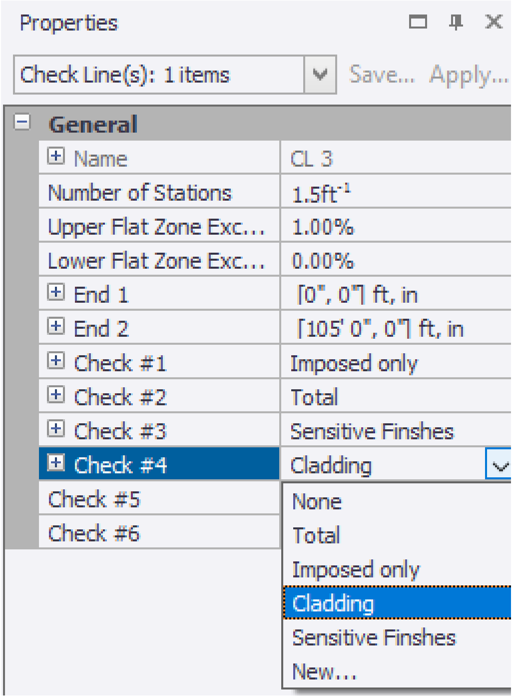
4. Return to the **Typical floor 2D** view.
5. Add three further check lines using the default deflection checks in the catalogue as follows:
    a. along grid line 1, from A/1 to E/1
    b. along grid line B, from B/1 to B/5
    c. half way between grid line A and B, starting at grid line 1 and finishing at grid line 5



6. Press **Esc** to end the command.

    If you select each Check Line in turn you are able to edit the deflection checks associated with it in the Properties Window.

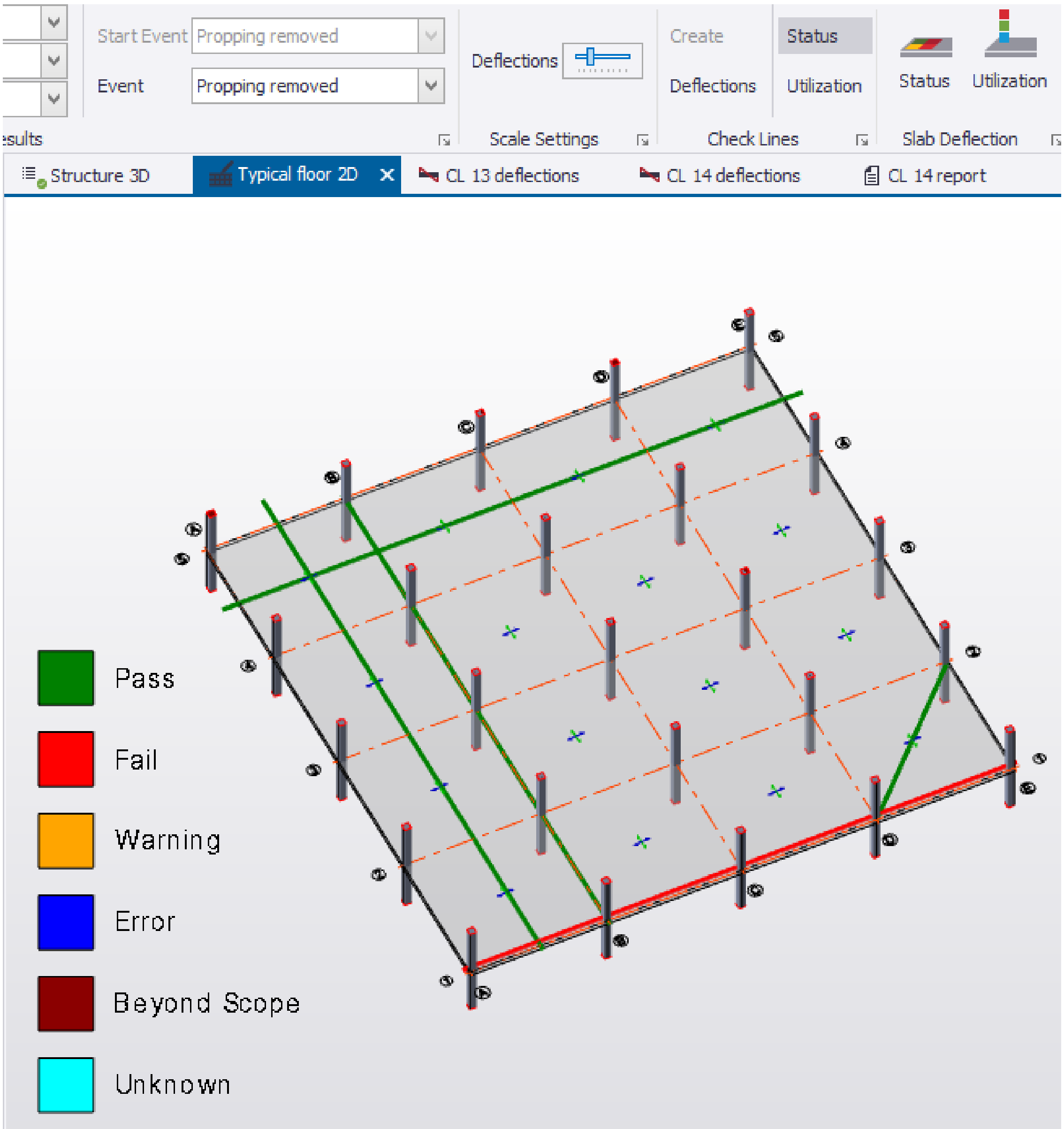
7. Select the check along grid line that runs along grid line 1 and ensure that it also has a **Check #3** defined as **Cladding**.



Review Check Line Status and Utilization

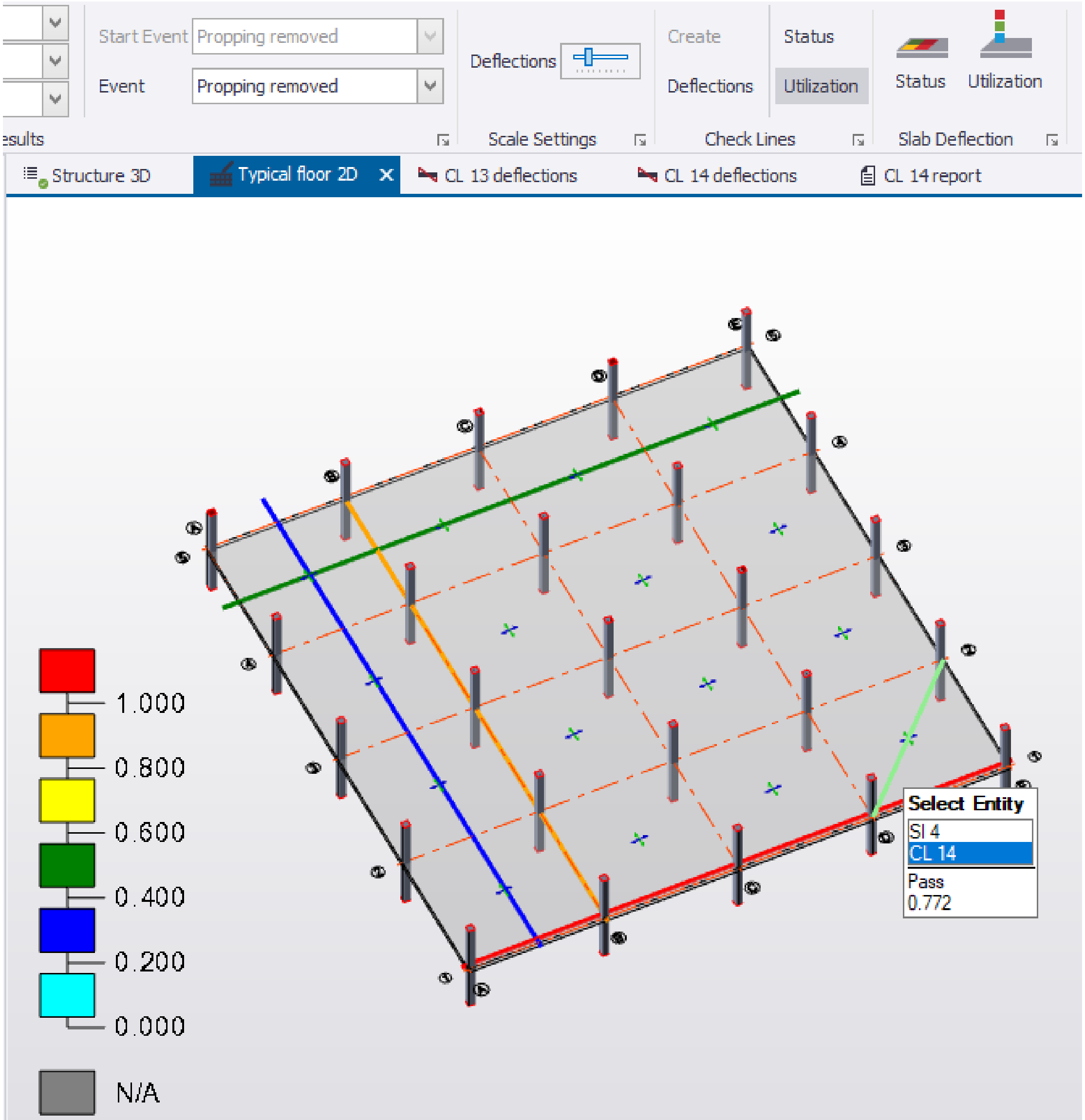
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the Typical floor 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



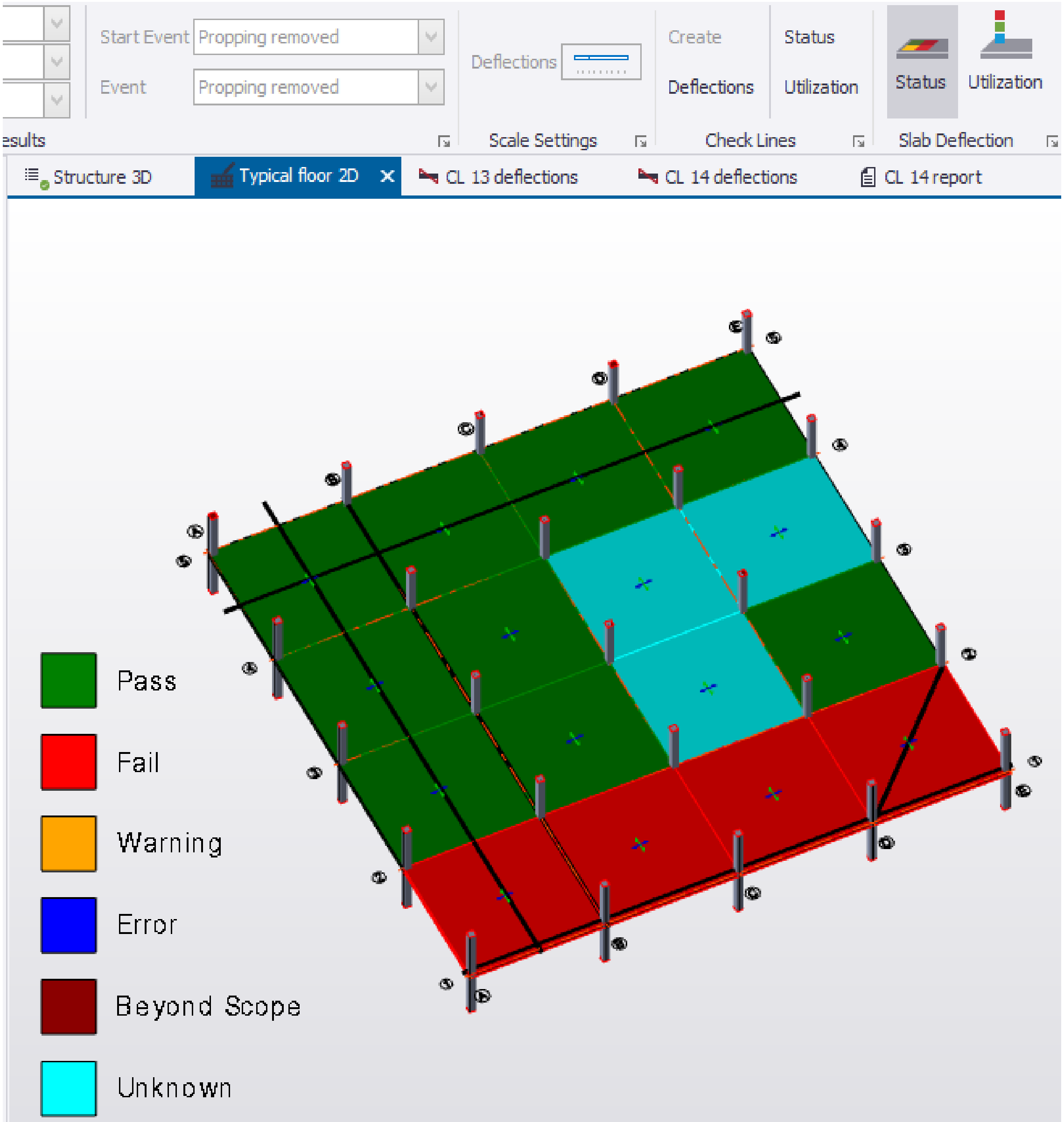
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/ fail status of each slab.

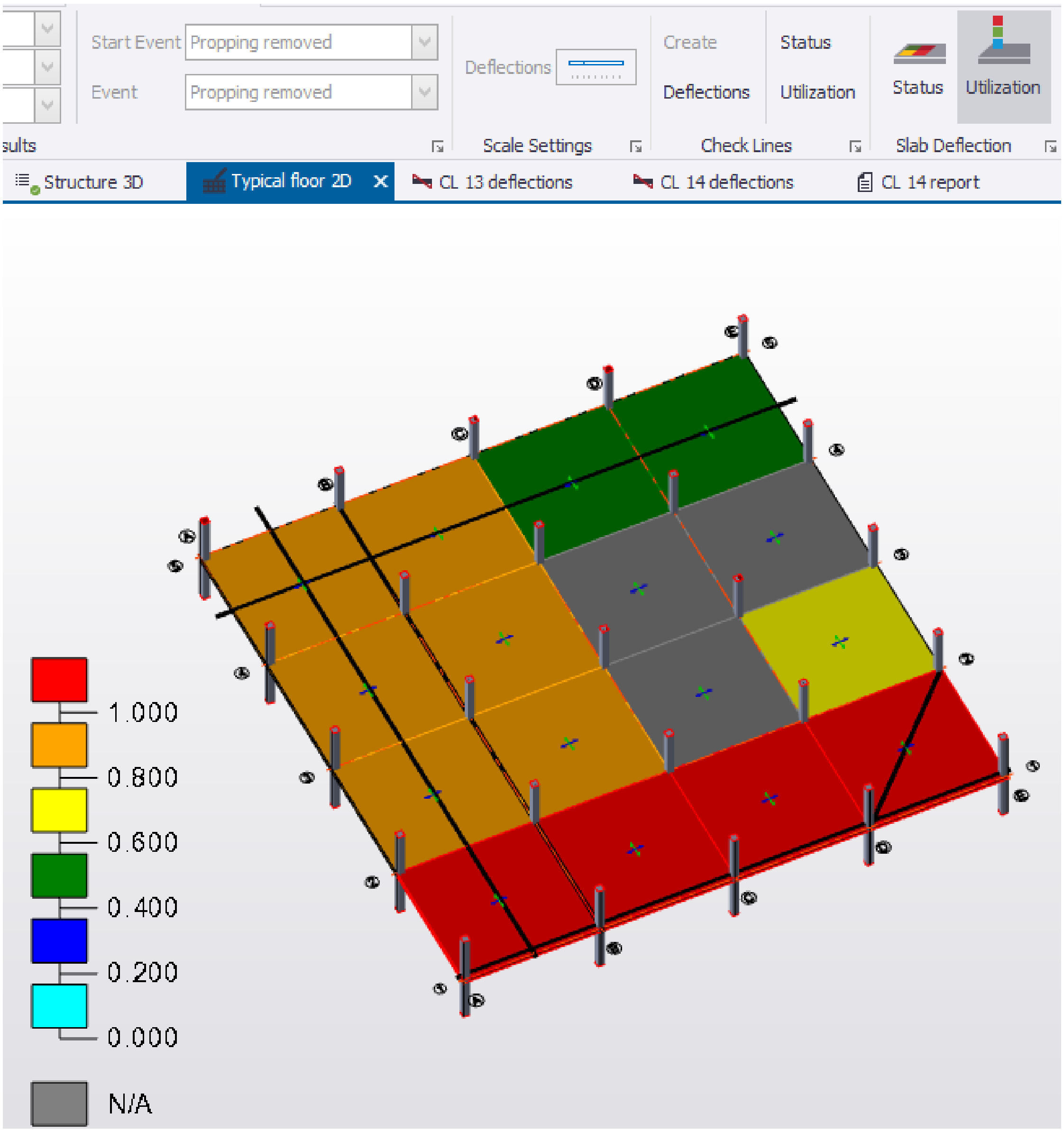


- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed.
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.

- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the slab items Fail where the check lines runs along A/1-E/1.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

   This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

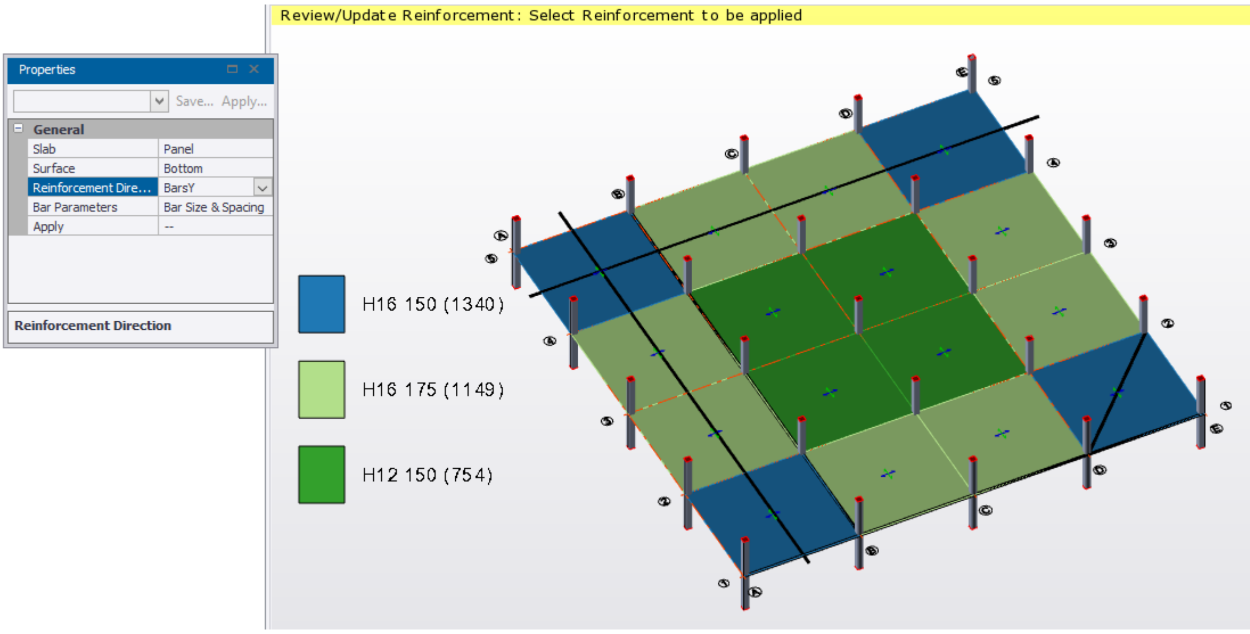
The analysis is extremely quick and since everything is contained within one model file, it allows “What If” scenarios to be considered to find the optimum solution.

In this exercise we will start by looking at the impact of adjusting the reinforcement.

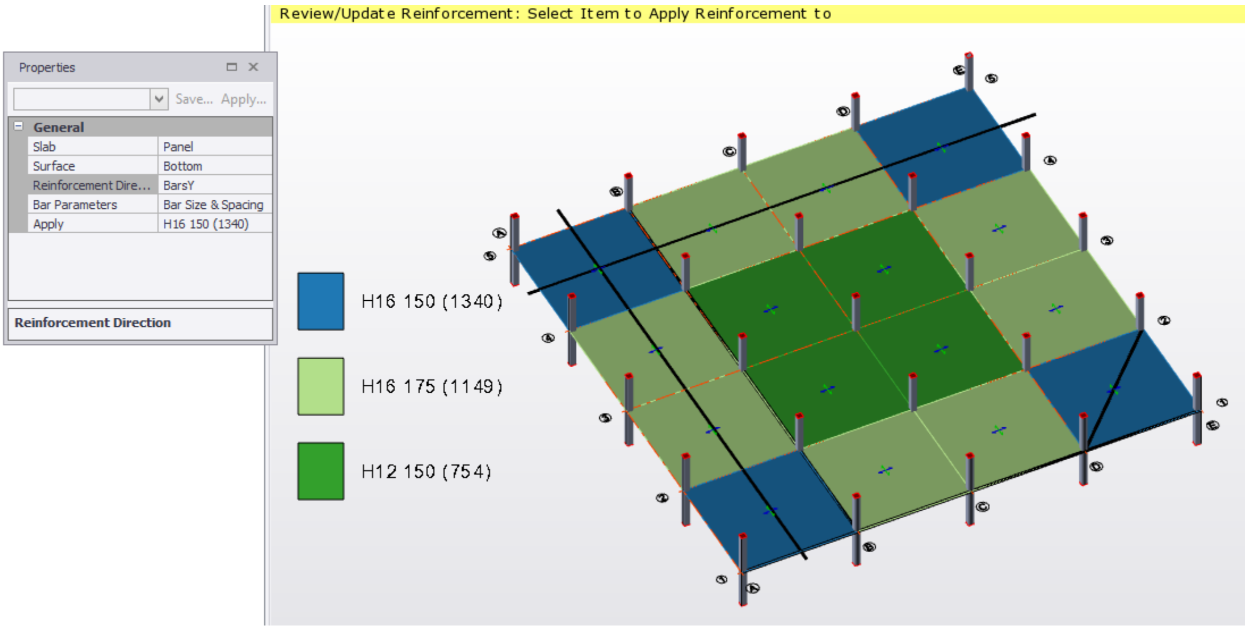
1. Highlight the check line running along grid line B and make a note of its utilization.

   TIP Press the tab key if necessary to highlight the check line when it is directly under the mouse cursor.

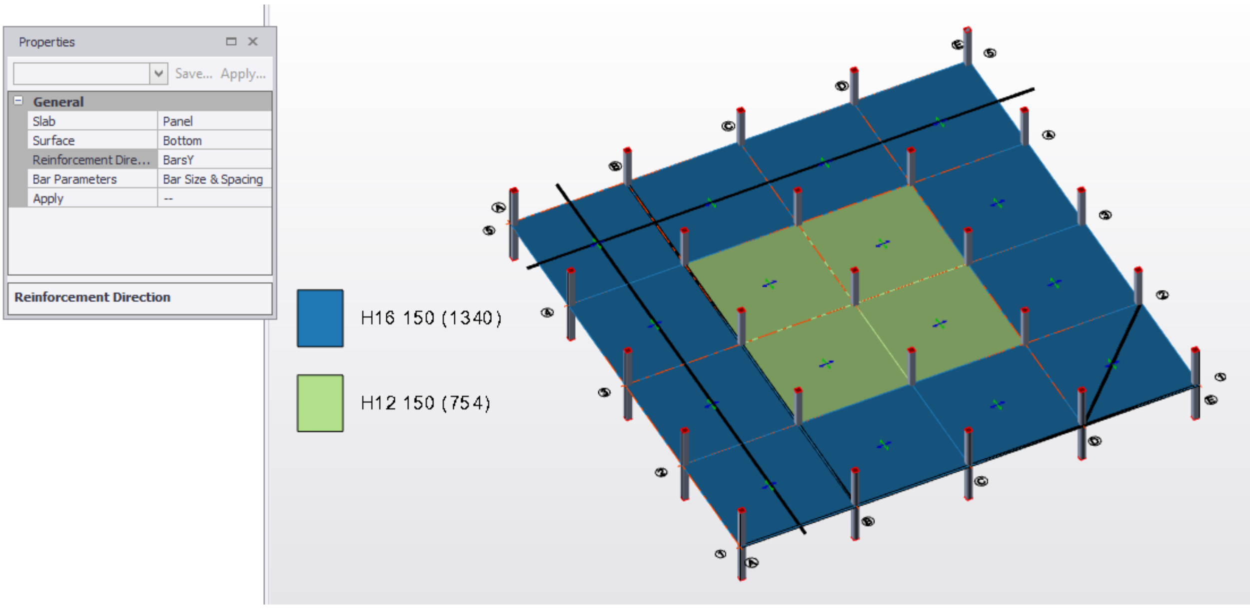
2. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.
3. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



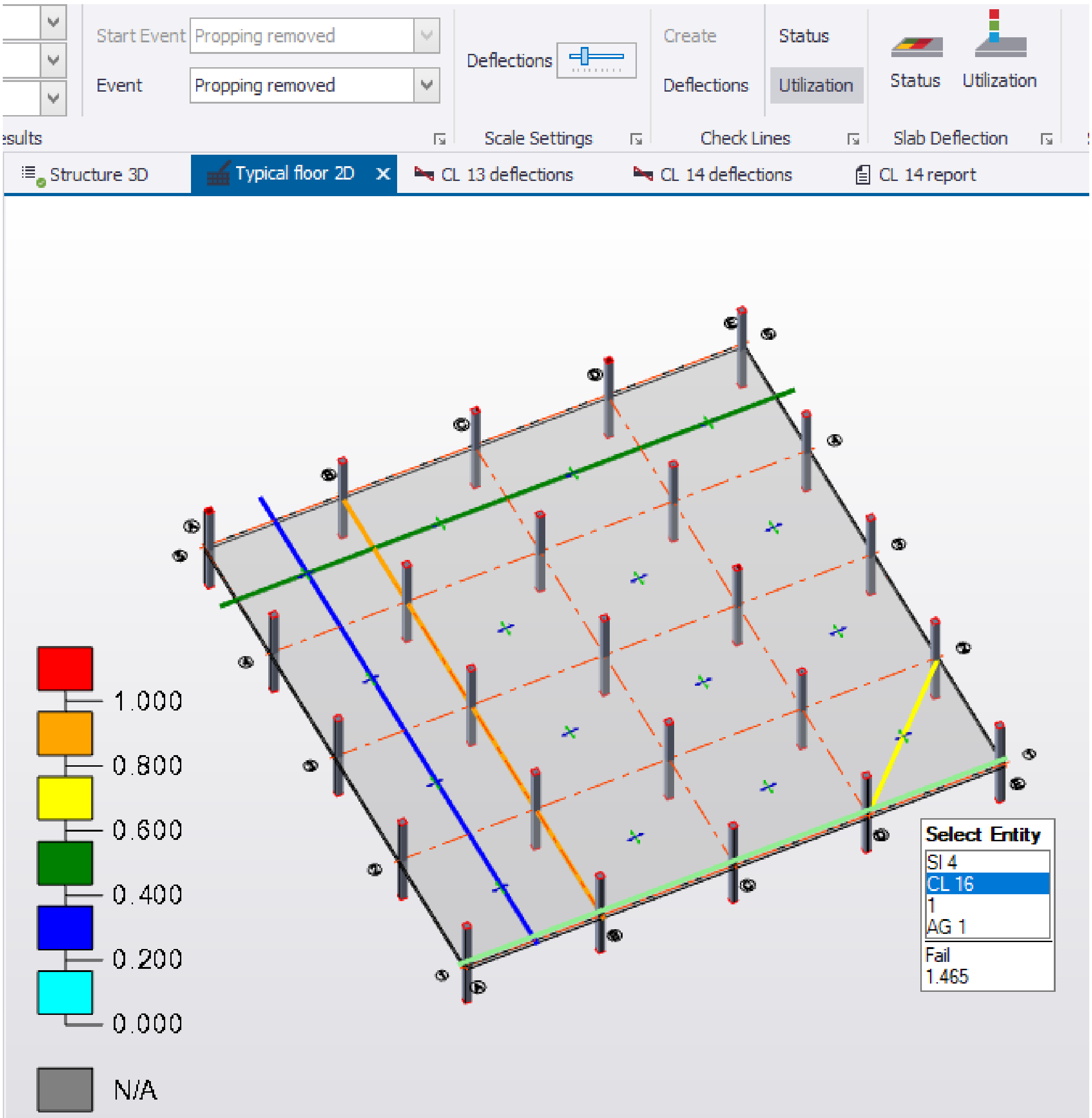
4. Click on one of the corner slab panels to select **H16 150** as the reinforcement to be applied.



5. Click on the eight slab panels currently showing H16 bars at 175 spacings to change them to the 150 spacings.



6. Change the Reinforcement Direction to **BarsX** to see the bars in that direction.
7. Edit the same 8 panels in exactly the same way, so that the bars in X match the bars in Y.
8. Click **Analyse Current** to update the results
9. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
10. Investigate the tooltip results

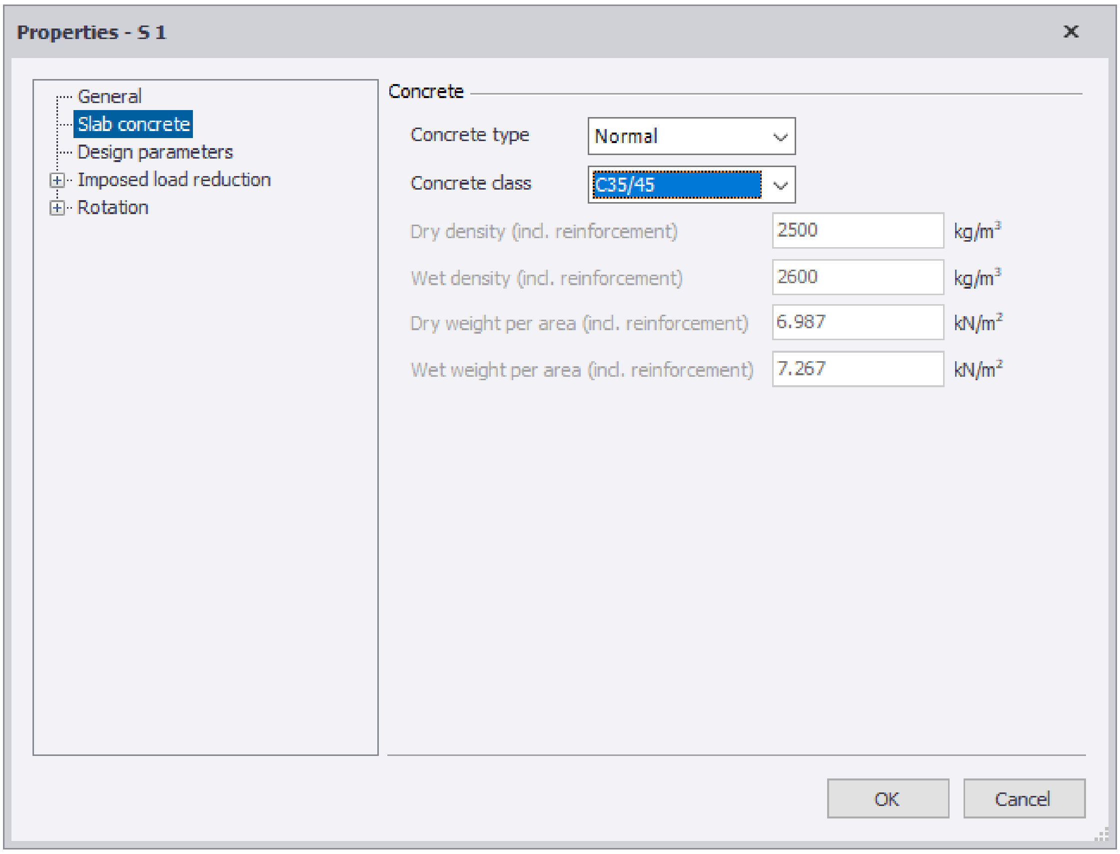


You should find that although some utilizations have reduced, the line along grid line B is still failing. At this point you could begin to look at the impact of the various other input parameters. For now, we will adjust the concrete grade of the typical floor slab group from C30/37 to C35/45.

11. Right click slab **Sl 4** between D-E/1-2 on the typical floor level and choose **Edit....** from the context menu.

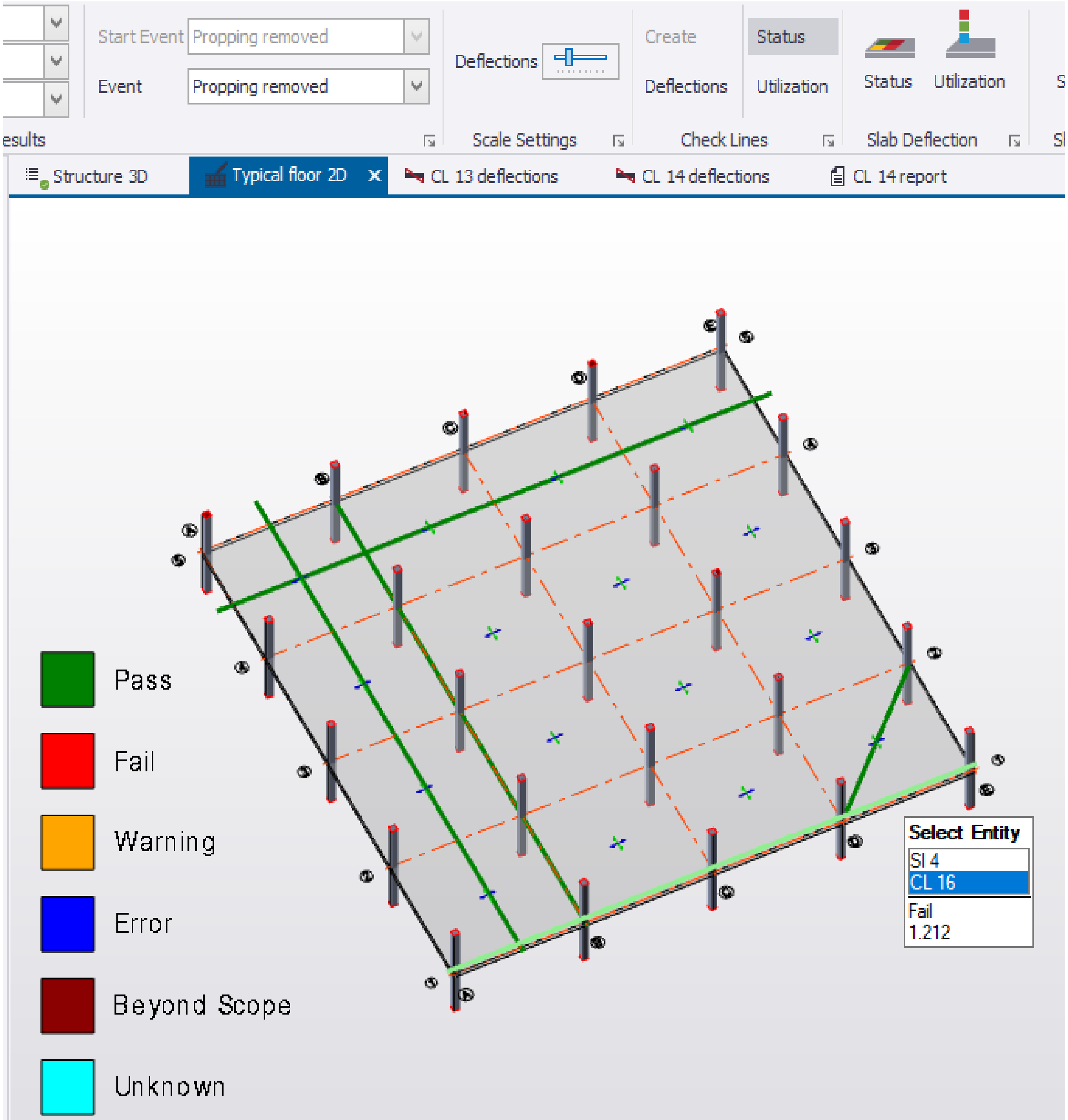
NOTE Editing a slab item via the right click context menu updates the parent slab properties, not just the individual slab item. Hence any changes will be applied to all the slab items in S 1.

12. On the **Slab concrete** page of the dialog, change the Concrete Strength to **C35/45** and click OK



To update the check line results we need to re-run the analysis. A chasedown analysis is automatically performed as part of the slab deflection analysis, however, it should be borne in mind that some edits could affect the element design i.e. reducing the concrete slab thickness would result in an increase to the required reinforcement and hence a Design Concrete (Static), Slab or patch design may be required again.

13. Click **Analyse Current** again to update the results.
14. Review the Check Lines **Status**.



The check line with the more onerous Cladding deflection limit is still failing, but we can clearly see improvements in the results. It has reduced from a Utilization ratio of 1.538 to 1.262.

In fact, to obtain a Pass we would have needed to increase the concrete grade to C45/55.

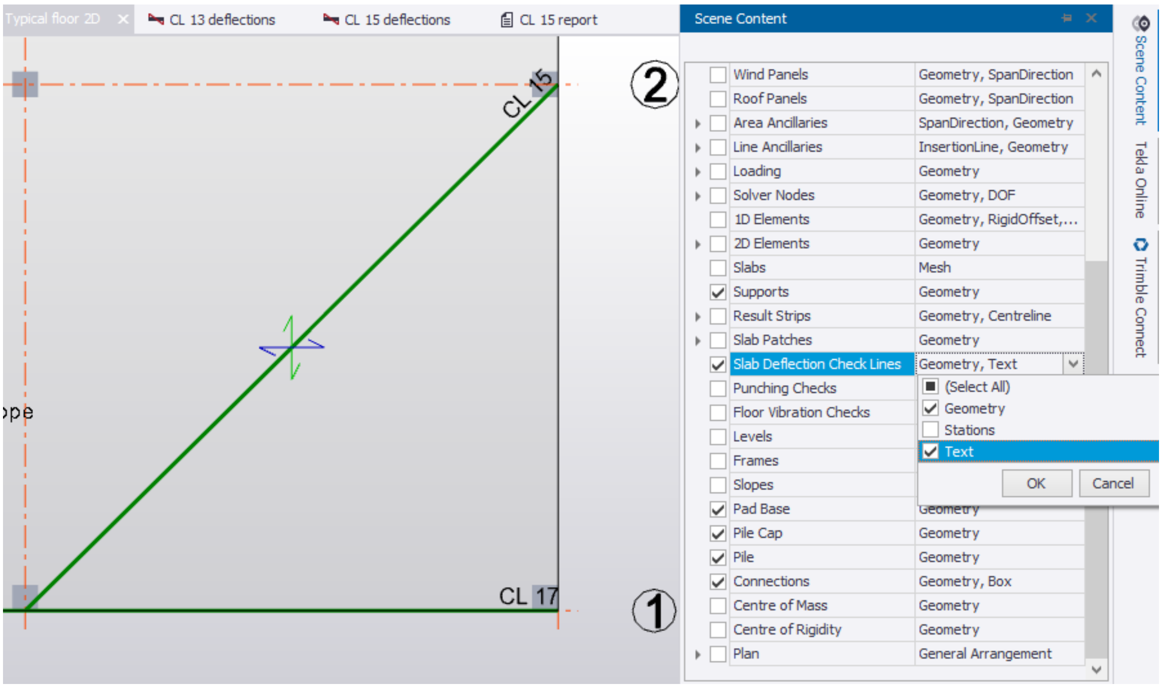
NOTE For the purpose of this exercise, as the cladding check is not a code requirement we will simply disable it before proceeding to generate the model report.

15. Select the check line along grid 1 and then in the **Properties window**, reset Check #3 to **None**.

Generate Model report

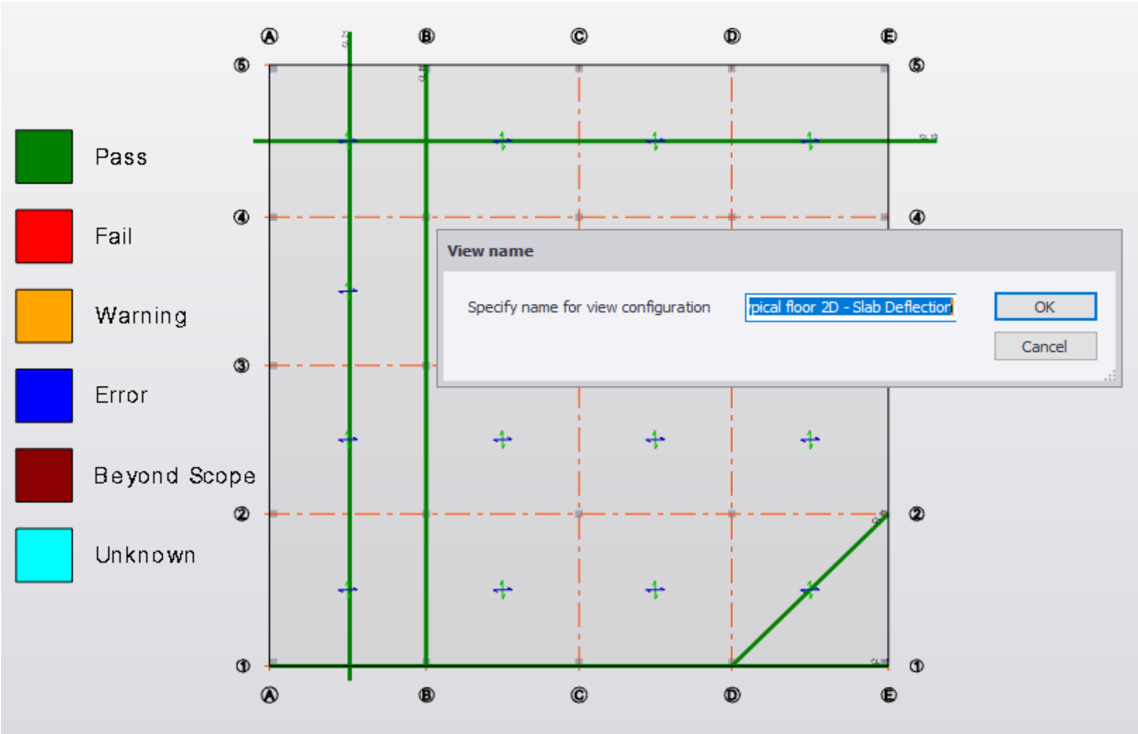
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



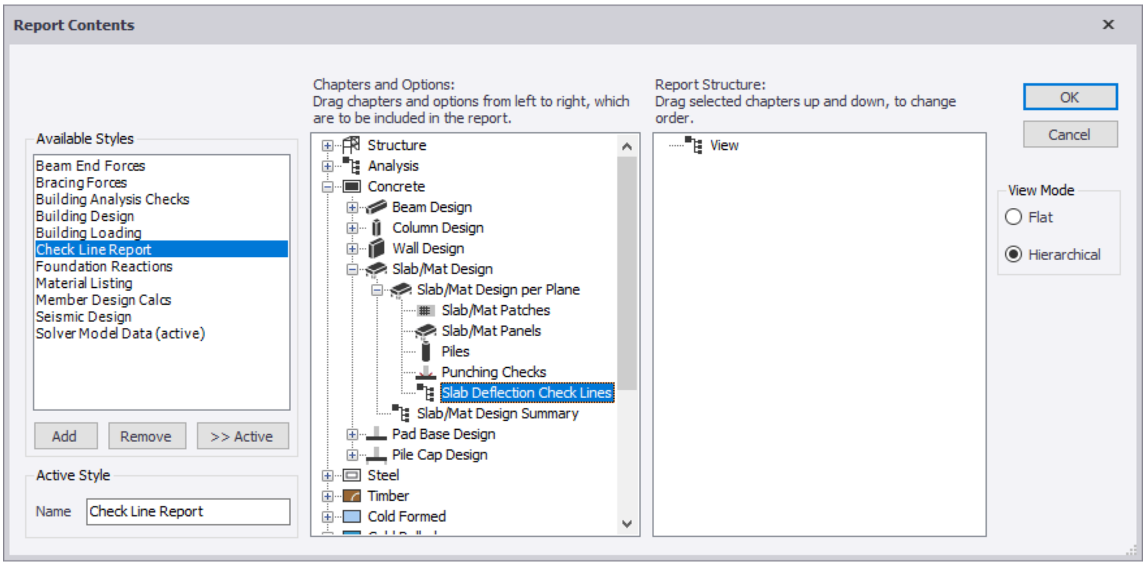
TIP The check line references can be customised using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

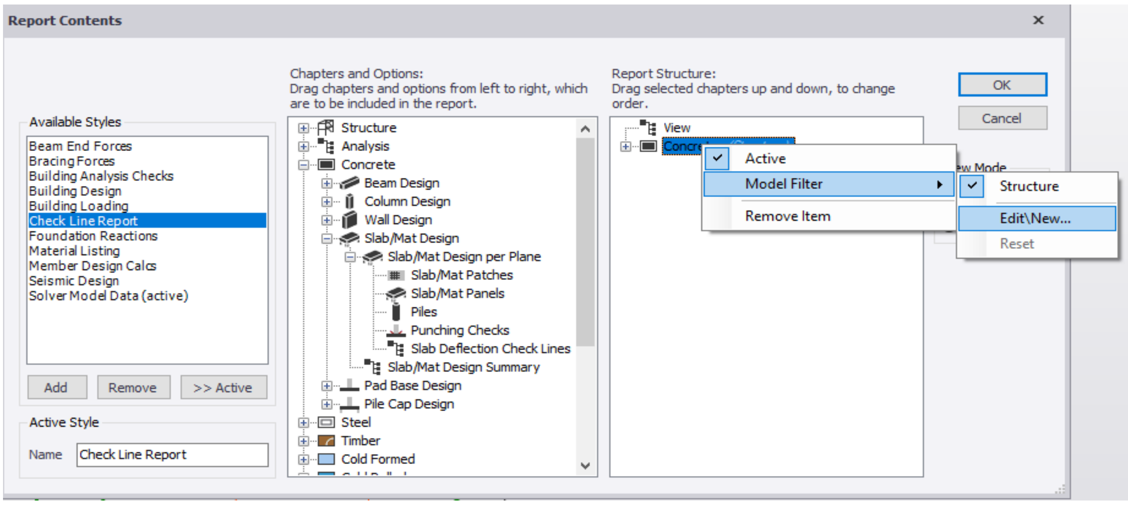


This saved view can be included in the Check line report.

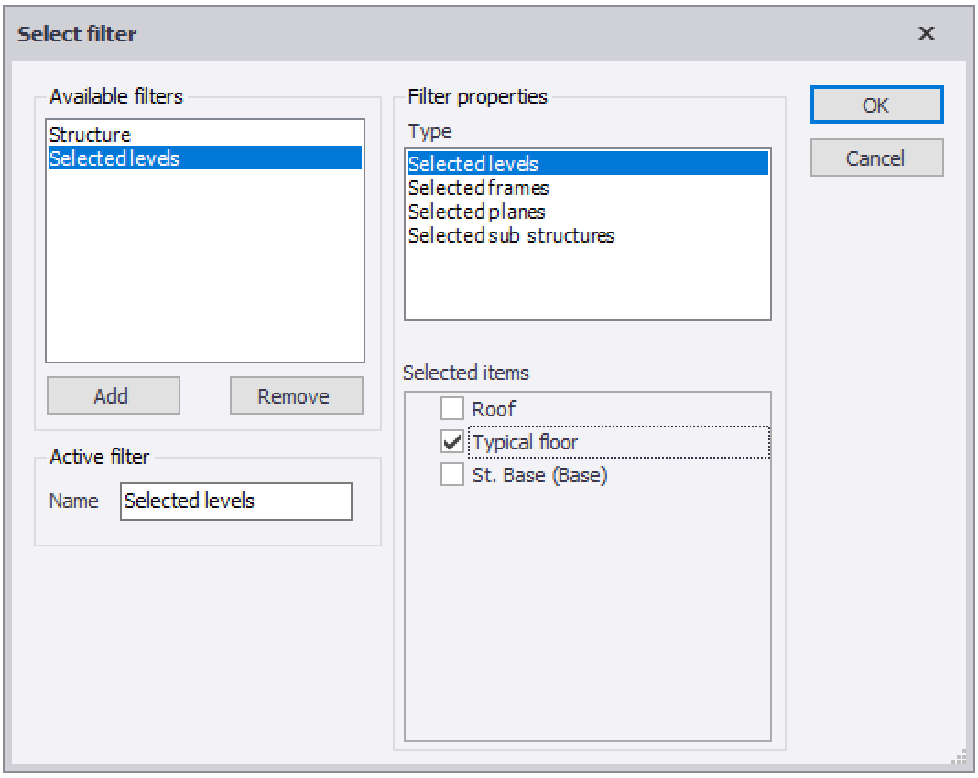
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



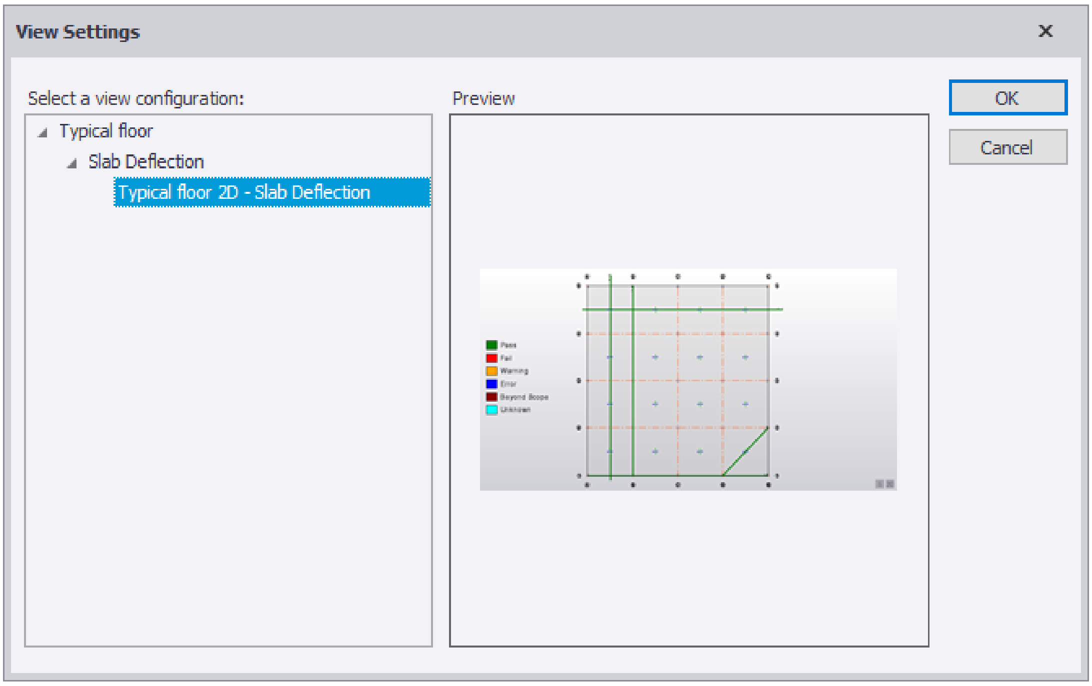
7. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**



8. In the Filter dialog, click **Add** and select Selected levels and ensure a check against Typical floor

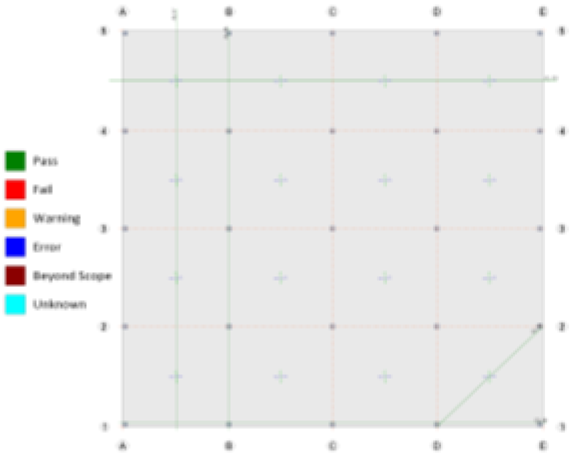


9. Click **OK** to return to the Report Contents dialog.
10. In the Report Structure, right click **View**, then choose **Settings...**
11. Select the Slab Deflection view you created earlier



12. Click **OK** to return to the Report Contents dialog
13. Click **OK** to exit and save the report.

    A report structure called Check Line Report has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select “Check Line Report”
    b. Click the Show Report command to open the report.



Typical floor 2D - Slab Deflection

<u>Concrete</u>

Slab/Mat Design

Slab/Mat Design per Plane

Typical floor

Slab Deflection Check Lines

CL 13

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	-3.9	3520.1	1 : 900	✓ Pass	0.278
Total	250	1 : 125	-10.7	3520.1	1 : 330	✓ Pass	0.379

CL 14

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	14.0	5515.4	1 : 394	✓ Pass	0.635
Total	250	1 : 125	-29.3	5515.4	1 : 188	✓ Pass	0.665

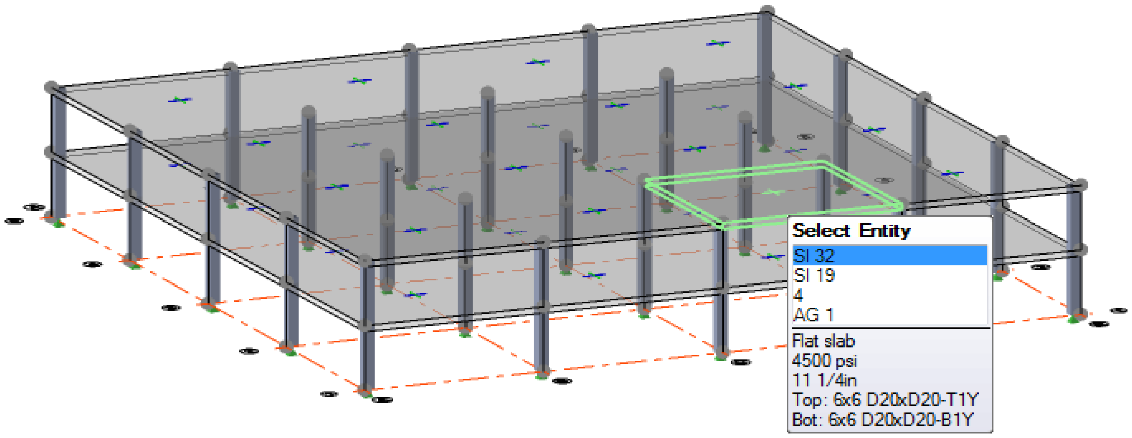
CL 15

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	8.7	3324.4	1 : 381	✓ Pass	0.656
Total	250	1 : 125	19.2	3324.4	1 : 173	✓ Pass	0.723

Slab deflection example (ACI)

In the following exercises deflections will be checked for the two story multi-bay flat slab tutorial model shown below.



Geometry:

- Bay centers 26’ 3”
- Floor to Floor spacing 9’ 10”
- Slab thickness 11 ¼ “, concrete grade 4500 psi
- 18” x 18” columns, concrete grade 4500 psi
- Analysis stiffness adjustment factors in accordance with ACI code guidance

Loading:

- Dead area load 30 psf
- Live area load 105 psf
- Cladding perimeter load 0.7klf

Deemed to satisfy checks (page 1463) provide one method of checking, however the main focus of these exercises will be to investigate rigorous slab deflection analysis (page 1465) and check lines (page 1501) as the method of checking.

Deemed to satisfy slab deflection checks example (ACI)

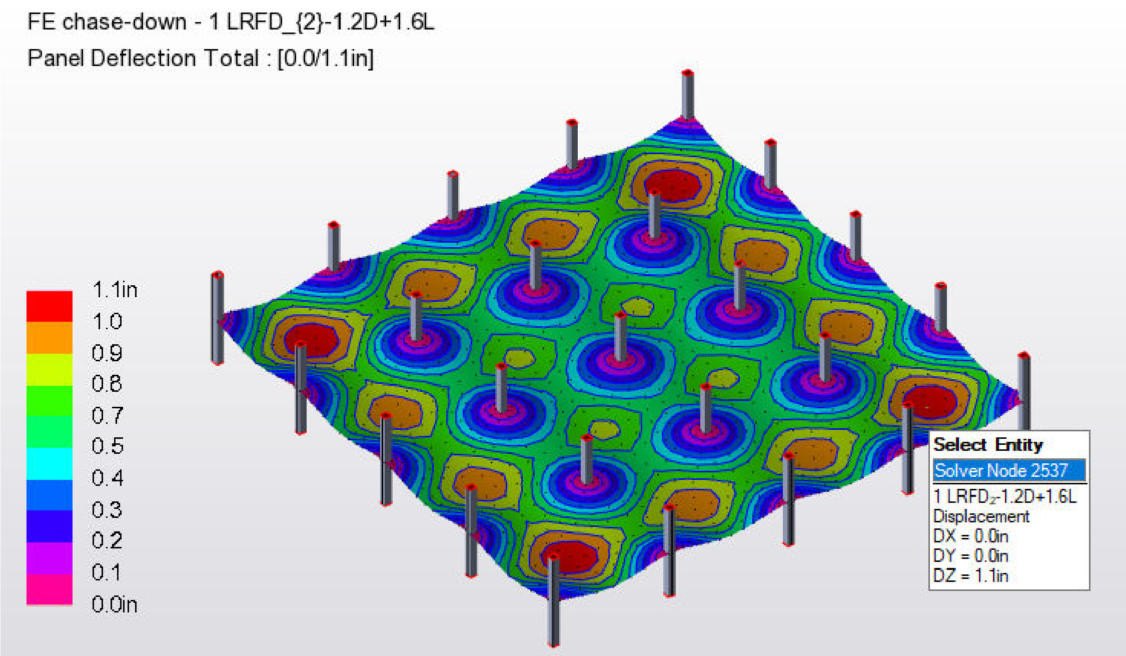
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results



Identify critical check locations

We can see that the maximum reported deflection is 1.1in, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs engineering judgment will be required to assess which deflections and span lengths require checking.

The deflection at the identified location now needs to be checked against a limiting span-to-depth ratio which we will assume for this example can be taken as span /240.

Taking the diagonal dimension across the columns, the deemed-to-satisfy span / 240 rule provides a deflection of [√(26' 3"^{2}+26' 3"^{2})] / 240 = 1.8in.

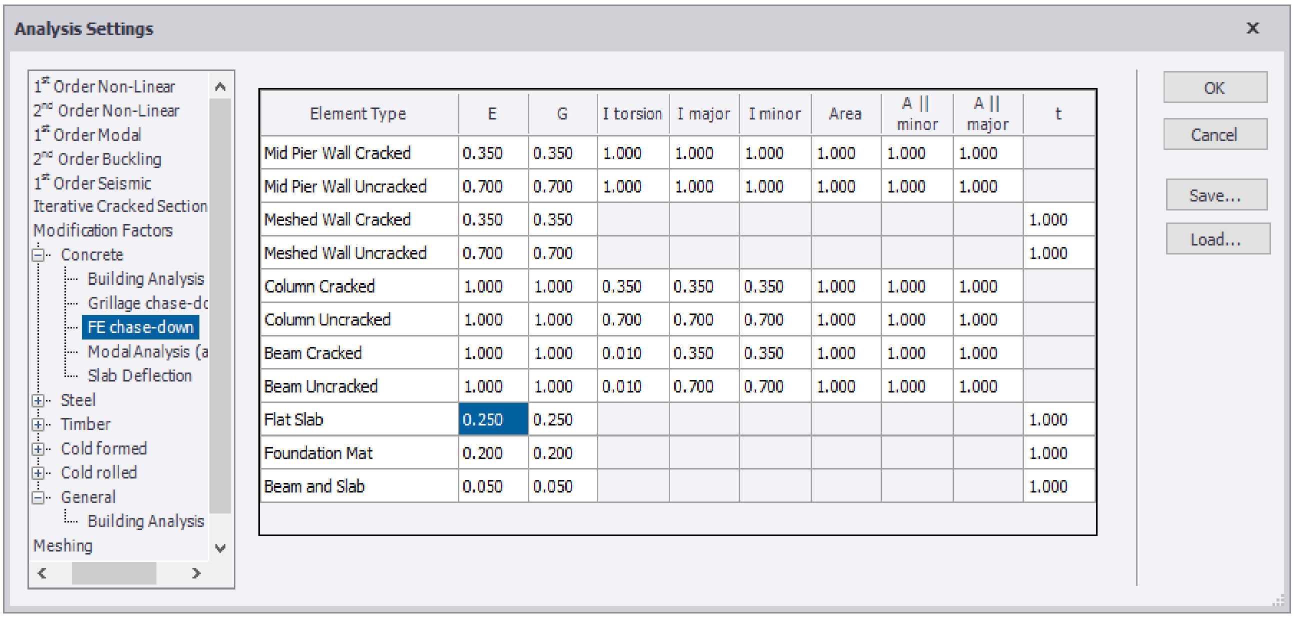
This compares unfavorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 240.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.25.



Element Type	E	G	I torsion	I major	I minor	Area	A || minor	A || major	t
Mid Pier Wall Cracked	0.350	0.350	1.000	1.000	1.000	1.000	1.000	1.000	
Mid Pier Wall Uncracked	0.700	0.700	1.000	1.000	1.000	1.000	1.000	1.000	
Meshed Wall Cracked	0.350	0.350							1.000
Meshed Wall Uncracked	0.700	0.700							1.000
Column Cracked	1.000	1.000	0.350	0.350	0.350	1.000	1.000	1.000	
Column Uncracked	1.000	1.000	0.700	0.700	0.700	1.000	1.000	1.000	
Beam Cracked	1.000	1.000	0.010	0.350	0.350	1.000	1.000	1.000	
Beam Uncracked	1.000	1.000	0.010	0.700	0.700	1.000	1.000	1.000	
Flat Slab	0.250	0.250							1.000
Foundation Mat	0.200	0.200							1.000
Beam and Slab	0.050	0.050							1.000

Rigorous slab deflection analysis examples (ACI)

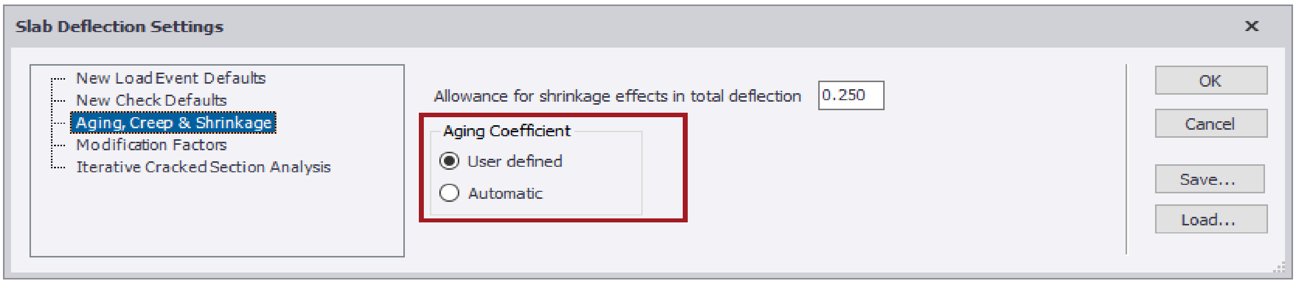
Three alternative methods of allowing for creep and shrinkage in the analysis are investigated.

Slab Deflection Property Choices

Aging Coefficient - User Defined or Automatic?

Tekla Structural Designer calculates a modulus of elasticity for each load event which accommodates both creep and aging.

The **Aging, Creep and Shrinkage** page on the Slab Deflection Settings dialog allows you to choose the method for this.



- **Automatic** - the modulus of elasticity used in the analysis (termed the composite modulus) is calculated rigorously according to the procedure defined in the Concrete Society Technical Report 58. This requires an early age event history to be defined by way of a detailed event sequence.
- **User defined** - the modulus of elasticity used in the analysis is calculated in accordance with ACI 435 Equation 3.33c:

  $\bar{E}_c(t, t_0) = E_c (t, t_0) / (1+ \chi C_t)$

  Where:

- The aging coefficient χ must be specified in the event sequence. The recommended value is 0.8. This is a user specified value.
- The creep coefficient C_t required in the above equation is automatically determined by *Tekla Structural Designer* using equation A-18 from ACI 209 taking account of shape and size effects.

The equation for C_t is $[(t_i-t_0) / (26\ e^{(0.36 \times V/S)}) + (t_i-t_0)] \times C_{ui}$

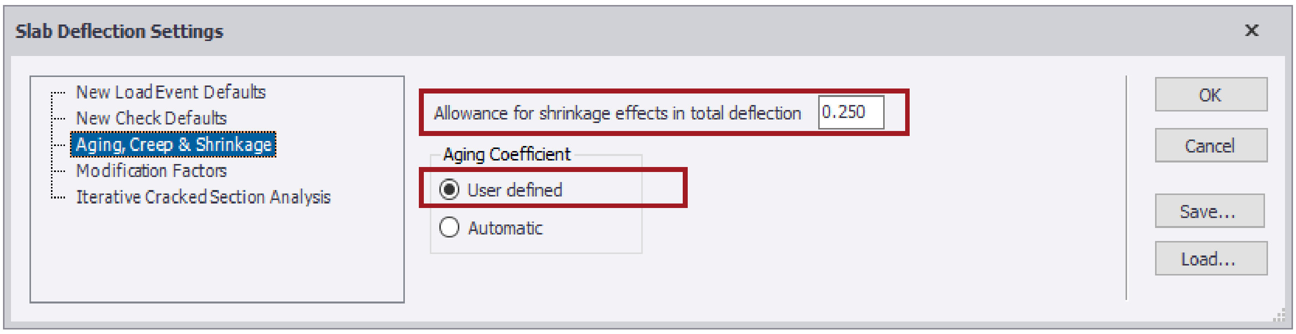
Where:

- For a slab the volume to exposed surface ratio V/S is taken as h/n where h is the slab thickness and n is the number of exposed surfaces. If n is set to zero then the creep coefficient is taken as zero thus eliminating creep from the analysis properties.
- Start of first event, t_0
- End time of event under consideration, t_i
- The Ultimate Creep coefficient, C_u must be specified in the event sequence. In normal conditions the recommended value from ACI 435R page 55 is $C_u = 2.35$. A different value can be specified according to guidance in ACI 209 if desired.

Shrinkage Allowance, or Combined Creep and Shrinkage Allowance?

When creep effects are included in the analysis, the Allowance for Shrinkage Effects multiplier can be set at a value which caters for shrinkage only. Alternatively you can increase the multiplier to allow for both creep and shrinkage, provided that you ensure that creep is not also accounted for in the analysis.

When using the latter approach it is important to ensure that the Aging coefficient is set to User defined.



Restraint Constant - Significant or Insignificant Restraint?

For each of the three methods considered in this example the Restraint constant (ACI) (page 1389) will be varied in order to model different restraint assumptions.

Restraint constant values from ACI 435 are:

- For situations with significant restraint - **4.0**
- For insignificant restraint - **7.5**

This will allow us to compare deflections estimations based on the assumption of significant restraint against those assuming insignificant restraint.

Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance

In this approach the simplest method suggested in ACI 318 is emulated. All creep and shrinkage effects are introduced as a single amplification factor and a cracked section analysis is run on a simplified event sequence without addressing the possibility of early age loading.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
    - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the Simplified Combined Creep and Shrinkage Allowance (ignoring ACI 435)

You can increase the **Allowance for Shrinkage Effects** multiplier to allow for both creep and shrinkage based on the multipliers from ACI 435 Table 4.1.

Using the **ACI code** recommendation (highlighted in the below table), multipliers are immediate **1.0**, creep and shrinkage **2.0**. Total = **3.0**.

Thus a creep and shrinkage contribution = 2.0/3.0 = 0.66 is to be applied which equates to 66% of the total deflection.

Table 4.1 - Multipliers recommended by different authors

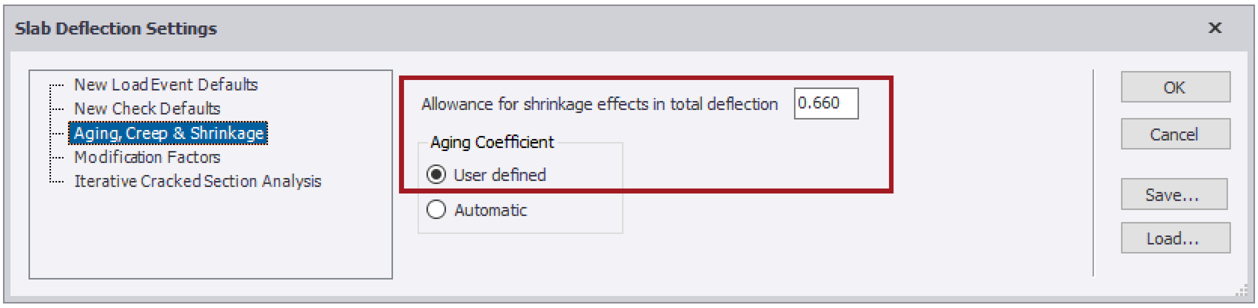
Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	$7.5 \sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	$7.5 \sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	$7.5 \sqrt{f_c'}$ $4 \sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	**$7.5 \sqrt{f_c'}$**	**1.0**	**2.0**		**3.0**

When using this approach:

- The automatic procedure using Technical Report 58 is not considered - this is ensured by setting the **Aging coefficient** to **User defined**.
- Creep is not also accounted for in the analysis - it can be excluded by setting the **Number of Exposed Faces** to **Zero** in the event sequence.

1. **To adopt the above ACI 318 Code multipliers for simplified creep and shrinkage:**

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.66**.



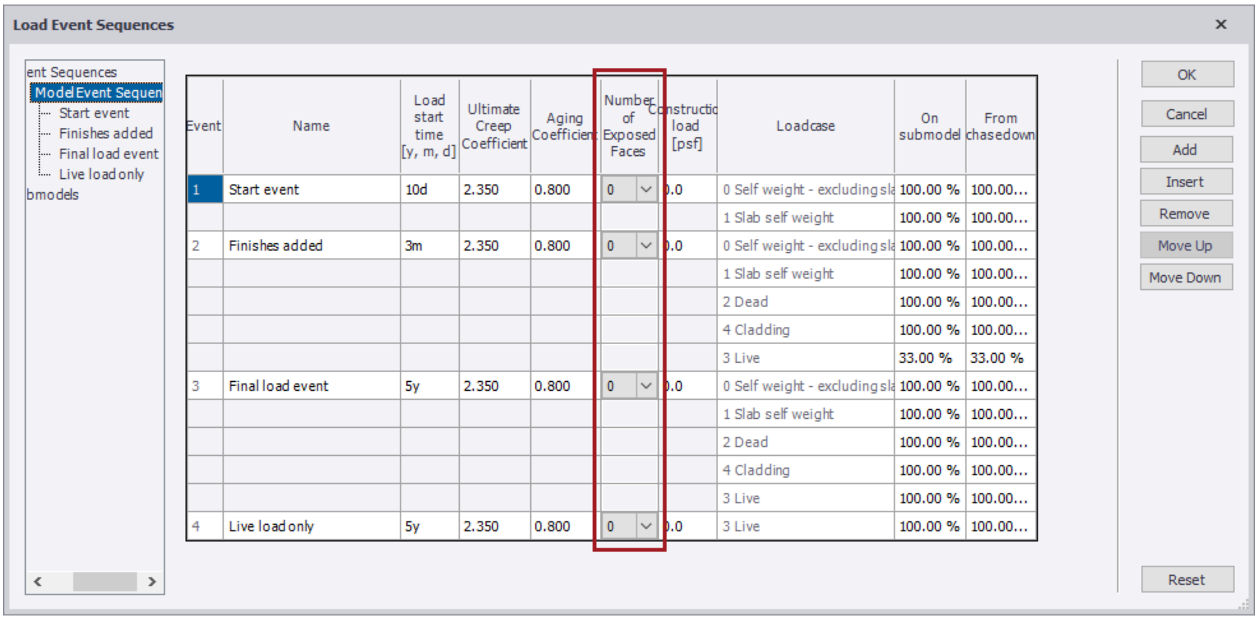
The above factor allows for both creep and shrinkage, (i.e. creep and shrinkage effects will account for 66% of the total deflection).

4. Click **OK** to close the dialog.

2. **To exclude additional creep effects from the analysis**

As creep is already being catered for by the amplification factor, we have to exclude additional creep effects from the analysis. This can be done by setting the number of exposed faces in each event to zero as follows:

5. From the **Slab Deflection** toolbar, click **Event Sequences**
6. Click **Model Event Sequence**
7. For each listed event in the model sequence, ensure the **Number of Exposed Faces** is set to **0**.



If the number of faces = 0 then the creep coefficient $C_t = 0$. See: Aging Coefficient - User Defined or Automatic? (page 1466)

Review the Individual Events in the Model Event Sequence

A simplified event sequence has been defined that does not include any construction stage propping events.

Various guidance documents discuss slab deflection analysis without addressing the possibility of significant early age loading events such as propping loads.

An early start event should always be created since time is measured from this start event. In this example we have assumed a start event of 10 days and allowed self weight only.

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Start event	10d	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
2	Finishes added	3m	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	33.00 %	33.00 %
3	Final load event	5y	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	100.00 %	100.00 %
4	Live load only	5y	2.350	0.800	0	0.0	3 Live	100.00 %	100.00 %

You will also note that:

- Each event has a load start time. The Final load event is set to the normal ACI requirement of 5 years.
- ACI requires that instantaneous deflection due to live loads only should be considered based upon a span/360 limit. To account for instantaneous deflection due to live load only, an end event has been included with the same load start time as the preceding event but only including live load.
- Ultimate Creep Coefficient, C_u is set with the default value of 2.350. This value can be set separately for each event.
- Aging Coefficient is set at the default value of 0.8. This value can be set separately for each event. It may be more logical to set higher values for the earlier event times, however, if your primary concern is differential deflection between later events then it will be conservative to use the same value everywhere.
- Number of exposed Faces is set to 0 - this has been done to exclude additional creep effects from the analysis, (as explained in the previous section).

Discussion:

- You need to think about the accumulation of deflection through time and hence the checks that you ultimately wish to consider.
- As an example, if you are interested in differential deflection between the final load event and the start of finishes (deflection at end of event 1), if you underestimate deflection to the end of event 1 then this check becomes more onerous.
- Is it reasonable to assume no construction load and no self weight from finishes during this period?
- How much load is reasonable to assume at this starting event is ultimately the responsibility of the engineer.

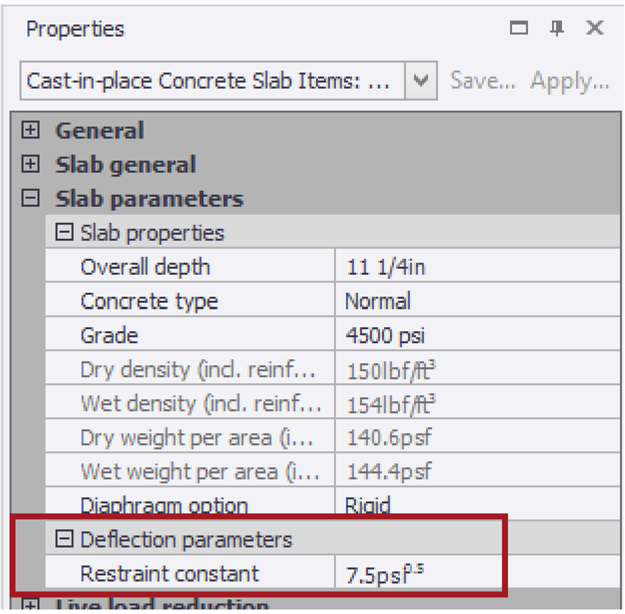
1. After reviewing the Event Sequence, click **OK** to close the dialog.

Set up the Restraint Constant

The modulus of rupture is set using the **Restraint Constant** slab deflection parameter. Since we are using the ACI code multipliers from Table 4.1 above this should be set to 7.5.

1. **To specify an appropriate restraint constant**

1. Open the **Structure** 3D view
2. Select all the slab items in the model and via the Properties Window, ensure the **Restraint Constant** is set to **7.5**



To establish some initial results:

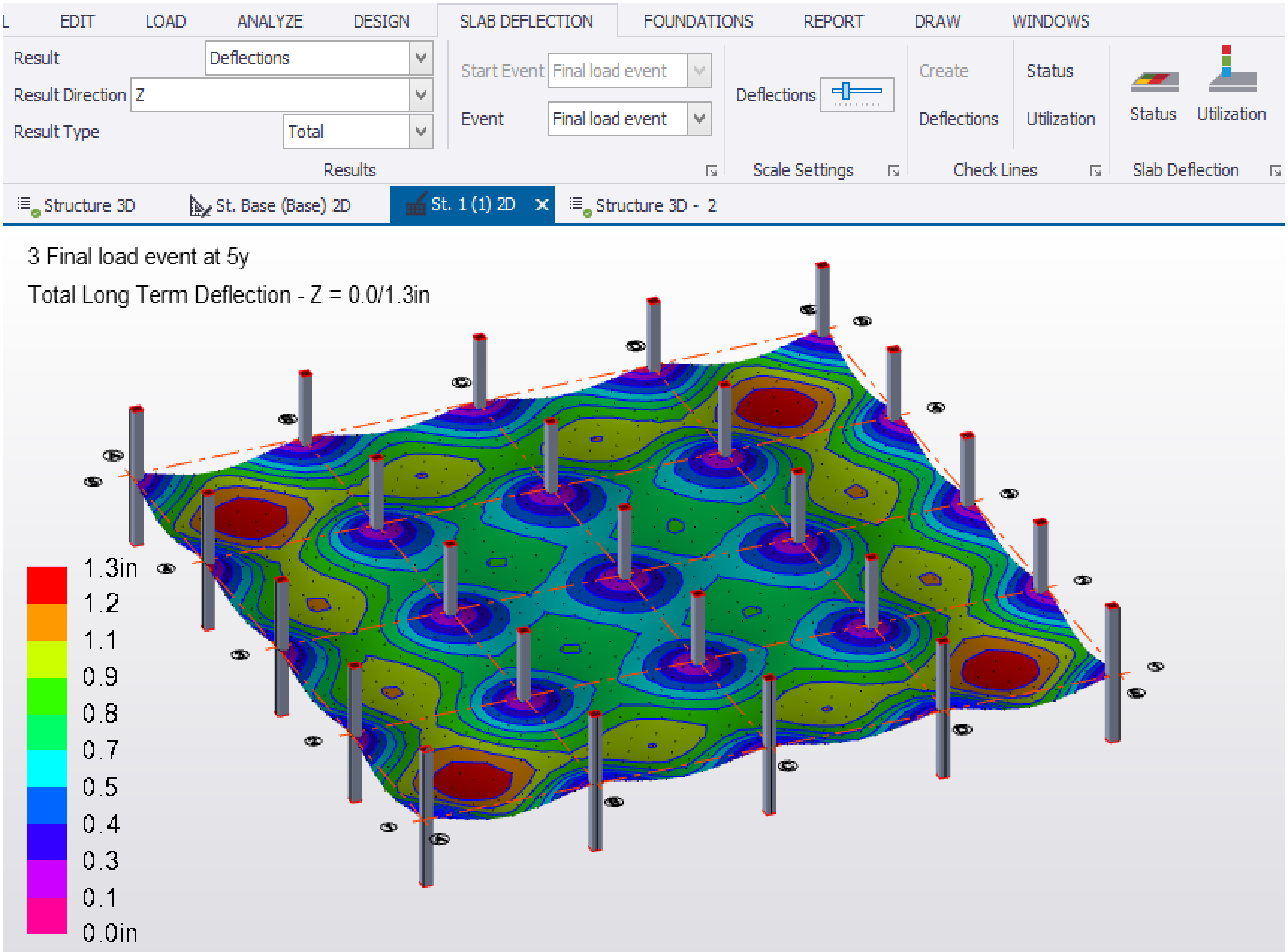
1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections for the events

   Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon. You are able to review:

   - Total deflection at the end of any event.
   - Differential deflection between any two events (Start of Event and End of Event).
   - Instantaneous deflection.

   The Total deflection at the final load event for the chosen location is 1.3”



TIP If slab patches are obscuring the above deflection contours, these can be switched in Scene Content.

Consider the Sustained Load Multiplier Effect from ACI 435

ACI 435 Clause 4.3.4.2 needs to be reviewed carefully at this stage, as this suggests that the above deflection estimation is unconservative.

- It states that if the restraint stresses are expected to be **insignificant**, (so that the restraint constant is set at 7.5) then:

- *"the multiplier for sustained-load deflection should be increased from 2 to 4, as recommended by Sbarounis (1984) and Graham and Scanlon [1986(b)]"*

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	**7.5 $\sqrt{f_c'}$**	**1.0**	**2.8**	**1.2**	**5.0**
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	**7.5 $\sqrt{f_c'}$** 4 $\sqrt{f_c'}$	**1.0** 1.0	**2.0** 1.5	**2.0** 1.0	**5.0** 3.5
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

Hence the combined creep and shrinkage contribution = 4.0/5.0 = **0.8** (i.e. 80% of the total deflection).

- Alternatively, if the restraint stresses are likely to have a **significant** effect then Clause 4.3.4.2 states that:
  - a reduced restraint constant of 4 can be used, along with a long-term sustained-load multiplier of 2.5.

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ **4 $\sqrt{f_c'}$**	1.0 1.0	2.0 **1.5**	2.0 **1.0**	5.0 **3.5**
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

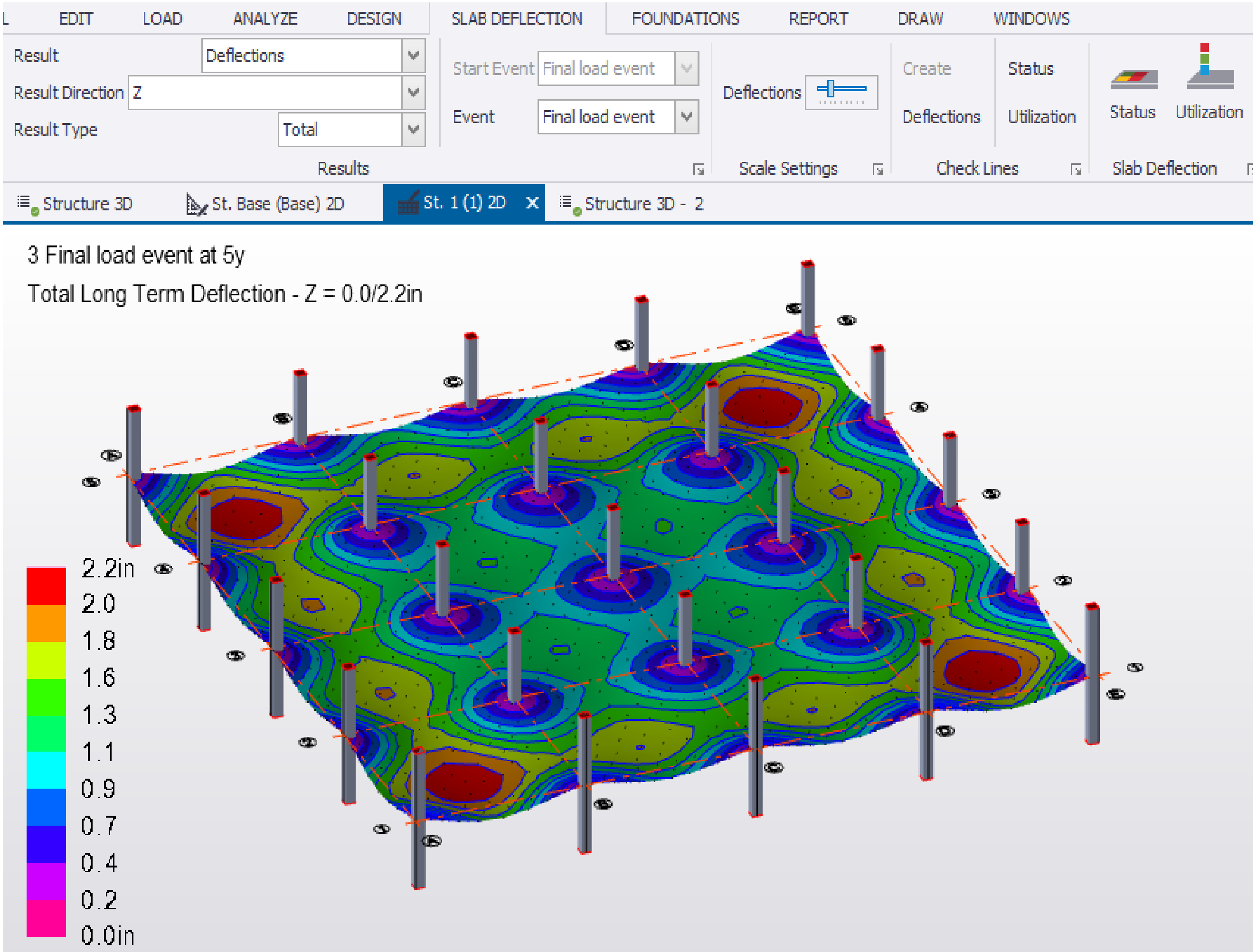
In this case the combined creep and shrinkage contribution should be reduced to 2.5/3.5 = **0.714** (i.e. 71.4% of the total deflection).

1. **To adopt the ACI 435 recommendation for insignificant restraint stresses:**

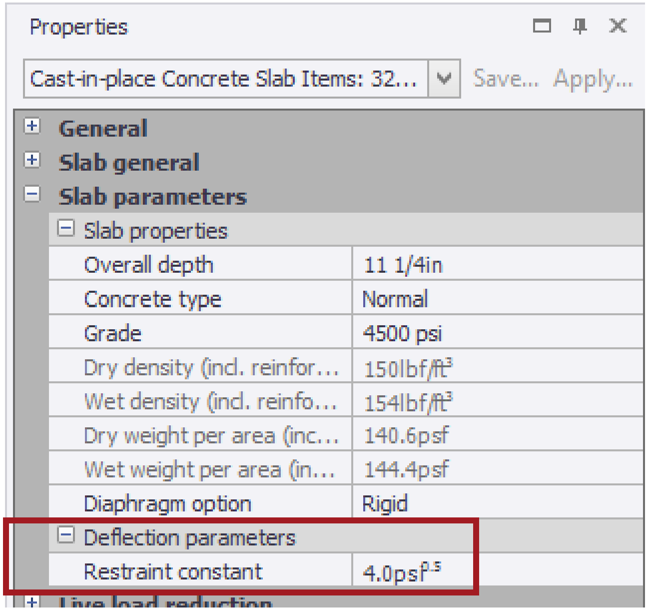
1. From the **Slab Deflection** toolbar, click **Settings**

2. In the dialog, click **Aging, Creep & Shrinkage**
3. Increase the **Allowance for shrinkage effects in total deflection** to **0.8** then click **OK**
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   Using this value, the revised deflection prediction increases to 2.2"

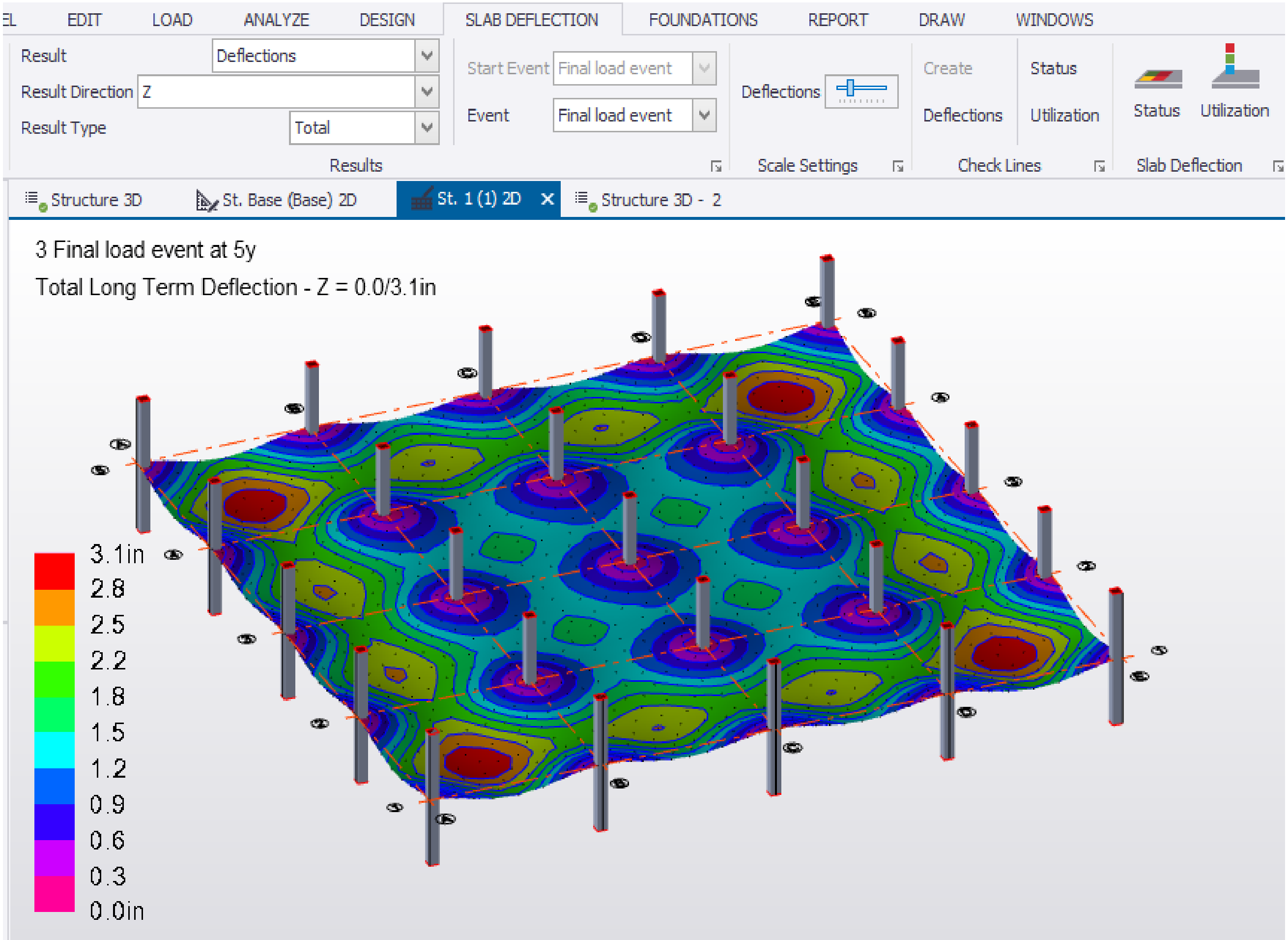


2. **To adopt the ACI 435 recommendation when restraint stresses are expected to be significant:**
5. Open the **Structure** 3D view
6. Select all the slabs in the model and via the Properties Window, adjust the **Restraint Constant** to **4.0**



7. From the **Slab Deflection** toolbar, click **Settings**
8. In the dialog, click **Aging, Creep & Shrinkage**
9. Change the **Allowance for shrinkage effects in total deflection** to **0.714** then click **OK**
10. Return to the **St.1 (1)** 2D view.
11. From the **Slab Deflection** toolbar, click **Analyze Current**

    Using this value, the revised deflection prediction increases further to 3.1"



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel > For the current slab item**

Composite Modulus Calculation

Event	Start [d]	To end of Event		
		Ultimate Creep Coefficient, c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.800	4000
2 Finishes added	91	2.350	0.800	4000
3 Final load event	1825	2.350	0.800	4000
4 Live load only	1825	2.350	0.800	4000

Slab_ S 3, Panel_ Sl 33

We can see from the report that the short term composite modulus, E_c is used.

Summary of Results

Using the Simplified event sequence + simplified combined creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Assumed Creep and Shrinkage combined allowance %	Total deflection (Final load event)
ACI 318 (ignoring ACI 435)	7.5	66%	1.3"
ACI 435 (simple approach - insignificant restraint)	7.5	80%	2.2"
ACI 435 (simple approach - significant restraint)	4.0	71.4%	3.1"

Next steps

- In Method 2 (page 1480) we will re-use the same model, but edit the simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.
- In Method 3 (page 1487) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 1496) are discussed.

- Finally, using results from one of the methods, deflection checks are performed using check lines (page 1501) and an output report is generated.

Method 2: Simplified Event Sequence + Rigorous Creep and Shrinkage Allowance

In Method 1 (page 1468) we used a simplified event sequence (without addressing the possibility of early age loading), to see how a single multiplier can be applied to allow for the combined effects of creep and shrinkage.

In **Method 2** we will re-use the same model, but edit the previous simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.

Download and open the tutorial model (if required)

NOTE If you are reusing the model from the Method 1 (page 1468) exercise you can skip this step.

1. Download the tutorial models from here.
2. Open the following tutorial model:
    - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

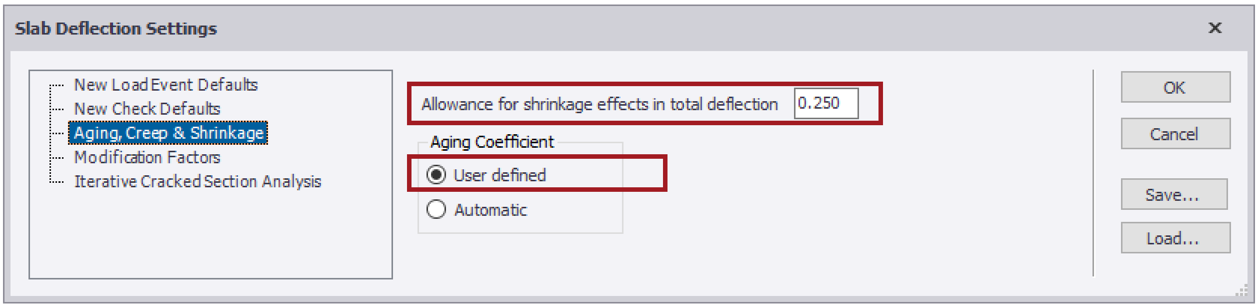
Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the rigorous creep and shrinkage allowance

In this method the **Allowance for Shrinkage Effects** multiplier is set to allow for shrinkage only.

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.25**.



The above factor allows for shrinkage only.

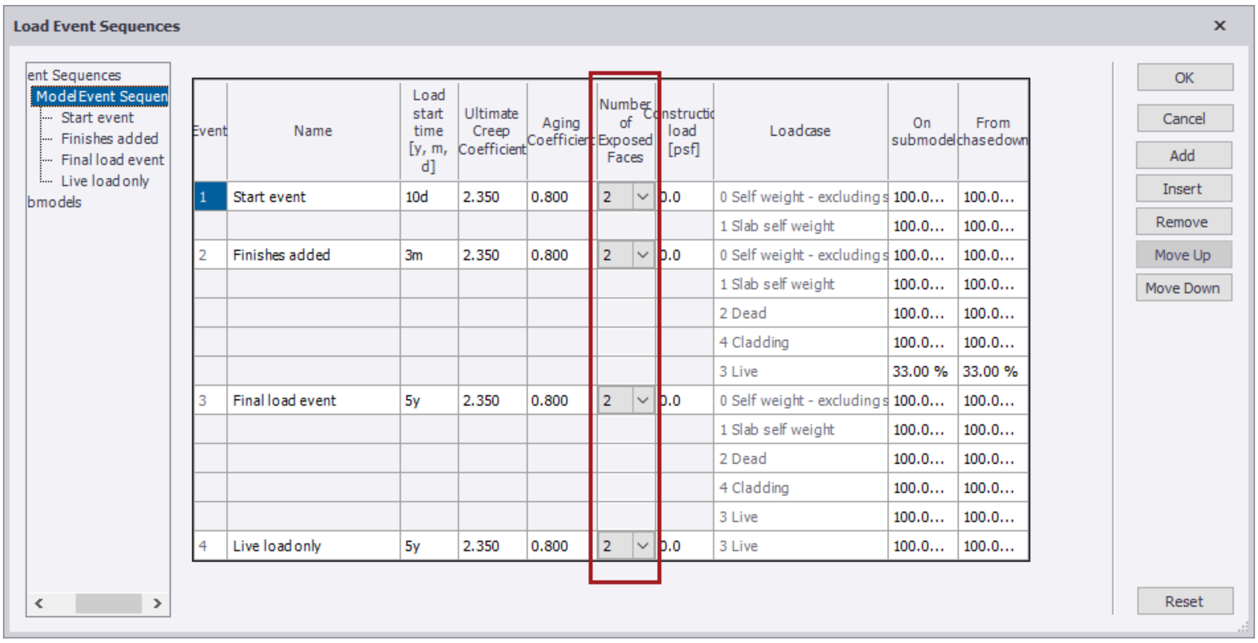
Refer to Shrinkage allowance (page 1417) for an explanation of where this value comes from.

4. Click **OK** to close the dialog.

Edit the Event Sequence to ensure that creep is accounted for in the analysis

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**
3. Ensure that the **Number of Exposed Faces** is set to **2** for each event.

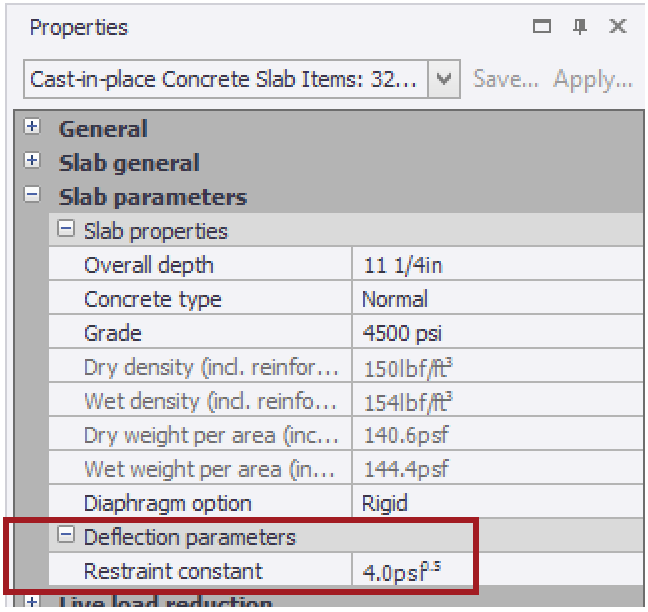
   The creep coefficient C_t will be calculated based on this, thus affecting the modulus of elasticity used in the analysis. For further details of this calculation, see: Aging Coefficient - User Defined or Automatic? (page 1466)



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

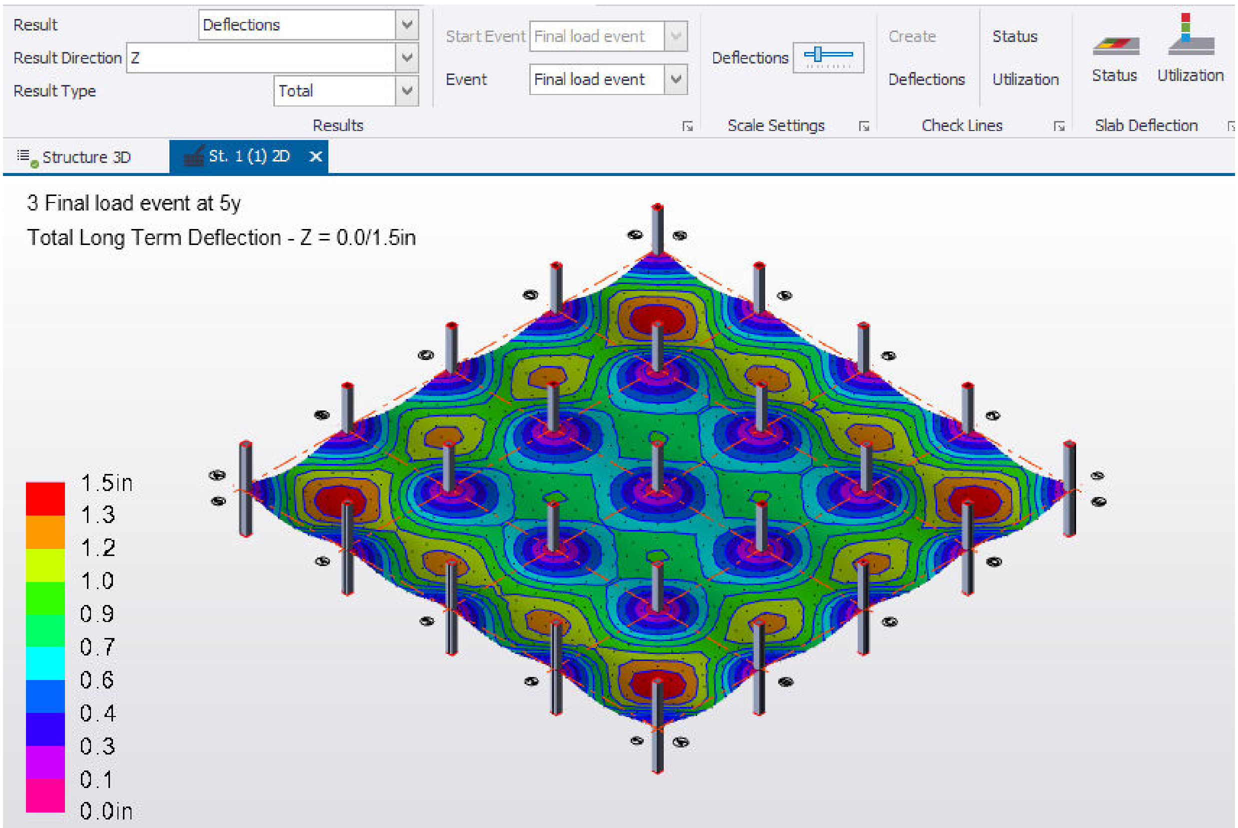


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

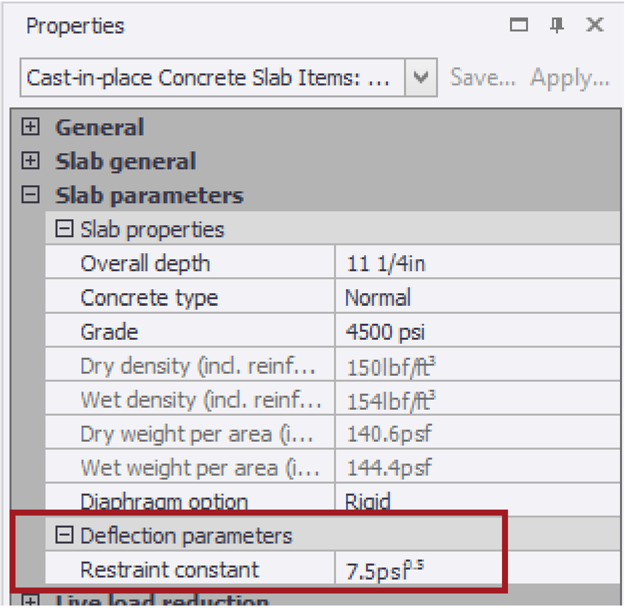
   The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



Adjust the Restraint Constant and Re-analyze

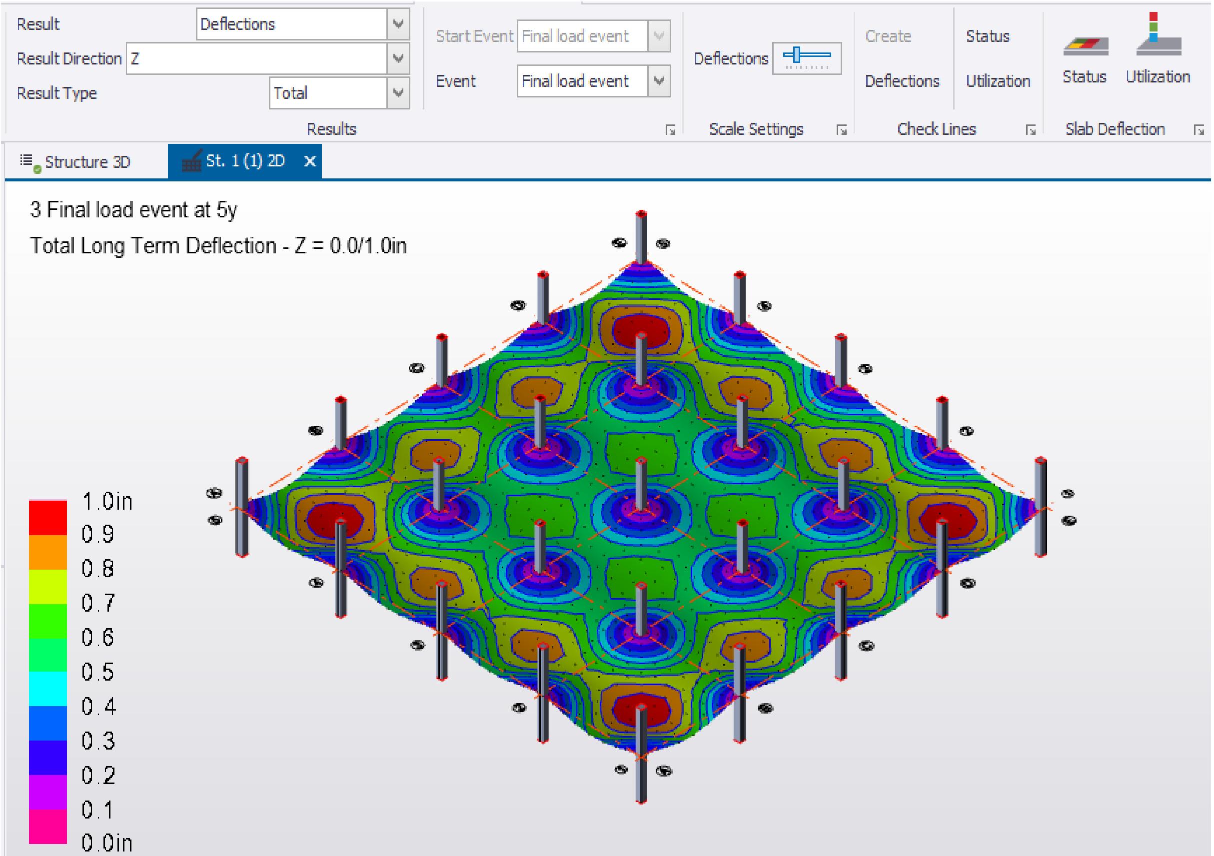
Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   With these settings the total deflection predicted at 5 years reduces to 1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**

Composite Modulus Calculation					
	Start	To end of Event			
Event	[d]	Ultimate Creep Coefficient, c_u	c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.684	0.800	2585
2 Finishes added	91	2.350	2.119	0.800	1484
3 Final load event	1825	2.350	2.119	0.800	1484
4 Live load only	1825	2.350	2.119	0.800	1484

Slab_ S 3, Panel_ Sl 33

Comparing this report to the report obtained when using the simplified creep and shrinkage allowance (page 1478), we can see a difference in the value of the Composite Modulus E_c used. In this method, the age of concrete is taken into account.

To verify the result, consider Event 2, 3 and 4 where the event ends at 1825 days:

- Start event t_0 = 10 days, Event being considered, t_i = 1825 days
- Time between events (t_i-t_0) = 1825-10 = 1815 days
- Modulus of Elasticity, E for 4500 psi concrete grade = 4000 ksi (from Material database)
- Assumed Ultimate Creep Coefficient, C_u = 2.35
- Assumed Aging coefficient χ = 0.8. This is typically in the range 0.7 to 0.9
- Slab depth, d = 11 ¼ in
- Number of exposed faces, h = 2.

  $C_t = [(t_i-t_0) / (26\ e^{(0.36 \times d/h)}) + (t_i-t_0)] \times C_{ui} = [1815 / (26\ e^{(0.36 \times 11\ 1/4\ /\ 2)} + 1815] \times 2.35 = 2.1199$

  Thus

  $\bar{E}_c = (t, t_0) = E_c\ (t, t_0) / (1 + \chi\ Ct)$

  For event 2, 3 and 4, Composite Modulus, E_c = 4000 / (1 + 0.8 x 2.1199) = 1484 ksi

Summary of Results

Using the Simplified event sequence plus rigorous creep plus shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = 0.8	25%	1.5”
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = 0.8	25%	1.0”

Next steps

- In Method 3 (page 1487) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 1496) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 1501) and an output report is generated.

Method 3: Detailed Event Sequence + Rigorous Creep and Shrinkage Allowance

In this approach, a detailed event sequence has been defined which includes propping loads at early events to create an overall load history. Creep is considered rigorously alongside this load history by setting the Aging coefficient to Automatic. This means that an effective composite modulus value to the end of each event is calculated in accordance with the Concrete Society Technical Report 58.

NOTE The aging coefficient method should only be set to Automatic when defining a detailed event sequence that includes a realistic assessment of early propping load events.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Detailed Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

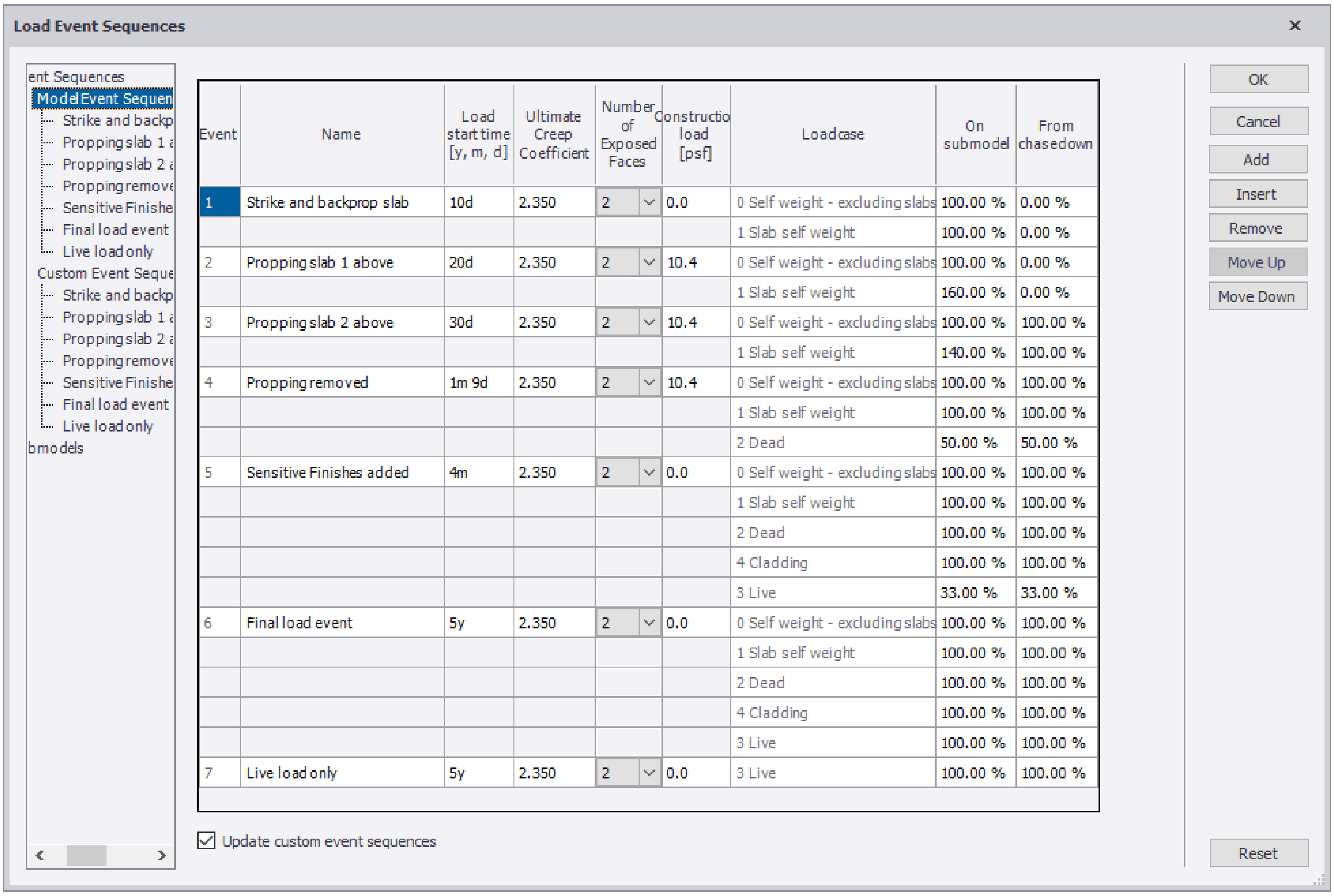
1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Detailed Event Sequence

1. **To display the Event Sequence**

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

   A detailed event sequence has already been created:



Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Number of Exposed Faces	Constructio load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	160.00 %	0.00 %
3	Propping slab 2 above	30d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	140.00 %	100.00 %
4	Propping removed	1m 9d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	50.00 %	50.00 %
5	Sensitive Finishes added	4m	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	33.00 %	33.00 %
6	Final load event	5y	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	100.00 %	100.00 %
7	Live load only	5y	2.350	2	0.0	3 Live	100.00 %	100.00 %

You will note that:

- Ultimate Creep Coefficient, C_u set at **2.35** for this example
- Number of exposed faces, h set at **2** for this example
- Construction load assumed set at **10.4 psf** in this example
- Propping load % assumed during early age propping events **160%** and **140%** assumed.

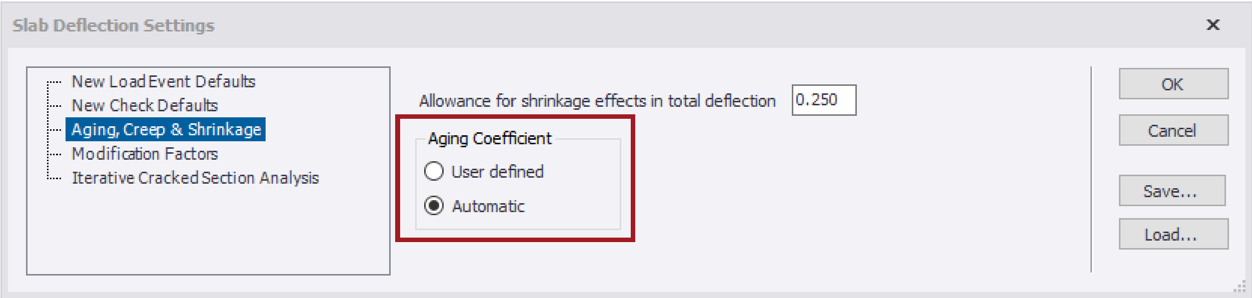
Set up Rigorous Creep

This method requires that the **Aging coefficient** is set to Automatic. Creep is then considered automatically as part of the analysis for each event sequence.

A user defined allowance for Shrinkage effects in the total deflection is also specified.

1. **To specify the multiplier for shrinkage effects only but also consider creep in the analysis**

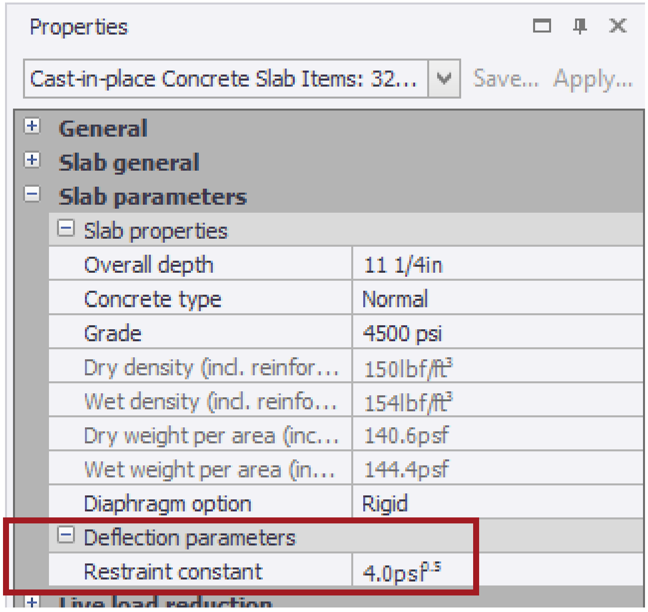
1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Allowance for shrinkage effects in total deflection** is set to **0.25**, and the **Aging Coefficient** is set to **Automatic**



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

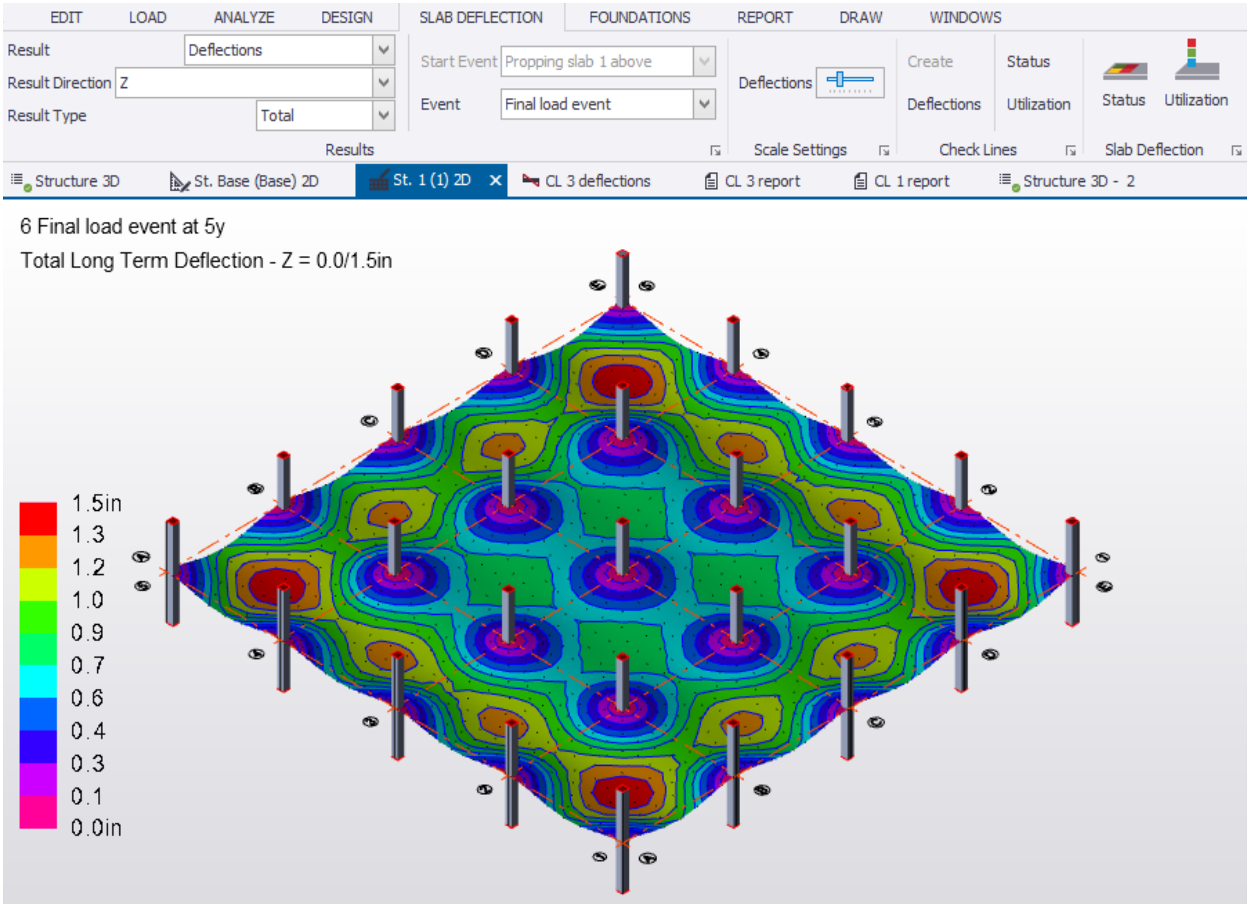


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

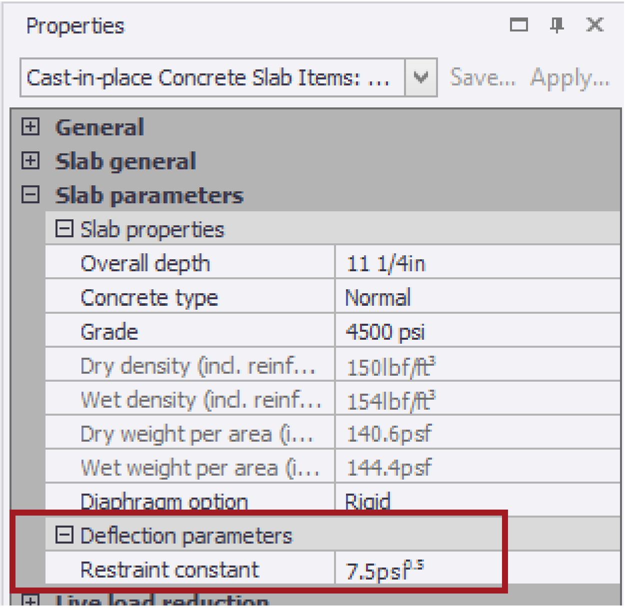
   The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



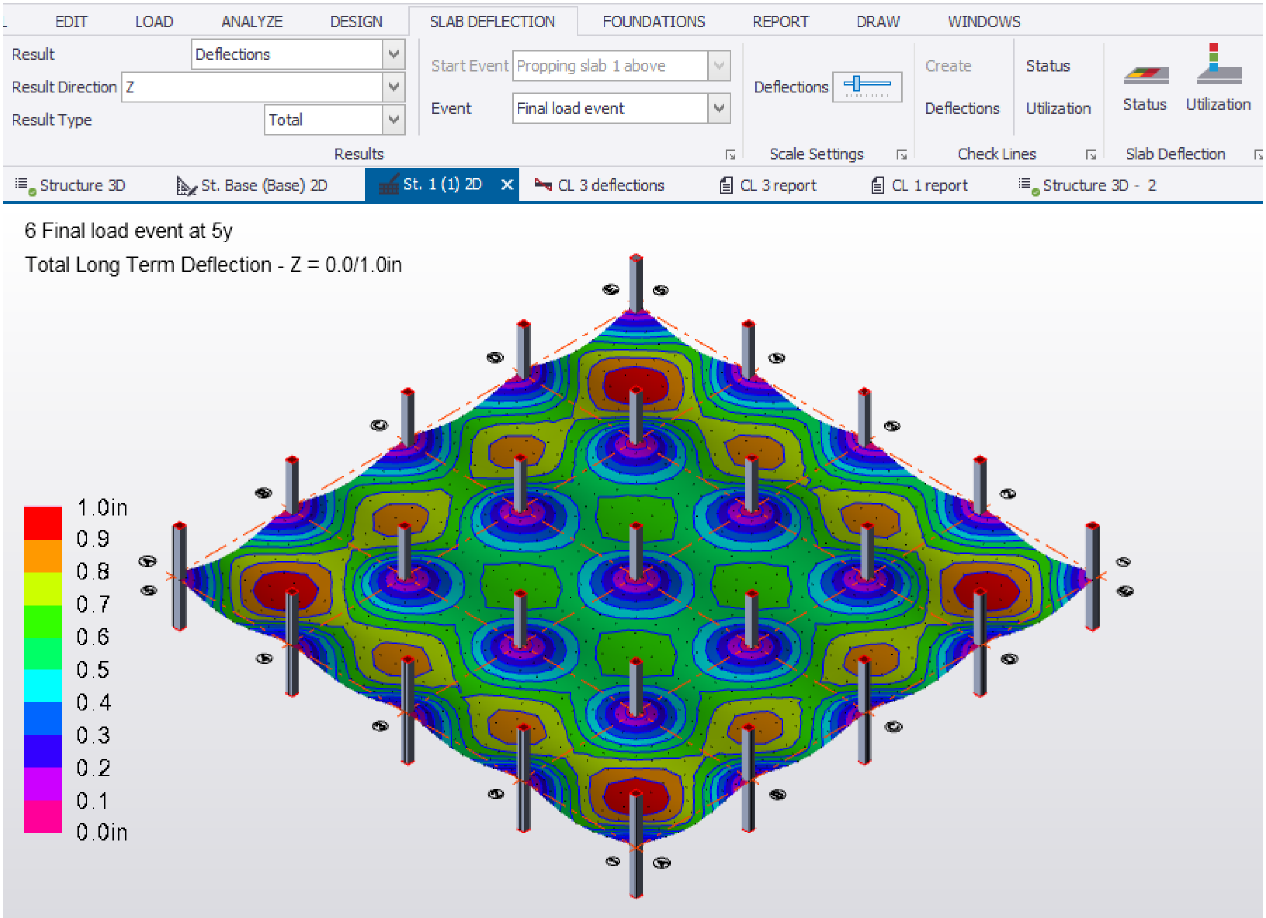
Adjust the Restraint Constant and Re-analyze

Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**
   With these settings the total deflection predicted at 5 years reduces to 1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.

2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**

Composite Modulus Calculation

Event	Start [d]	Incremental load factor, λ	To end of Event 1			To end of Event 2			To end of Event 3			To end of Event 4		
			φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]
1 Strike and backprop slab	10	6.735	0.113	3593	0.00187	0.216	3290	0.00205	0.304	3067	0.00220	0.848	2164	0.00311
2 Propping slab 1 above	20	4.539	-	-	-	0.113	3593	0.00126	0.210	3305	0.00137	0.798	2225	0.00204
3 Propping slab 2 above	30	-1.349	-	-	-	-	-	-	0.107	3614	-0.00037	0.744	2293	-0.00059
4 Propping removed	39	-1.976	-	-	-	-	-	-	-	-	-	0.690	2366	-0.00083
5 Sensitive Finishes added	122	2.831	-	-	-	-	-	-	-	-	-	-	-	-
6 Final load event	1825	3.367	-	-	-	-	-	-	-	-	-	-	-	-
7 Live load only	1825	-9.120	-	-	-	-	-	-	-	-	-	-	-	-
Total of $\lambda / E_{c,eff}$ [1 / ksi]					0.00187			0.00331			0.00320			0.00373
E_c to end of Event [ksi]					3593			3406			3105			2132

Event	To end of Event 5			To end of Event 6			To end of Event 7		
	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	$\lambda / E_{c,eff}$ [1 / ksi]
1 Strike and backprop slab	2.119	1282	0.00525	2.119	1282	0.00525	2.119	1282	0.00525
2 Propping slab 1 above	2.118	1283	0.00354	2.118	1283	0.00354	2.118	1283	0.00354
3 Propping slab 2 above	2.117	1283	-0.00105	2.117	1283	-0.00105	2.117	1283	-0.00105
4 Propping removed	2.116	1284	-0.00154	2.116	1284	-0.00154	2.116	1284	-0.00154
5 Sensitive Finishes added	2.106	1288	0.00220	2.106	1288	0.00220	2.106	1288	0.00220
6 Final load event	-	-	-	0.000	4000	0.00084	0.000	4000	0.00084
7 Live load only	-	-	-	-	-	-	0.000	4000	-0.00228
Total of $\lambda / E_{c,eff}$ [1 / ksi]			0.00840			0.00924			0.00696
E_c to end of Event [ksi]			1284			1531			722

Slab_ S 3, Panel_ Sl 33

Comparing this report to the report for the more simplified ACI "rigorous" approach to creep (page 1485) undertaken earlier.

- ACI composite modulus calculation for the Final Load Event = 1484 ksi (also verified by hand calculation earlier)
- TR58 composite modulus calculation for the Final Load Event = 1531 ksi
- These values compare favorably.

Note, however, that if we compared event 5 above where the composite modulus, E_c = 1284 ksi using the ACI method, we can establish the aging coefficient as:

$E_c = E / (1 + X * Ct)$

$1284 = 4000 / (1 + \chi \times 2.1199)$

$\chi = [(4000 / 1284) - 1] / 2.1199 = 0.9978$

This increased aging coefficient occurs due to the significant propping loads at early events, which creates an overall load history that is almost equivalent to constant loading.

Summary of Results

Using the Detailed event sequence plus rigorous creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic	25%	1.5"

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
		based on TR58		
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58	25%	1.0"

Next steps

- Having obtained results for all three methods, observations on the different methods (page 1496) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 1501) and an output report is generated.

Observations on the Different Methods

Combined Table of Results

Approach		Restraint Constant for Modulus of Rupture	Creep approach	Assumed Creep and Shrinkage combined allowance %	Assumed Shrinkage %	Total deflection (Final load event)
Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance	ACI 318 (ignoring ACI 435)	7.5		66%		1.3"
	ACI 435 (simple approach - insignificant restraint)	7.5		80%		2.2"
	ACI 435 (simple approach	4.0		71.4%		3.1"

Approach		Restraint Constant for Modulus of Rupture	Creep approach	Assumed Creep and Shrinkage combined allowance %	Assumed Shrinkage %	Total deflection (Final load event)
	- significant restraint)					
Method 2: Simplified Event Sequence + Rigorous Creep + Shrinkage Allowance	**ACI 435 (creep included in analysis)**	**4.0**	**C_u = 2.35, h = 2 and χ = 0.8**		**25%**	**1.5”**
		7.5	C_u = 2.35, h = 2 and χ = 0.8		25%	1.0”
Method 3: Detailed Event Sequence + Rigorous Creep + Shrinkage Allowance	ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.5”
		7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.0”

Discussion

When the guidance in ACI 435 about the use of a combined allowance for creep and shrinkage is taken into account, the simplified approach using a combined creep and shrinkage multiplier seems to determine very conservative deflection estimations.

Looking at table 4.1 in ACI 435, it is clear that the creep contribution is the most significant part of the overall deflection estimate. It is also the area where there seems to be greatest variation in opinion on the contribution level.

The creep contribution can be dealt with more rigorously by including it in the analysis. So rather than using a short term E value and then amplifying the result to account for creep, the analysis at the end of each event uses an effective E value which includes for creep up to that point. This impacts on the analysis properties of every shell in the FE analysis and even on the extent of cracking.

It is suggested that when the creep is included in the analysis the use of a modulus of rupture which assumes low restraint (restraint constant = 7.5) be used cautiously.

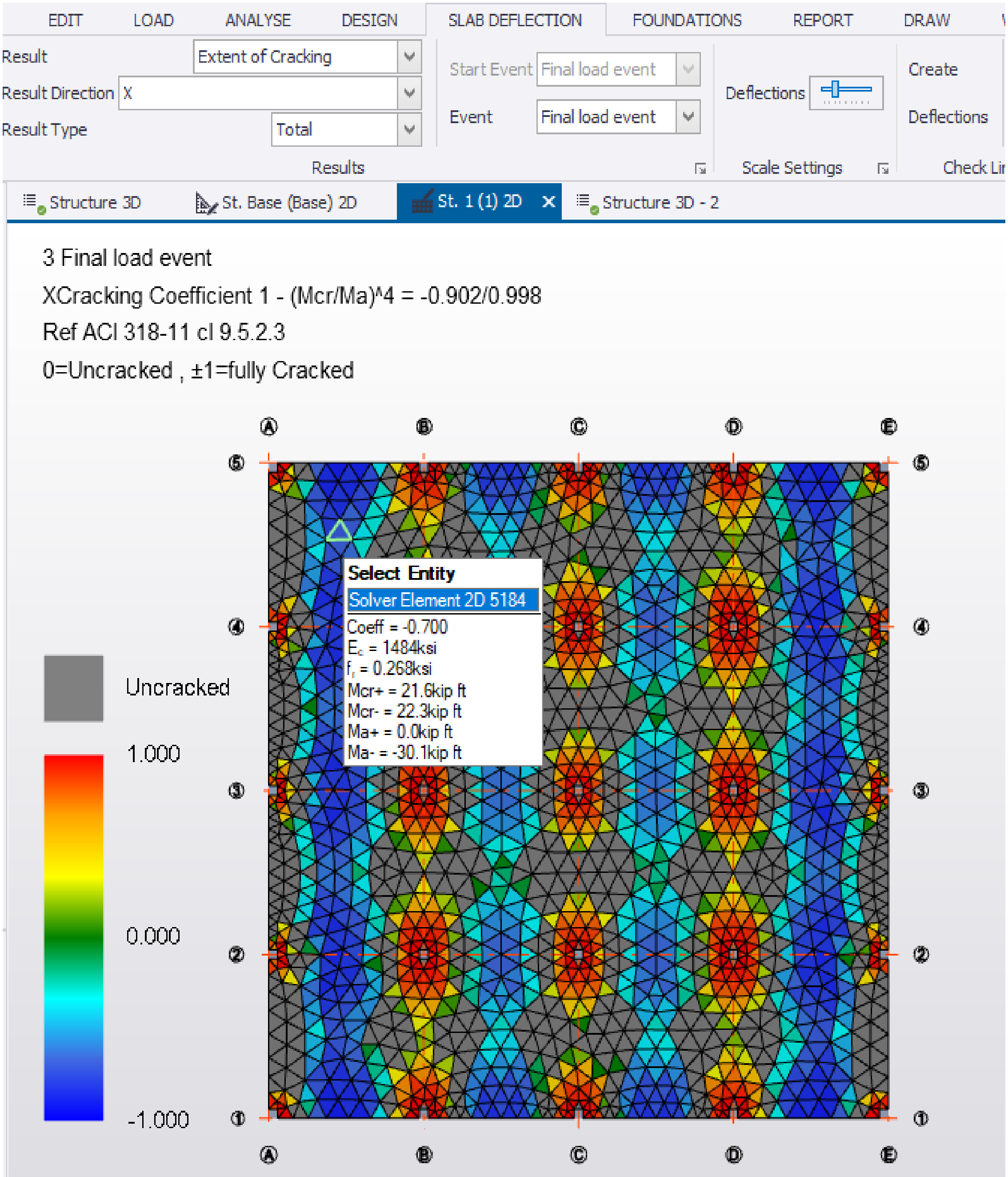
The option to calculate the effective E value based on some UK guidance in TR58 makes little difference in this example. It should be borne in mind that the aging coefficient χ is user defined and 0.8 is just a default for “normal” situations. In situations such as a transfer slab where load will accrue more slowly over time 0.8 is likely to be conservative and it may be of interest to consider the TR58 alternative.

Overall, it is expected that most engineers would use the settings highlighted in bold in the table above as a default starting point and then make refinements if necessary.

Extent of Cracking

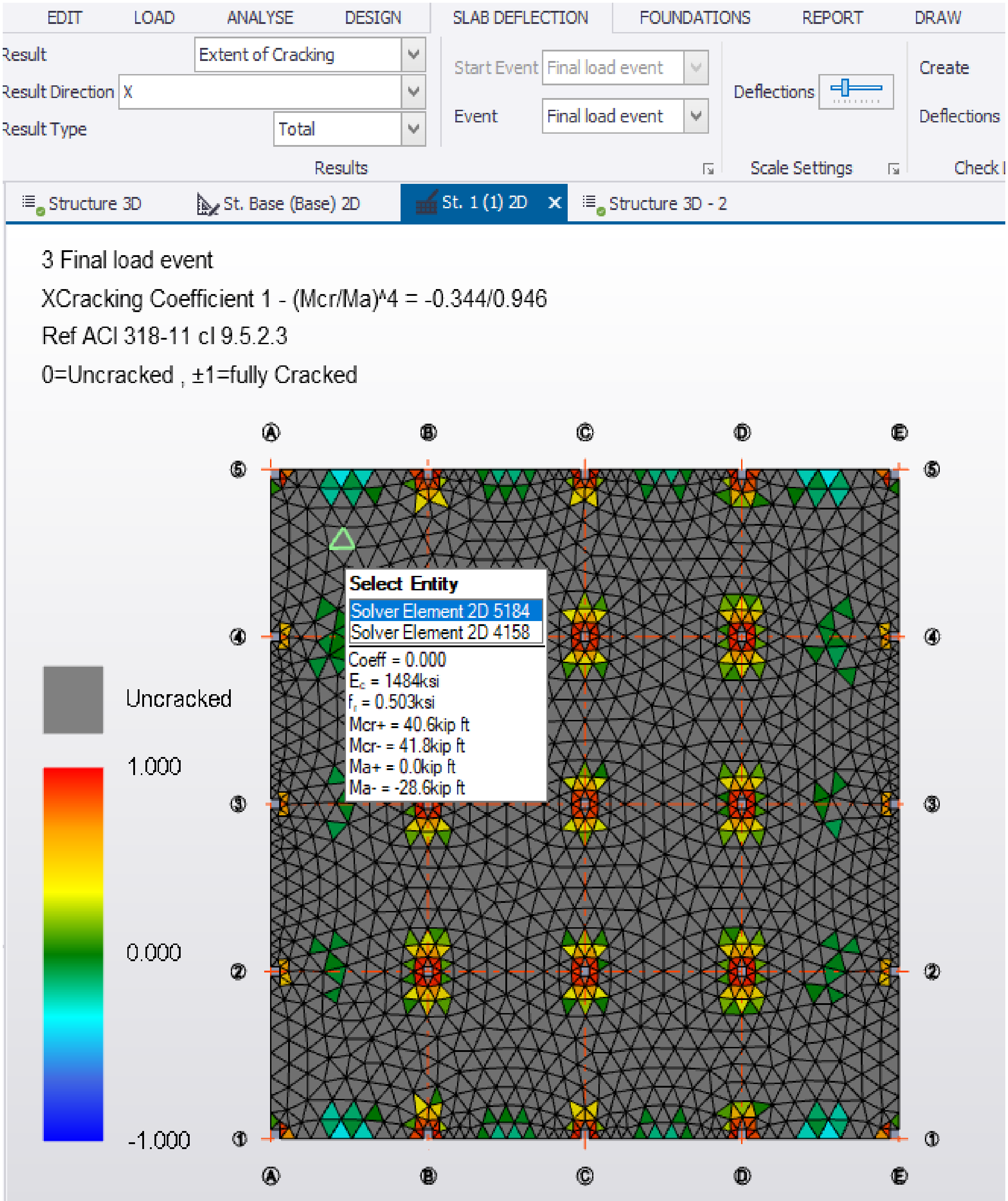
It is clear from the **Combined Table of Results**, that the Restraint Constant and hence the Modulus of Rupture has a huge impact on the predicted deflections. This is due to the impact the modulus of rupture has on the extent of cracking that develops. This is illustrated below where the extent of cracking for the final load event is displayed for the Method 2 model.

Extent of cracking resulting from a restraint constant of 4.0



You can clearly see the majority of the slabs have cracked where the restraint constant is 4.0.

Extent of cracking resulting from a restraint constant of 7.5



The majority of the slabs remain uncracked where the restraint constant is 7.5.

Next steps

- Using results from one of the methods, deflection checks are performed using check lines (page 1501) and an output report is generated.

Use of check lines to check deflections (ACI)

Once the input parameters have been set as required to allow for creep and shrinkage, and the analysis has been performed, check lines can then be placed in order to check the deflections.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Check Lines.tsmd

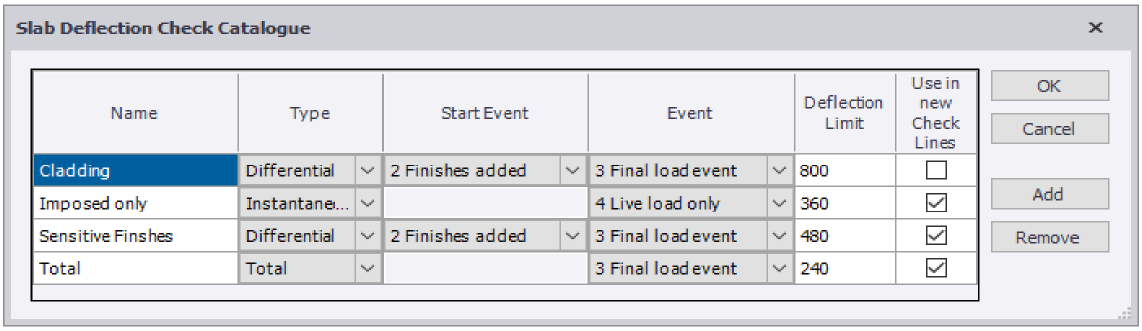
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the Slab Deflection ribbon, click **Deflection Checks**



Name	Type	Start Event	Event	Deflection Limit	Use in new Check Lines
Cladding	Differential	2 Finishes added	3 Final load event	800	☐
Imposed only	Instantane...		4 Live load only	360	☑
Sensitive Finshes	Differential	2 Finishes added	3 Final load event	480	☑
Total	Total		3 Final load event	240	☑

Whilst four checks have been defined above, only three of these have been set to be used in new Check Lines. The more onerous cladding check which does not come from code requirements is not applied by default:

- **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/480
- **Total** will check the total deflections to the final load event against a deflection limit of 1/240

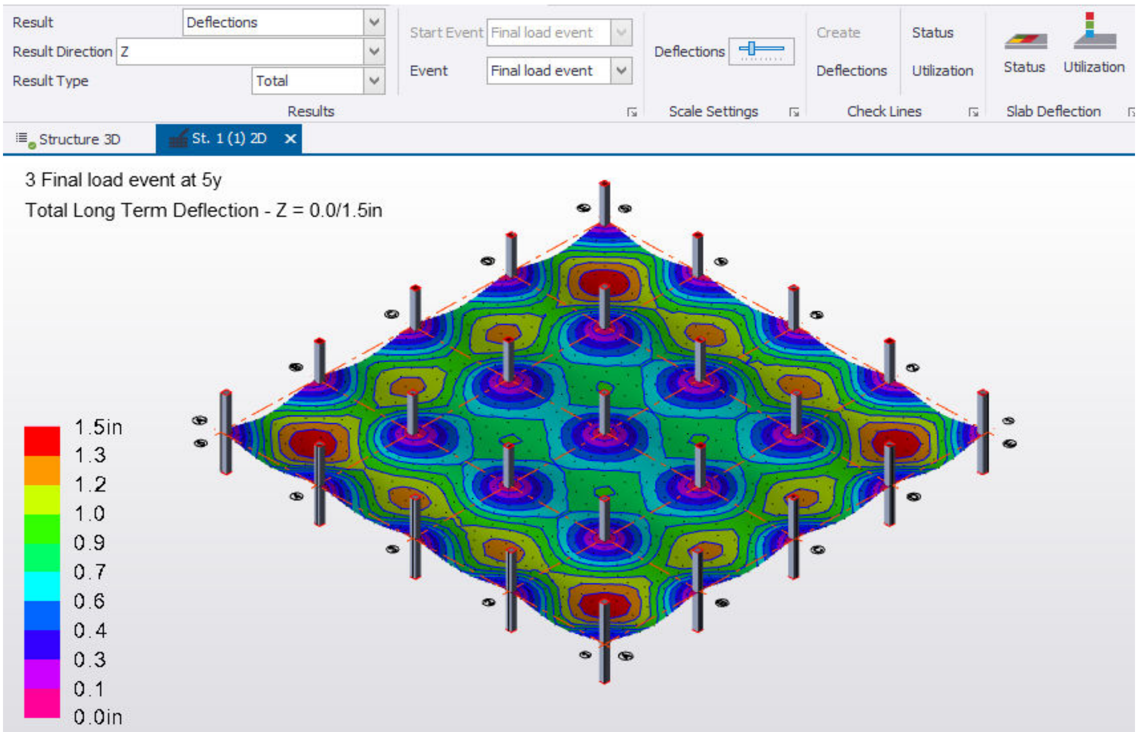
- **Live load only** will check the instantaneous deflections for the live load only event against a deflection limit of 1/360

2. Click **OK** to close the dialog.

Place Check Lines

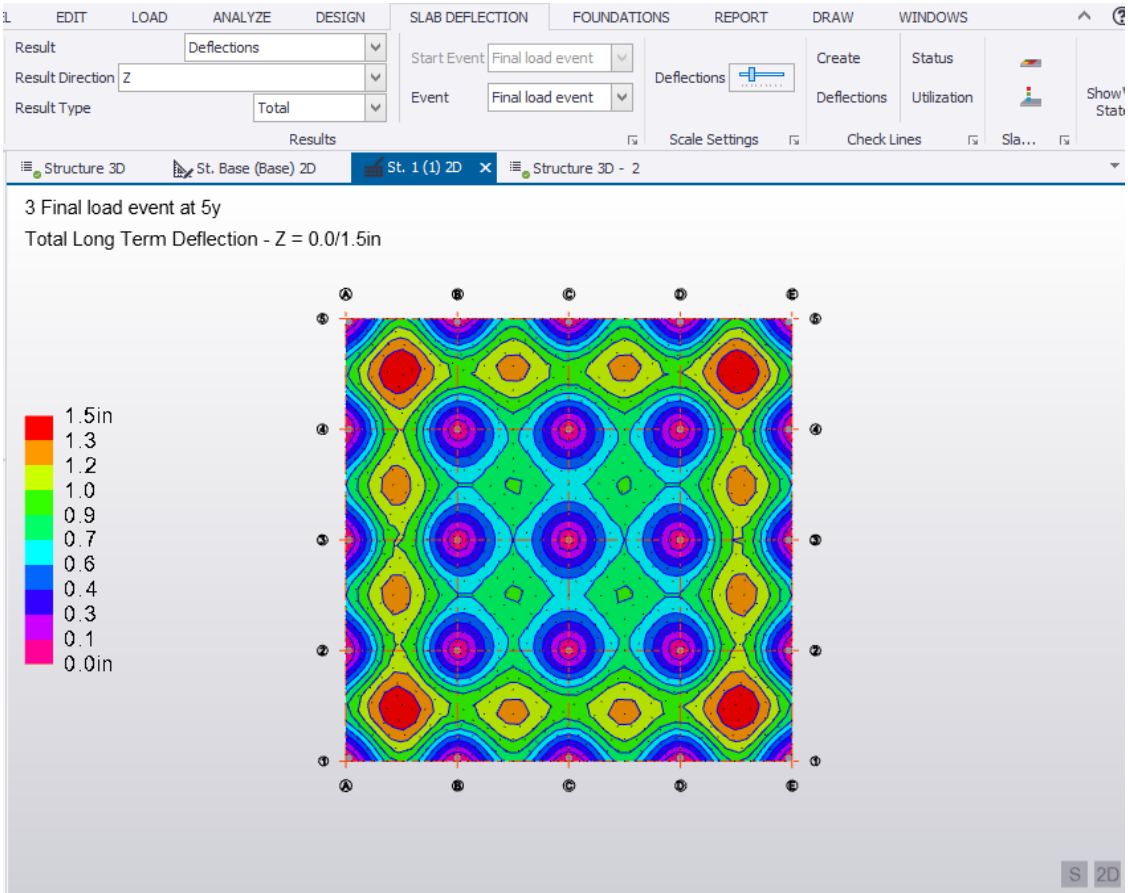
We will define two check lines in this example:

1. Open the **St.1 (1)** 2D plan view



NOTE When the 2D plan view is being viewed in 3D as above, the **Create** button on the **Slab Deflection** toolbar is grayed out.

2. Click 3D in the bottom right corner of the view to toggle the view until it is displayed in 2D.



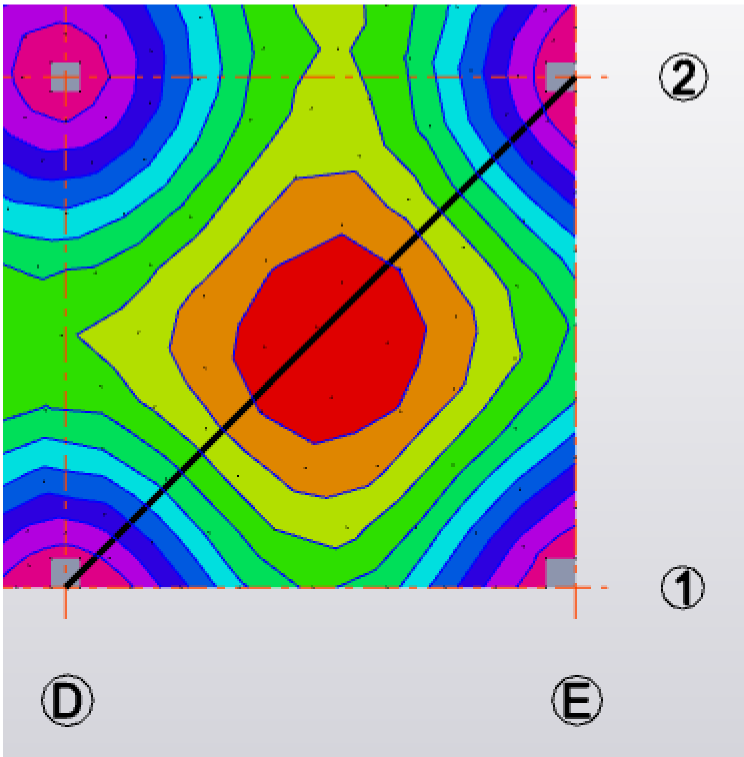
3. From the **Slab Deflection** toolbar, ensure the Results are set as above:
    a. Result - **Deflections**,
    b. Result Direction - **Z**,
    c. Result Type - **Total**,
    d. Event - **Final load event**
4. Click **Create**

NOTE Check lines can only be created in a 2D view.

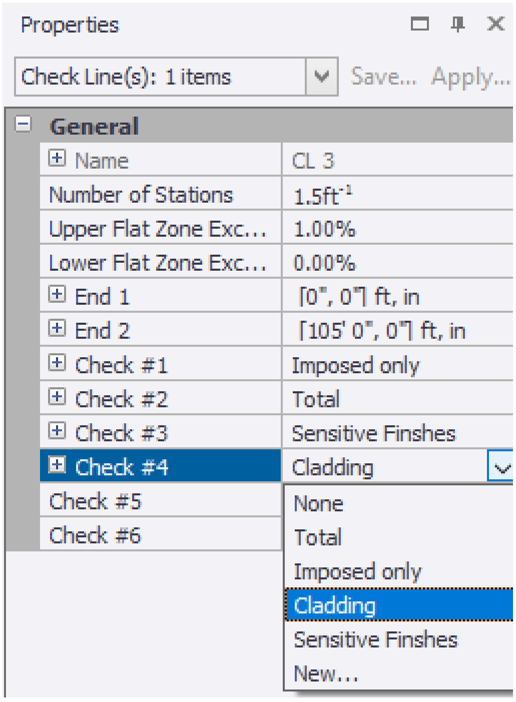
NOTE When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which "Use in new Check Lines" was checked.

---

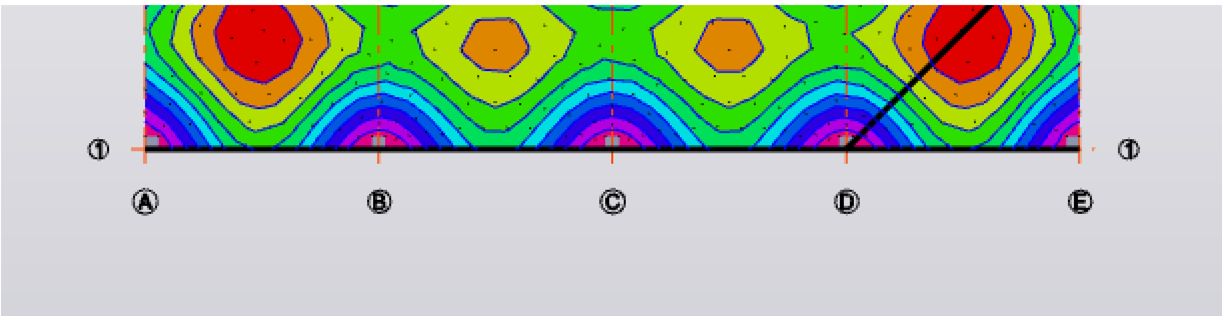
5. To place a check line running diagonally between columns in bottom right corner panel where the peak deflection occurs:
   a. Pick the start point as grid line intersection D/1.
   b. Pick the end point as grid line intersection E/2.



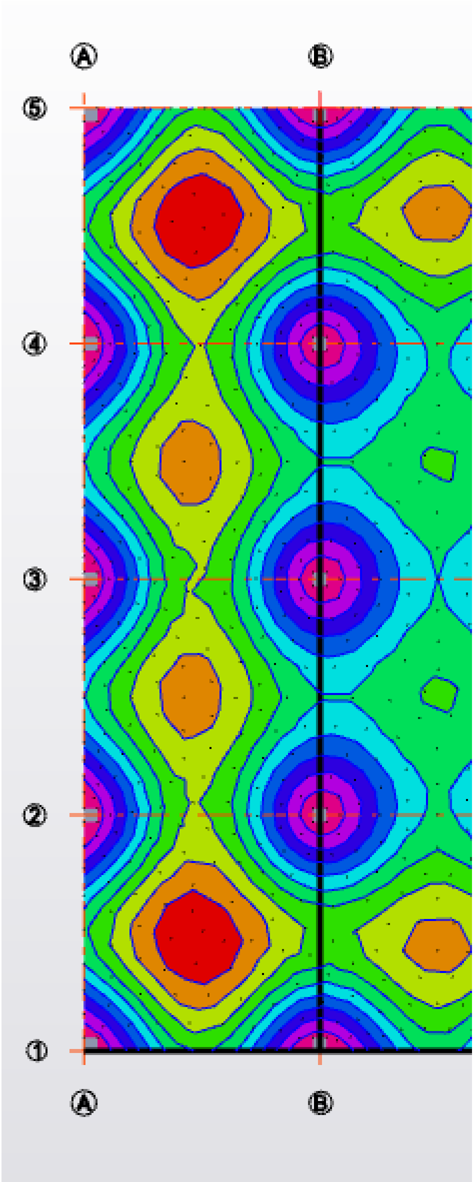
Before creating a second check line, in the Properties Window add a check #3, by selecting Cladding from the droplist.



6. Create the second check line along grid line 1.
   a. Pick the start point as grid line intersection A/1.
   b. Pick the end point as grid line intersection E/1.

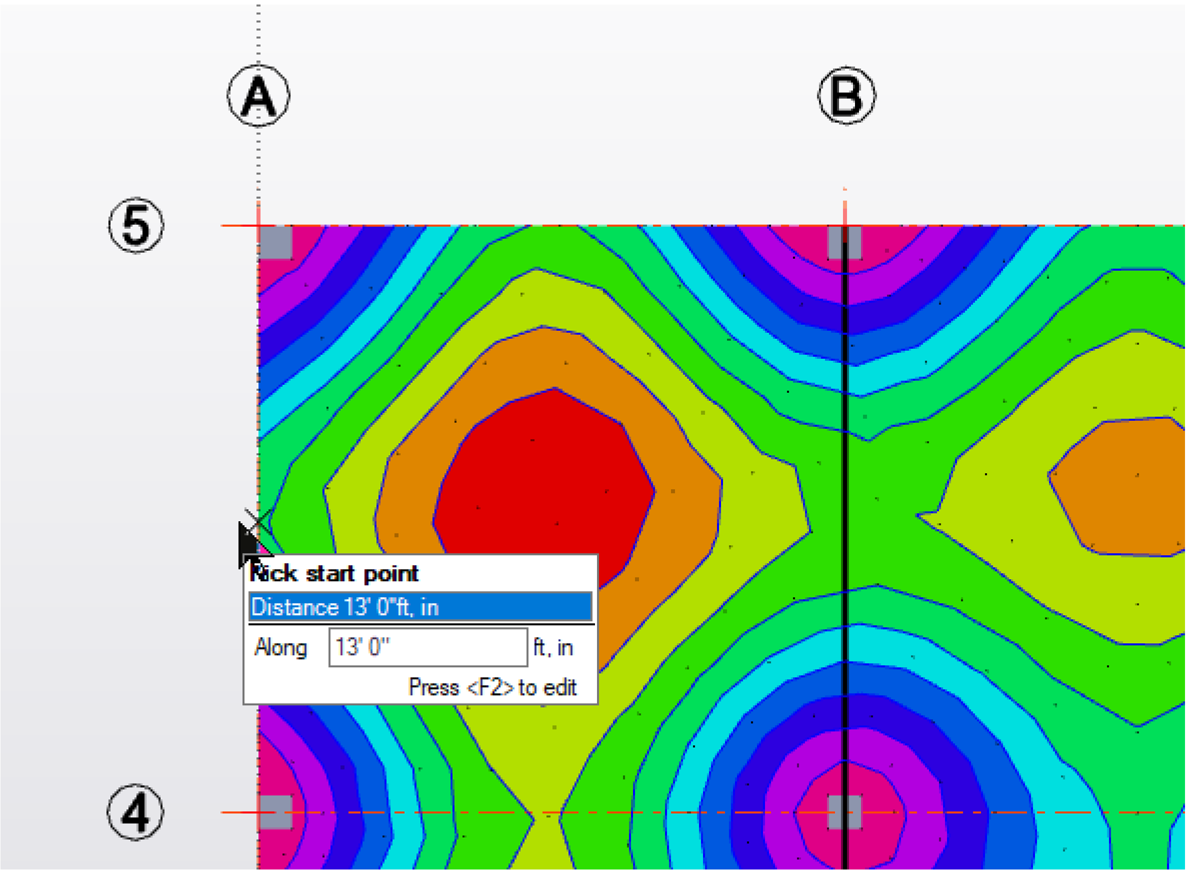


7. Create the third check line along grid line B.
   a. Pick the start point as grid line intersection B/1.
   b. Pick the end point as grid line intersection B/5.

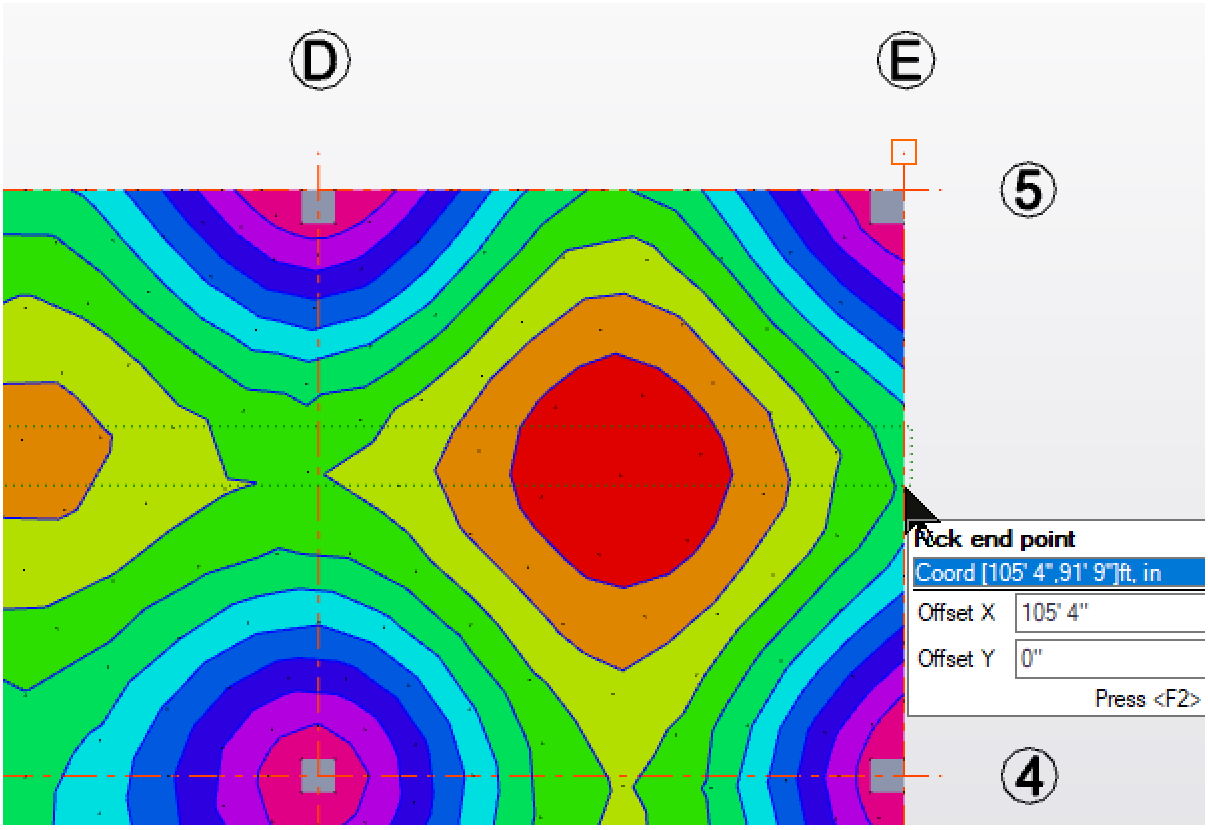


8. Create the final check line mid-way between grid lines 4-5 from grid line A to E.

   a. Pick the start point approximately as shown:



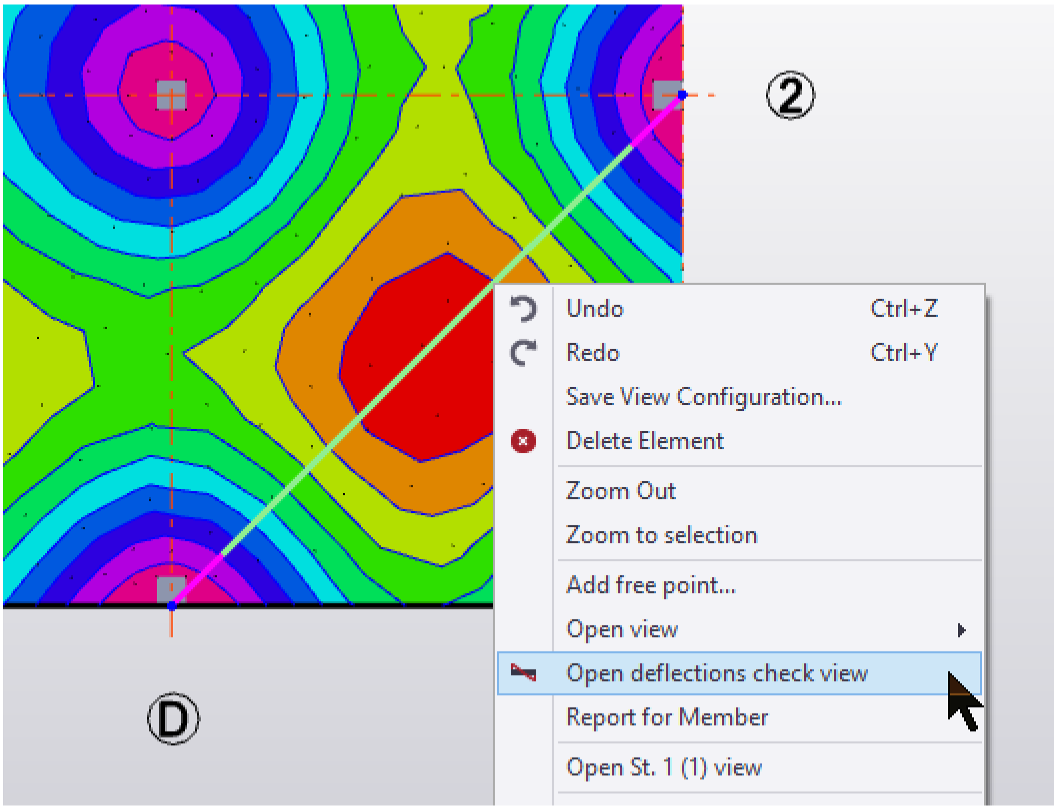
b. Pick the end point approximately as shown:

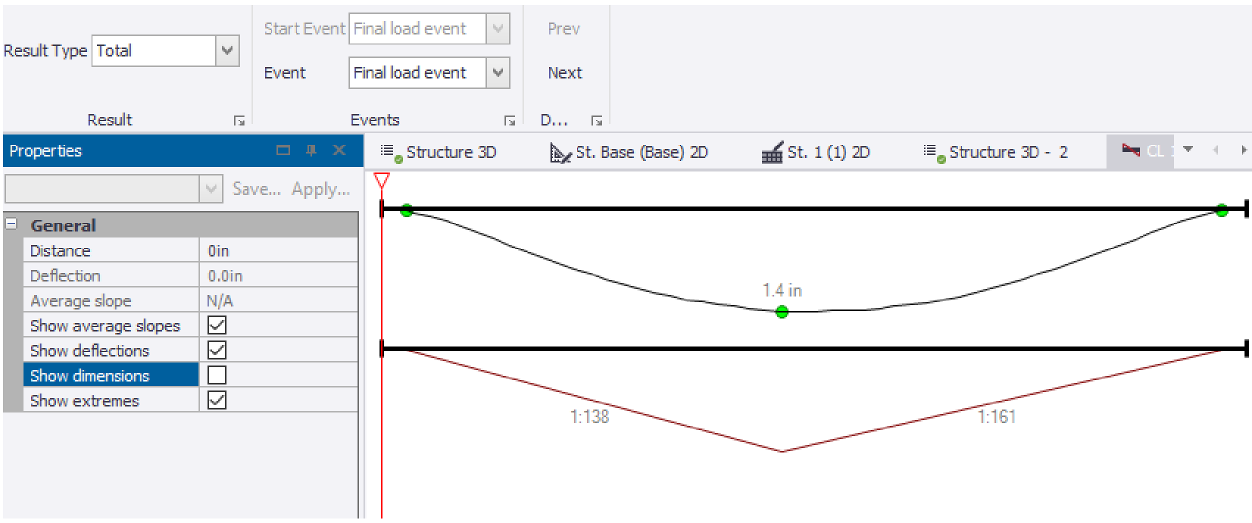


9. Press **Esc** to end the command.

NOTE In a real model you can add as many check lines to the model as you consider appropriate.

10. Right click on the diagonal check line and from the context menu select **Open deflections check view**.



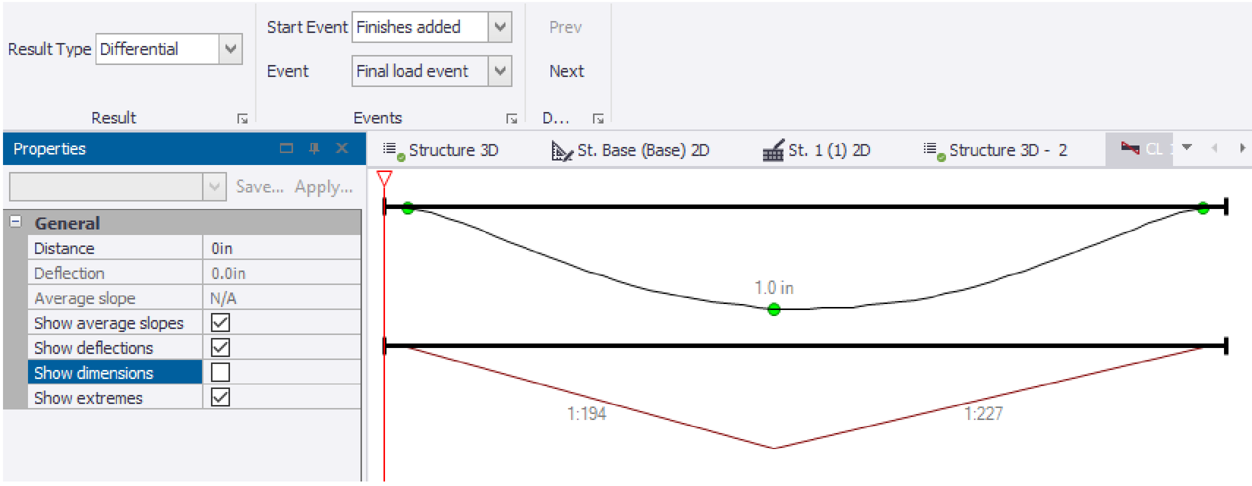


The ribbon allows you to specify the total (as shown above), or differential or instantaneous results for the selected events. Tekla Structural Designer then draws average slopes between maximum and minimum points.

A total deflection limit of Span / 240 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 120. In the view above the average slope between these points is 1 / 138, so the check passes.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

11. Click within the Deflection load analysis view and change the Result type to **Differential** and check deflection and slopes between the **Finishes added** and **Final load event**. The resulting differential deflection is as shown:



For the Sensitive Finishes check of differential deflection between finishes added and the final load limit had a limit of Span / 480, which is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 240. In the view above the average slope between these points is 1 / 194, so this check fails.

Generate Check Line Reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **St.1 (1) 2D** view, right click on the diagonal check line and select **Report for Member**

Deflection Checks Summary

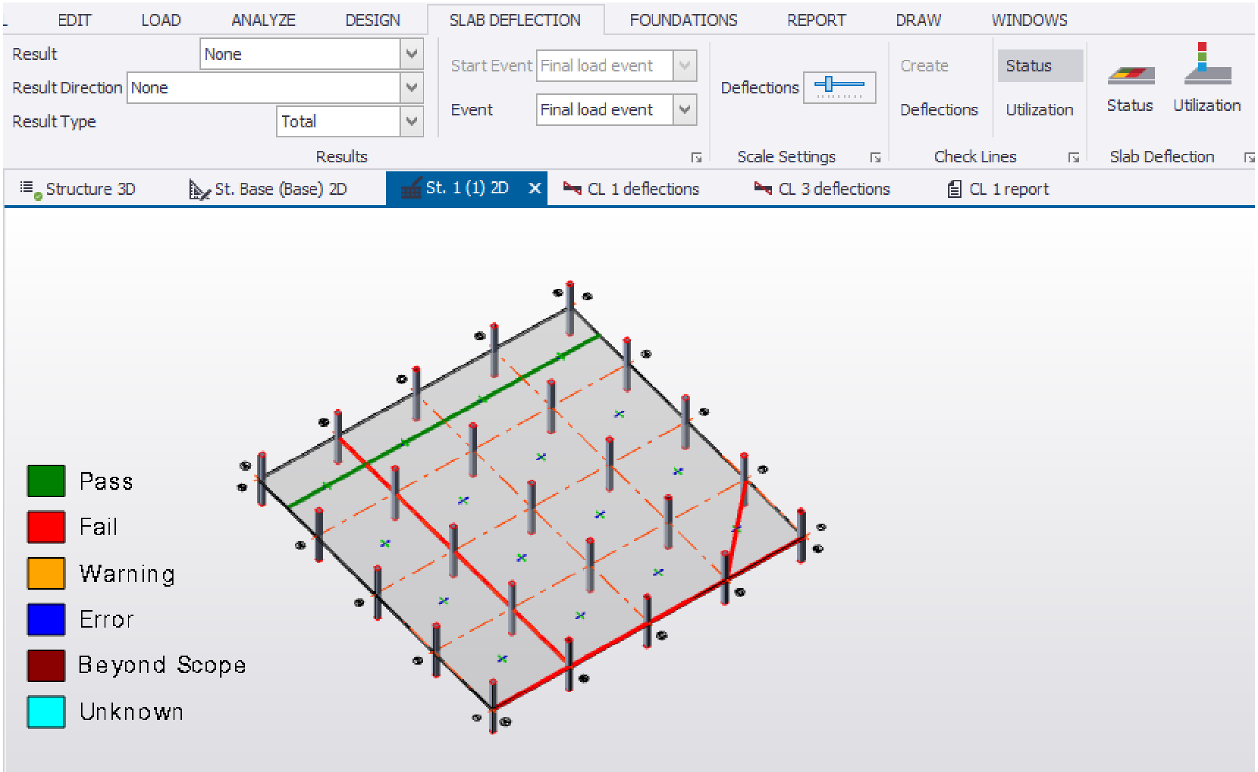
Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	193 33/64	1 : 811	✓ Pass	0.222
Sensitive Finshes	480	1 : 240	1.0	193 33/64	1 : 194	✗ Fail	1.237
Total	240	1 : 120	1.4	193 33/64	1 : 138	✓ Pass	0.870

As previously stated, the Sensitive Finishes check fails since the slope for differential deflection is reported as 1:206 which is greater than the allowable slope limit of 1:240.

Review Check Line Status and Utilization

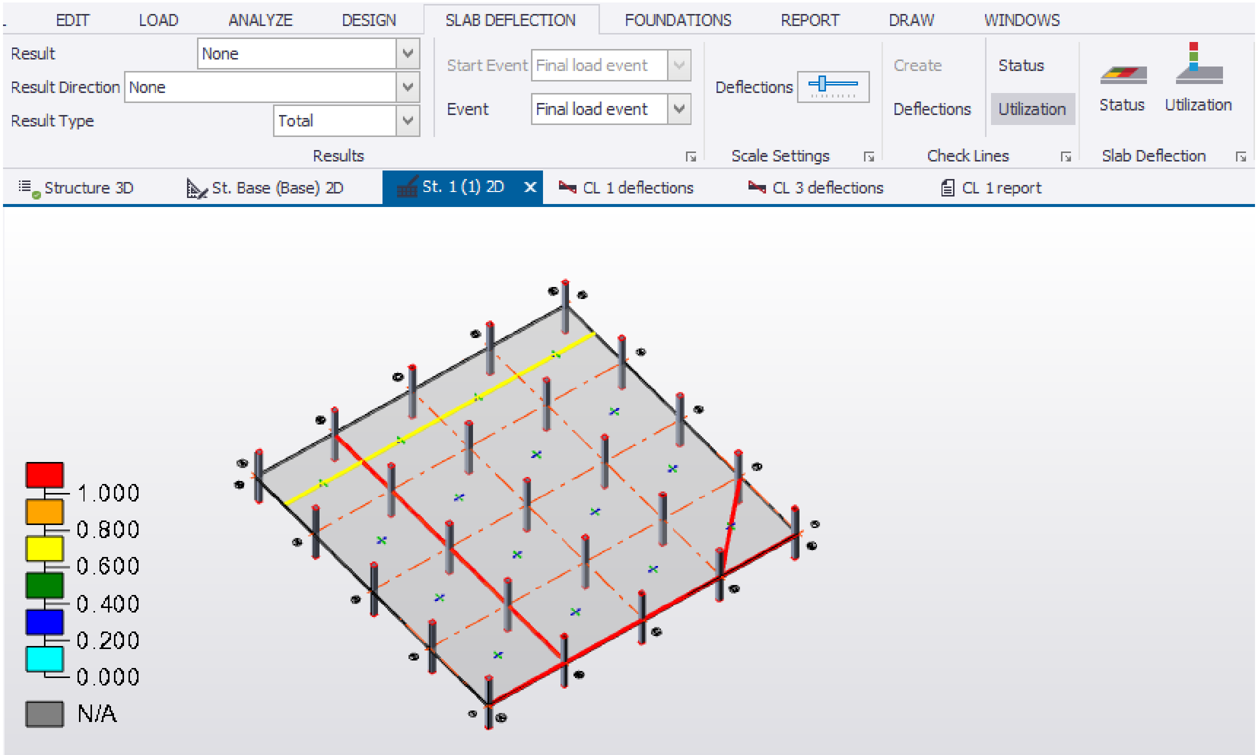
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the **St.1 (1) 2D** 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



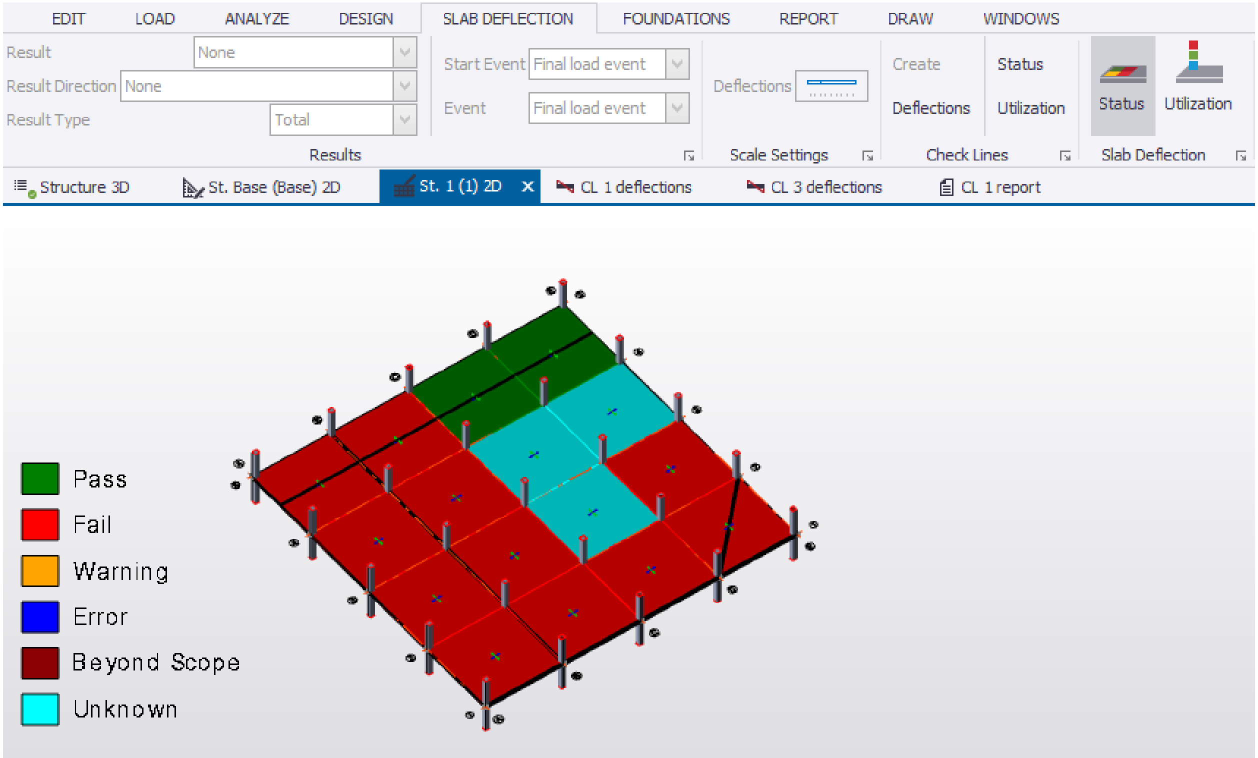
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

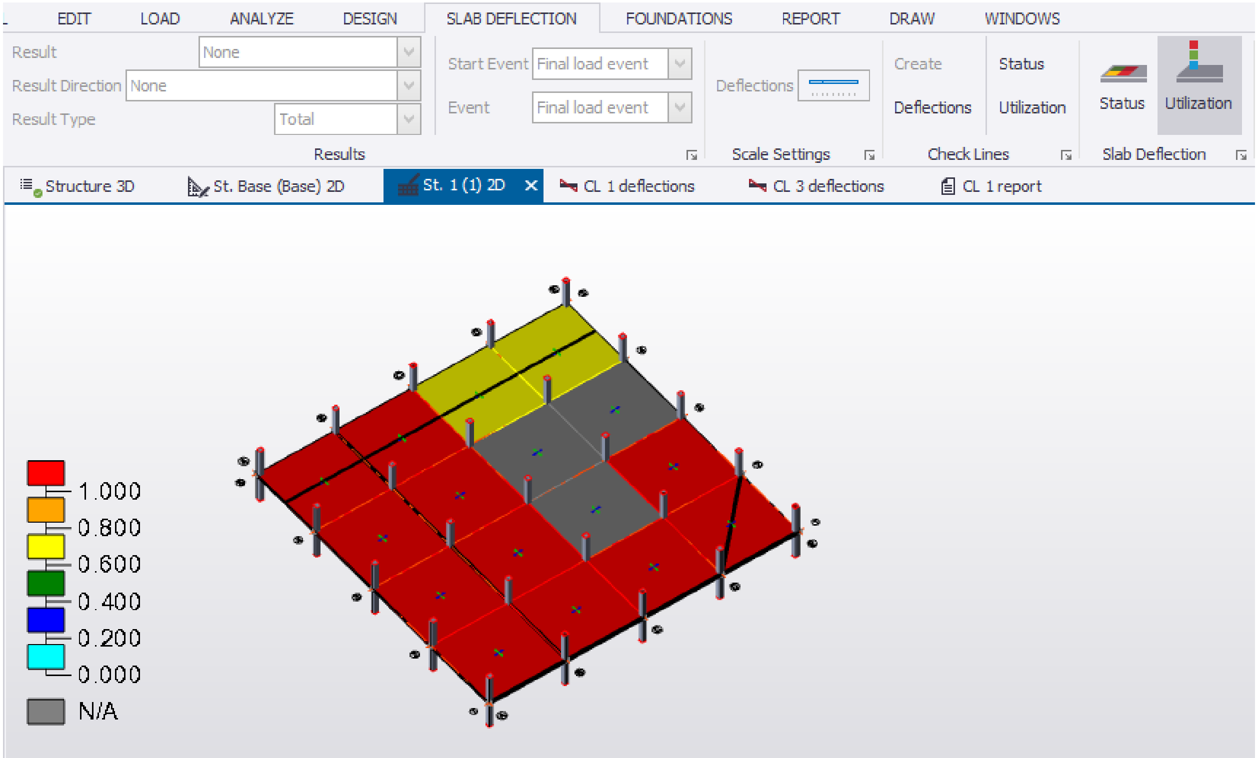
1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/ fail status of each slab.



- No check lines cross the majority of the slab items so they currently have a status of Unknown.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the 4 slab items crossed by the horizontal check line Fail.
- One slab item is being crossed (only just, in one corner) by the passing diagonal check line, this is the only slab to have a pass status.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

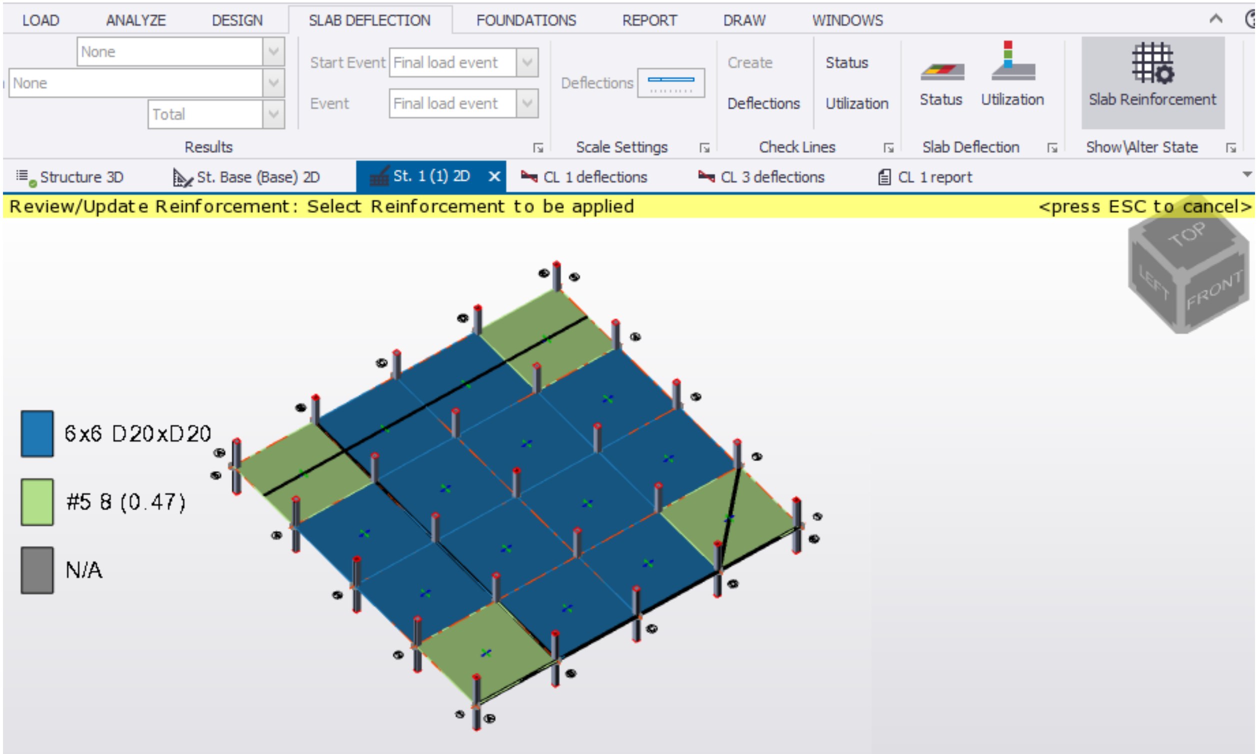
Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

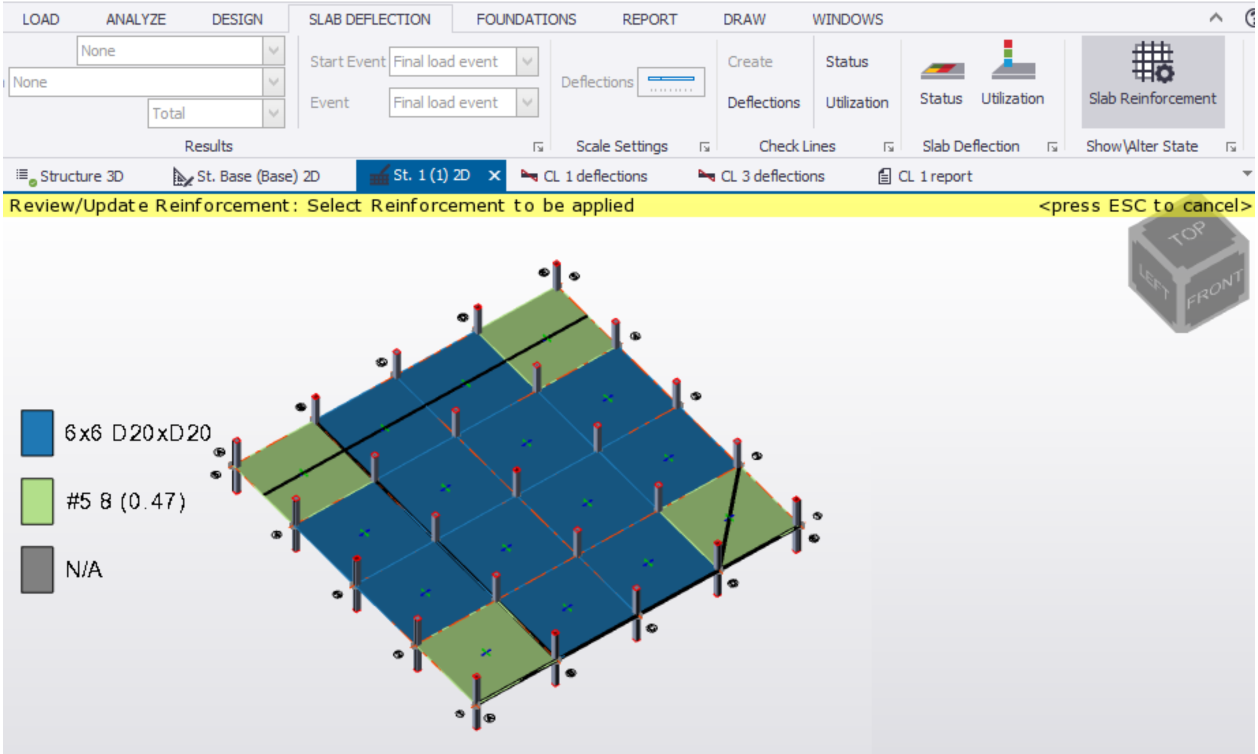
The analysis is extremely quick and since everything is contained within one model file, it allows “What If” scenarios to be considered to find the optimum solution.

In this exercise we will start by looking at the impact of adjusting the reinforcement.

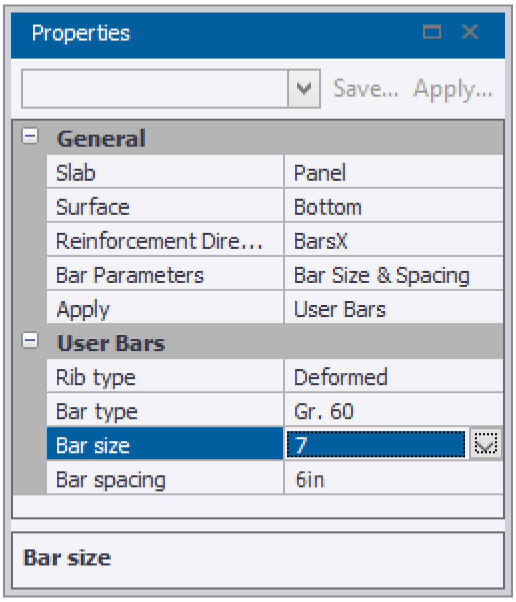
1. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.



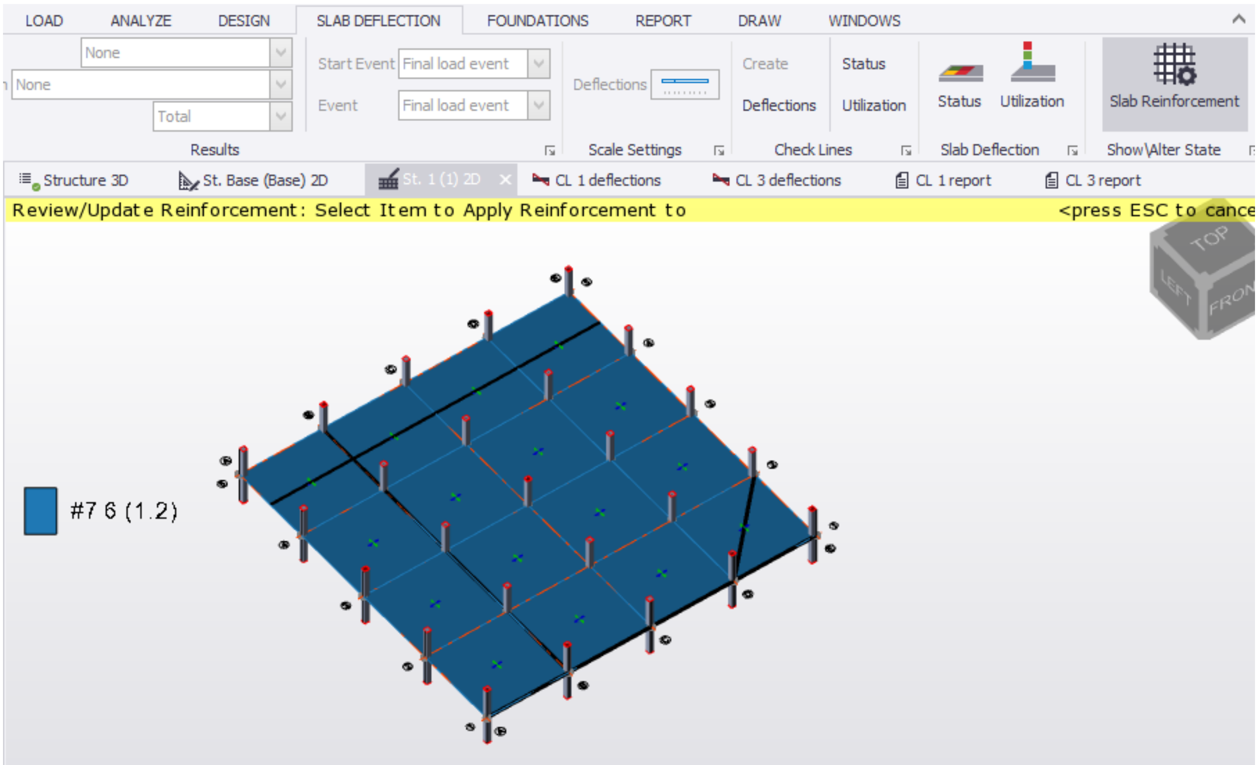
2. In the Properties Window:
    a. Leave the Slab Reinforcement to modify as **Panel**
    b. Leave the Slab Layer to modify as **Bottom**
    c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



3. In the Properties Window:
    a. Leave the Slab Reinforcement to modify as **Panel**
    b. Leave the Slab Layer to modify as **Bottom**
    c. Change Apply to **User Bars**
    d. Change Bar size to **7**
    e. Change Bar spacing to **6in**

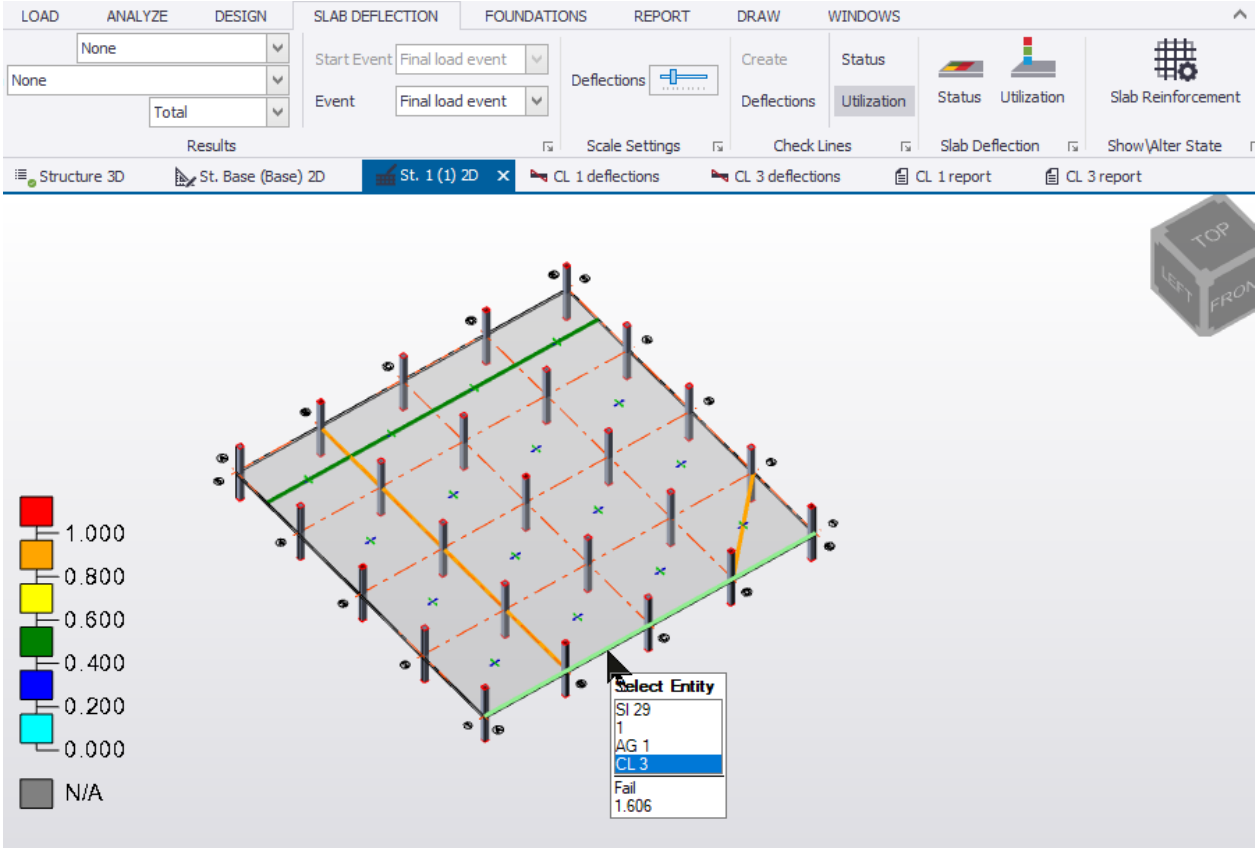


4. Click on each panel to apply the new reinforcement.



5. Repeat the above process to apply the same bars in the X reinforcement direction also.
6. Click **Analyze Current** to update the results

7. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
8. Investigate the tooltip results

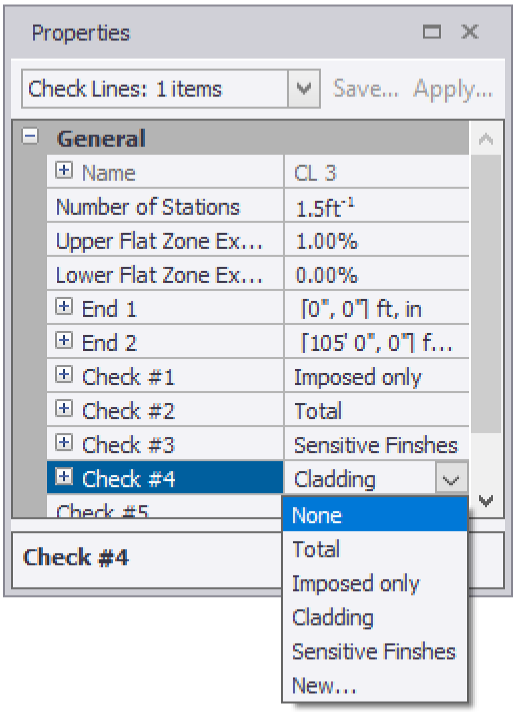


We can clearly see improvements in the results, however the check line along grid 1 with the more onerous Cladding deflection limit is still failing.

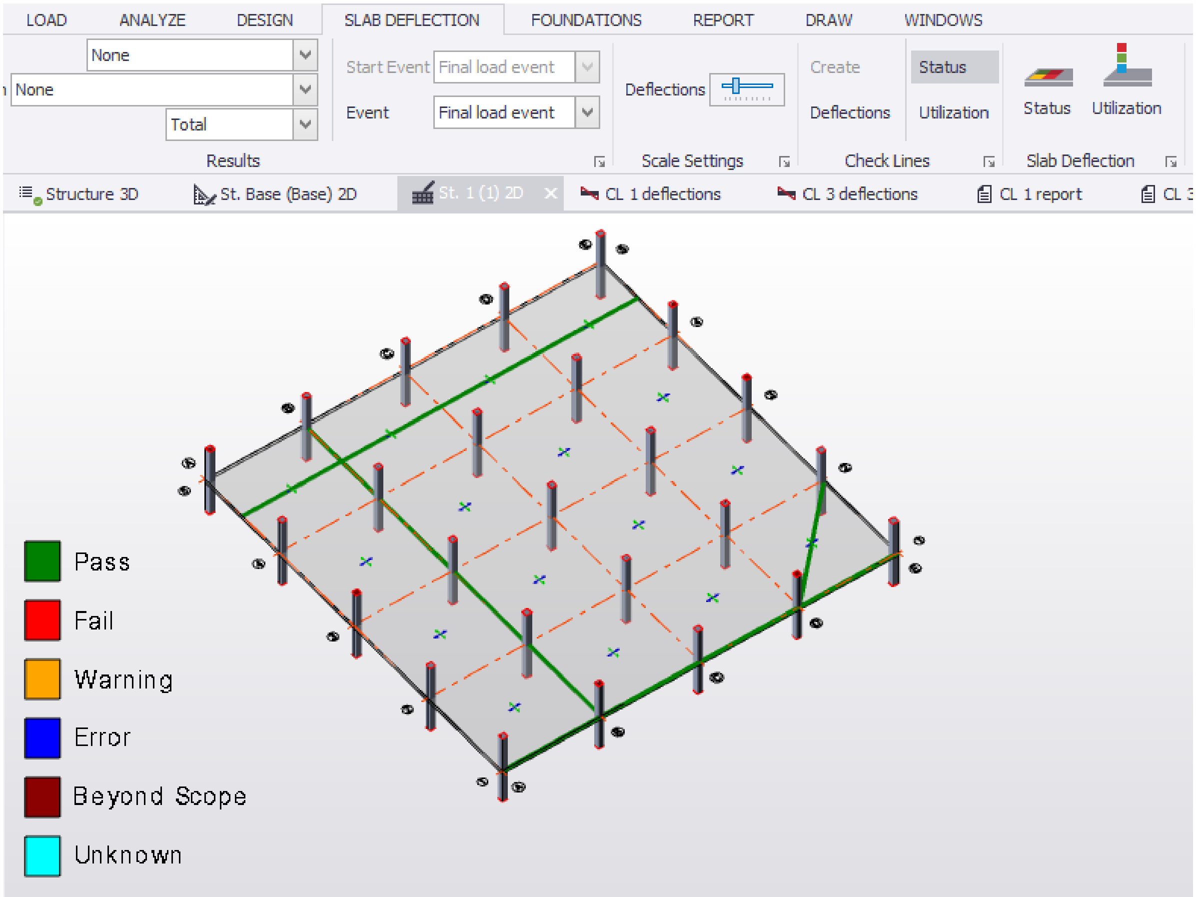
At this point in a real model you could begin to look at the impact of the various other input parameters, such as changing the concrete grade.

For the purpose of this exercise, and as the cladding check is not a code requirement, we will simply disable it, as follows:

9. Select the check line along grid 1
10. In the Properties window, reset Check #4 to **None**.



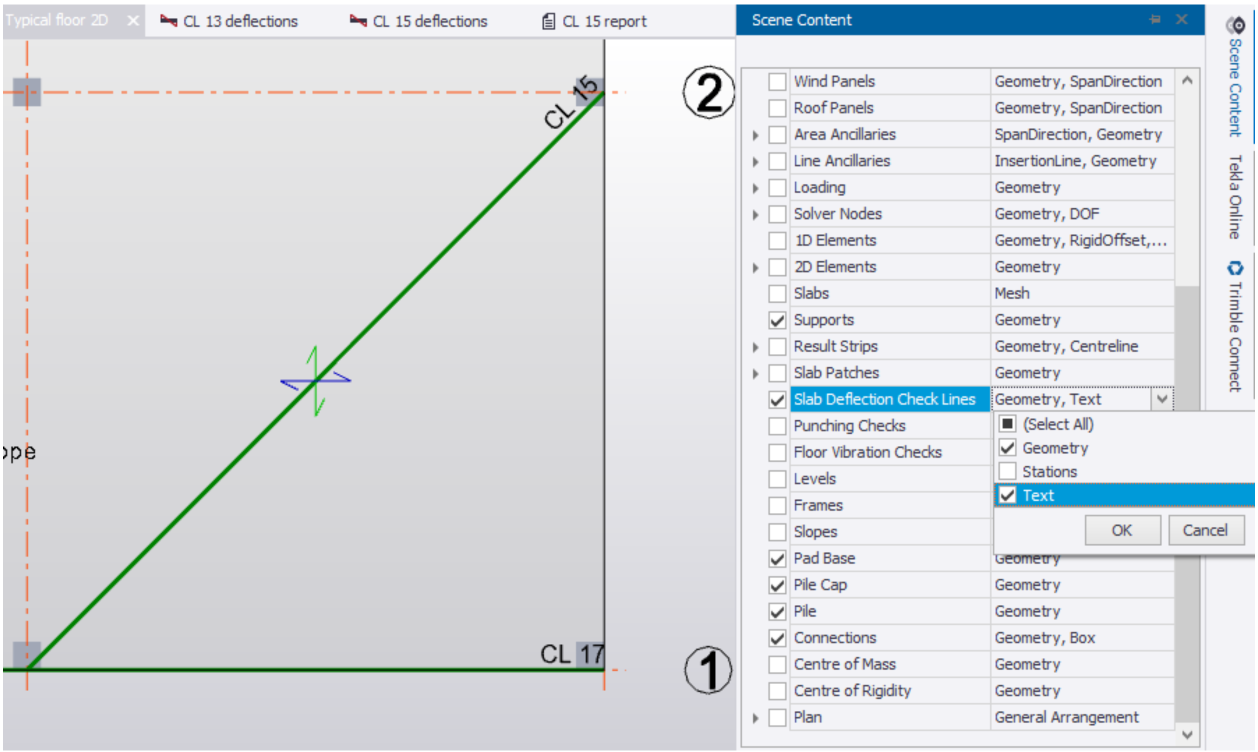
11. Click **Analyze Current** to update the results
12. Click **Status** in the Check Lines group. All of the check lines should now pass.



Generate Model report

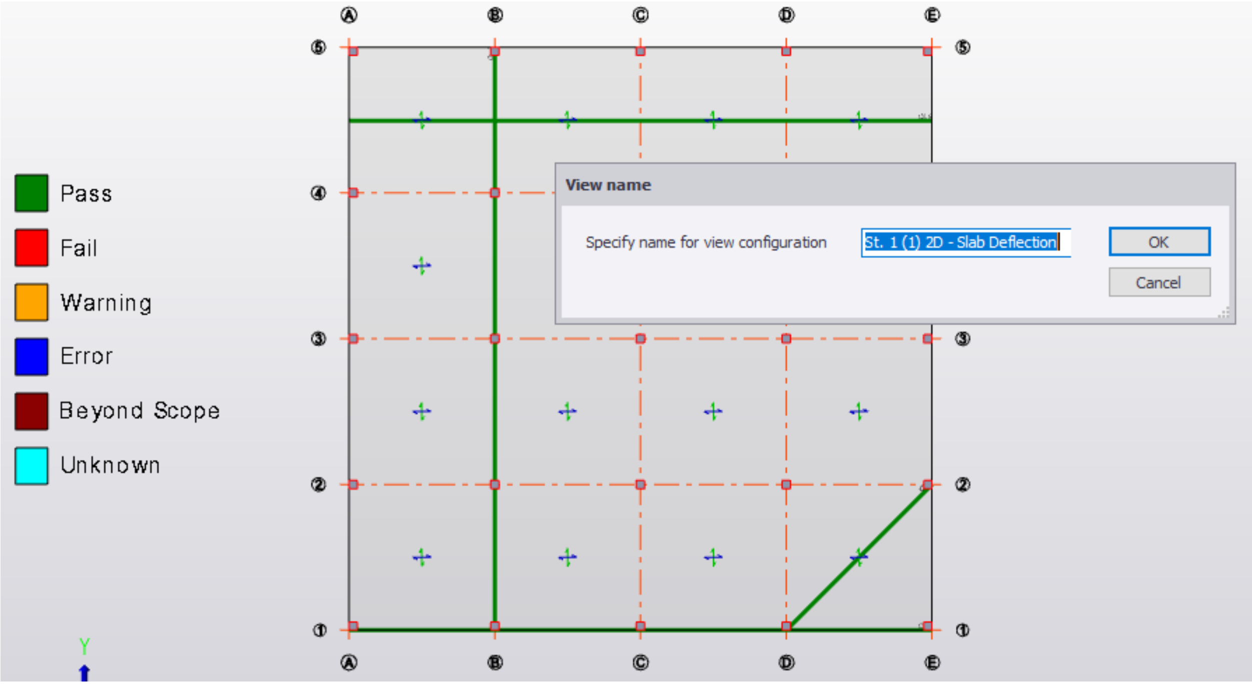
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



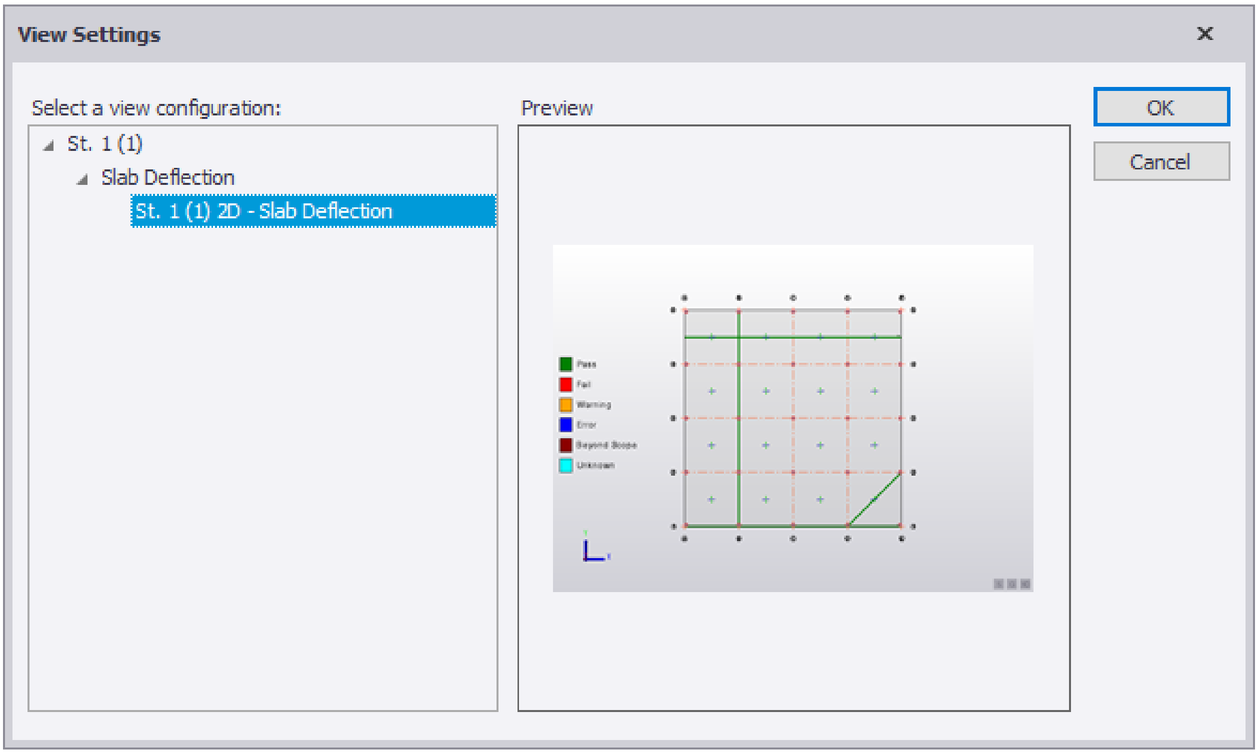
TIP The check line references can be customized using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

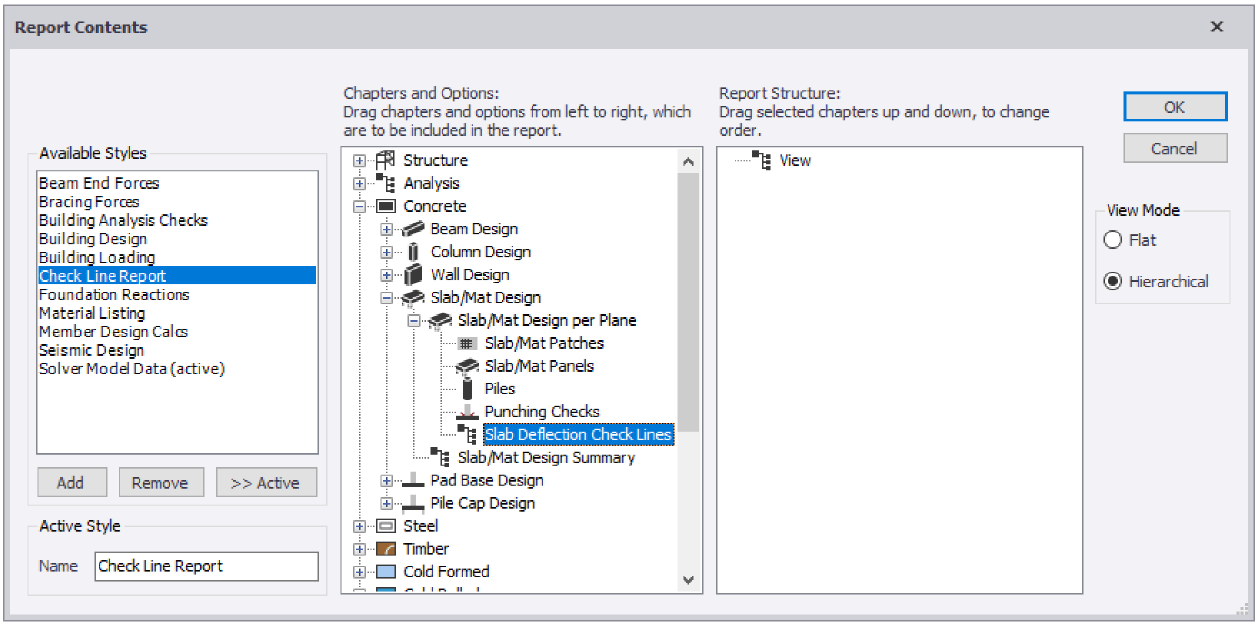


This saved view can be included in the Check line report.

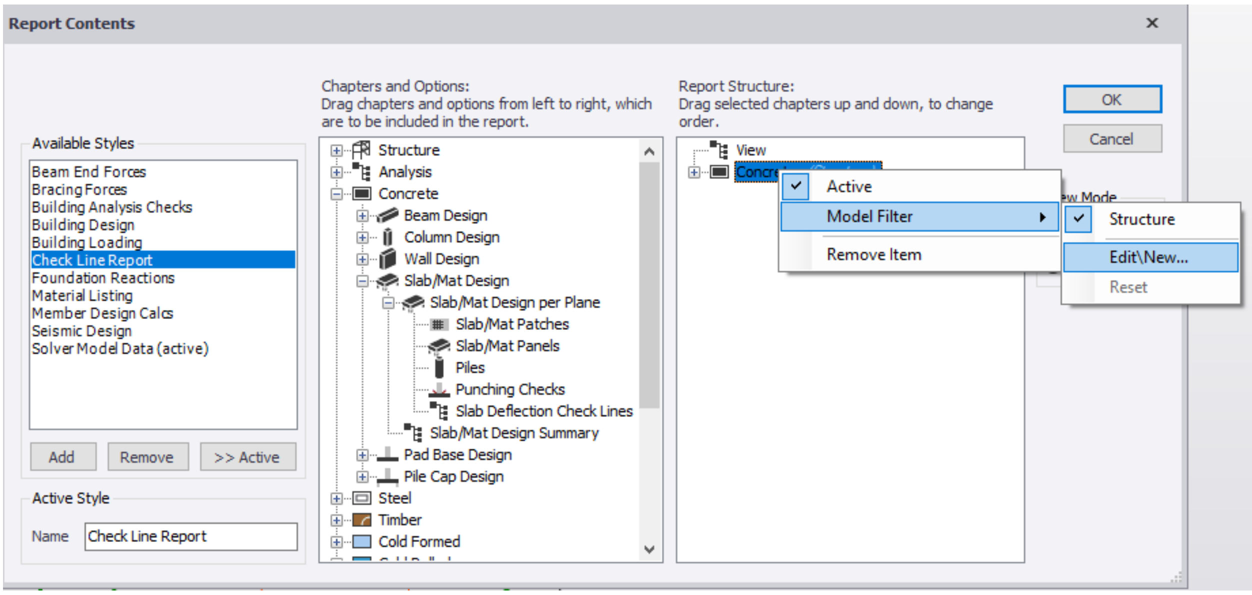
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In the Report Structure, right click **View**, then choose **Settings...**
7. Select the Slab Deflection view you created earlier



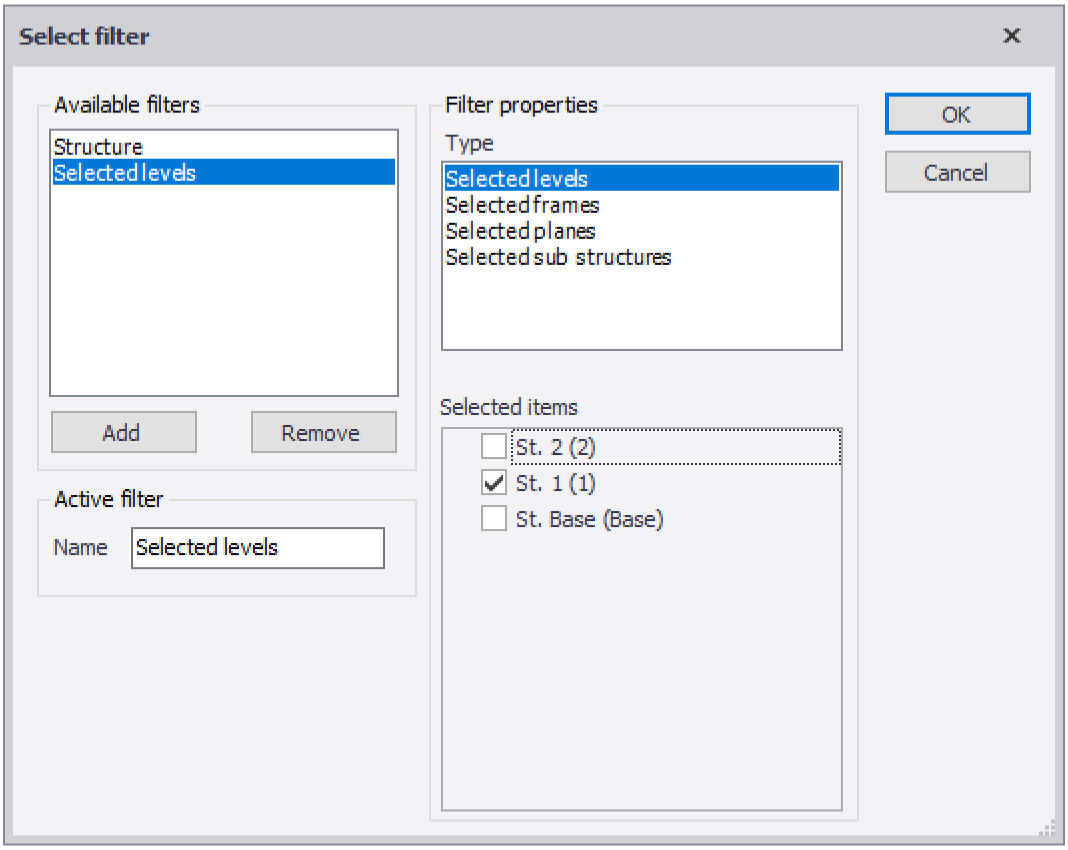
8. Click **OK** to return to the Report Contents dialog
9. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



10. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**

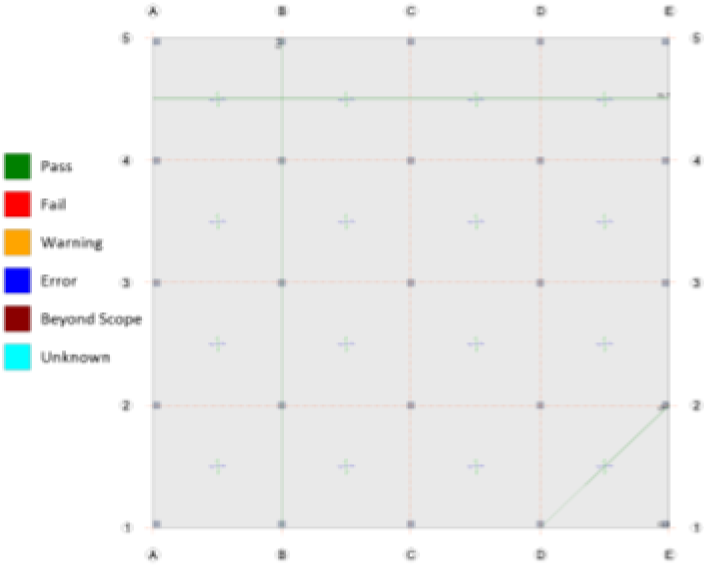


11. In the Filter dialog, click **Add** and select Selected levels and ensure a check against **St. 1 (1)**



12. Click **OK** to return to the Report Contents dialog.
13. Click **OK** to exit and save the report.

    A report structure called **Check Line Report** has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select **Check Line Report**
    b. Click **Show Report** to open the report.



St. 1 (1) 2D - Slab Deflection

<u>Concrete</u>

Slab/Mat Design per Plane

St. 1 (1)

Slab Deflection Check Lines

CL 1

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	201 49/64	1 : 1087	✓ Pass	0.166
Sensitive Finshes	480	1 : 240	0.7	201 49/64	1 : 273	✓ Pass	0.879
Total	240	1 : 120	1.1	201 49/64	1 : 185	✓ Pass	0.649

CL 3

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	-0.1	144 15/32	1 : 1232	✓ Pass	0.146
Total	240	1 : 120	-0.8	144 15/32	1 : 183	✓ Pass	0.656
Sensitive Finshes	480	1 : 240	-0.6	144 15/32	1 : 254	✓ Pass	0.945

CL 5

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.1	135 15/32	1 : 1059	✓ Pass	0.170
Sensitive Finshes	480	1 : 240	0.5	135 15/32	1 : 284	✓ Pass	0.845

13.8 Precast member design handbook

Precast structures can be modelled and analysed in Tekla Structural Designer. Precast beams and columns can then be designed, provided that you have access to a licence of Telka Tedds.

This handbook describes the workflow required in Tekla Structural Designer for running the Telka Tedds precast calculations.

NOTE The following limitations apply:

- Only design of normal weight precast concrete beams and columns to Eurocodes is supported.
- Design of precast concrete slabs and walls is beyond scope for all head codes.
- The process is only applicable to precast structures that don't make use of in-situ structural toppings.
  - Precast construction which adopts a more hybrid construction involving the use of in-situ toppings exceeds the limitations of the Tekla Tedds calculations and should therefore be considered beyond scope.

The following topics are covered:

- Precast member design workflow (page 1528)
- Precast member design groups (page 1538)
- Precast beam design (page 1540)
- Precast column design (page 1554)
- Precast column connection eccentricity moments (page 1558)
- Precast member design commands (page 1563)

Related video

Precast modelling, analysis and design using Tekla Tedds

Precast member design workflow

The basic workflow for precast design in Tekla Structural Designer is described in the sections below:

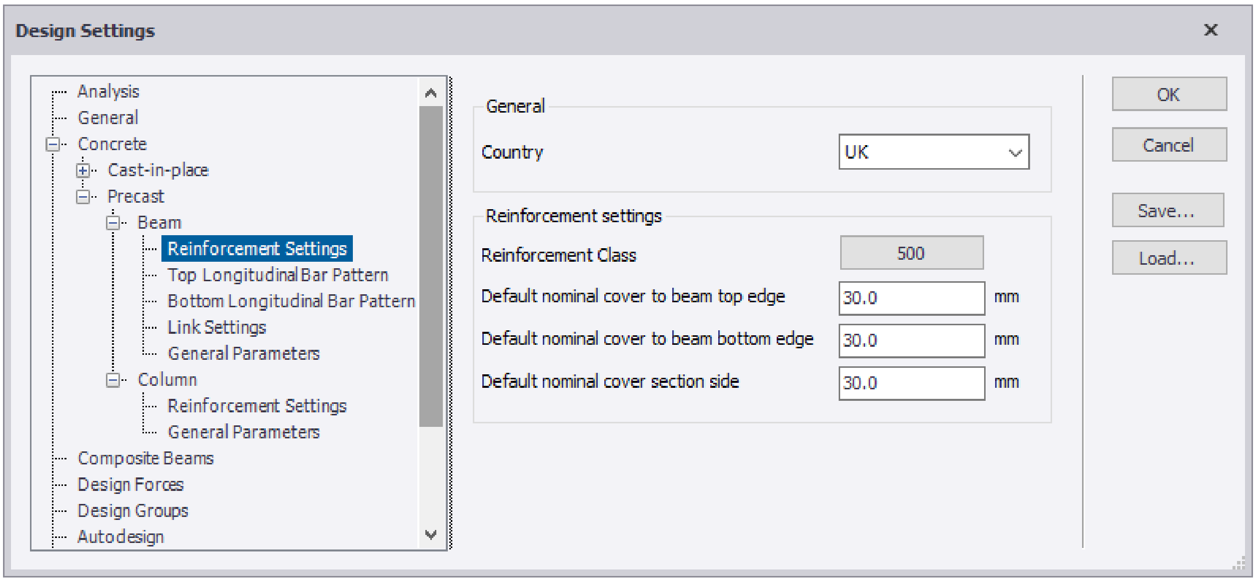
Related video

Precast modelling, analysis and design using Tekla Tedds

Configure precast beam and column design settings

Specific precast beam design settings (page 2320) and precast column design settings (page 2322) that apply to all the beams and columns in the model should be specified prior to running the designs.

These are set on the repective **Concrete>Precast** pages of the **Design Settings** dialog.

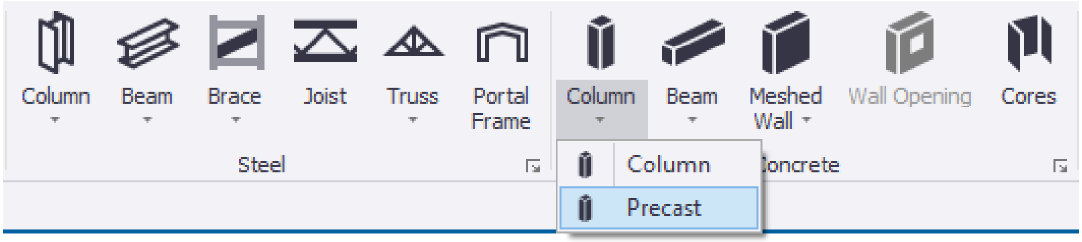


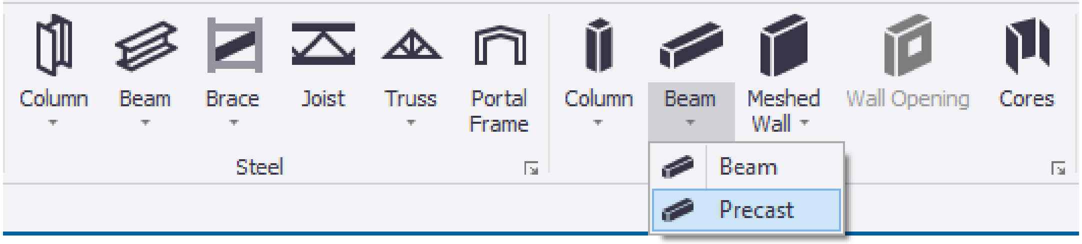
By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds precast calculation as it is run.

You still have control within individual calculations to adjust these settings on a member by member or group basis.

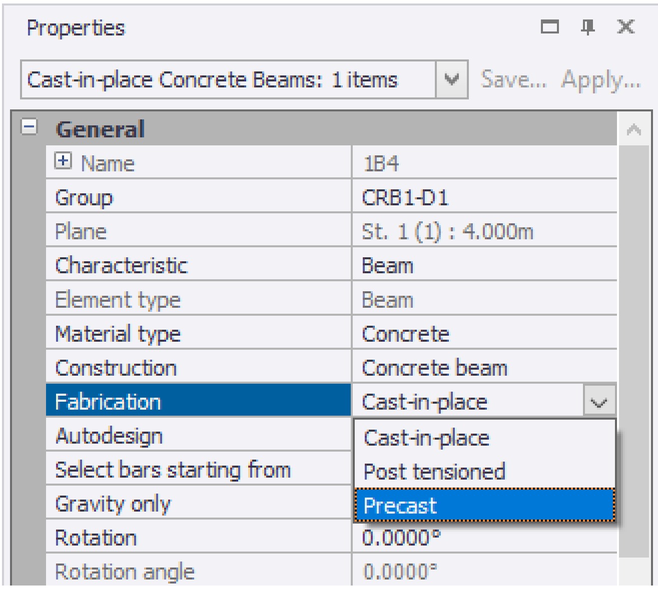
Define and place precast members

To place precast members, simply use the **Concrete Column > Precast**, or **Concrete Beam > Precast** commands on the Model tab.



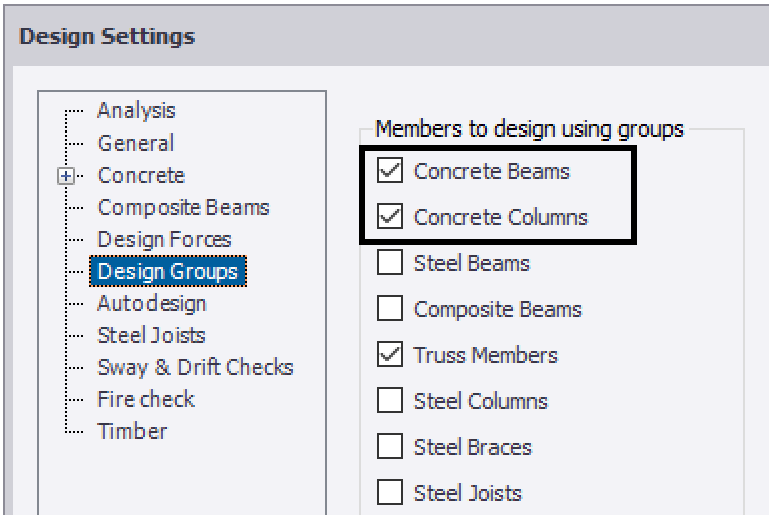


An alternative to this would be to use the **Concrete Column > Column**, or **Concrete Beam > Beam** commands and to subsequently change the Fabrication parameter to be Precast.



Configure precast groups

Customizable member groups are created automatically. You can choose whether to utilize them for design purposes via the **Design Groups** page of the **Design Settings** dialog.



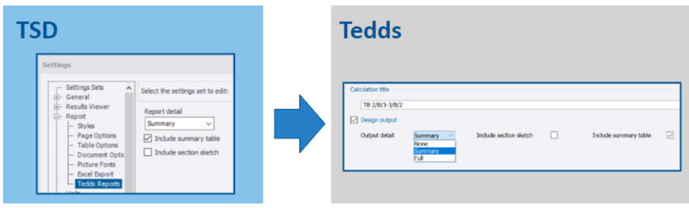
Using precast member design groups (page 1538) can help speed the overall design process, particularly so in the case of medium to large size models.

- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

The initial precast member design groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can move members into new groups as required.

Set the Tedds results output level

You can choose the output level for the Tedds precast calculations in advance by clicking **Home > Settings > Report > Tedds Reports**



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds precast calculation as it is run.

The setting applies to all Tedds linked calculations (precast and timber).

Establish design forces by running the analysis

Precast members can only be designed provided a set of analysis results exist. These can be generated from the **Analyze** ribbon by running Analyze All (Static) (page 626).

A design force envelope is established, and all critical load combinations are considered in one Tedds calculation.

Provided load combinations have been created, once analysis has been performed the **Design using Tekla Tedds** options become available.

Design using Tekla Tedds

NOTE We recommend that the Tekla Structural Designer model is developed as much as possible prior to considering the design of the precast members. This should ensure that the correct distribution of forces is carried through into the Tekla Tedds calculations.

Design a Selection

To design several precast members or groups in one go:

1. Select the precast members you want to design.
2. Right click and select Design using Tekla Tedds> Selection (page 2260)
3. Design the selected members.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whther ot not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

4. When changes are made to the model, you can check if the existing sections are still sufficient by by running
5. Check In Tedds (page 2182) from the Design tab.

   NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data (page 2247) from the right-click context menu.

Design a group

If you have activated concrete member design groups, each group can be designed as follows:

1. Highlight any member in the group, right click and select Design using Tekla Tedds> Group (page 2259)

   This design gathers the analysis results for all members in the group and collates them in to one design. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

2. Design the selected member for these loads.
3. Click **Finish**.

   If the section size was changed during the design, all members of the group will be updated to the new section. If the reinforcement data was changed this is also copied to all group members.

   Irrespective of whether the section or reinforcement has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

4. In the Review View, review the utilization ratios (page 850) for all members in the group - if these all indicate an efficient design, the process can stop at this point.

   NOTE For columns in particular, the envelope of design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

5. If a group member has a lower than desired utilization, right click on it and select Design using Tekla Tedds> Member (page 2258)
6. In the Review View, review the resulting utilizations for the group members - note that some of might now fail.
7. Using the new utilizations to better inform you choice, add extra groups (page 268) as necessary and re-allocate the members between the groups.
8. Run Design using Tekla Tedds> Group (page 2259) for one member in each of the new groups.
9. Iterate the process, starting from step 5 above.
10. When changes are made to the model, you can check if the existing designs are still sufficient by by running Check In Tedds (page 2182) from the Design tab. Alternatively you can check groups individually by running Check using Tekla Tedds> Group (page 2245) for one member in each group.

NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data (page 2247) from the right-click context menu.

Ungrouped Design

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member (page 2258)
2. Design the selected member.
3. Continue to design additional members in the same way as required.
4. When changes are made to the model, you can check if the existing sections are still sufficient by by running Check In Tedds (page 2182) from the Design tab.

NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data (page 2247) from the right-click context menu.

Design model

If you want to design every precast (and timber) member in the model in one go:

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Design using Tekla Tedds > Model**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check the design after changes

If changes are made to the model you can run a 'check' design to determine if the existing sections are still sufficient. A check is quicker to perform than a design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

The updated utilizations can then be reviewed in a Review View.

Check the whole model

To check all the existing Tedds member designs,

1. Click Check In Tedds (page 2182) from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.
2. Right click and select **Check using Tekla Tedds> Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds> Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

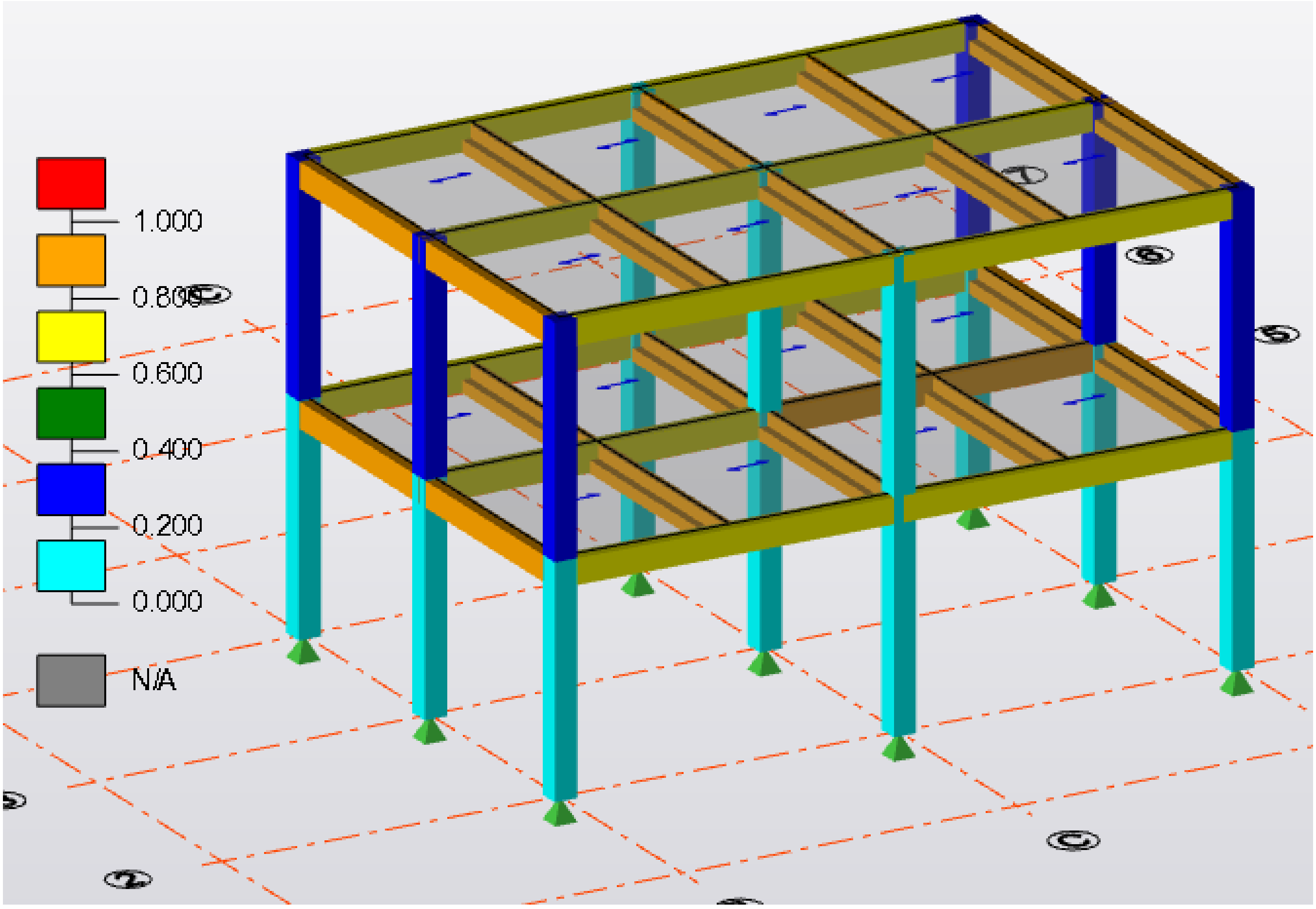
To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds> Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

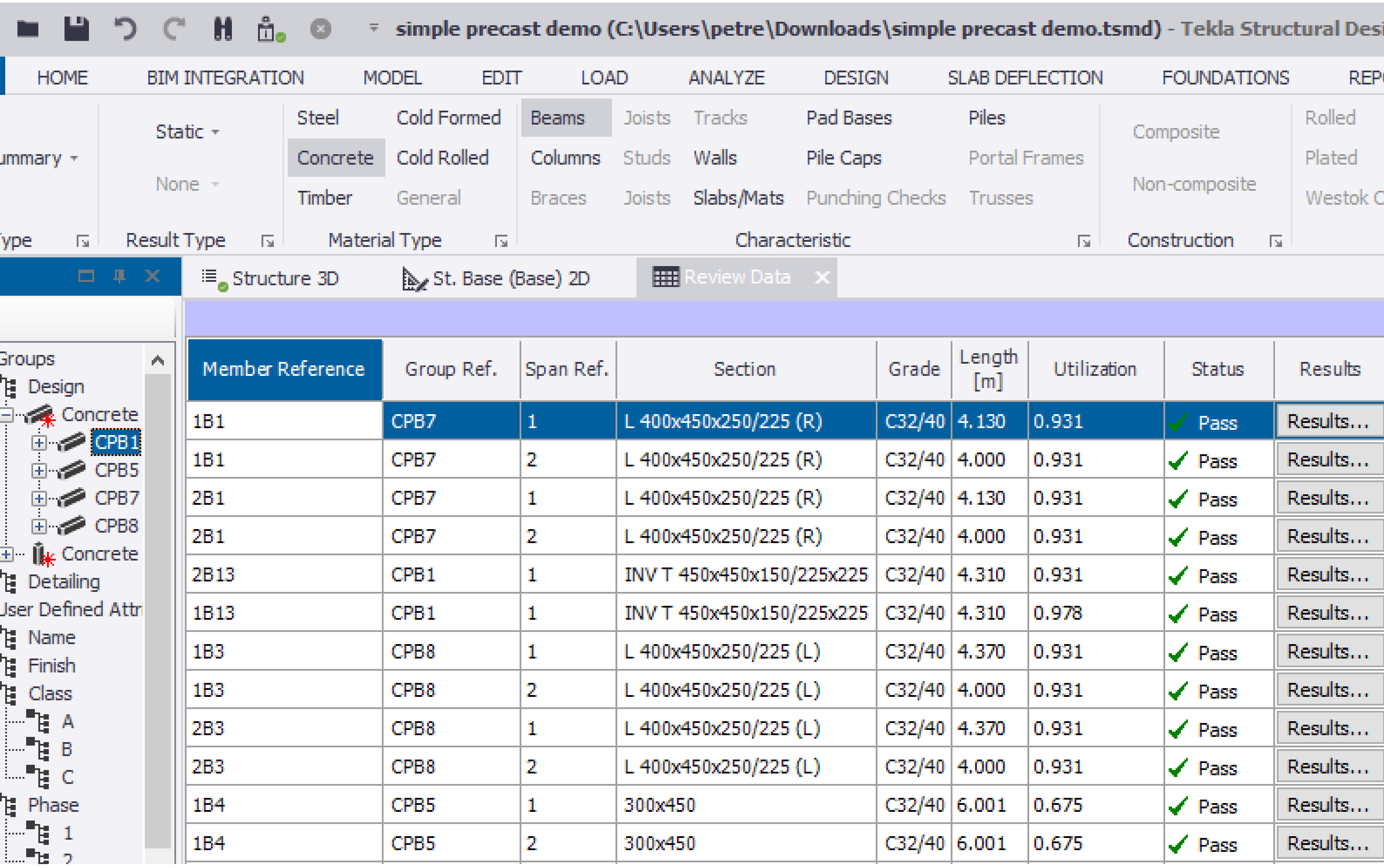
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

At this stage you can display member design status and utilization ratios (page 849) from a Review View.

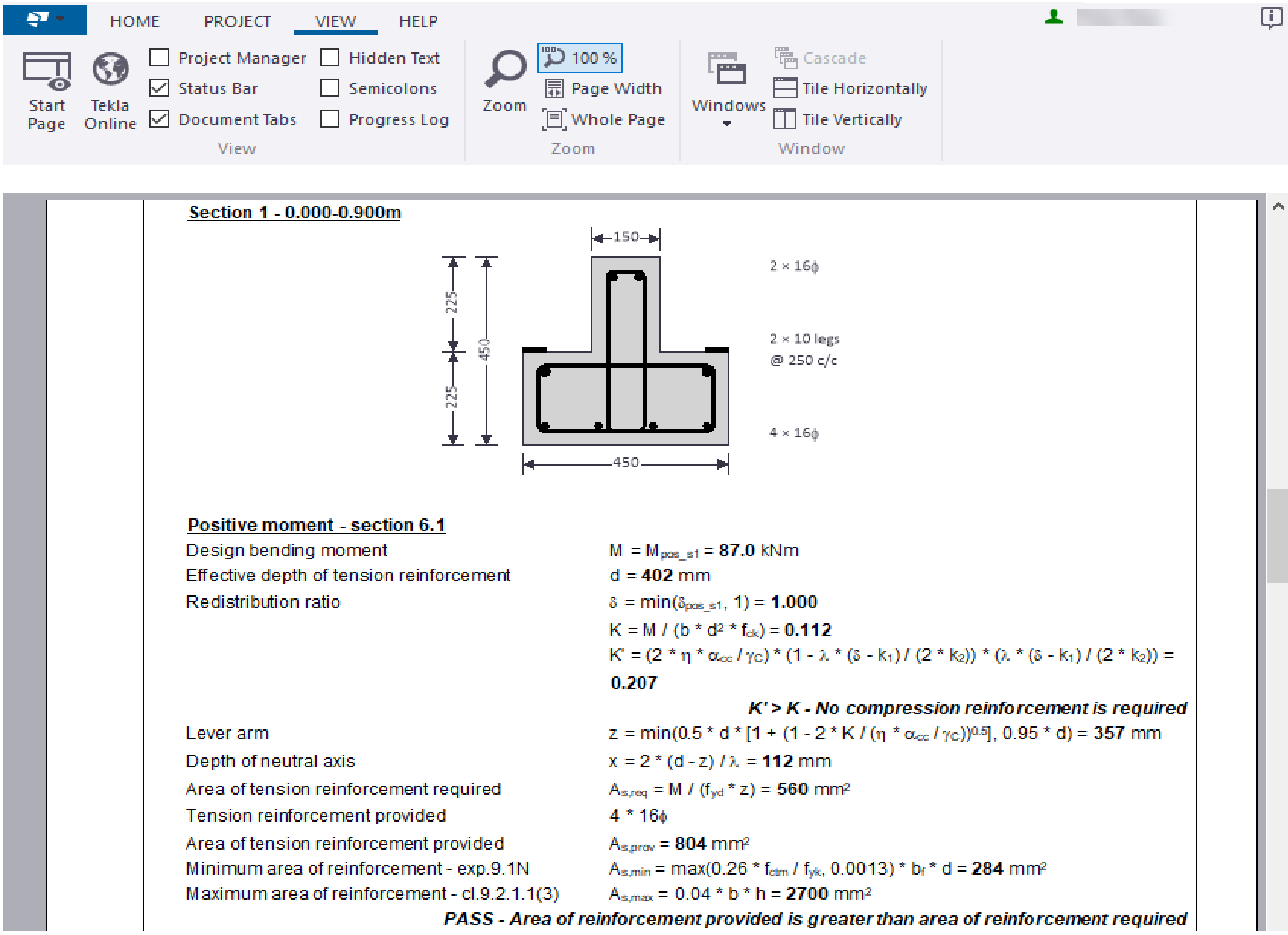


You can also display the design status tabular results (page 898).



Member Reference	Group Ref.	Span Ref.	Section	Grade	Length [m]	Utilization	Status	Results
1B1	CPB7	1	L 400x450x250/225 (R)	C32/40	4.130	0.931	Pass	Results...
1B1	CPB7	2	L 400x450x250/225 (R)	C32/40	4.000	0.931	Pass	Results...
2B1	CPB7	1	L 400x450x250/225 (R)	C32/40	4.130	0.931	Pass	Results...
2B1	CPB7	2	L 400x450x250/225 (R)	C32/40	4.000	0.931	Pass	Results...
2B13	CPB1	1	INV T 450x450x150/225x225	C32/40	4.310	0.931	Pass	Results...
1B13	CPB1	1	INV T 450x450x150/225x225	C32/40	4.310	0.978	Pass	Results...
1B3	CPB8	1	L 400x450x250/225 (L)	C32/40	4.370	0.931	Pass	Results...
1B3	CPB8	2	L 400x450x250/225 (L)	C32/40	4.000	0.931	Pass	Results...
2B3	CPB8	1	L 400x450x250/225 (L)	C32/40	4.370	0.931	Pass	Results...
2B3	CPB8	2	L 400x450x250/225 (L)	C32/40	4.000	0.931	Pass	Results...
1B4	CPB5	1	300x450	C32/40	6.001	0.675	Pass	Results...
1B4	CPB5	2	300x450	C32/40	6.001	0.675	Pass	Results...

Detailed Tekla Tedds calculations for precast members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds (page 324).



Precast member design groups

NOTE The use of suitable design groups for precast members is recommended. Using design groups will help speed the overall design process and allow for identical reinforcement amounts to be provided for each element within the group.

Why use precast design groups?

Precast beams and columns are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design - to standardise designs, reduce processing time, and reduce the volume of output created.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same number of stacks. Members in all member groups must be of similar lengths and cross-section.

NOTE Although precast members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect concrete beams and/or concrete columns as required.

Activating precast member design groups

Precast beam and column design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups
2. Select concrete beams and/or concrete columns to activate precast beam and/or precast column design groups.

Group management

Automatic Grouping

Precast beams and columns are grouped automatically according to a fixed set of requirements.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See: Manage groups in the Project Workspace (page 267)

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Precast design group requirements

Precast member design groups are formed according to the following rules:

Member type	Design group rules
Precast beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties.
Precast column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties.

Precast beam design

Specific aspects of the precast beam design workflow in Tekla Structural Designer are described below:

Section shapes

The following section shapes can be used within the Tekla Structural Designer - Tekla Tedds workflow.

- Rectangular
- L-Section
- Inverted L-Section
- T-Section
- Inverted T-Section

Beam arrangement

The Tekla Tedds calculations have no concept of the model geometry within Tekla Structural Designer and so there are no limitations on the design of curved beams in both major and minor axes. Continuous beams, cantilevered beams etc. are similarly supported by the workflow.

Concrete type

While you can apply both normal and lightweight concrete in the beam properties, precast beam design using lightweight concrete is currently beyond scope.

Nominal cover

The default nominal top, bottom and side cover values set in **Design Settings> Precast> Beam> Reinforcement Settings** are automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

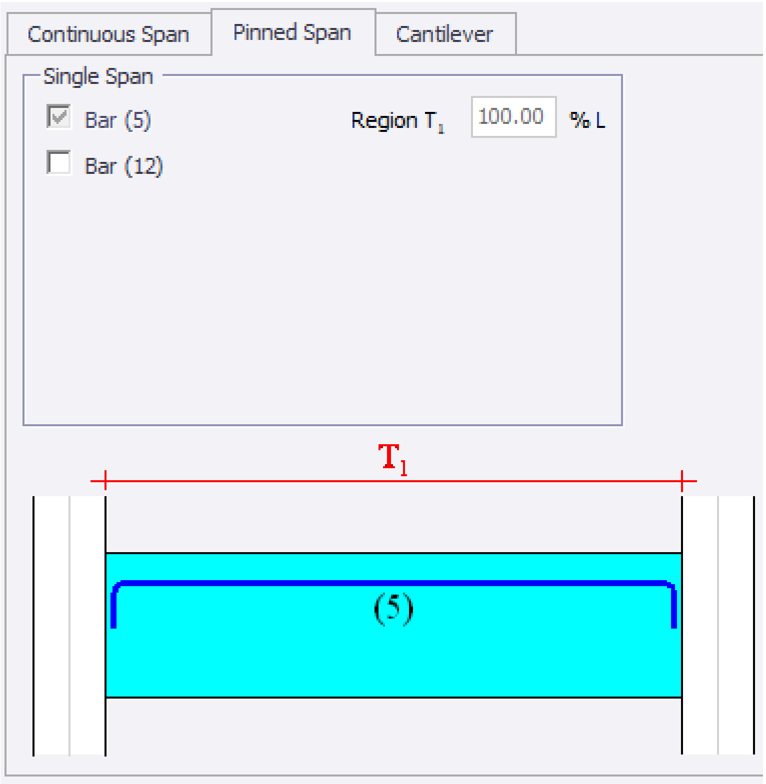
Reinforcement - longitudinal bar patterns

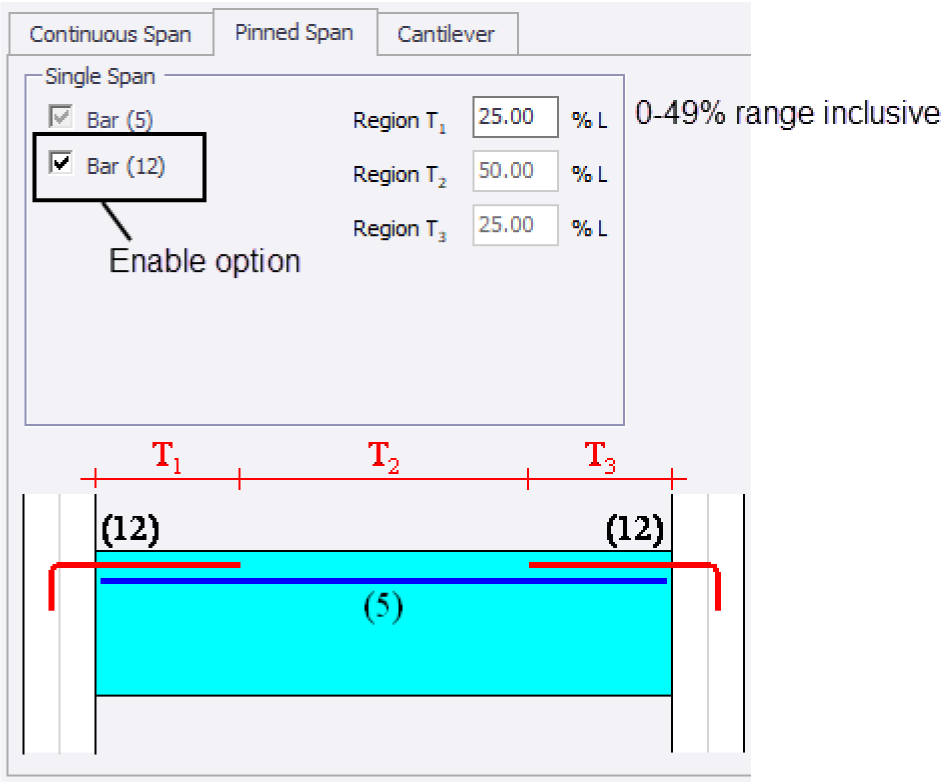
NOTE The reinforcement regions are considered to be the extent of the effective reinforcement. Anchorage and anchorage lengths are not considered in precast beam design within Tekla Tedds.

In **Design Settings> Precast> Beam** there is a one Standard Pattern available for top longitudunal reinforcement, Precast Top 1, and one Standard Pattern available for bottom longitudunal reinforcement, Precast Bottom 1 as illustrated in the figures below.

Standard Pattern of Top Reinforcement - Precast Top 1

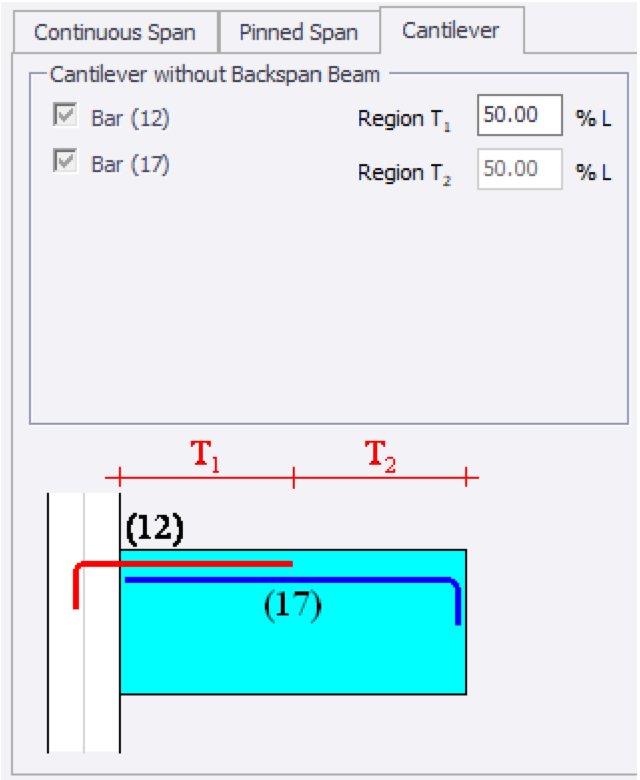
Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (5)
Bar (12)
Region T1
100.00
% L
T1
(5)

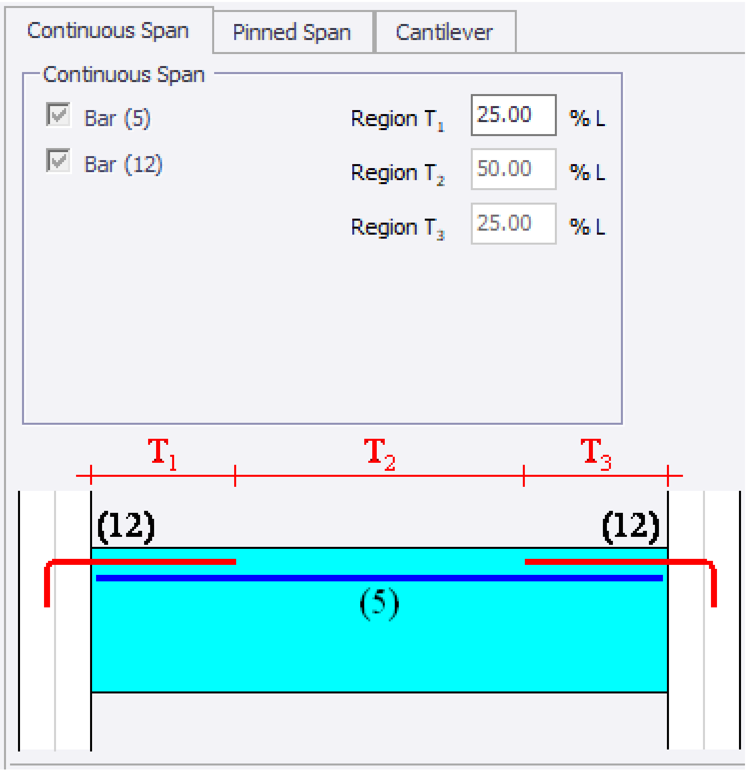


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (12). When enabled, the end region length is allowed to be 0%.

Cantilever span settings are applied to all spans with cantilever selected.

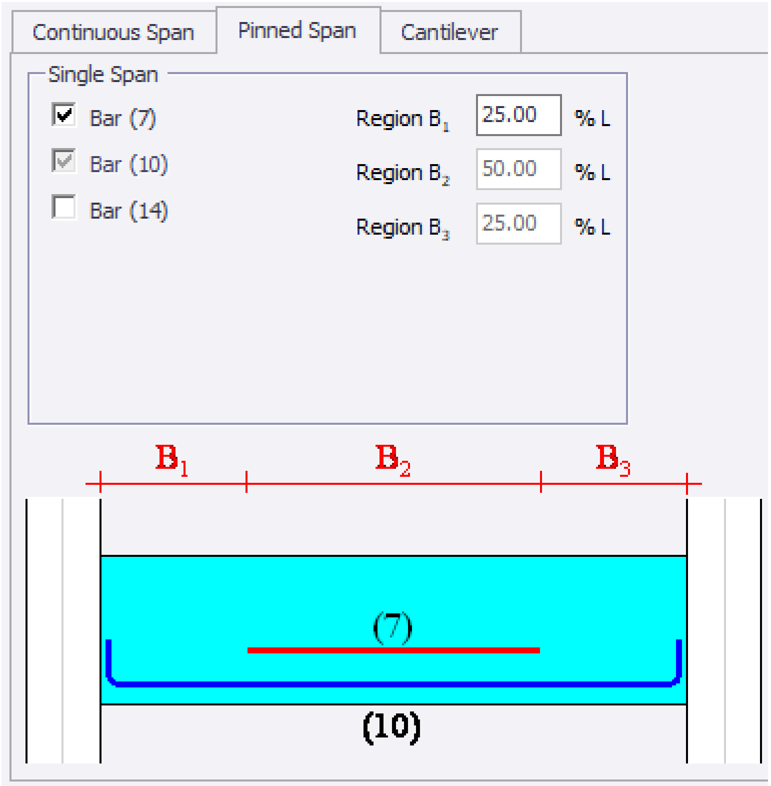


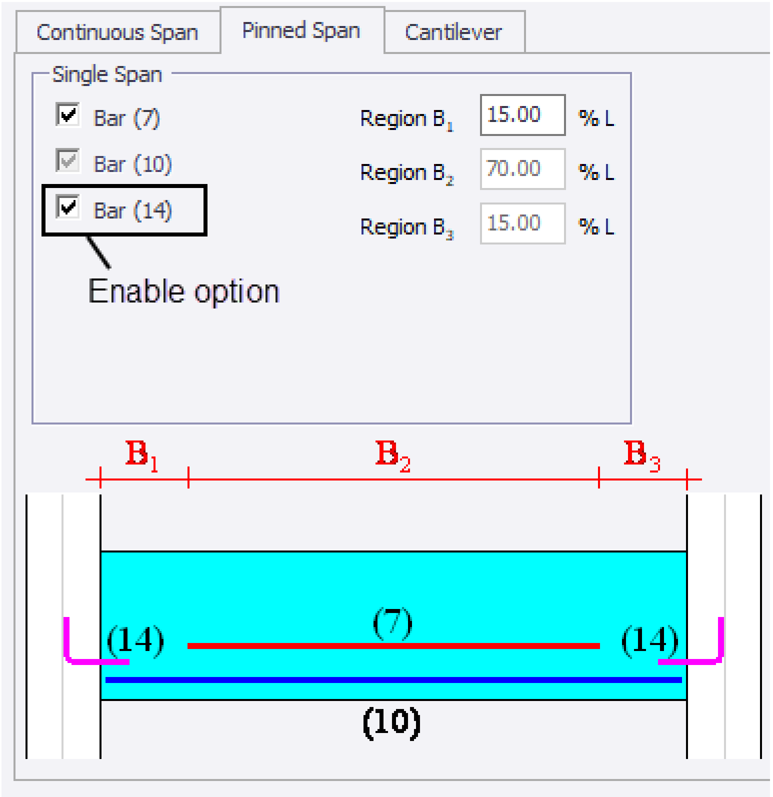
Continuous span settings apply to all other spans.



Standard Pattern of Bottom Reinforcement - Precast Bottom 1

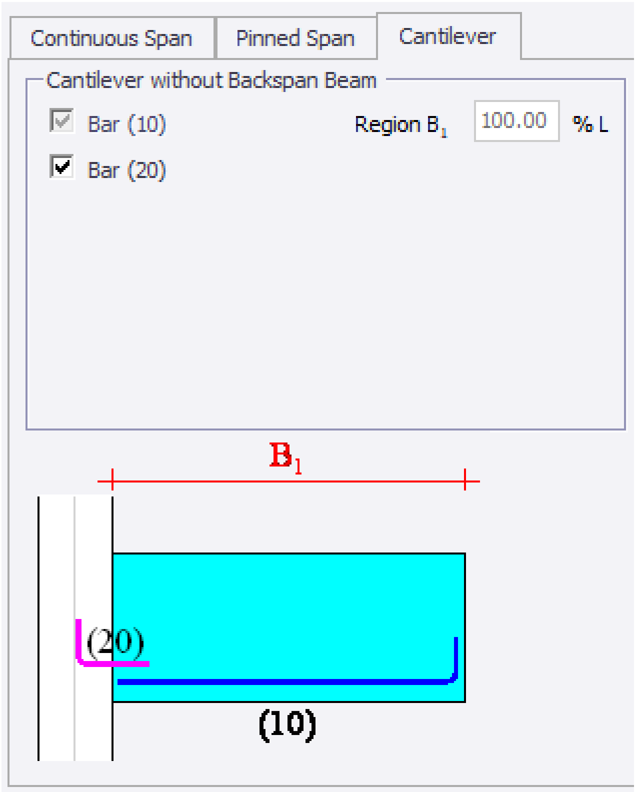
Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (7)
Bar (10)
Bar (14)
Region B1
25.00
% L
Region B2
50.00
% L
Region B3
25.00
% L
B1
B2
B3
(7)
(10)

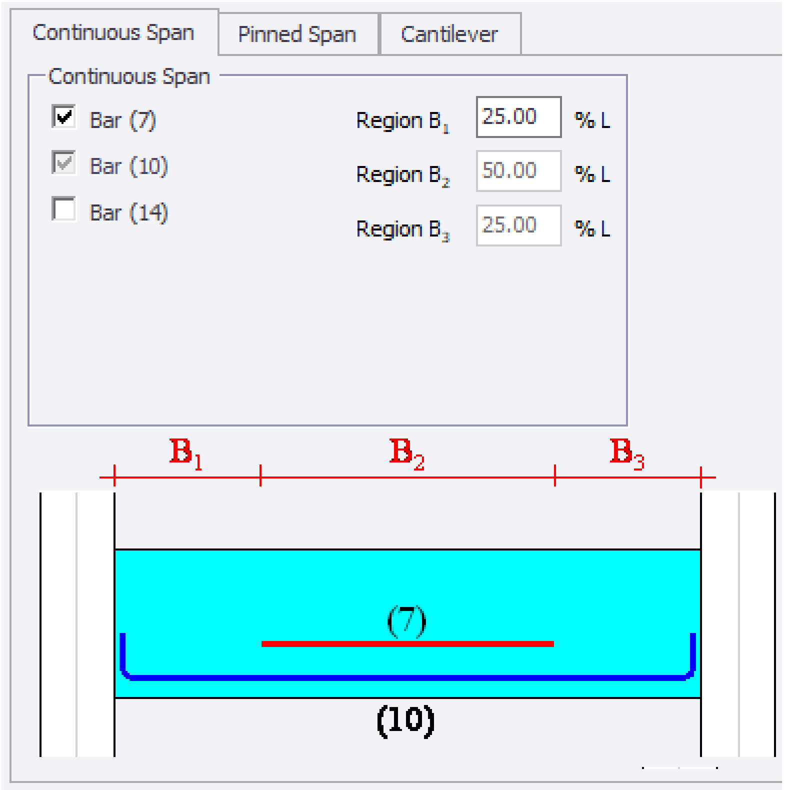


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (14).

Cantilever span settings are applied to all spans with cantilever selected.



Continuous span settings apply to all other spans.



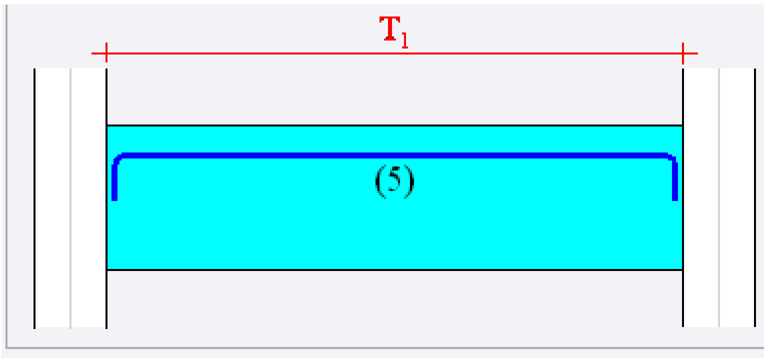
Design sections

An individual design section is created for each length of beam with similar reinforcement definitions. Each design section is then designed for the maximum forces in that region of similar reinforcement.

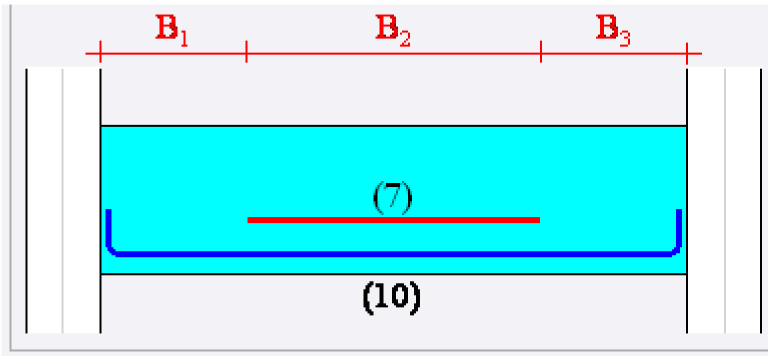
The numbers of design sections and their locations are determined by aggregating the different regions specified in the **Design Settings> Precast> Beam**: **Top Longitudinal Bar Pattern**, **Bottom Longitudinal Bar Pattern** and **Link Settings**.

Example: The default **Precast Top 1** and **Precast Bottom 1** longitudinal bar patterns for a Pinned Span are adopted without any continuity bars added as follows:

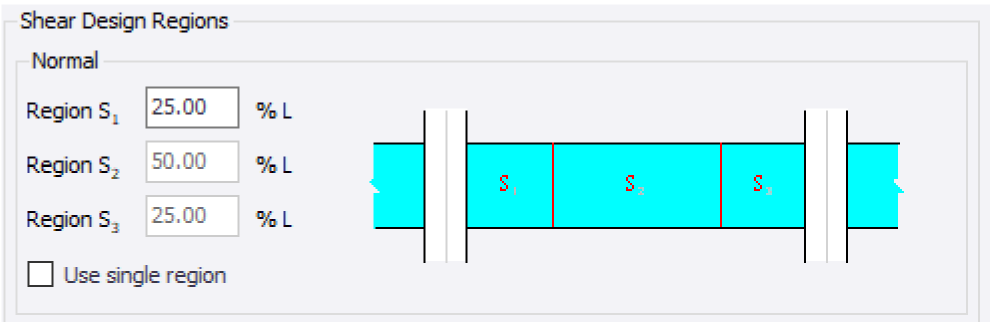
- **Precast Top 1**: single region 100% of the beam span.



- **Precast Bottom 1**: B1 and B3 regions set as 15% of the beam span.

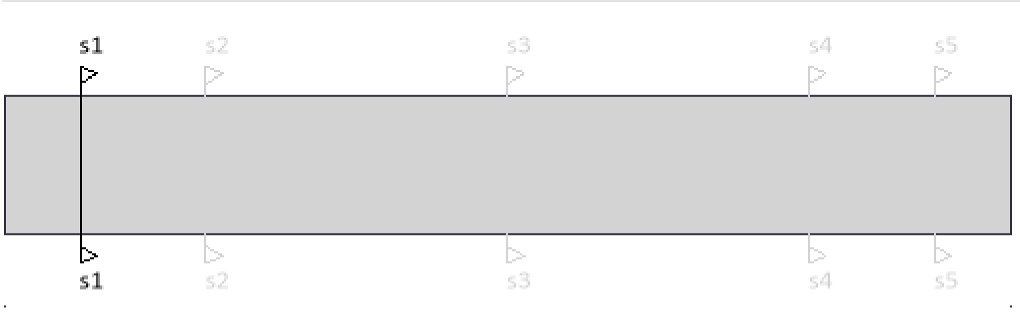


- Default **Link Settings** are also adopted: S1 and S3 regions set as 25% of the beam span.



For a pinned single span precast beam, different region boundaries would therefore exist at 15%, 25%, 75% and 85% of the span.

Consequently, when the span is designed in Tekla Tedds, 5 design sections would be created:



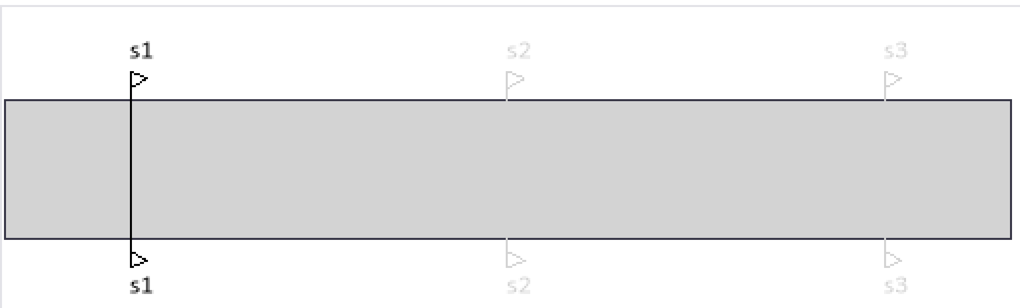
- s1 - positioned half way along the first bottom span region (7.5% of the span)
- s2 - positioned half way between the first and second region boundaries (20% of the span)
- s3 - positioned half way along the second and third region boundaries (50% of the span)
- s4 - positioned half way between the third and forth region boundaries (80% of the span)
- s5 - positioned half way along the last bottom span region (92.5% of the span)

NOTE In this example the first design section lies in the B1 bottom span region, so the positive design moment at s1 would be the maximum that occurs within B1; the second, third and forth design sections all lie in B2, so the positive design moment at s2, s3 and s4 would be the maximum that occurs within B2.

The engineer can of course adjust the standard bar patterns in the Tekla Structural Designer design settings to suit their requirements.

For instance, if in this example the **Precast Bottom 1** pinned span pattern were to be amended so that B1 and B3 regions were set as 25% of the span; because the bottom and link regions would then coincide, region boundaries would only exist at 25% and 75% of the span.

Consequently, when the span is designed in Tekla Tedds, only 3 design sections would be required:



- s1 - positioned half way along the first region (12.5% of the span)

- s2 - positioned half way between the first and second region boundaries (50% of the span)
- s3 - positioned half way along the last region (75% of the span)

NOTE In the above elevation diagrams:

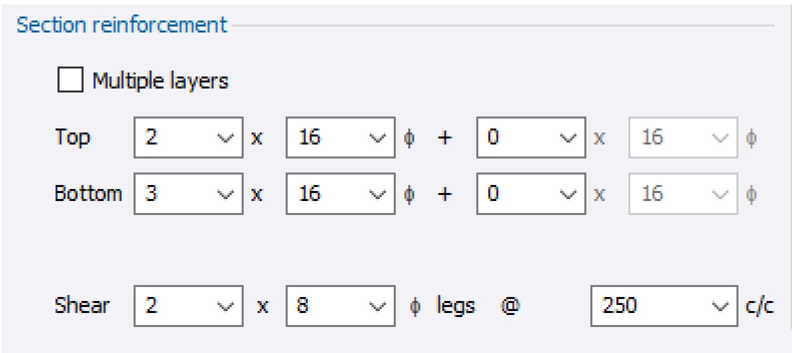
- A black section mark indicates the selected design section is passing the design criteria.
- Grey section marks indicate unselected design sections which are passing.
- A red section mark would indicate a design section which is failing.

Default reinforcement in the Tekla Tedds calculation

Regardless of beam shape and size imported from Tekla Structural Designer, the reinforcement in each beam member is always defaulted to the same values Tekla Tedds calculation.

This default reinforcement is:

- Top: 2x 16 dia bars
- Bottom: 3x 16 dia bars
- Shear: 2 legs of 8 dia bars at 250 cross centres



NOTE The Tekla Tedds calculation contains some limitations to the error trapping when it comes to maximum reinforcement values. Although the main bars in both the top and bottom of the beam have a check allocated to prevent unrealistic distances being entered, the shear links have no such limitation.

The engineer is expected to use their own knowledge to place in shear link values and distances which are both realistic and constructible.

NOTE The Tekla Tedds calculation contains some limitations to the section diagram. It may be noted that the diagram is never altered if the number of shear legs is increased from the default value of 2 legs. It should be noted that shear link legs are indicative only within Tekla Tedds and in reality there are a variety of ways that reinforcement can be detailed. The Tekla Tedds calculation therefore chooses to omit additional shear reinforcement in the section diagram and to flag it instead in the notes and the calculations.

Lifting checks

Lifting checks are disabled by default. This has been done to ensure a rapid design of grouped beams can take place. If lifting checks are required then simply enable the switch within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check for beam members makes allowance for reinforcement for both bending and shear checks.

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Positive moment
    - Positive design moment
    - Positive quasi-permanent moment
    - Redistribution ratio
- Negative moment
    - Positive design moment
    - Negative quasi-permanent moment
    - Redistribution ratio
- Shear
    - Maximum Design shear force
    - Design shear force
- Torsion
    - Design torsional moment

NOTE Minor axis forces (as defined in Design > Settings > Design Forces) are not used in the design of the member. Where minor axis forces are present within a beam, a warning is displayed within the process dialog of Tekla Structural Designer.

Other precast beam properties

The following properties which are common to both cast-in-place and precast beams, are not transferred to the Tekla Tedds calculation, but still require consideration as they will impact the analysis in Tekla Structural Designer.

- Concrete member cracked or uncracked status (page 1286)
- Design parameters (page 1293)

Precast column design

Specific aspects of the precast column design workflow in Tekla Structural Designer are described below:

NOTE The current Tekla Tedds calculation makes no allowance for any shear checks.

Section shapes

Whist a variety of different section shapes can be defined in Tekla Structural Designer, the Tekla Tedds calculation only supports the design of rectangular and circular column sections.

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Nominal cover

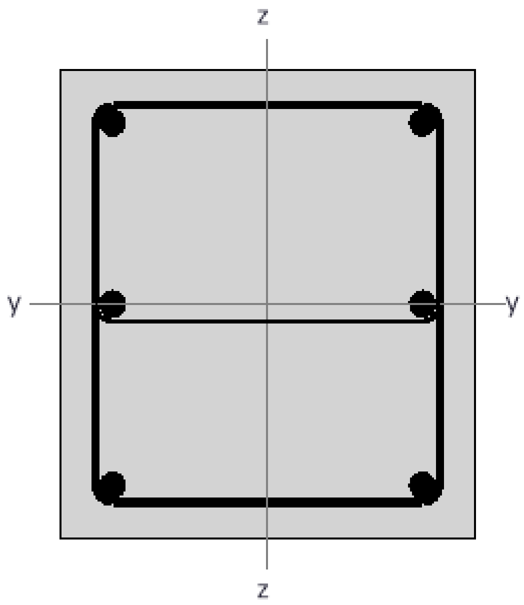
The default nominal cover value set in Design Settings> Precast> Column> Reinforcement Settings is automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

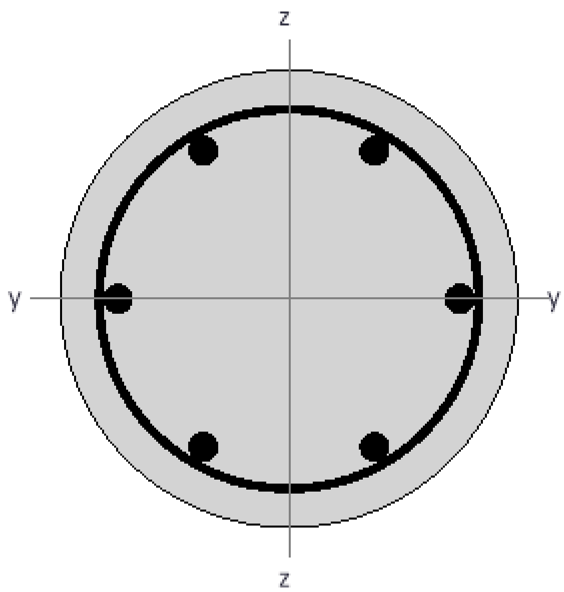
Default reinforcement in the Tekla Tedds calculation

Regardless of the column size imported from Tekla Structural Designer, the initial default reinforcement for all stacks in the Tekla Tedds calculation is as follows:

- Rectangular columns: a 2x3 formation of 25mm dia rebars with 8mm shear links.



- Circular columns: 6x 25mm dia rebars with 8mm shear links.



NOTE Tekla Tedds has no concept of multiple stack columns and Tekla Structural Designer has to associate separate Tekla Tedds calculations with each stack. It is the responsibility of the engineer to ensure that rebar on adjacent stacks are of appropriate quantities and sizes to ensure correct detailing can take place at the construction stage. It should be noted that Tekla Structural Designer does not hold any reinforcement information relating to precast members. Reinforcement is completely managed by the Tekla Tedds calculations.

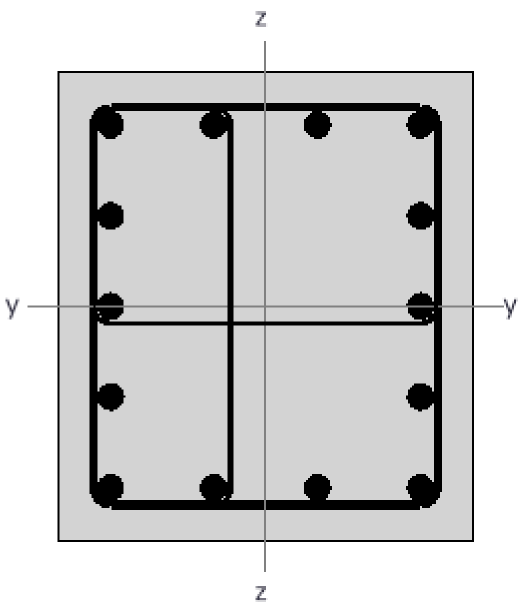
The maximum number of bars which can be entered in either direction for rectangular columns is 10. For circular columns, the maximum total number of bars is 20. Please note that these restrictions will have an effect on the maximum allowable column size for the design workflow.

There is a basic minimum distance calculation within the Tekla Tedds document relating to bar amounts to avoid unrealistic values of rebar from being selected. This amount is to be the minimum of:

- K1 x bar dia
- 20mm
- k2 x max aggregate size

NOTE For values of coefficients k1 and k2, refer to clause 8.2 of the Eurocode.

The number of link legs shown in the Tekla Tedds calculation will increase or decrease depending on the number of reinforcing bars assigned within the document.



NOTE The link legs are displayed to indicate required confinement reinforcement for vertical bars in compression, but have no effect on the overall design process.

Lifting Checks and Splice Design

Lifting checks and splice checks are disabled by default. This has been done to ensure a rapid design of grouped columns can take place. If these checks are required then simply enable the switches within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check is performed against the unreinforced area of the column, hence there is no input for the shear link centres within the calculation. (Bending checks (for splice design) do include reinforcement allowances within the design process.)

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Axial load, 1st case (Max value)
- Axial load, 2nd case (Min value)
    - Where tension exists in the column, the lowest compression value is populated into this parameter and the tension value ignored. A warning will be displayed in the process log within Tekla Structural Designer.

- Moment about y axis at top
- Moment about z axis at top
- Moment about y axis at bottom
- Moment about z axis at bottom

NOTE Tekla Structural Designer populates values from all design combinations. For columns subjected to low axial loading, the moment capacity will not be as high as a column subjected to larger axial loading. Low axial load and high moment can easily be a critical combination for concrete column design.

Other precast column properties

The following properties which are common to both cast-in-place and precast columns, are not transferred to the Tekla Tedds calculation, but require consideration as they impact the overall performance of the model.

- Concrete member cracked or uncracked status (page 1286)
- Design parameters (page 1300)
- Stiffness (page 1302)
- Sway/Drift Checks (page 1303)

Precast column connection eccentricity moments

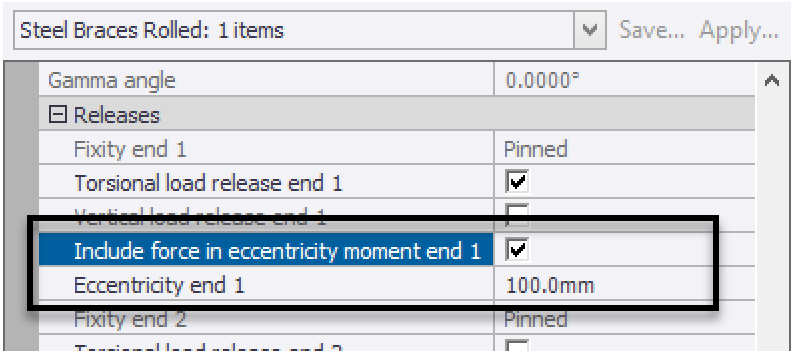
Overview

Nominal eccentricity moments arise in precast columns due to the reactions from pin ended beams being applied at eccentricities to the column centerline. The eccentricity moments are calculated for all headcodes, and can be interrogated in the Load Analysis View.

If working to Eurocodes, these eccentricity moments are also taken into account when precast columns are transferred to Tedds for design.

These moments do not come directly from the global analysis but instead are calculated at the 'load analysis' post-processing stage as follows:

- At each level the eccentricity of each connection is specified as a user defined offset from the column face. If rigid zones are not being used this is increased by half the depth of the supporting column.
- At each level the pinned beam end reactions connecting to the column at each face are determined.
- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

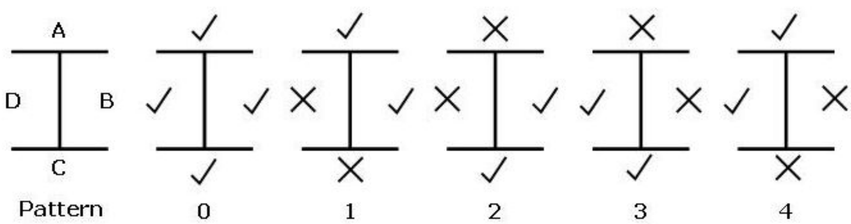
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are 'true' patterns switching live load 'on' and 'off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design to Eurocodes

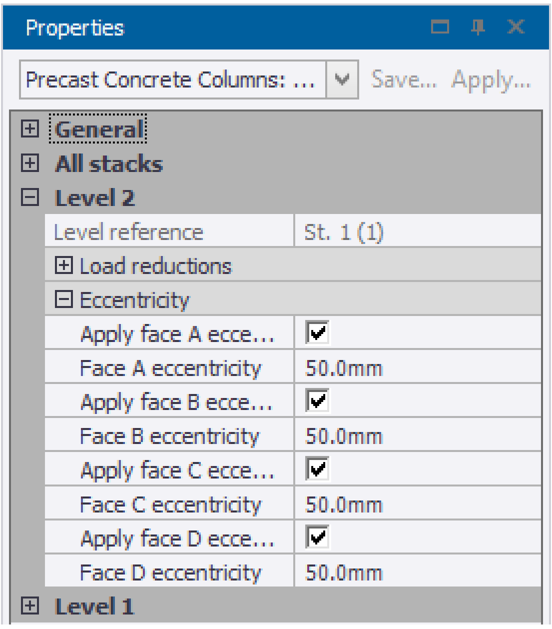
In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.

Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



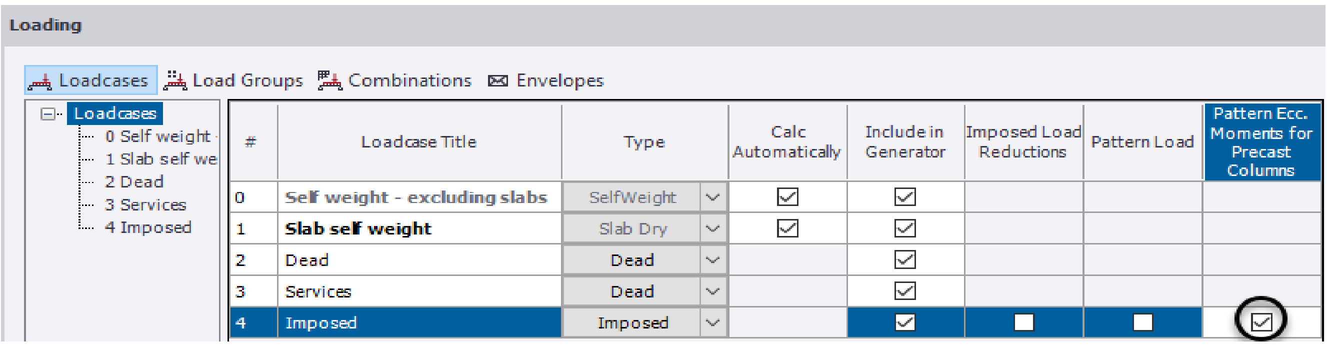
If you uncheck the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 1250)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
    a. Click **Model Settings > Loading > General**
    b. Select **Use patterning of eccentricity moments for precast columns**
    c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
    a. Select a live loadcase that you want to be patterned
    b. Select **Pattern Ecc. Moments for Precast Columns**
    c. When patterning has been selected for each of the required loadcases, click **OK**

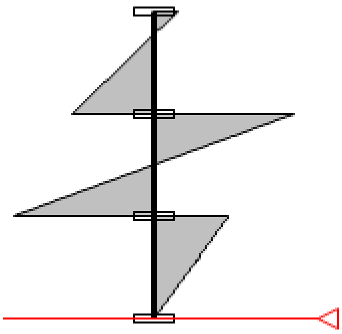


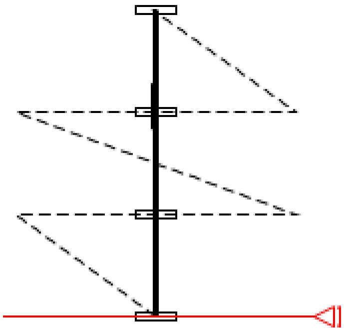
Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View (page 716).

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:



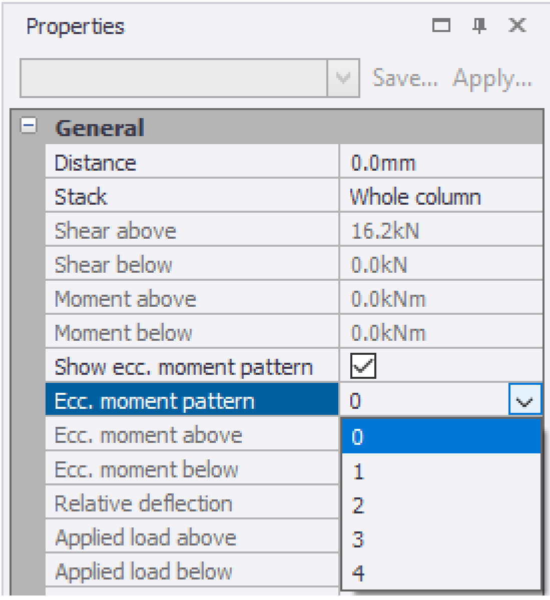


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.



Precast member design commands

By right clicking over the required member in a view or appropriate branch of the **Project Workspace**, the following interoperative design commands can be accessed:

- Design using Tekla Tedds (page 2255)
- Check using Tekla Tedds (page 2242)
- Export to Tekla Tedds (page 324)
- Clear Tekla Tedds Data (page 2247)

NOTE Initially only Design using Tekla Tedds (page 2255) command is shown, but once this has been run the other commands then become available.

Each of the above offers a sub-menu of choices, depending on context:

- > Model
- > Member
- > Group
- > Selection
- > *<Substructure name>*

NOTE Check using Tekla Tedds (page 2242) > Model is repeated on the Design ribbon tab as  Check in Tedds (page 2182)

13.9 Timber member design handbook

Timber structures can be modelled and analyzed in Tekla Structural Designer. If a licence of Tekla Tedds is available, timber beams, columns and braces can then also be designed.

NOTE The following limitations and assumptions apply:

- The user should ensure the locale is set correctly in Tekla Tedds before attempting to run the design from Tekla Structural Designer.
- Design is supported for NDS LRFD, NDS ASD, base Eurocode, and the following Eurocode National Annexes: UK; Ireland; Norway; Finland; Sweden.
  - If the Singapore or Malaysia Eurocode National Annex is selected, the calculation will adopt base Eurocode recommended values.
- All timber sections should be consistent with the Timber property assumptions (page 1021) appropriate to the selected head code.
- If working to the NDS code, size classifications taken from the NDS supplement 'Design Values for Wood Construction' are used in the Tedds calculation to distinguish which grades are valid for particular timber species. The same classification rules are not implemented in Tekla Structural Designer itself. Therefore, after a member has been designed in Tedds, if the member size or grade is subsequently changed in Tekla Structural Designer the associated Tedds calculation is deleted, so that the member cannot be checked and a redesign is required.
- Column design: If designing a multi-stack column the user must indicate a splice in the stack properties **before** running the design if a change in cross section is required between stacks. If a splice has not been specified the change in section size will not be correctly returned to the Tekla Structural Designer model.

The following topics are covered in this handbook:

Timber member design workflow

The essentials of design of timber members using Tekla Tedds can be thought of as being very similar to Interactive concrete member design (page 1310) within the program:

- Just like the concrete beam/ column/ wall interactive design dialog, the Tedds Timber member design calculation interface lists the design forces and settings for a selected member/ group, populated from the model and analysis, the design pass/fail status and individual check results and utilization ratios. You can make changes in the interface - such as to section size and/ or grade - and immediately see the design results for these. All the data you input and changes you make are updated to and stored in the model. You can then re-analyze and run a check on all designed members to ensure the design results are up to date. You can perform design and analysis loops as necessary so that all members are passing, and to cater for any other changes to the model or loading.

- Design Groups (page 1577) work in a similar manner also - these are automatically created for Timber members and listed in the Project Workspace Groups Tree. The resulting groups can be reviewed and customized as necessary. Group design is enabled by default (in **Design Settings > Design Groups > Timber Beams/ Columns/ Braces**) and can be run both via the model and from the Groups Tree.

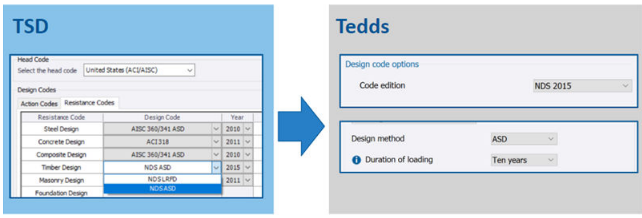
Key aspects of the Tedds integrated timber design process and workflow in Tekla Structural Designer are:

Set the timber design code

The timber design code is set from the **Home** ribbon by clicking **Model Settings > Design Codes > Resistance Codes**.

- If working to **US codes**, there are two different methods, ASD or LRFD to choose from and also a choice of code year to apply. The design forces consider only the selected design method.
- If working to **Eurocodes**, the timber design code will be applied with a national annex that is appropriate to the head code that has been selected.

The selected design code is automatically applied in all Tedds timber member designs.



Define and place timber members

To place timber members, select the appropriate member type from the **Timber** group on the **Model** ribbon, ensure the values in the **Properties** window are as required and then pick the points to locate the members in position.

Note the following with regard to the properties:

- **Section shapes** - When defining a timber model in Tekla Structural Designer it may appear that the range of timber sections is somewhat limited. This is because Tekla Structural Designer doesn't currently have a facility to specify timber sections by simply inputting a breadth and depth. Instead you have to select the required size from Tekla Structural Designer's database of timber sections.

  While this might seem to be a significant limitation, it is actually not too great an issue, because the Tedds timber calculation allows you to override the original size to input the breadth and depth you require. Then when the member data is transferred back from Tedds to Tekla Structural Designer the new size is automatically added to the timber section database.

  Once the new section is in the section database it becomes available for future Tekla Structural Designer models also.

- **Timber fabrication and grade** - You can specify timber, glulam, or structural composite lumber fabrication types.

  Initially the grade must be selected from the existing grades in the Tekla Structural Designer materials database, although if the grade is changed in the the Tedds timber calculation, when the member data is transferred back from Tedds to Tekla Structural Designer the new grade is automatically added to the materials database.

  NOTE If a situation arises such that the material properties in the programs don’t align, Tedds ignores unmatched data and selects calculation defaults.

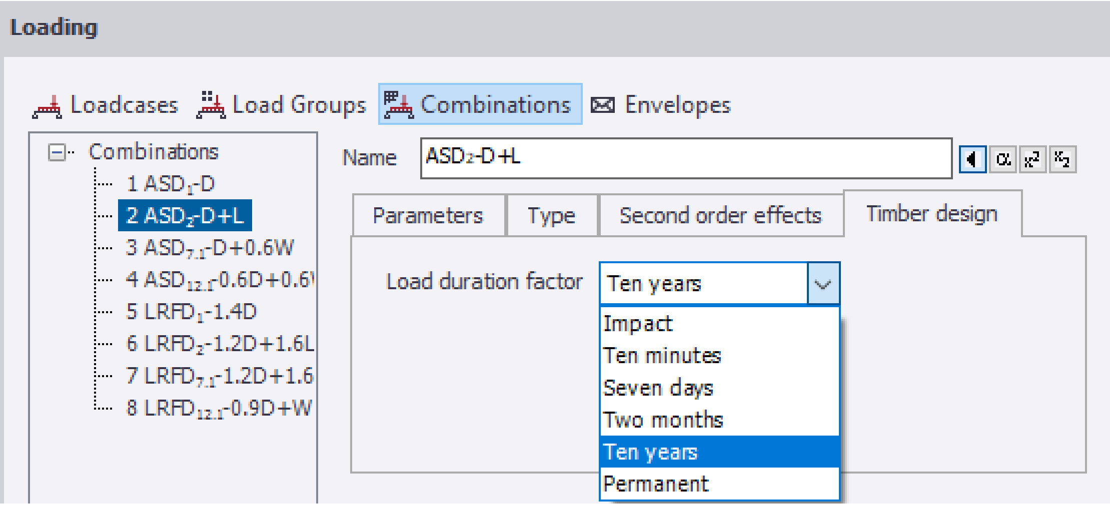
- **Multi-stack columns** - splices should be specified in the stack properties when the column is being defined if a change in cross section is required between stacks. If a splice has not been specified before the column is designed in Tedds the change in section size will not be correctly returned to the Tekla Structural Designer model.

Create load combinations and set load duration/time effect factors

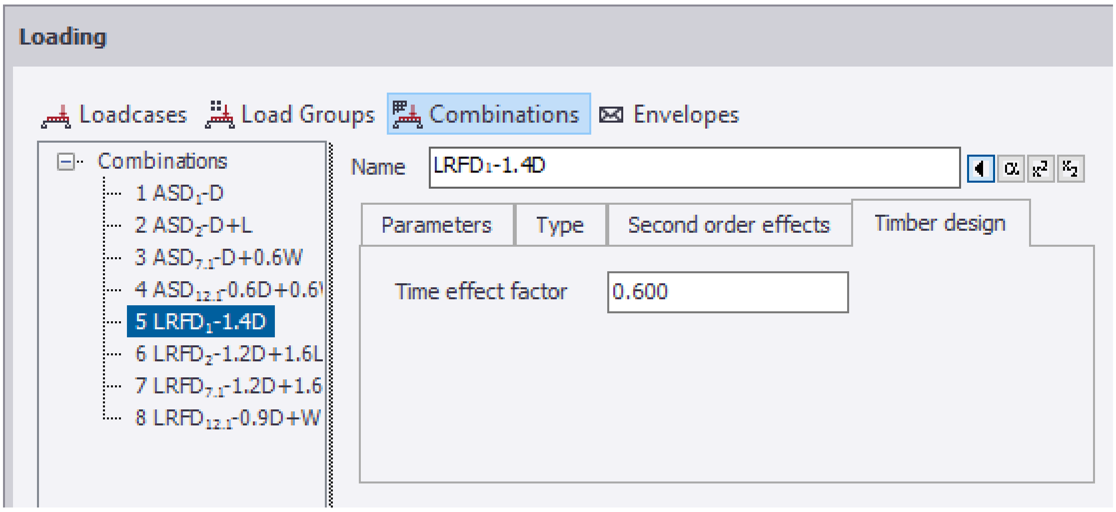
Loads are applied in the standard manner. Once load combinations have been created you should review their load duration/ time effect factors.

These factors are set on the **Timber design** tab on the **Combinations** page of the **Loading** dialog once you have selected an individual combination from the list in the dialog. The value is set automatically according to the constituent loadcases.

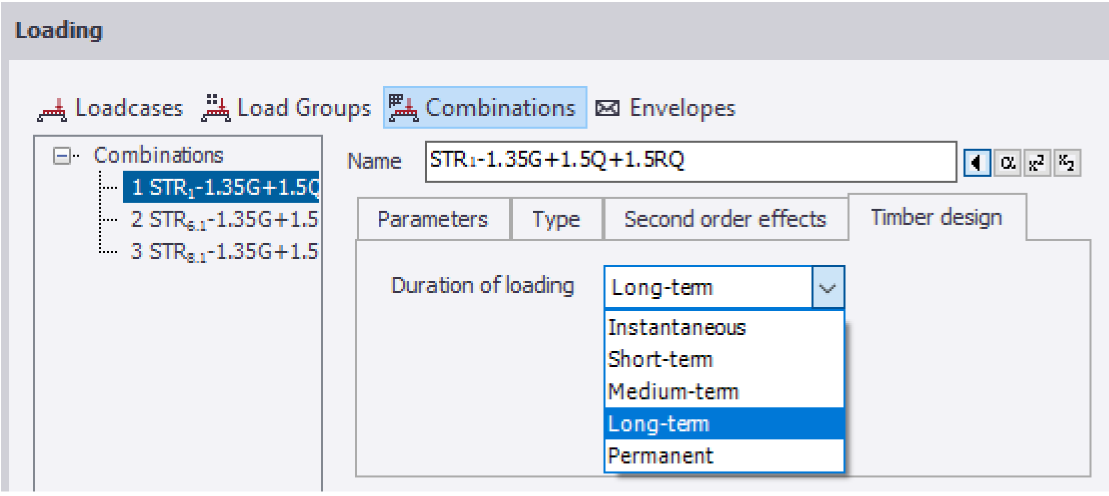
- **ASD combination:**



- **LRFD combination:**



- **Eurocode combination:**



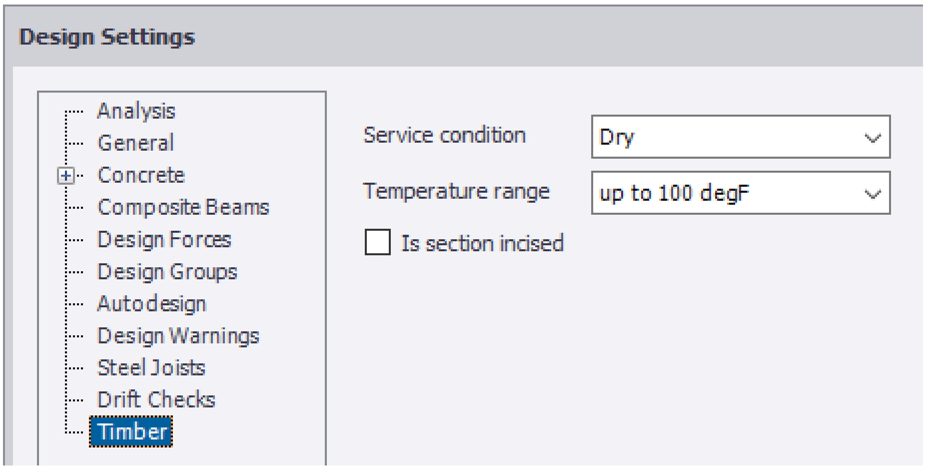
You have control to adjust the default value on a combination by combination basis.

NOTE The actual factor that gets applied in the Tedds calculation will be the one that is associated with the worst case load combination.

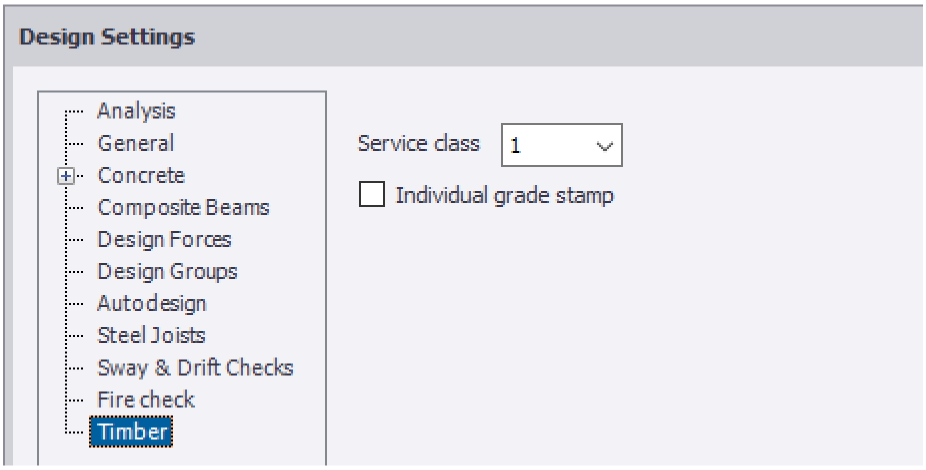
Configure timber design settings

Certain timber design settings which apply to the entire building can be specified on the (page 2347).

- If working to **US codes**:



- If working to **Eurocodes**:



By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds timber calculation as it is run.

You still have control within individual calculations to adjust these settings on a member by member or group basis.

NOTE Tedds data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established.

If you change timber design settings after a member has already been designed you would need to manually clear the Tedds data from the design in order to have it use the revised design settings. To do this highlight the member, right-click and select Clear Tekla Tedds Data (page 2247) from the context menu.

Configure timber groups

When timber members are placed in the model, to make editing easier they are automatically grouped according to a set of rules (page 1579).

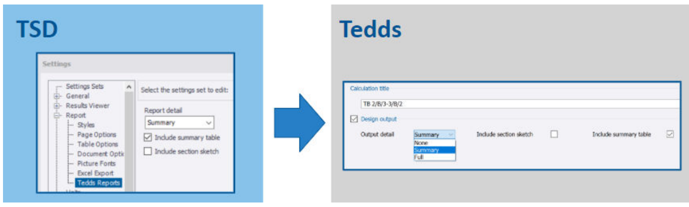
These initial groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can add new groups (page 268) and move members between groups (page 269) as required.

Groups are not only useful for editing, they can (optionally) be activated for design purposes also. There are a number of reasons why you would choose to do this.

- Why use timber design groups? (page 1577)
- Activating timber member design groups (page 1577)

Set the Tedds results output level

You can choose the output level for the Tedds timber calculations in advance by clicking **Home > Settings > Report > Tedds Reports**



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds timber calculation as it is run.

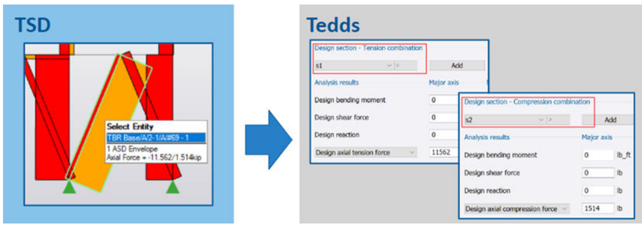
The setting applies to all Tedds linked calculations (precast and timber).

Establish design forces by running the analysis

Timber members can only be designed provided a set of analysis results exist. These can be generated from the **Analyze** ribbon by running Analyze All (Static) (page 626).

A design force envelope is established, which if designing to the US head code considers the selected design method (ASD or LRFD) only.

All critical load combinations are considered in one Tedds calculation.



Provided load combinations have been created, once analysis has been performed the **Design using Tekla Tedds** options become available.

Design using Tekla Tedds

After Analysis, Design or Check using Tekla Tedds can be initiated from both the Project Workspace and any graphical view of the model:

You can right-click over any individual member in the Project Workspace, or make a graphical selection, then right-click to open the context menu listing the Tekla Tedds design options. You can then select Design of the Member, Group or Selection as appropriate. Design of a Selection can also be used for example to design all members of a truss - which can be selected with one click - in one operation.

- Design a selection (page 1572)
- Design a group (page 1573)
- Design a grouped member (page 1575)
- Design an ungrouped member (page 1575)
- Design model (page 1576)

Check the design after changes

If changes are made to the model you can run a 'check' design to determine if the existing sections are still sufficient. A check is quicker to perform than a design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

You can check the whole model from the **Design** tab, or right-click over any individual member in the Project Workspace, or you can make a graphical selection, then right-click to open the context menu listing the Tekla Tedds design options as shown below. You can then select Design of the Member, Group or Selection as appropriate.

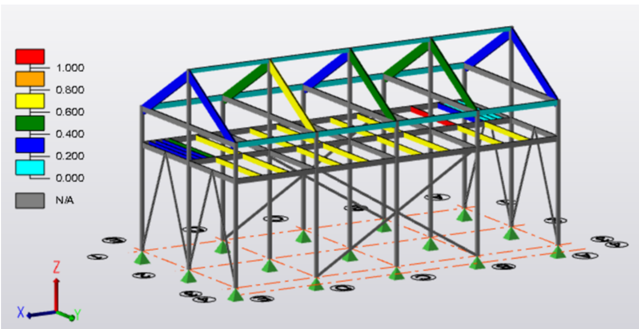
- Check model (page 1576)
- Check a selection (page 1576)
- Check a member (page 1576)
- Check a group (page 1577)

The updated utilizations can then be reviewed in a Review View.

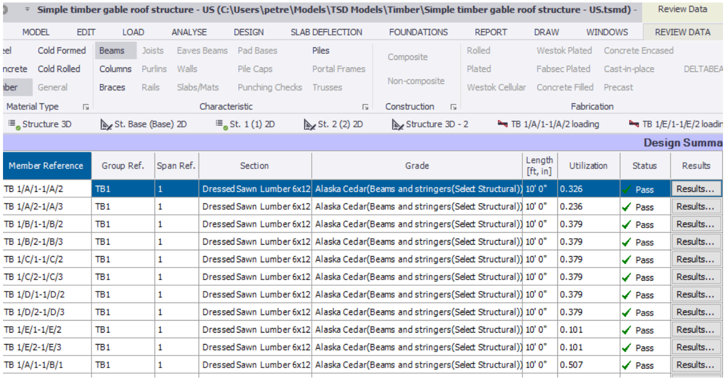
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

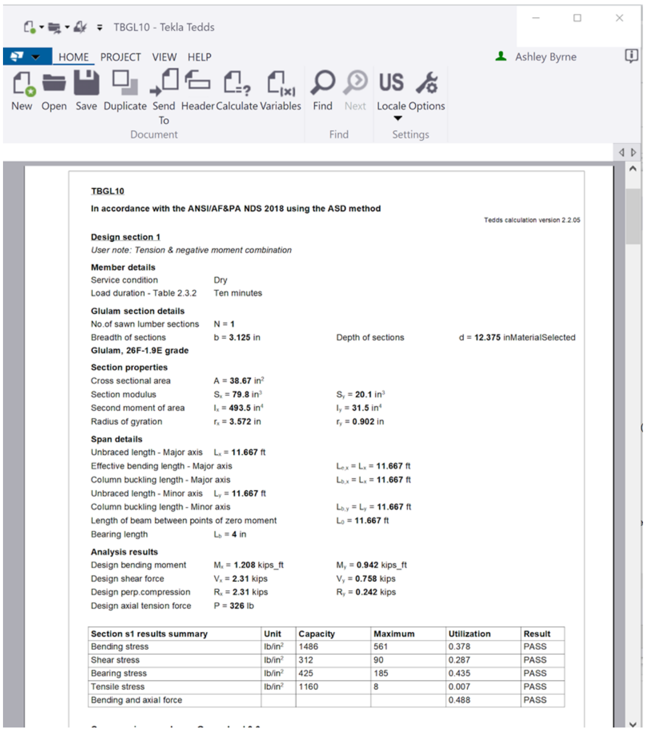
At this stage you can display member design status and utilization ratios (page 849) from a Review View:



You can also display the design status tabular results (page 898):



Detailed Tekla Tedds calculations for timber members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds (page 324).



Design timber members using Tekla Tedds

Design a selection

To design several timber members or groups in one go:

1. Select all the timber members you want to design.

2. Right click and select Design using Tekla Tedds> Selection (page 2255)

   Tekla Structural Designer highlights the first member or group in the selection and a Tedds dialog opens displaying the design forces at the member's first design section.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whether or not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.
4. When all sections are satisfactory, click **Finish**.

   Tekla Structural Designer highlights the next member or group in the selection and the Tedds dialog re-opens to allow it to be designed.

   Continue in the same way until all the selected members/groups have been designed.

   NOTE If a section is changed during the design to a size that does not exist in Tekla Structural Designer's Timber Section database, it will be automatically added to the local section database that exists for the current model only.

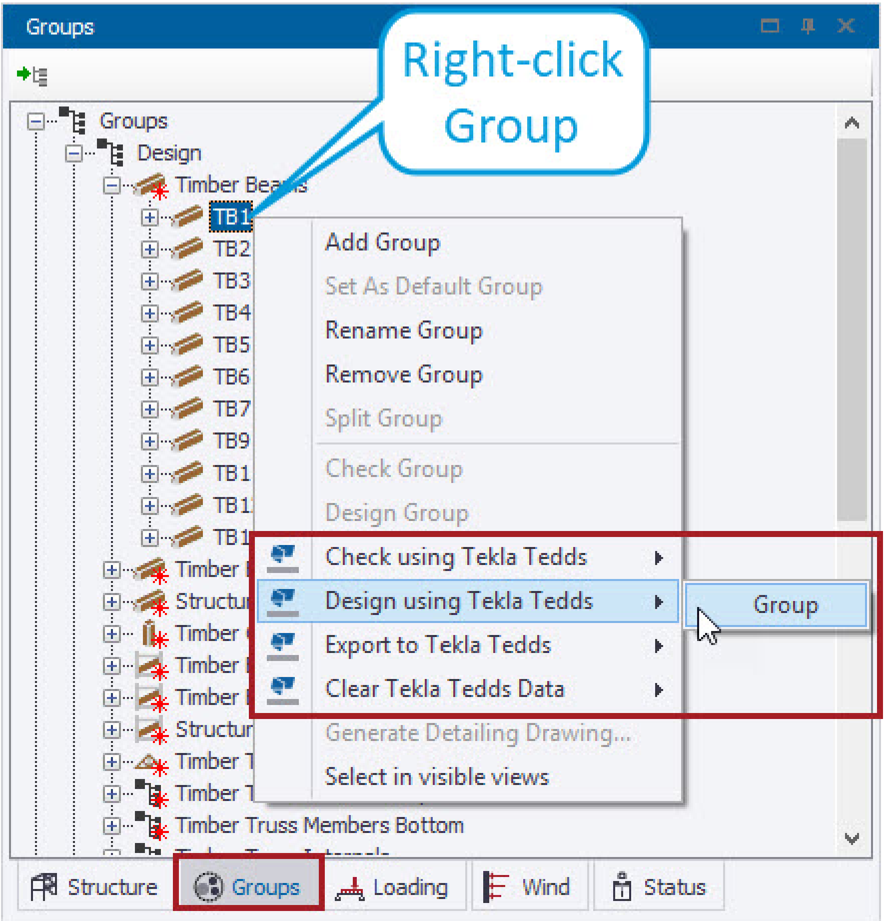
Design a group

If you have activated timber member design groups, each group can be designed as follows:

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, expand **Timber Beams**, **Timber Columns**, or **Timber Braces** as required.

   NOTE When designing timber brace pairs, the groups for these are always located under **Timber Braces** and not **Timber X Braces, A Braces, V Braces, K Braces.** These latter groups are used for editing purposes only, but not design.

3. In the **Design** tree, right-click the timber member group you want to design.
   a. It can be useful at this point to click **Select in visible views** to confirm all the group members.
4. In the context menu, select Design using Tekla Tedds> Group (page 2255).



This design gathers the analysis results for all members in the group and collates them into one set of critical design forces. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

5. Using the results preview, design the first design section in the group for its critical design forces, then if there is a second section design that also.

6. When all sections are satisfactory, click **Finish**.

   If the section size was changed during the design, all members of the group are updated to the new section.

   Irrespective of whether the section has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

7. In the Review View, review the utilization ratios (page 850) for all members in the group - if these indicate an efficient design, the design can stop at this point.

NOTE For columns in particular, the envelope of critical design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

If the utilizations indicate the existing grouping is not very efficient, you can investigate alternative grouping arrangements by adding extra groups (page 268) as necessary and re-allocating members between the groups (page 269). You might also consider individually designing grouped members (as described below) when optimizing the groups.

Design a grouped member

If after running a grouped design some group members have a lower than desired utilization, you can re-design them individually to investigate the effect optimizing them would have on the rest of the group.

1. Highlight a group member which has a lower than desired utilization, right click and select Design using Tekla Tedds> Member (page 2255)

   The analysis results of the selected member are used to establish the set of critical design forces.

2. Optimize the design of the member for its forces.
3. Click **Finish**.
4. If the section size was changed during the design, all members of the group are updated to the new section and checked against only the loads that they see individually.
5. In the Review View, review the new utilizations for the group members - note that some of the group might now be failing.
6. Using the new utilizations to better inform you choice, add extra groups (page 268) as necessary and re-allocating members between the groups (page 269).
7. Run Design using Tekla Tedds> Group (page 2255) for each of the new groups.
8. Iterate the process as necessary until the groups are fully optimized.

Design an ungrouped member

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member (page 2255)
2. Tekla Structural Designer highlights the member and a Tedds dialog opens displaying the design forces at the member's first design section.

3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.
4. When all sections are satisfactory, click **Finish**.
5. If the section size was changed during the design, it is updated to the new section in the Tekla Structural Designer model.
6. The member status and utilization are displayed in the member tooltip.

Design model

If you want to design every timber (and precast) member in the model in one go:

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Design using Tekla Tedds > Model**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check timber members using Tekla Tedds

Check model

To check all the existing Tedds member designs,

1. Click Check In Tedds (page 2182) from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.
2. Right click and select **Check using Tekla Tedds> Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds> Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds> Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

Timber member design groups

Why use timber design groups?

Timber beams, columns and braces are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design -
   a. to standardize designs,
   b. to reduce the volume of output created,
   c. to speed up the design process, particularly so in the case of medium to large size models.

When groups are used for design:

- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

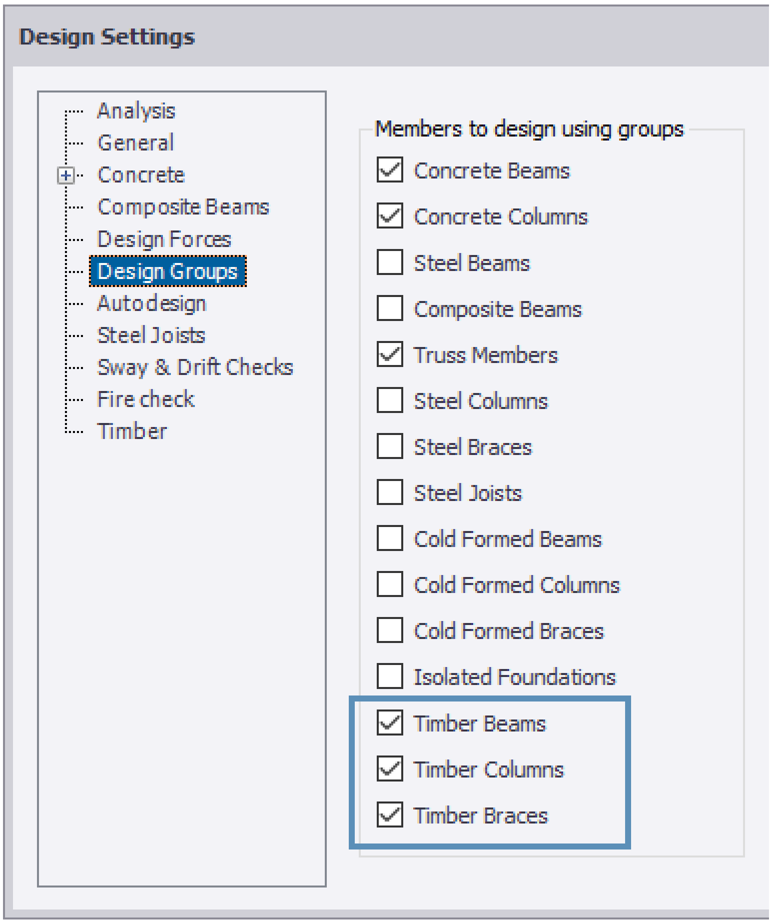
NOTE Although timber members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

Activating timber member design groups

Timber beam, column and brace design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups

2. Select **Timber Beams**, **Timber Columns**, **Timber Braces** as required.



Group management

Automatic Grouping

Timber beams, columns and braces are grouped automatically according to a fixed set of requirements, including section size, grade, and member length.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are of sufficiently similar length to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See: Manage groups in the Project Workspace (page 267)

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup Members

If you have made changes in the model that impact on grouping of a specific member type, you can update the affected groups from the Groups tab of the Project Workspace by right-clicking **Timber Beams**, **Timber Columns**, or **Timber Braces** as required and selecting **Regroup Members**.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Regroup ALL Model Members

If you have made changes in the model that impact on grouping, you can update all affected groups accordingly from the Groups tab of the Project Workspace, by clicking Re-group ALL Model Members. (Located at the top of the Groups tab.)

NOTE Any manually applied grouping will be lost if you elect to re-group!

Timber design group requirements

Timber member design groups are formed according to the following rules:

Member type	Design group rules
Timber beam	• A beam may be in only **one** design group. • All beams in the group must have an identical cross section. • All beams in the group must have an identical grade. • All beams in the group must have an identical number of spans. • Individual span lengths must be the same across the group.
Timber column	• A column may be in only **one** design group. • All columns in the group must have an identical cross section. • All columns in the group must have an identical grade. • All columns in the group must have an identical number of stacks. • Any splices must be located in the same stacks for all columns of the group. • Individual stack lengths must be the same across the group.

Member type	Design group rules
Timber brace	• A brace may be in only **one** design group. • All braces in the group must have an identical cross section. • All braces in the group must have an identical grade. • All braces in the group must have an identical span length.

Timber member design commands

By right clicking over the required member in a view or appropriate branch of the **Project Workspace**, the following interoperative commands can be accessed:

- Design using Tekla Tedds (page 2255)
- Check using Tekla Tedds (page 2242)
- Export to Tekla Tedds (page 324)
- Clear Tekla Tedds Data (page 2247)

NOTE Initially only Design using Tekla Tedds (page 2255) command is shown, but once this has been run the other commands then become available.

Each of the above offers a sub-menu of choices, depending on context:

- > Model
- > Member
- > Group
- > Selection
- > *<Substructure name>*

NOTE Check using Tekla Tedds (page 2242) > Model is repeated on the Design ribbon tab as  Check in Tedds (page 2182)

13.10 Foundation design handbook

This handbook contains information relevant to the design of foundations in Tekla Structural Designer.

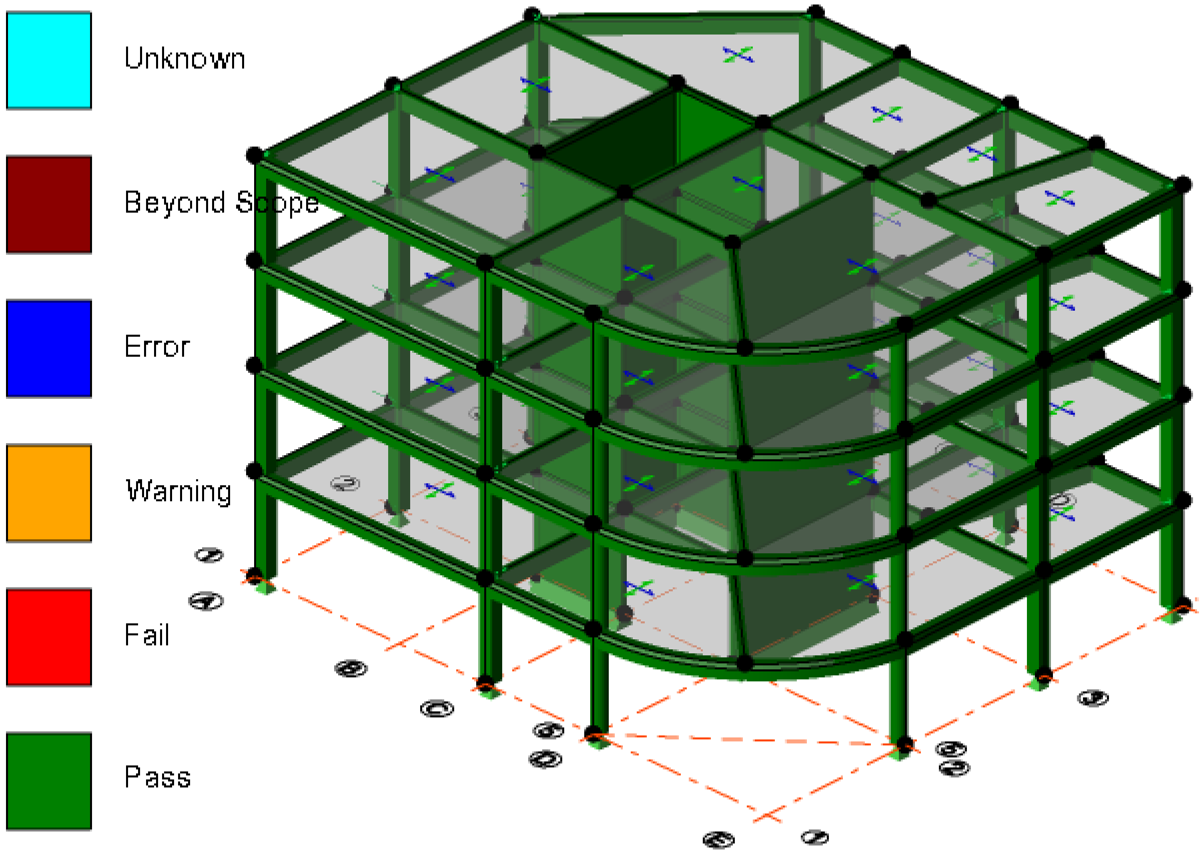
You can find the following information in this handbook:

- Pad base design workflow (page 1581)

- Pile cap design workflow (page 1587)
- Pad base, strip base and pile cap design forces (page 1593)
- Mat foundation design workflow (US customary units) (page 1608)
- Mat foundation design workflow (metric units) (page 1594)
- Piled mat foundation design workflow (US customary units) (page 1622)
- Piled mat foundation design workflow (metric units) (page 1633)
- Concrete slab design aspects (page 1373)

Pad base design workflow

The small concrete building model shown below will be used to demonstrate the pad base design process.



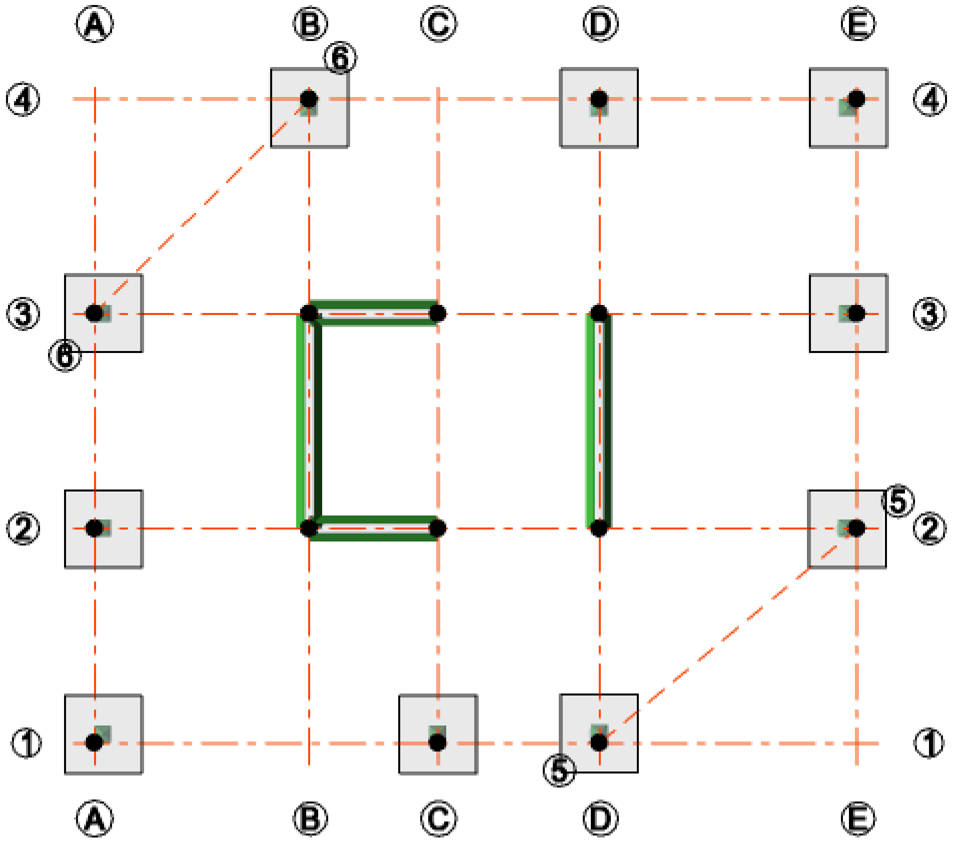
The model has already been designed prior to placing the bases.

Apply pad bases under supported columns

Pad bases can only be placed under, and be loaded by supported columns; strip bases can only be placed under and be loaded by supported walls.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the base design.

At this stage, as you are not aware of the individual base size and depth requirements; you can simply choose to place the bases where required, accepting the default size/depth offered.



Auto-size pad bases individually for loads carried

To obtain an idea of the range of potential sizes, bases should initially be designed individually for their respective loads, as follows:

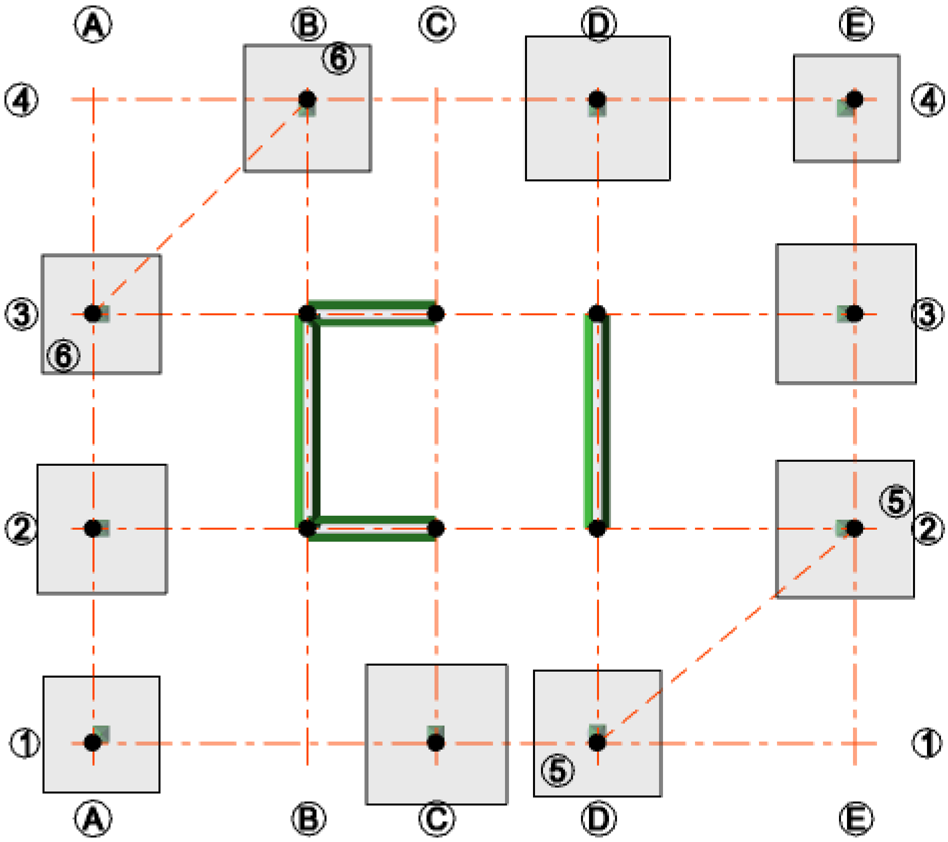
1. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.

2. Select the bases to be auto-sized and in the Properties Window and choose to auto-design both the size and depth; In this way the program establishes suitable base dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

3. From the **Foundations** tab click **Design Pad Bases**.

   Each base will be sized accordingly (any that are not in auto-design mode will simply be checked).



4. With the auto-design options cleared, you can then adjust individual base dimensions and re-check if required (by right-clicking the base that has been edited and choosing Check Member).

   The site boundary may impose restrictions on the positioning of an isolated foundation relative to the column/wall it supports. This restriction may result in a requirement for an offset base, this can be achieved by specifying the eccentricity required in the base properties.

NOTE The overall auto-design procedure is summarized as follows:

1. Bearing Design: - increase size

2. Bending Design: - increase reinforcement - If max allowable reinforcement is reached then increase depth, set reinforcement back to start point, and go back to step 1.
3. Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
4. Punching Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
5. Sliding Checks - increase depth, set reinforcement back to start point, and go back to step 1
6. Uplift Checks - increase size, set reinforcement back to start point, and go back to step 1

At every stage, if the max allowable depth is reached the design fails.

Apply grouping to rationalize pad base sizes

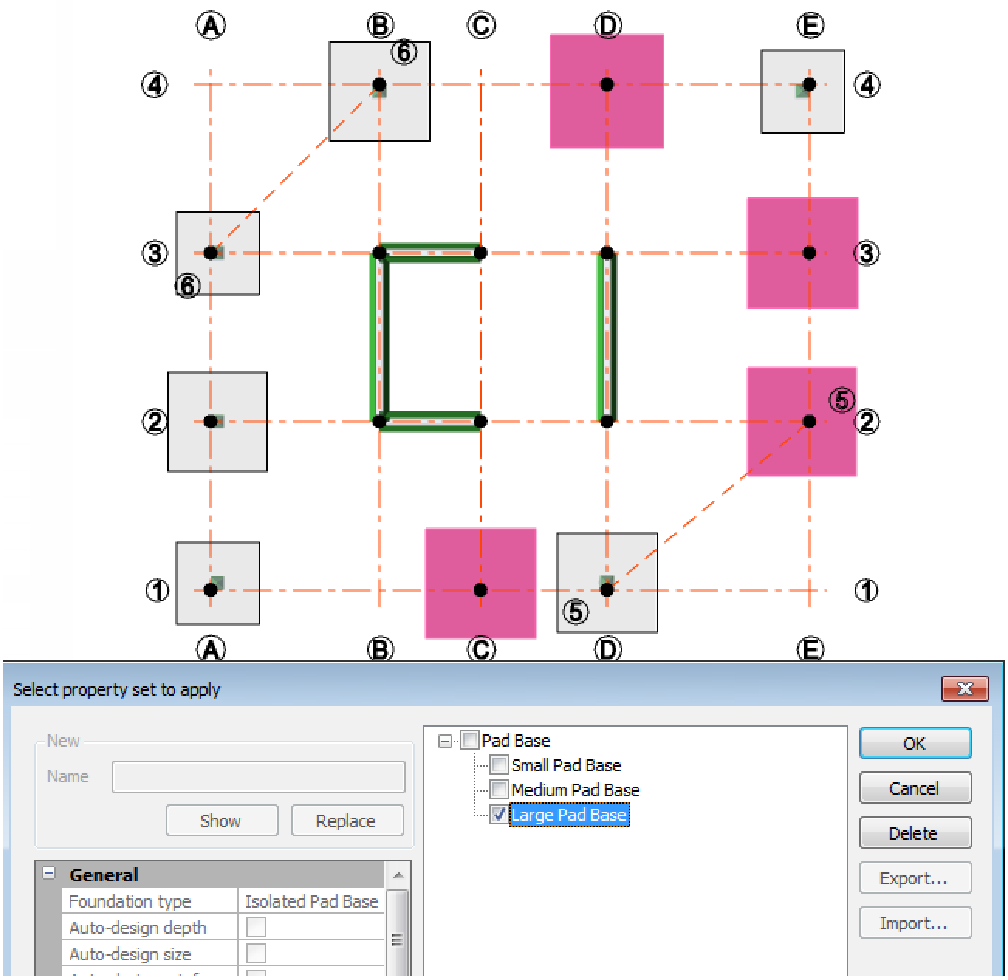
NOTE Grouping can only be applied to pad bases - not to strip bases.

Once pad bases have been sized individually, the designs can be rationalized by activating grouping, in order to obtain one design per group sufficient for all bases within the group.

This is done as follows:

1. Select a base that you want to be in a particular group.
2. In the Properties Window, ensure it is set to auto-design.
3. Right-click the same base and from the context menu choose Create Property Set...
4. Select all the other bases that you want to be in the same group.

5. In the Properties Window, click **Apply...** to apply the property set you have just created to the selected bases.

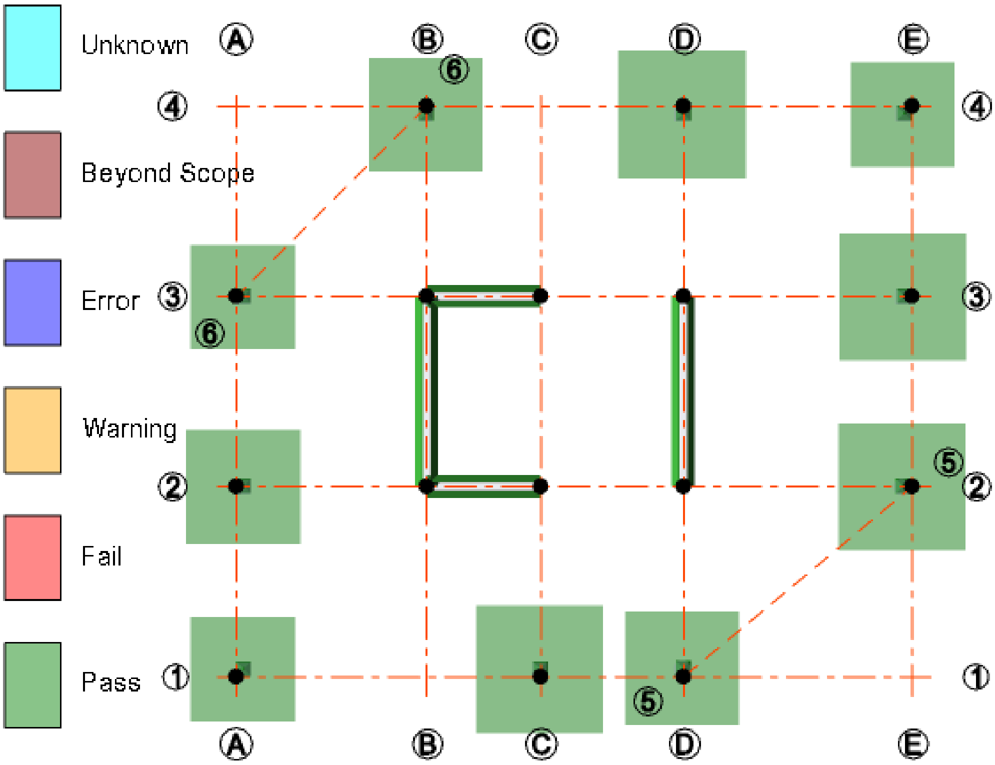


NOTE Ensure you click **Apply...** from the Properties Window and not from the right-click menu, otherwise the properties will only be applied to the last base selected.

6. From the Groups page of the Project Workspace, right-click Pad Bases (under the Design branch) and choose Regroup Members - this will put those bases that share similar properties into the same group.

7. Open **Design Settings** dialog box, and from the Design Groups page select the option to design isolated foundations using groups.

8. Click **Design Pad Bases** - the results obtained will reflect the grouping that has been applied.



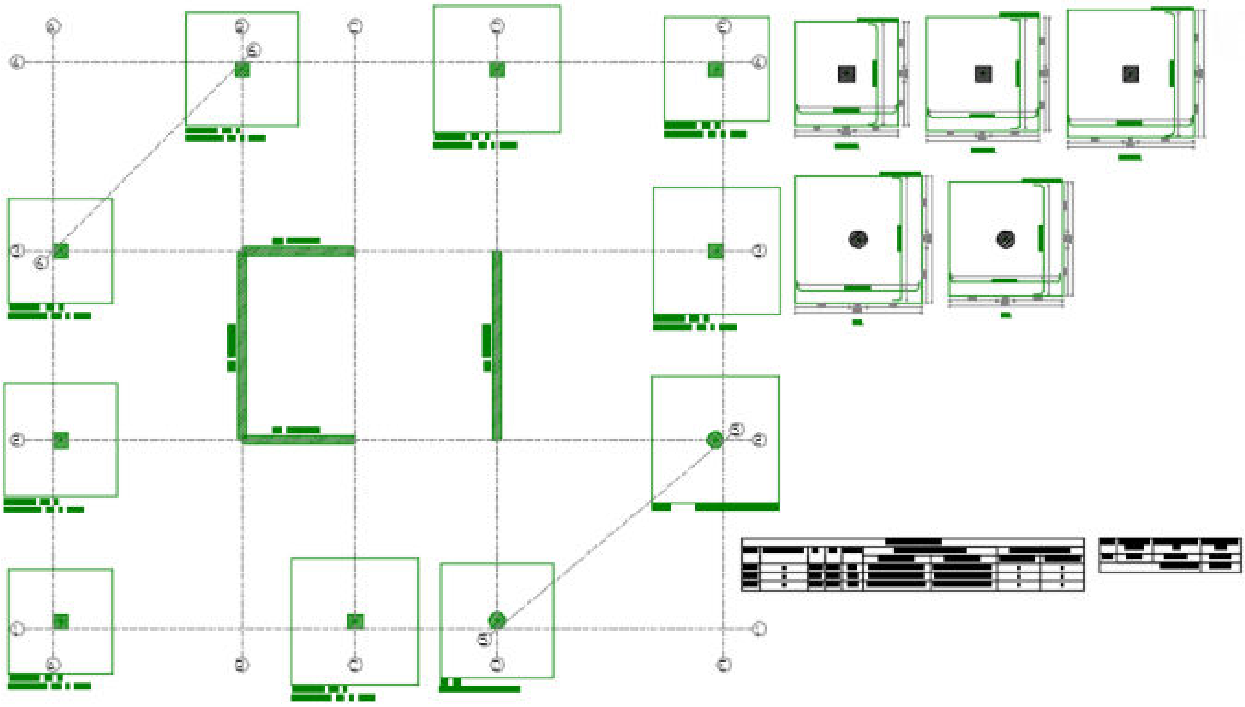
Review/optimize base design

In the Review View you can examine the design efficiency by switching from Foundations Status to Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in **Design Options> Concrete> Foundations**.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer



Print calculations

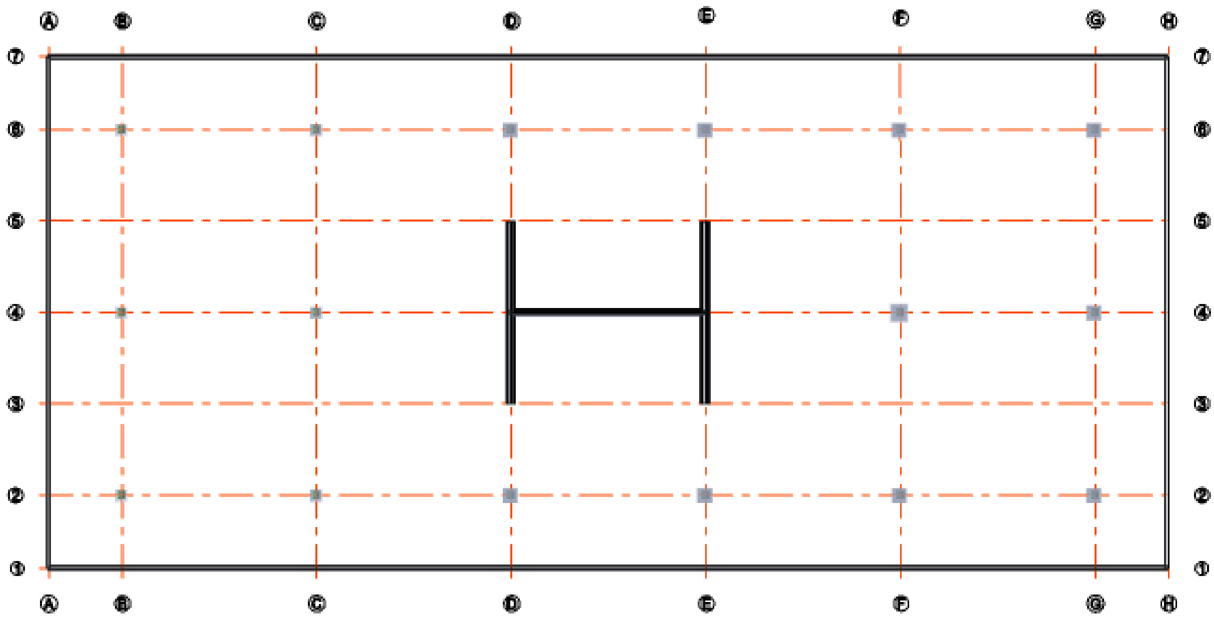
Create a model report that includes the concrete pad base design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

See also

Apply user defined utilization ratios (page 786)

Pile cap design workflow

The small concrete building model shown below will be used to demonstrate the pile cap design process.



The model has already been designed prior to placing the pile caps.

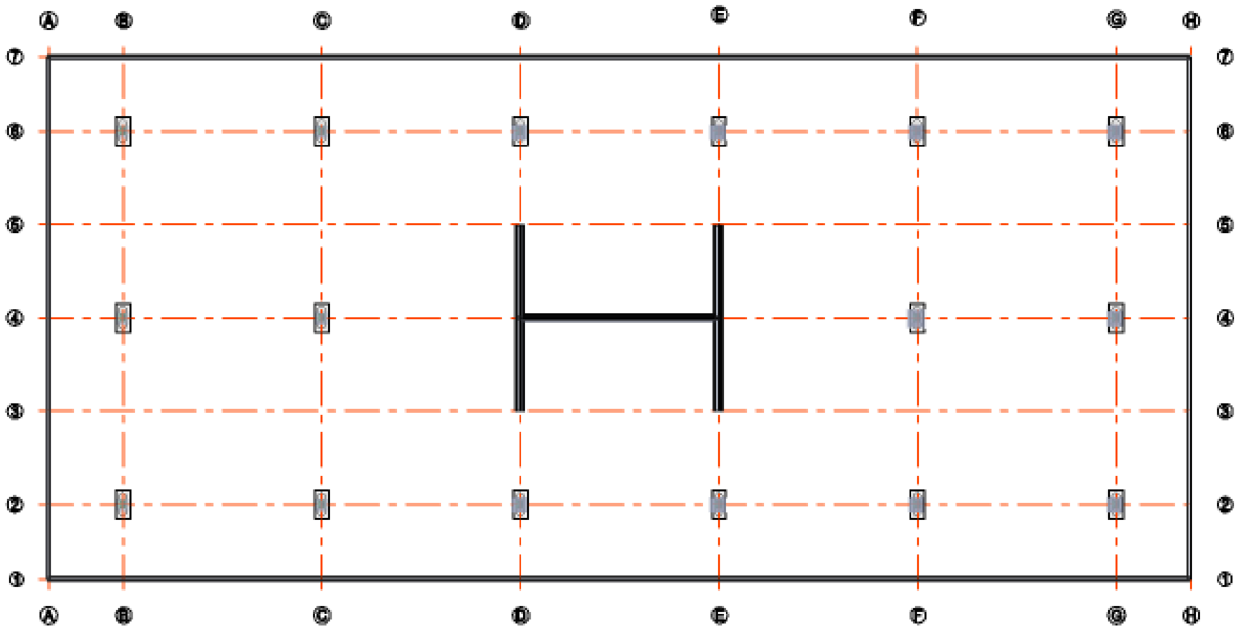
Apply pile caps under supported columns

Before a pile cap can be placed the **Pile Type Catalogue** must contain at least one pile type.

Pile caps can only be placed under and be loaded by supported columns.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the pile cap design.

At this stage, as you are not aware of the individual pile cap size and depth requirements; you can simply choose to place pile caps where required, accepting the default size/depth offered.



Auto-size pile caps individually for loads carried

To obtain an idea of the range of potential sizes, pile caps should initially be designed individually for their respective loads

Note that when piles are auto-designed the outcome will depend on the auto-design method selected; the pile cap size will either be based on the minimum number of piles required, or on the minimum pile capacity.

To individually size the pile caps:

1. In the **Design Settings** dialog box got to **Concrete> Foundations> Isolated Foundations > Piles** to choose the pile auto-design method required: (minimize pile capacity, or minimize number of piles).
2. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.
3. Select the pile caps to be auto-sized and then in the Properties Window choose to auto-design both the piles and depth; In this way the program will establish suitable pile cap dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.
4. From the **Foundations** tab, click **Design Pile Caps** and all the pile caps set in auto-design mode will be sized accordingly. (Those not in auto-design mode will simply be checked). Similarly the piles beneath the pile caps will either be designed (if pile auto-design mode is enabled) or checked.

At any point you can switch to a user defined arrangement, modify the pile cap configuration and have the design re-checked.

One example where you might choose a user defined arrangement is where the site boundary imposes restrictions on the positioning of the pile cap relative to the column/wall it supports. Switching to a user defined arrangement allows you to specify an eccentricity and create an offset pile cap.

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

Apply grouping to rationalize pile cap sizes

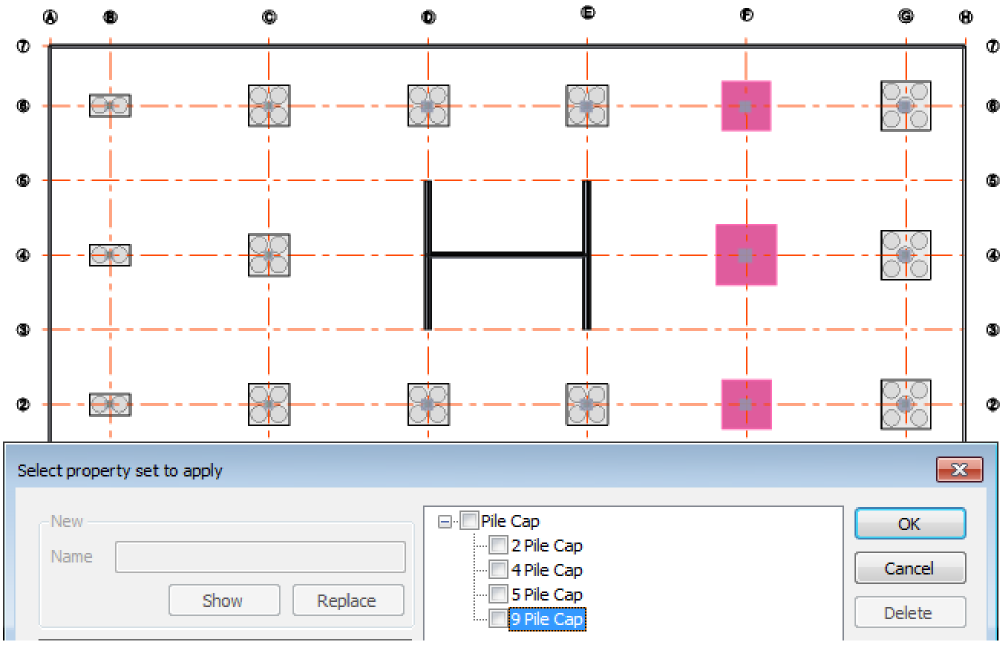
Once pile caps have been sized individually, the designs can be rationalized by activating grouping, in order to obtain one design per group sufficient for all pile caps within the group.

This is done as follows:

1. Select a pile cap that you want to be in a particular group.
2. In the Properties Window, ensure it is set to auto-design.
3. Right-click the same pile cap and from the context menu choose Create Property Set...
4. Select all the other pile caps that you want to be in the same group.

   NOTE When applied moments are significant, be cautious when grouping pile caps where auto-design has initially determined different principal directions.

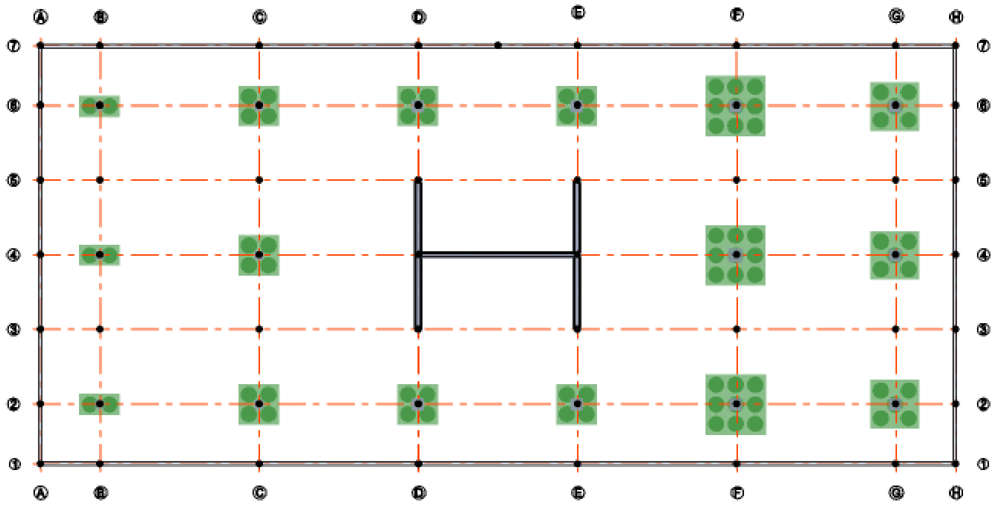
5. In the Properties Window, click **Apply...** to apply the property set you have just created to the selected pile caps.



NOTE Ensure you click **Apply...** from the Properties Window and not from the right-click menu, otherwise the properties will only be applied to the last pile cap selected.

6. From the Groups page of the Project Workspace, right-click **Pile Caps** (under the Design branch) and choose Regroup Members - this will put those pile caps that share similar properties into the same group.
7. Open the **Design Settings** dialog box, and from the Design Groups page select the option to design isolated foundations using groups.

8. Click **Design Pile Caps** - the results obtained will reflect the grouping that has been applied.



Review/optimize pile cap design

In the Review View you can graphically examine the pile cap design status & efficiency by switching between Foundations Status and Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

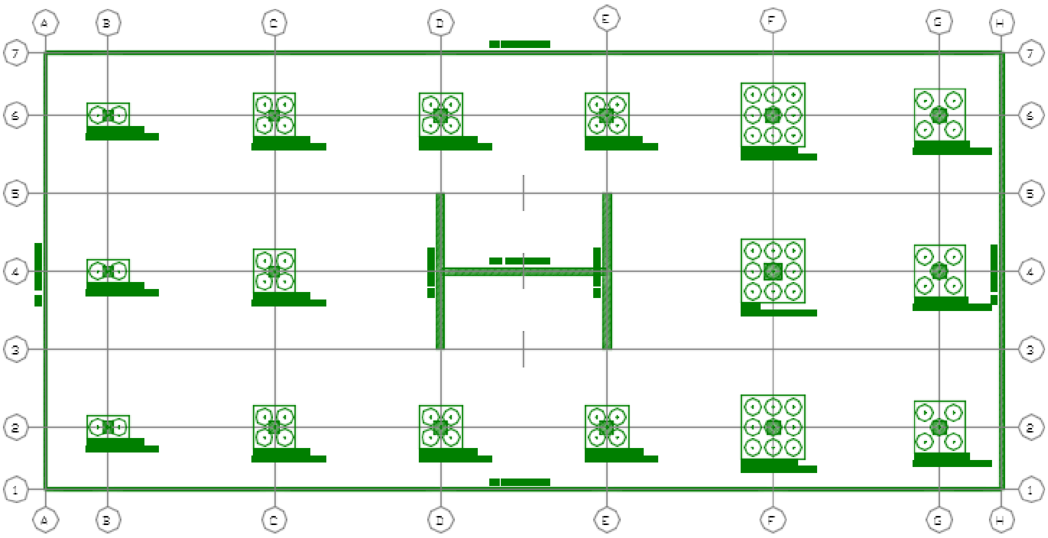
Similarly you can graphically examine the pile design status & efficiency by switching between Piles Status and Piles Ratio.

If the selections are unacceptable you may need to review the settings in **Design Options> Concrete> Foundations**.

Design results can be examined in more detail by selecting Tabular Data and setting the View Type to Design Summary. By selecting the Pile Caps or Piles as the Characteristic (and the appropriate material) the full results for each pile cap or pile can be examined.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.



Print calculations

Create a model report that includes the concrete pile cap design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

See also

Apply user defined utilization ratios (page 786)

Pad base, strip base and pile cap design forces

Forces acting on supports

The following forces and moments on the supports are determined from the analysis of the active load combinations:

- Vertical force in direction Z
- Horizontal forces in directions Y and X
- Moments around X and Y axis

NOTE A torsional moment around the Z axis is also determined, but the base/pile cap is not designed for this in the current release.

Foundation self weight

The foundation self-weight is automatically calculated and applied as an added load, F_{swt}

Soil self weight

The surcharge depth and soil unit weight that have been specified in the base/ pile cap properties are used to determine the soil self weight. This is applied as an added load, F_{soil}

NOTE In the current release, horizontal pressure caused by soil is not considered.

Additional surcharge loads

For isolated foundations user can apply additional surcharge loads: acting in the global Z direction.

- Permanent (dead) surcharge load
- Variable (live) surcharge load

Design Forces

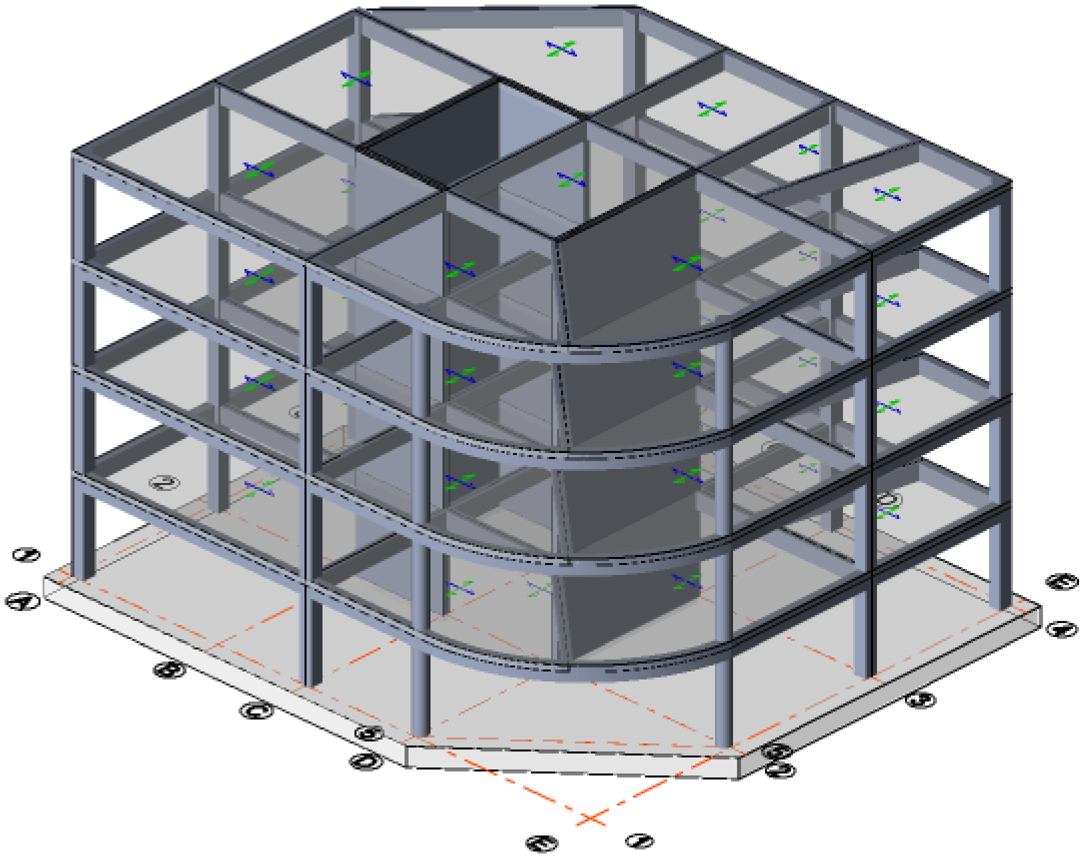
To obtain the design forces, the loads due to foundation self weight, soil self weight and additional surcharge are added to the forces acting on the supports.

These design forces (axial load and bi-axial shear and moment) are then applied to the base/pile cap at the foundation level.

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

Mat foundation design workflow (metric units)

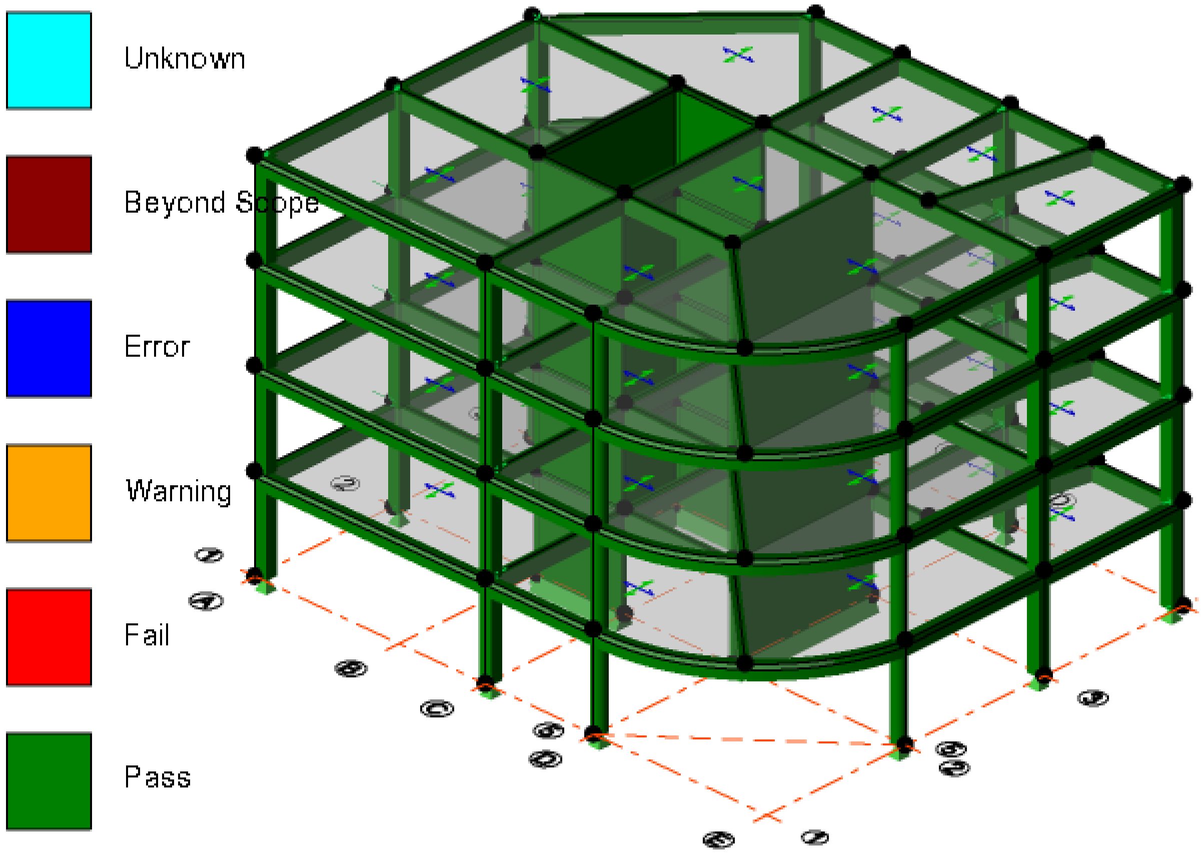
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
    - Linear ground stiffness, or,
    - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear spring are applied:

- Allowable bearing pressures are checked
- Uplift (tension) is checked
- If no problems then linear springs are sufficient

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit (kN/m3)	Upper Limit (kN/m3)
Loose Sand	4,800	16,000
Medium Dense Sand	9,600	80,000
Dense Sand	64,000	128,000
Clayey Medium Dense Sand	32,000	80,000
Silty Medium Dense Sand	24,000	48,000
Clayey Soil (qa<200kPa)	12,000	24,000
Clayey Soil (200<qa<800 kPa)	24,000	48,000
Clayey Soil (qa>800kPa)	48,000	200,000

1. Reference: "Foundation Analysis and Design", Joseph E. Bowles, 1995 (Table 9-1)

Horizontal support

The degree of horizontal support provided by the ground springs can be set as:

- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to "Free" a mechanism will result unless you provide additional discreet supports.

Determine the remaining mat properties

You are required to manually specify the Reduce live loads by mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

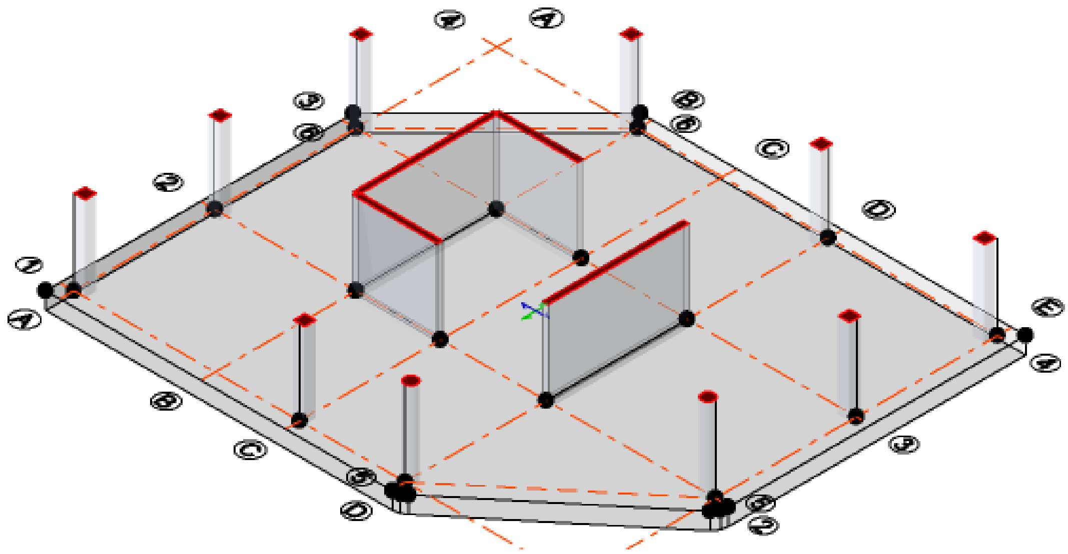
In this example the minimum area method is used to create a mat with the following properties:

- Imposed loads reduced by 30%
- Default overhang
- Mat thickness 600mm
- Ground bearing springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)
3. Click on those columns (or walls) that define its perimeter,
4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

When not supported by a mat, columns and walls typically have supports at their bases.

When a mat is introduced these supports must be removed - as the mat now supports the whole building on ground bearing springs. Consequently adding a mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

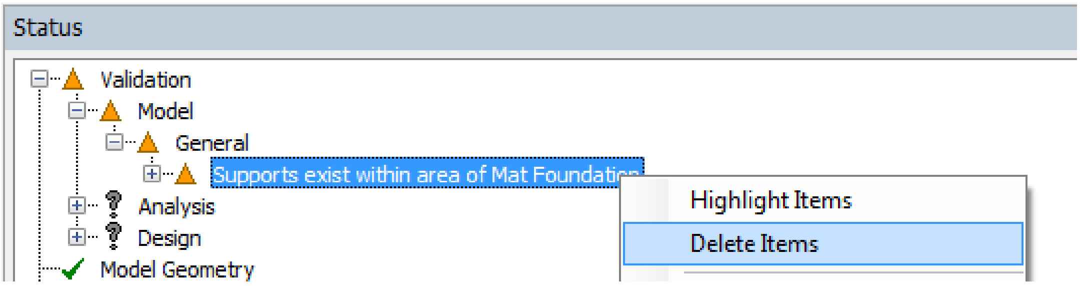
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Design All (Static), and any seismic RSA combinations by running Design All (RSA). These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

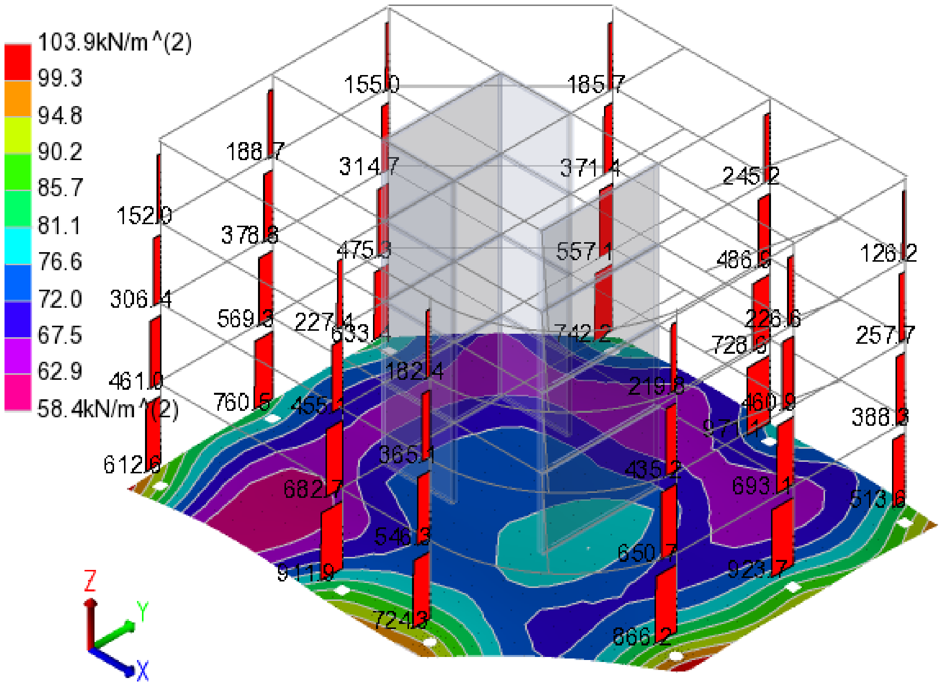
In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

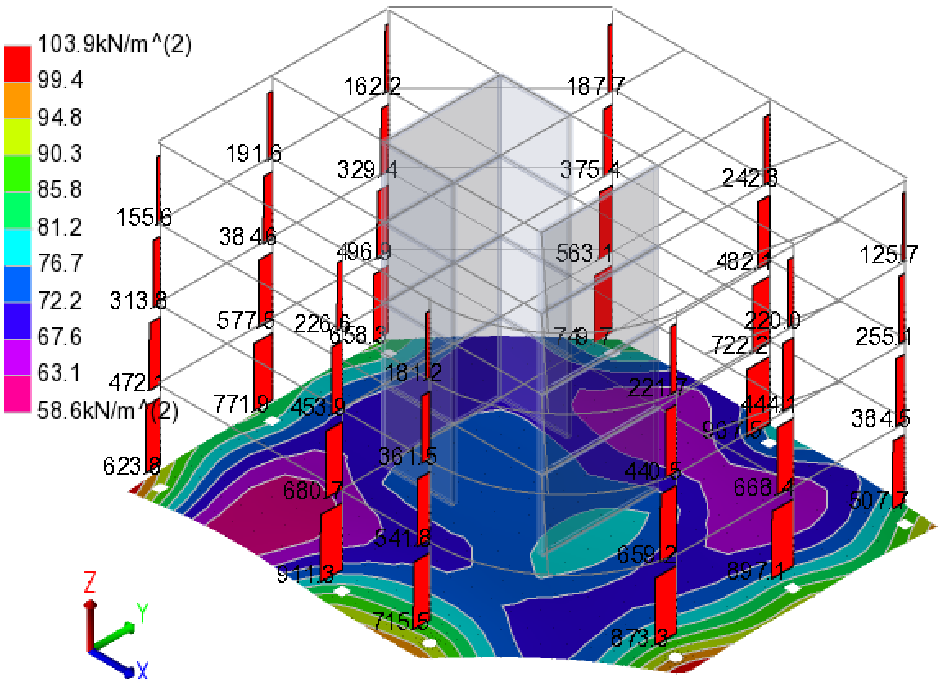
NOTE In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run the analyses that are required from the Analyze ribbon then re-run the member design at a later stage.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

The default stiffness adjustments are dependent on the design code. For design to EC2 the default adjustment factor applied is 0.2. For design to ACI the default adjustment factor applied is 0.25.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking Design All (Static) from the Design toolbar.

NOTE Similarly if an RSA design has previously been performed, but is now out of date Design All (RSA) should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

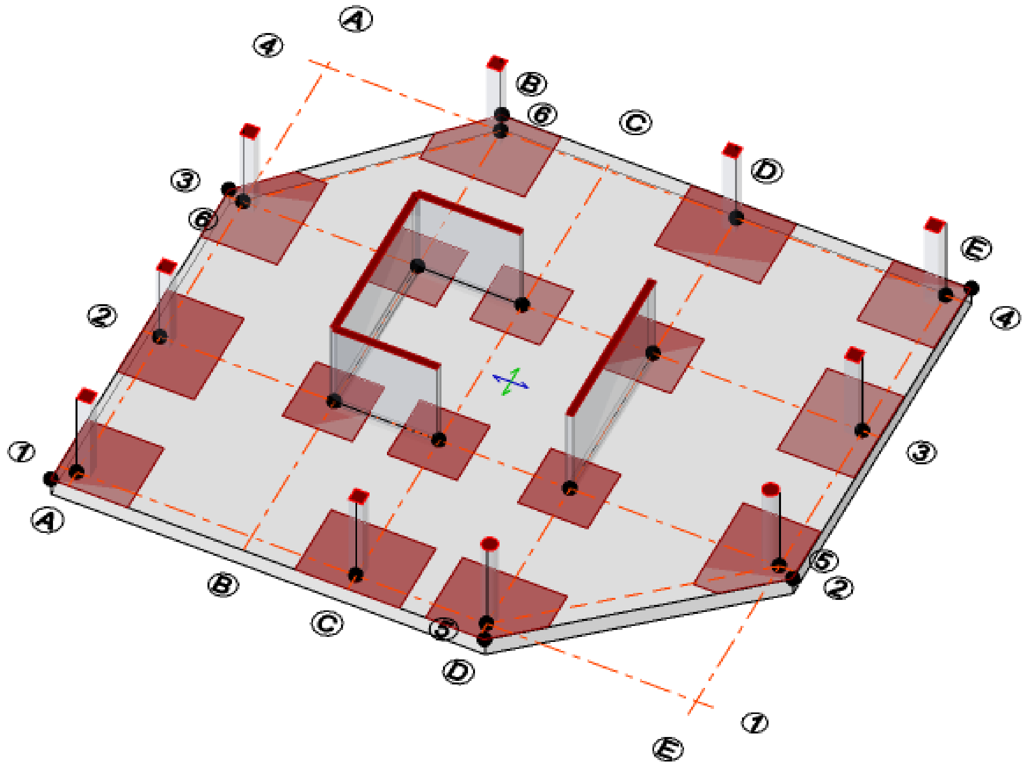
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgement.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run Design Mats in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either Design Slabs or Check Slabs.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Slabs will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Slabs will check the current reinforcement in slabs and mats regardless of the current autodesign setting.

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.

- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run Design Patches in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

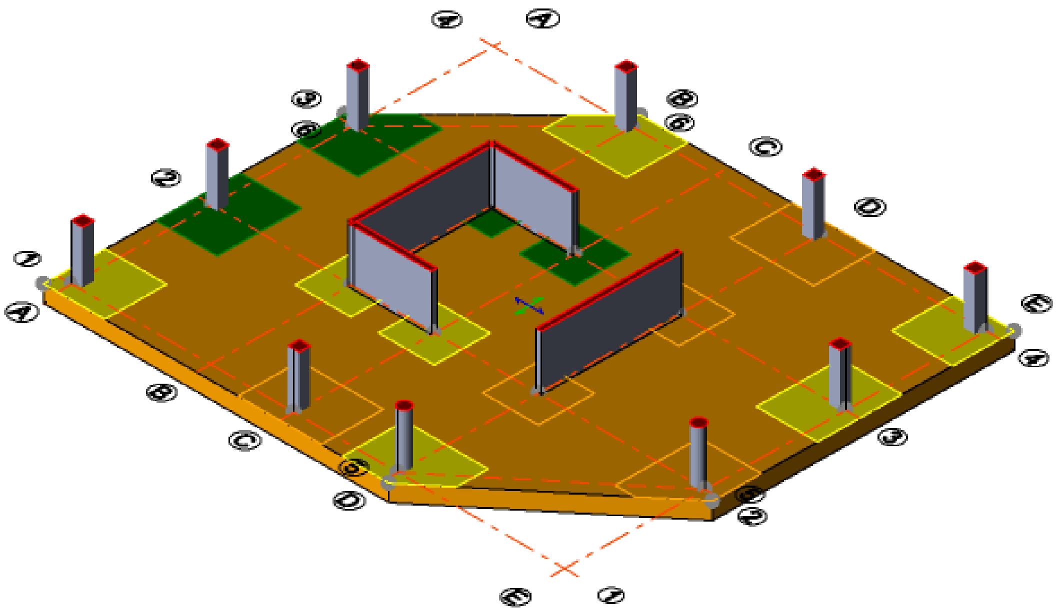
- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardise the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has

H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

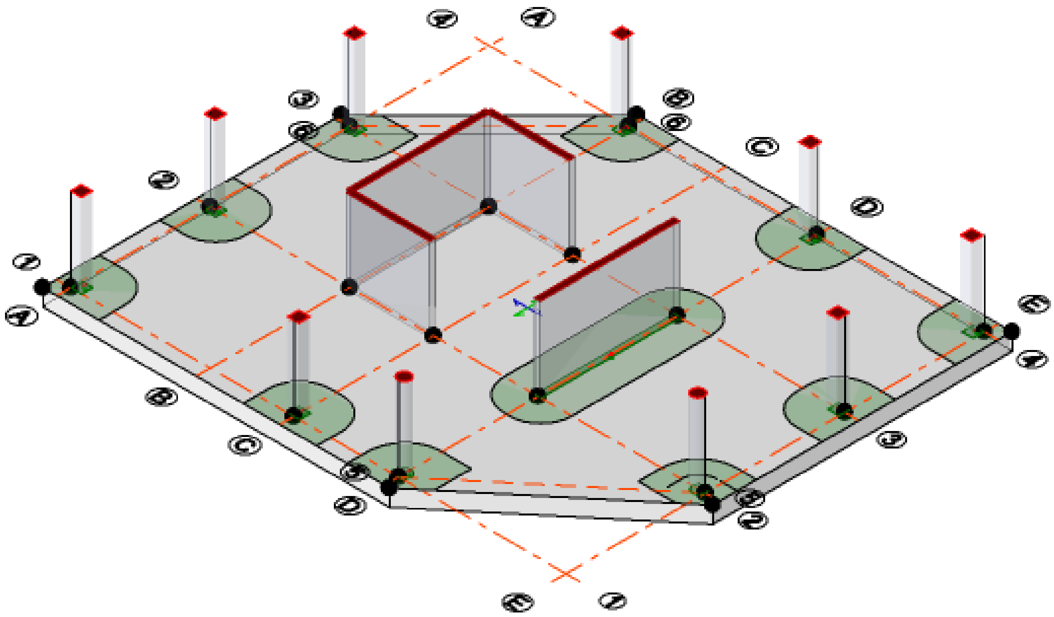


Add and run punching checks

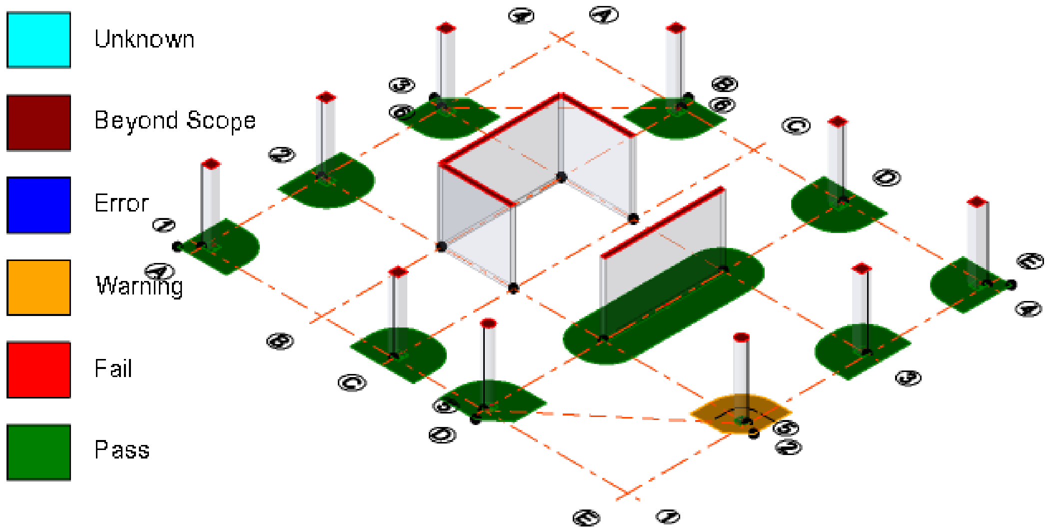
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See: Create punching shear checks (page 801)

You can then select any check and review the properties assigned to it.



Once added click Design Punching Shear. See: Design and check punching shear (page 803)



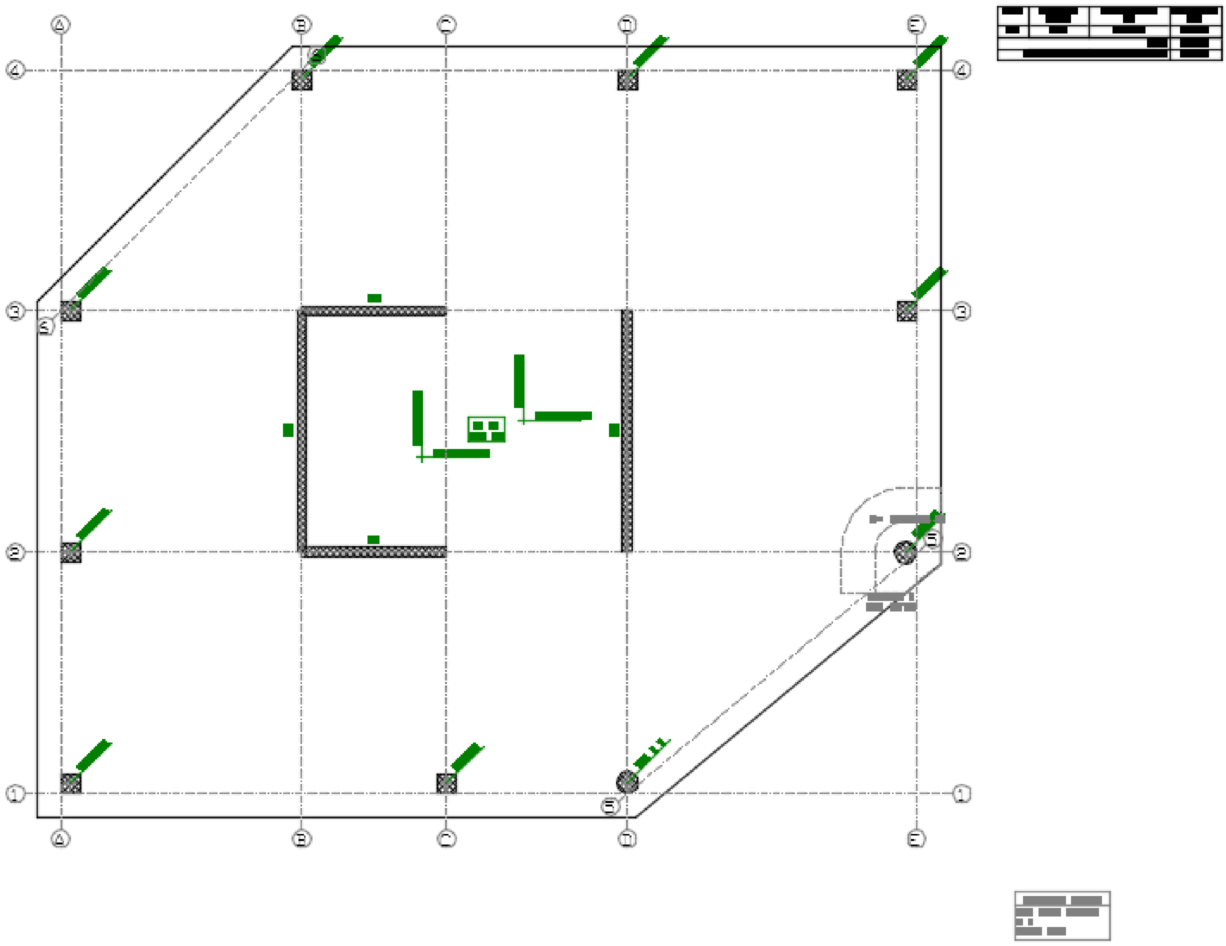
The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet

- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

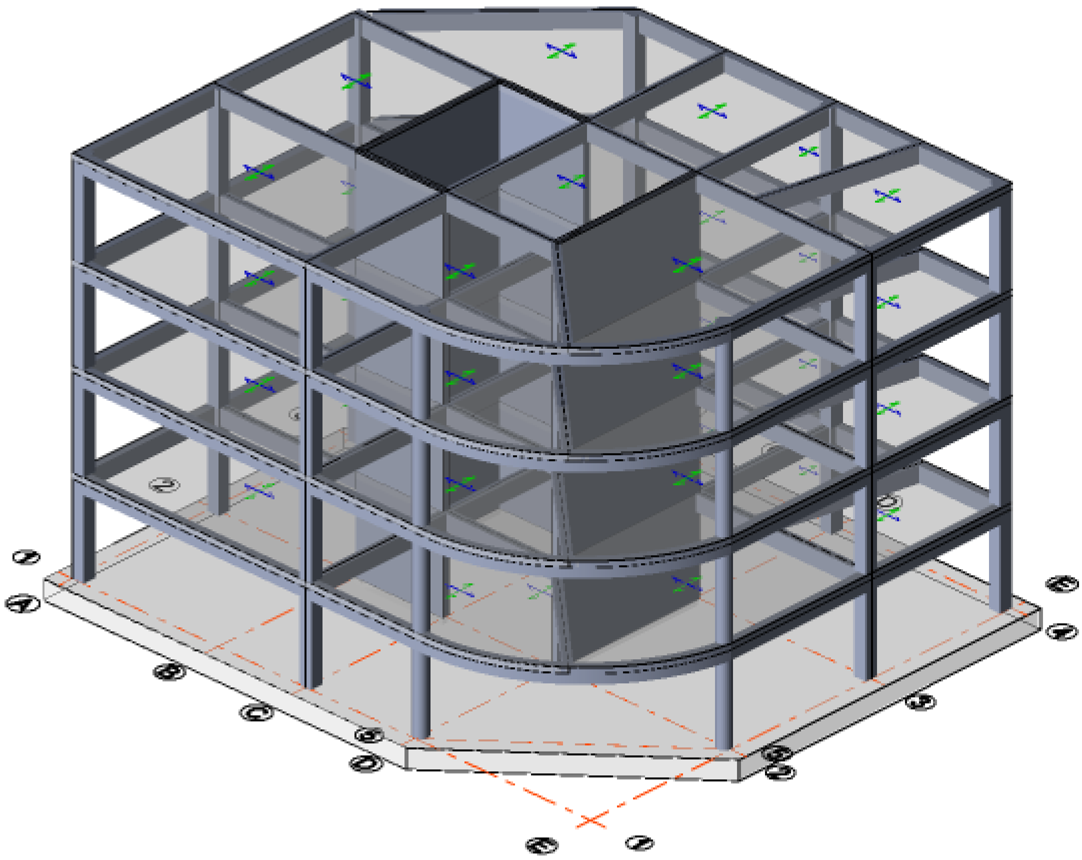


Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Mat foundation design workflow (US customary units)

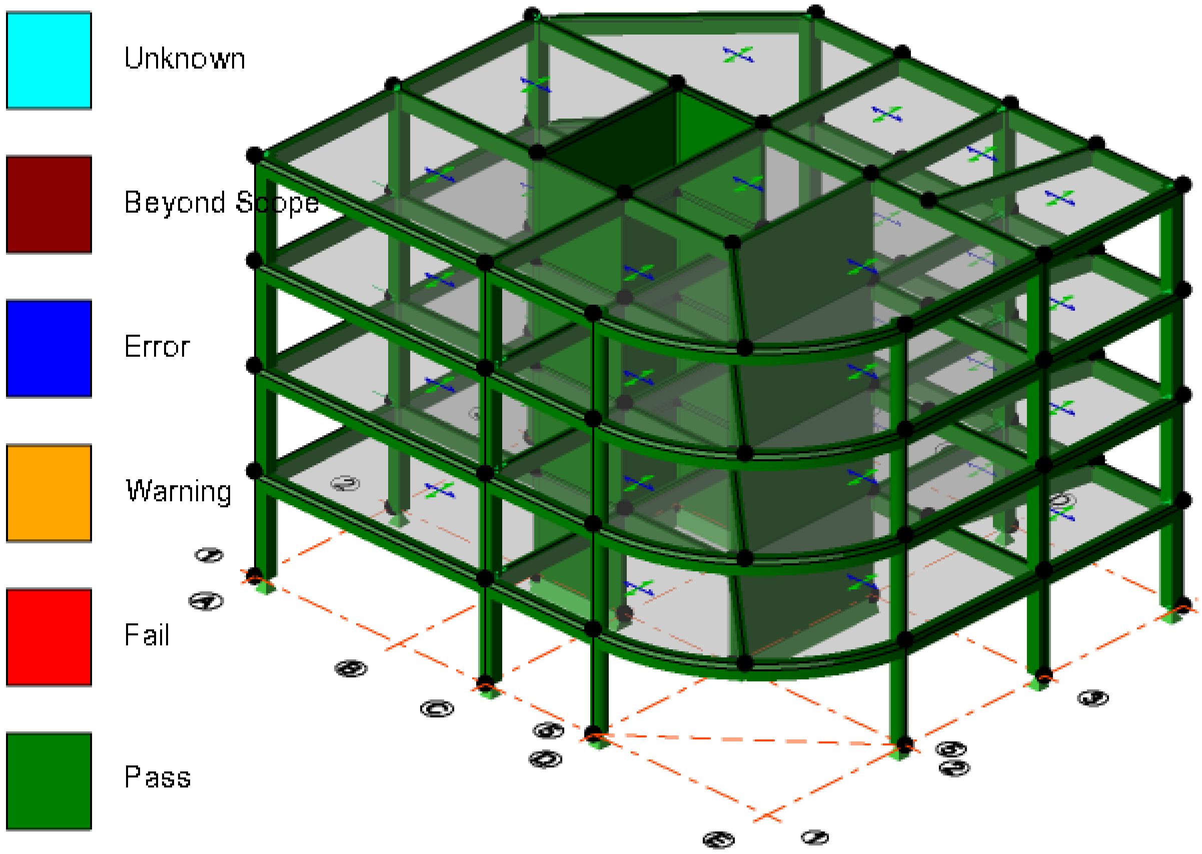
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
    - Linear ground stiffness, or,
    - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear springs are applied:

- Allowable bearing pressures are checked
- Uplift (tension) is checked
- If no problems then linear springs are sufficient

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit		Upper Limit	
	(kN/m3)	(kip/ft2/ft) approx.	(kN/m3)	(kip/ft2/ft) approx.
Loose Sand	4,800	31	16,000	102
Medium Dense Sand	9,600	61	80,000	509
Dense Sand	64,000	407	128,000	815
Clayey Medium Dense Sand	32,000	204	80,000	509
Silty Medium Dense Sand	24,000	153	48,000	306
Clayey Soil (qa<200kPa)	12,000	76	24,000	153
Clayey Soil (200<qa<800kPa)	24,000	153	48,000	306
Clayey Soil (qa>800kPa)	48,000	306	200,000	1273

1. Reference: "Foundation Analysis and Design", Joseph E. Bowles, 1995 (Table 9-1)

Horizontal support

The degree of horizontal support provided by the ground springs can be set as:

- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to "Free" a mechanism will result unless you provide additional discreet supports.

Determine the remaining mat properties

You are required to manually specify the Reduce live loads by mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

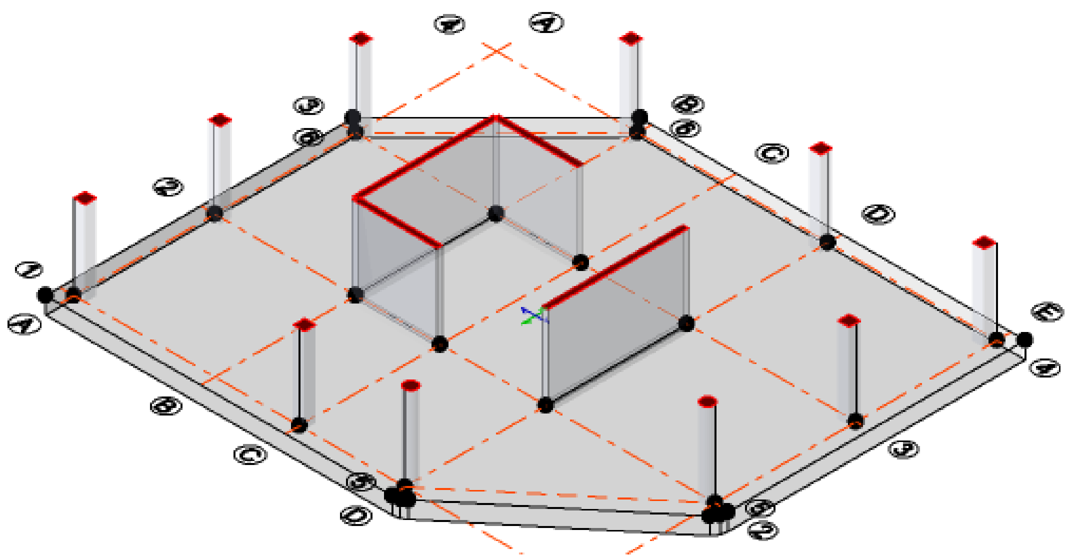
In this example the minimum area method is used to create a mat with the following properties:

- Live loads reduced by 30%
- Overhang, 20in.
- Mat thickness, 24in.
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)
3. Click on those columns (or walls) that define its perimeter,
4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

When not supported by a mat, columns and walls typically have supports at their bases.

When a mat is introduced these supports must be removed - as the mat now supports the whole building on ground bearing springs. Consequently adding a mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

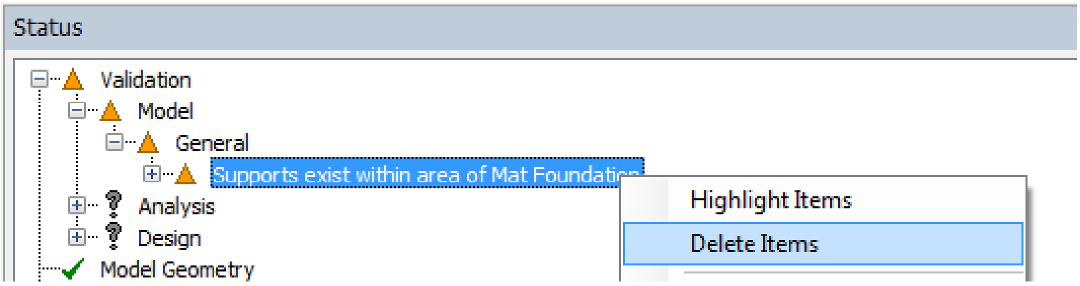
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Design All (Static), and any seismic RSA combinations by running Design All (RSA). These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

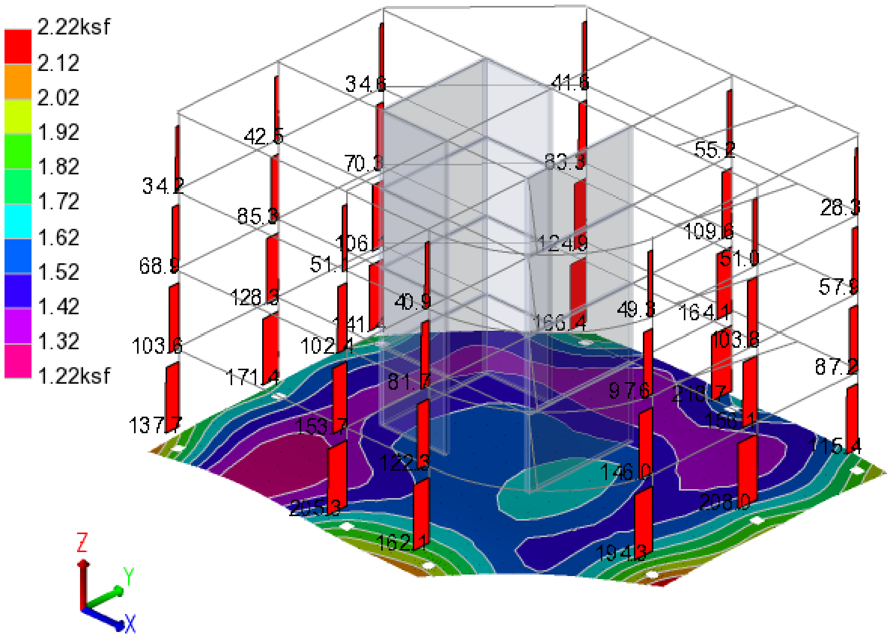
In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

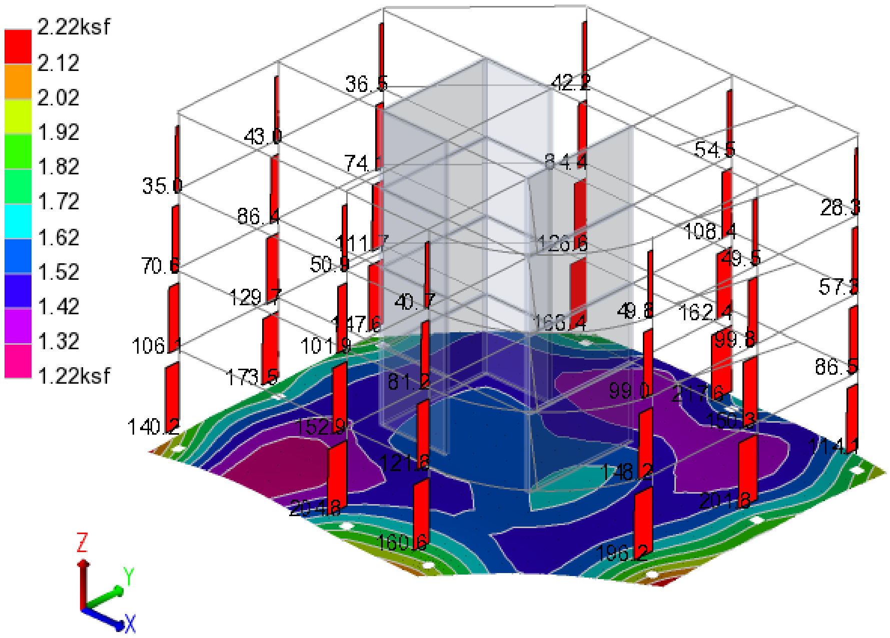
NOTE In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run the analyses that are required from the Analyze ribbon then re-run the member design at a later stage.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via

Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

For the above analysis Tekla Structural Designer default stiffness adjustment factors were used, the default factor for foundation mats being 0.2 for both ACI and EC2 design codes.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking Design All (Static) from the Design toolbar.

NOTE Similarly if an RSA design has previously been performed, but is now out of date Design All (RSA) should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

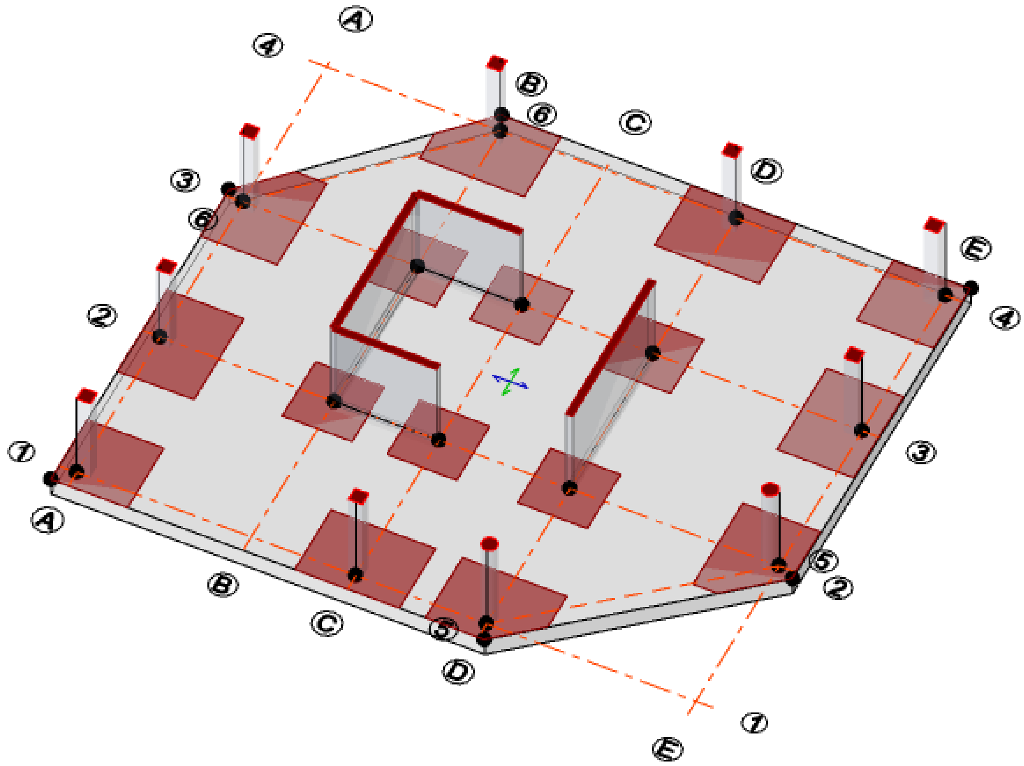
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgement.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run Design Mats in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either Design Slabs or Check Slabs.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Slabs will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Slabs will check the current reinforcement in slabs and mats regardless of the current autodesign setting.

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 3 size bars are selected and you would just never use anything less than a 4 size bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 6".

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.

- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run Design Patches in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

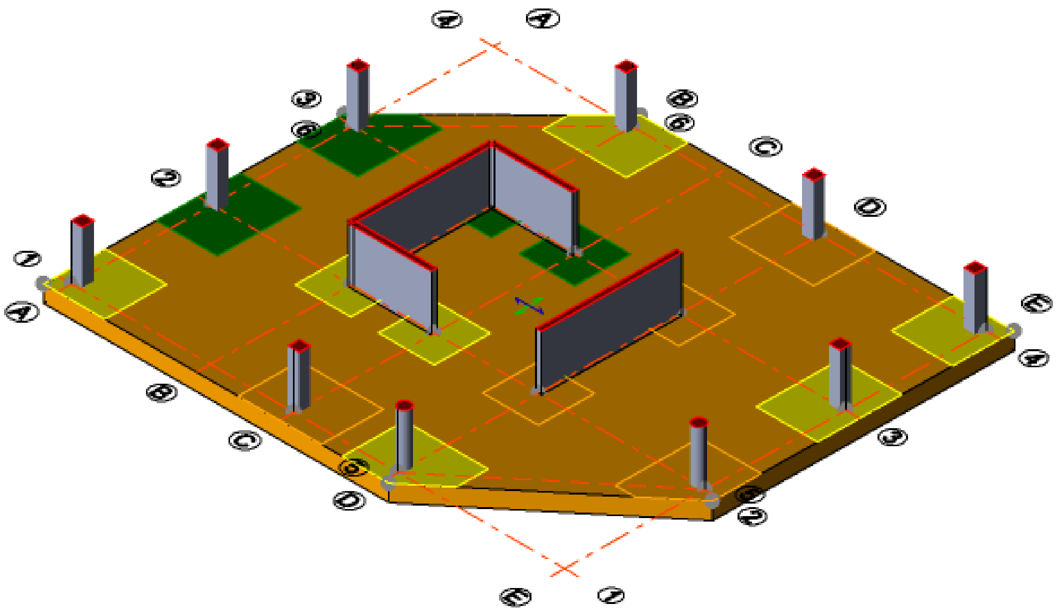
- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has

size 4 at 8in, then in the patch don't add size 4 at 5in, swap to size 5 at 8in - then there will be alternate bars at 4in crs in this strip of the patch.

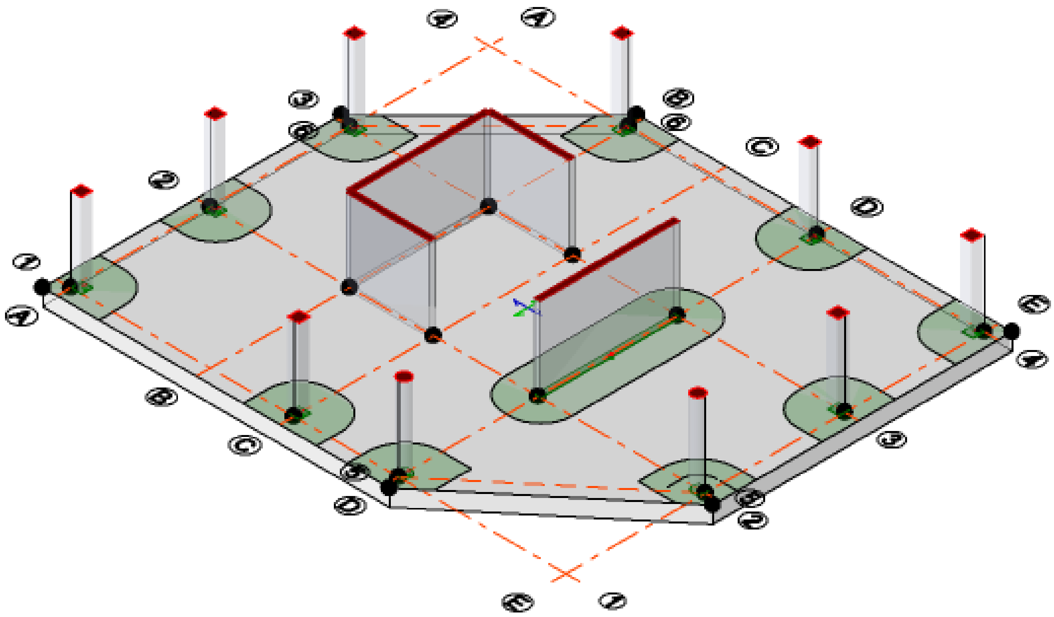


Add and run punching checks

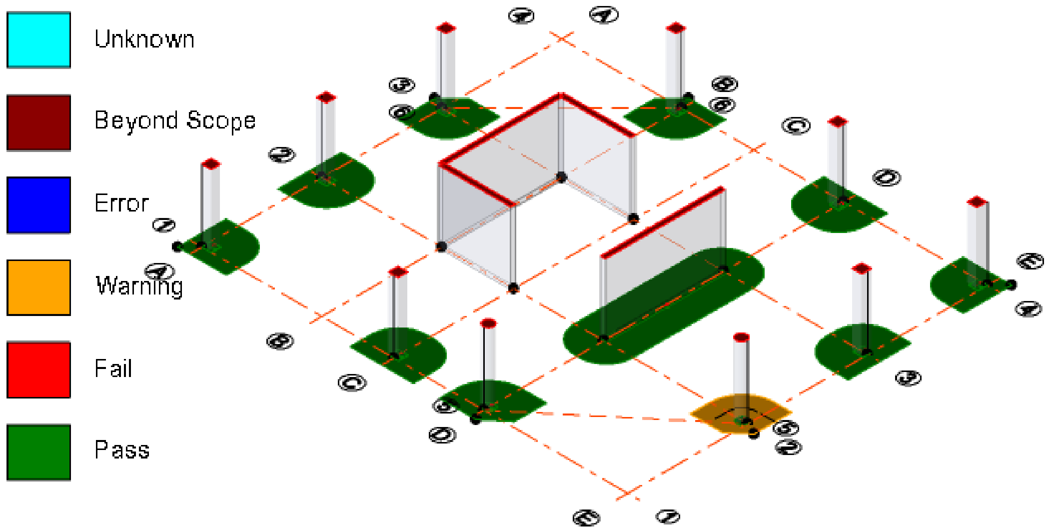
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See: Create punching shear checks (page 801)

You can then select any check and review the properties assigned to it.



Once added click Design Punching Shear. See: Design and check punching shear (page 803)



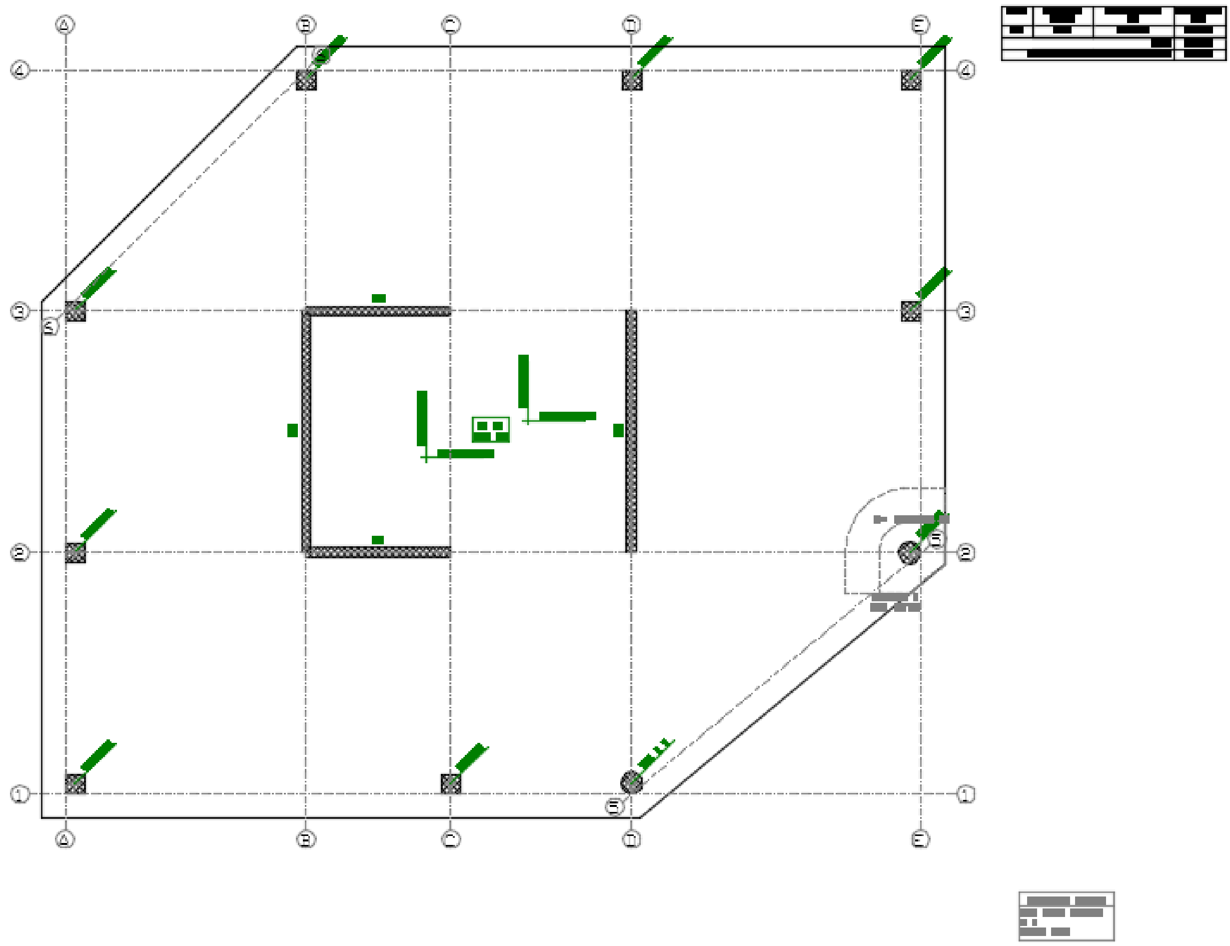
The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet

- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

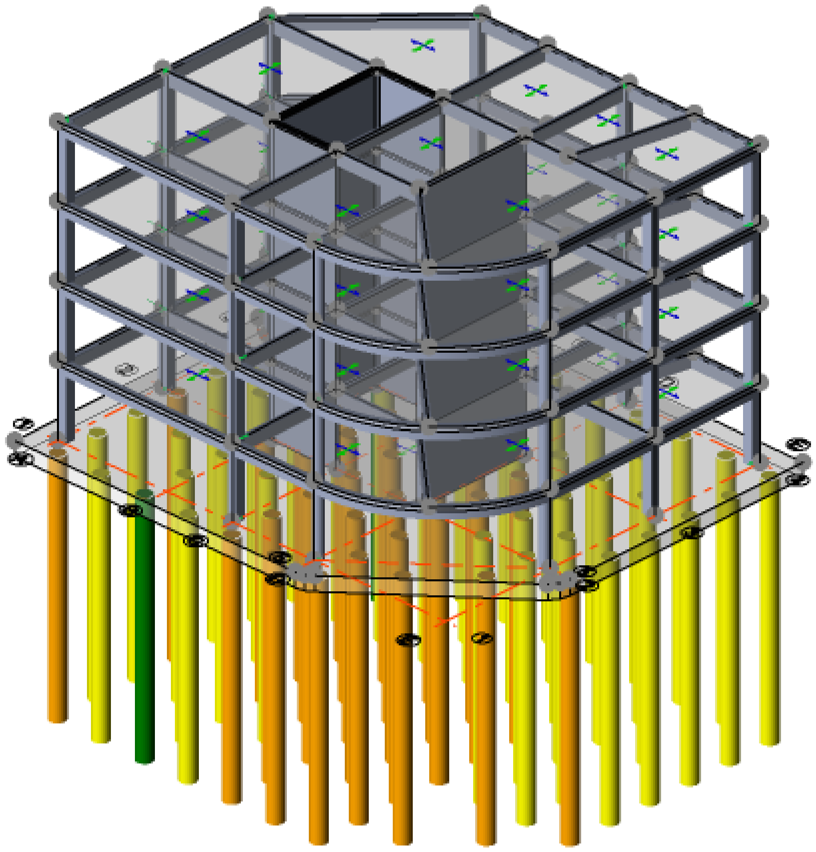


Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

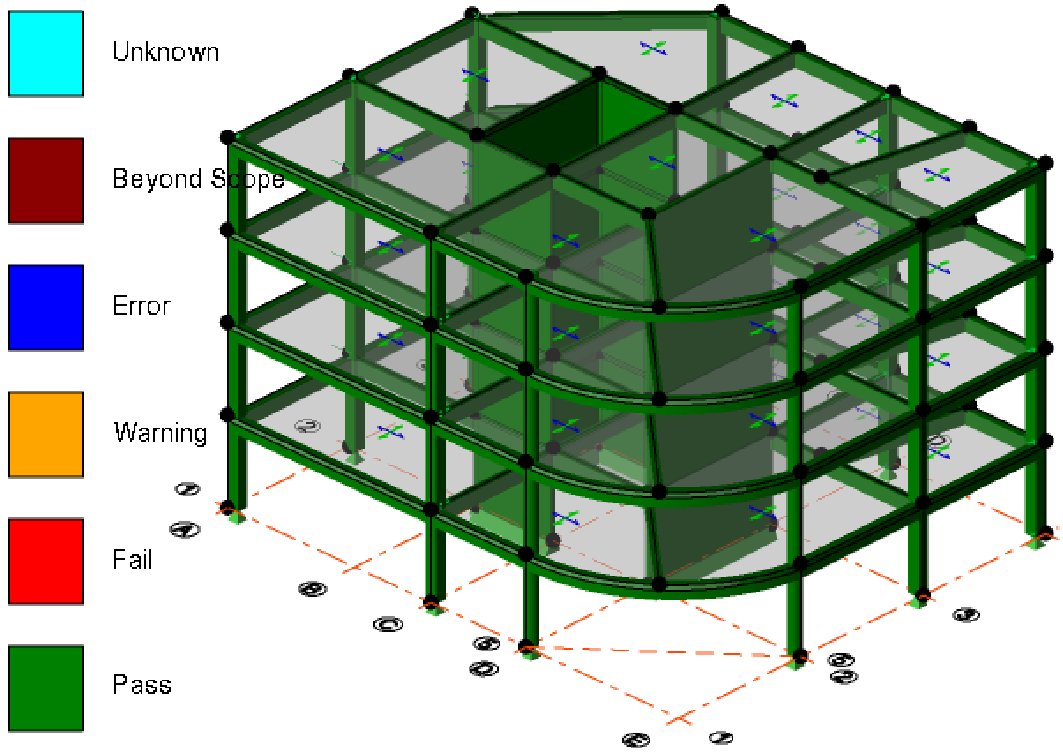
Piled mat foundation design workflow (US customary units)

The following example illustrates the typical process to model and design piles in a piled mat foundation.



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

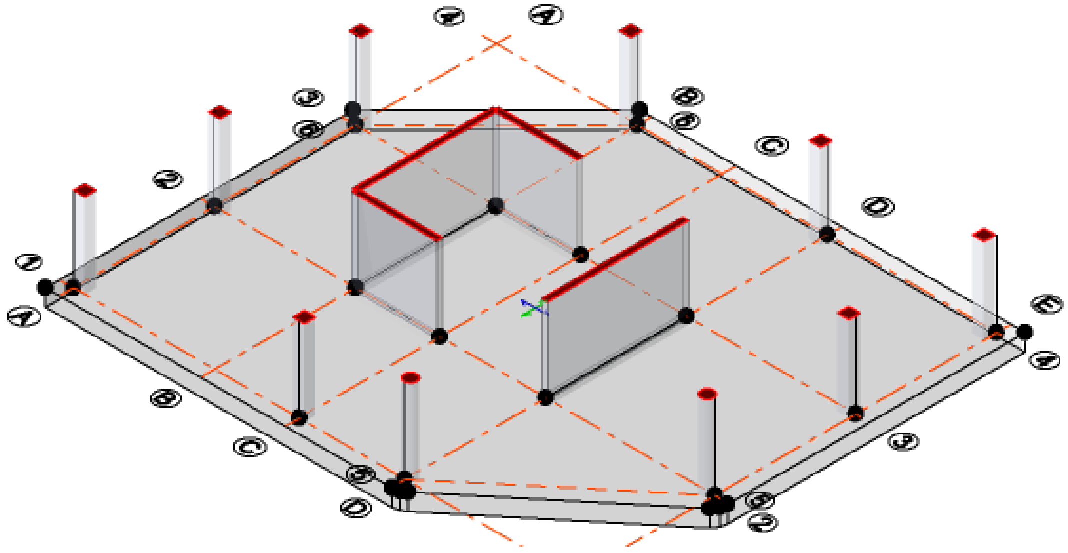
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 40in.
- Mat thickness 24in.
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

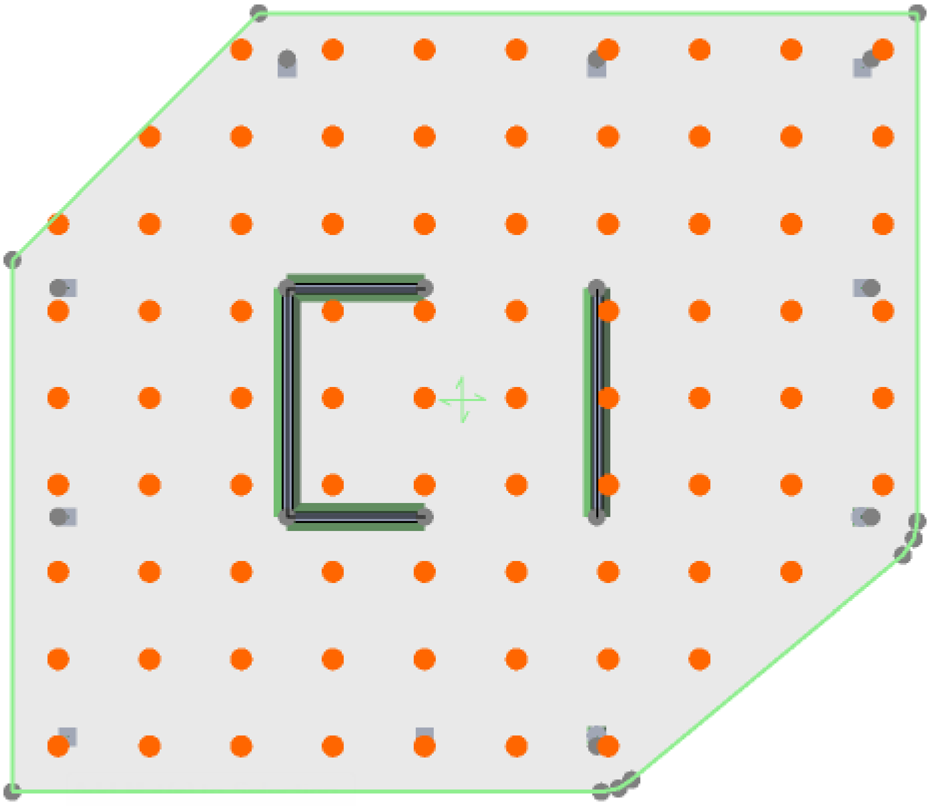
NOTE In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

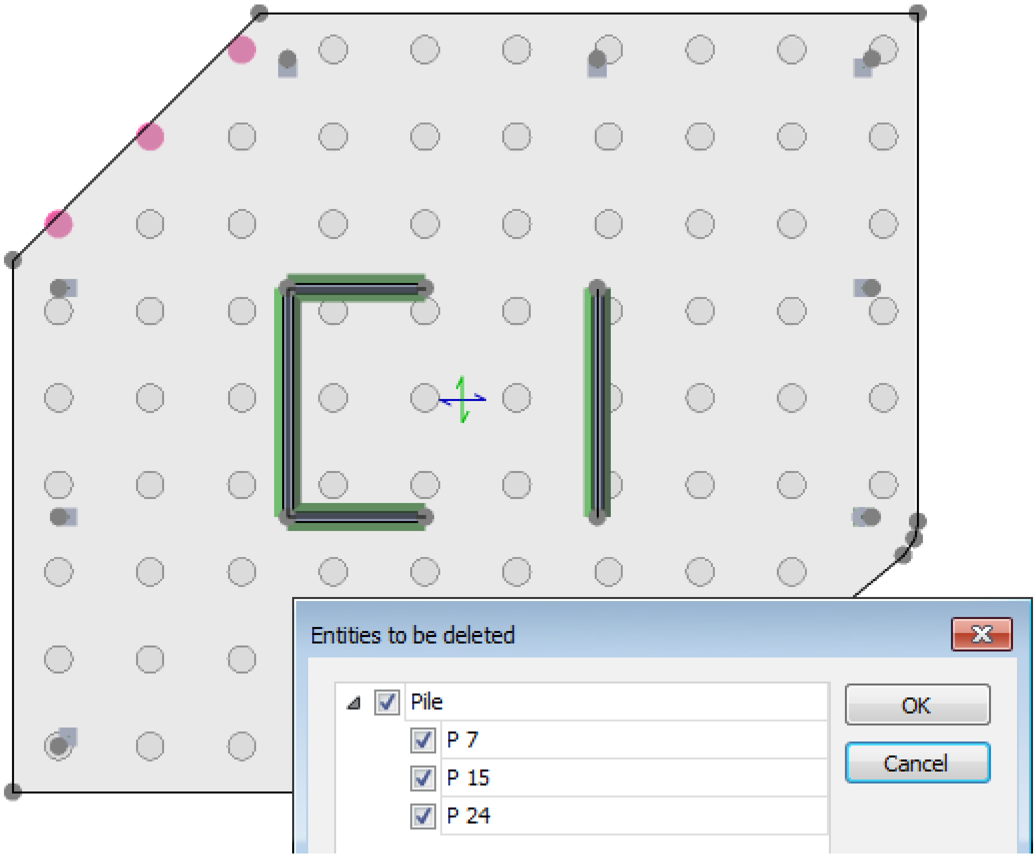
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

When not supported by a mat, columns and walls typically have supports at their bases.

When a piled mat is introduced these supports must be removed - as the mat now supports the whole building on pile springs. Consequently adding a piled mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

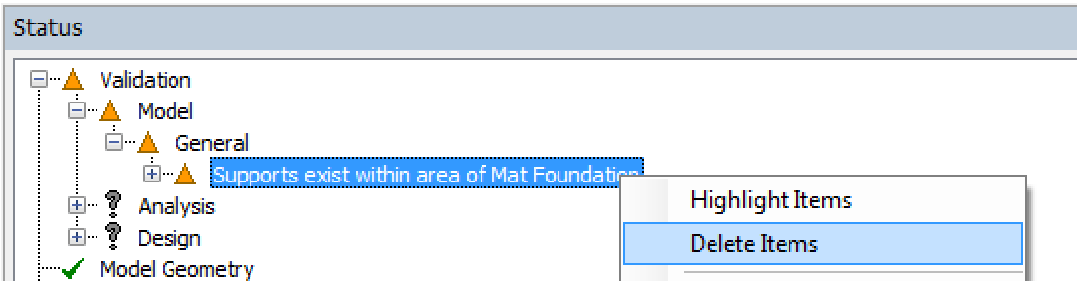
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

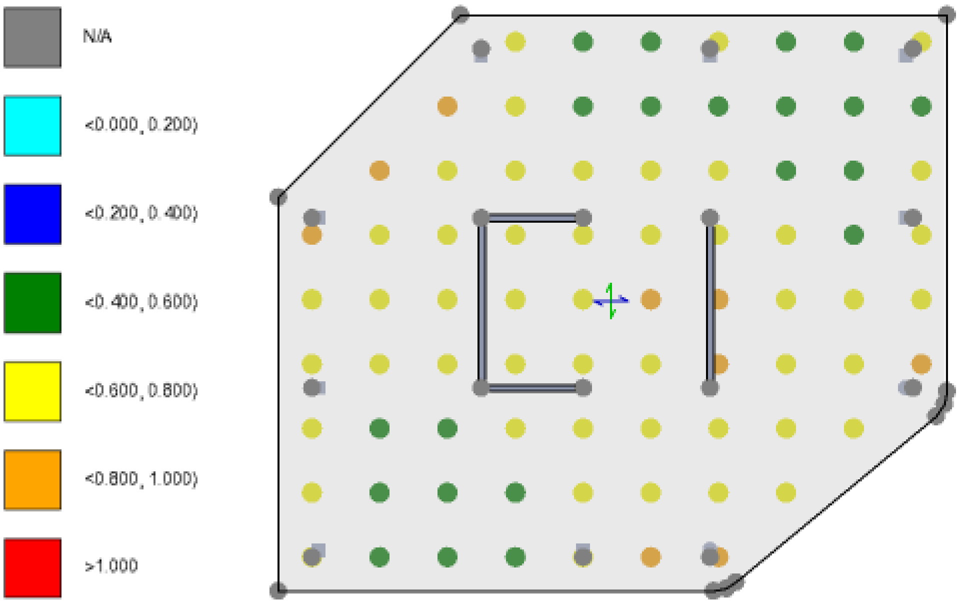
Perform the pile design

The piles are checked (and the mat is designed) by running Design Mats from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process

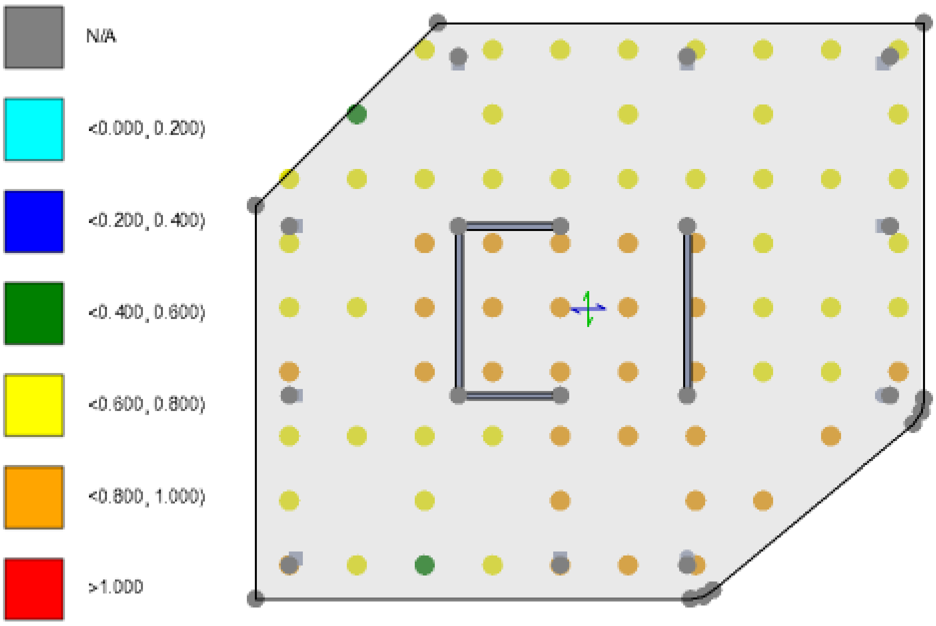
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyse All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

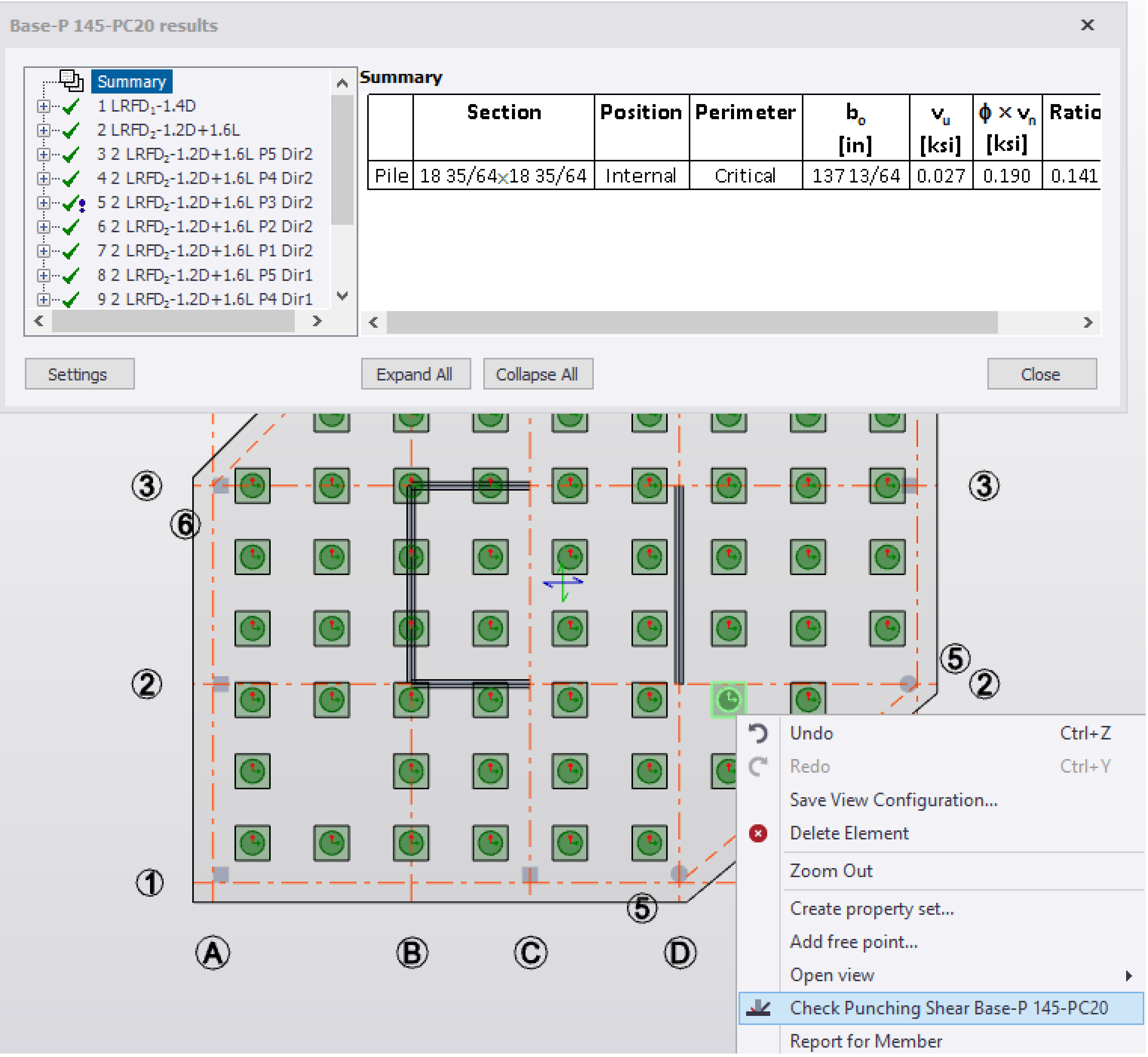
Add and run pile punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

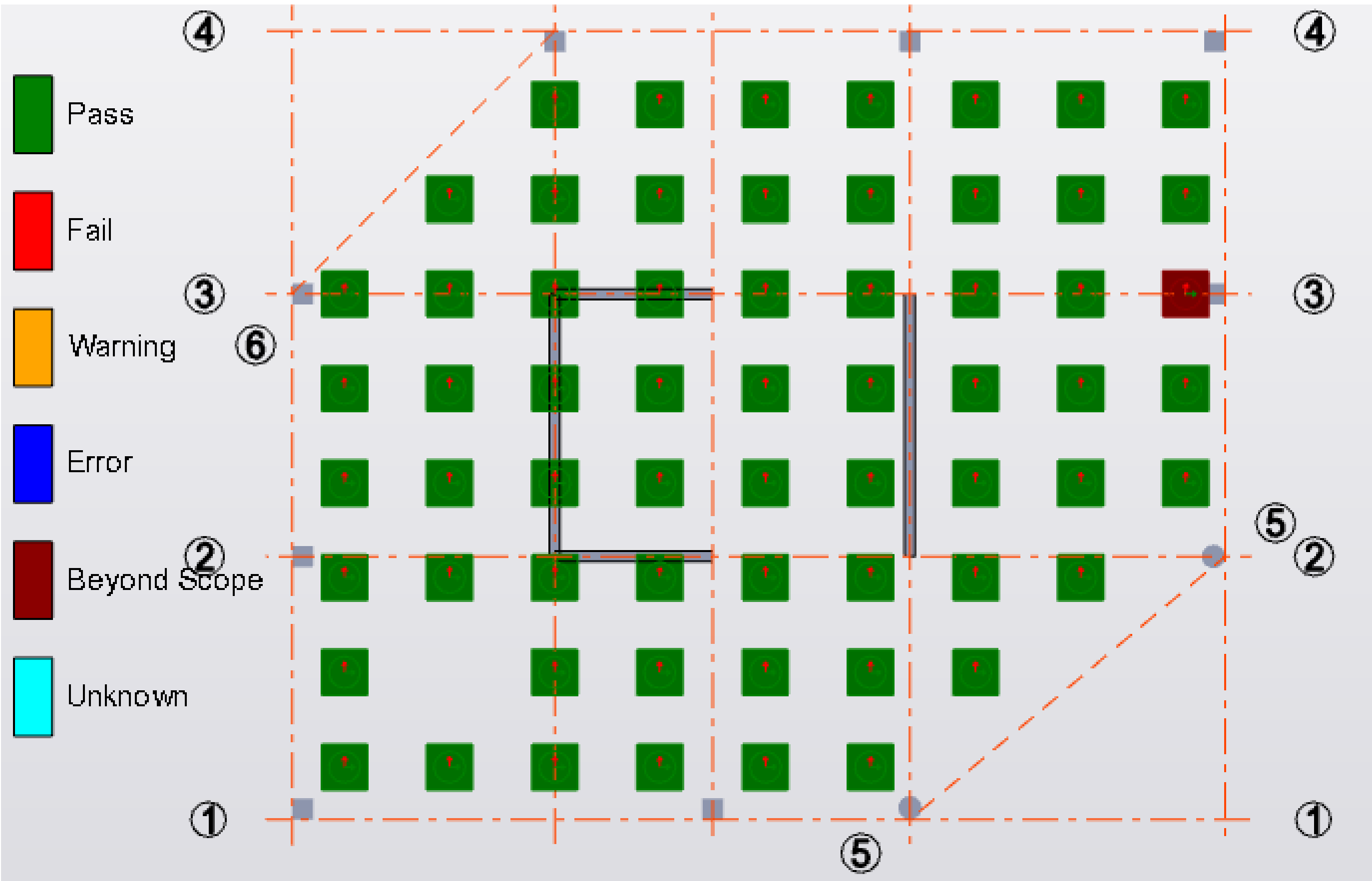
Punching checks can be added individually, or by windowing over an area. See: Create punching shear checks (page 801)

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See: Design and check punching shear (page 803)



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.

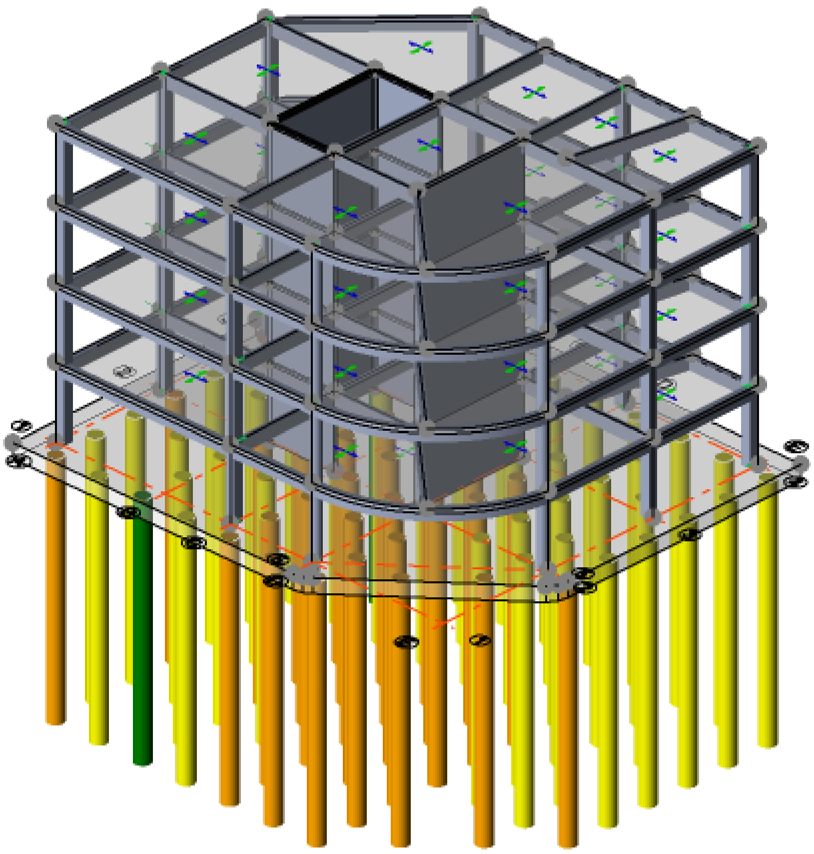
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).
- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (US customary units) (page 1608) topic.

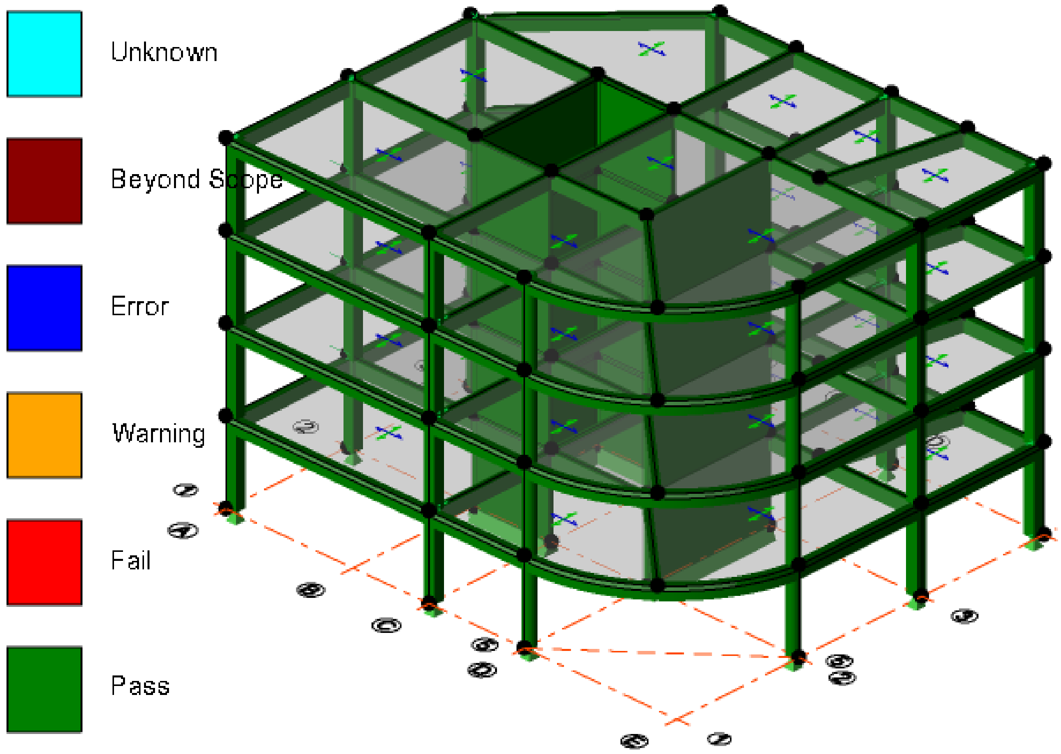
Piled mat foundation design workflow (metric units)

The following example illustrates the typical process to model and design piles in a piled mat foundation.



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

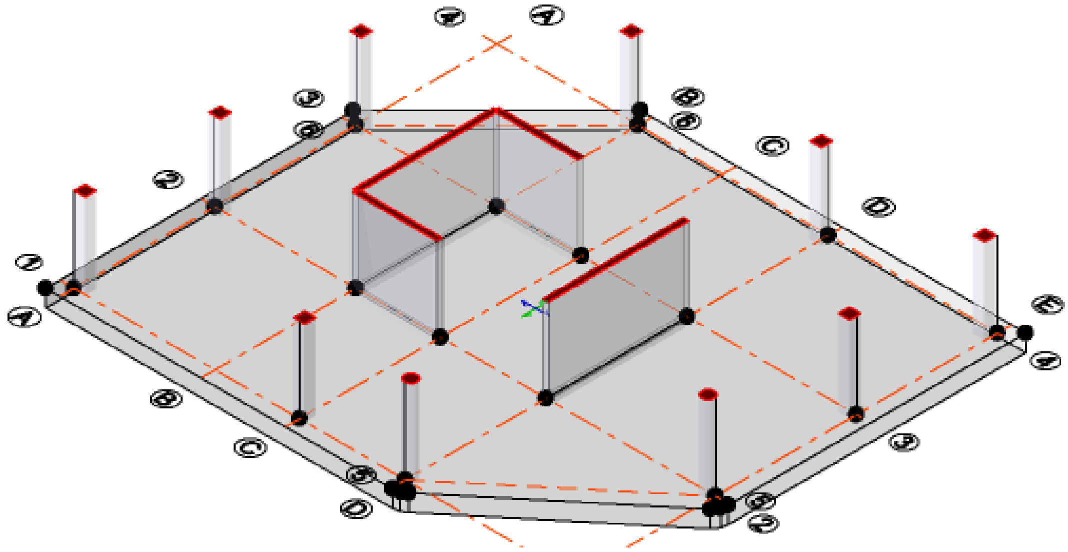
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 1.0m
- Mat thickness 600mm
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

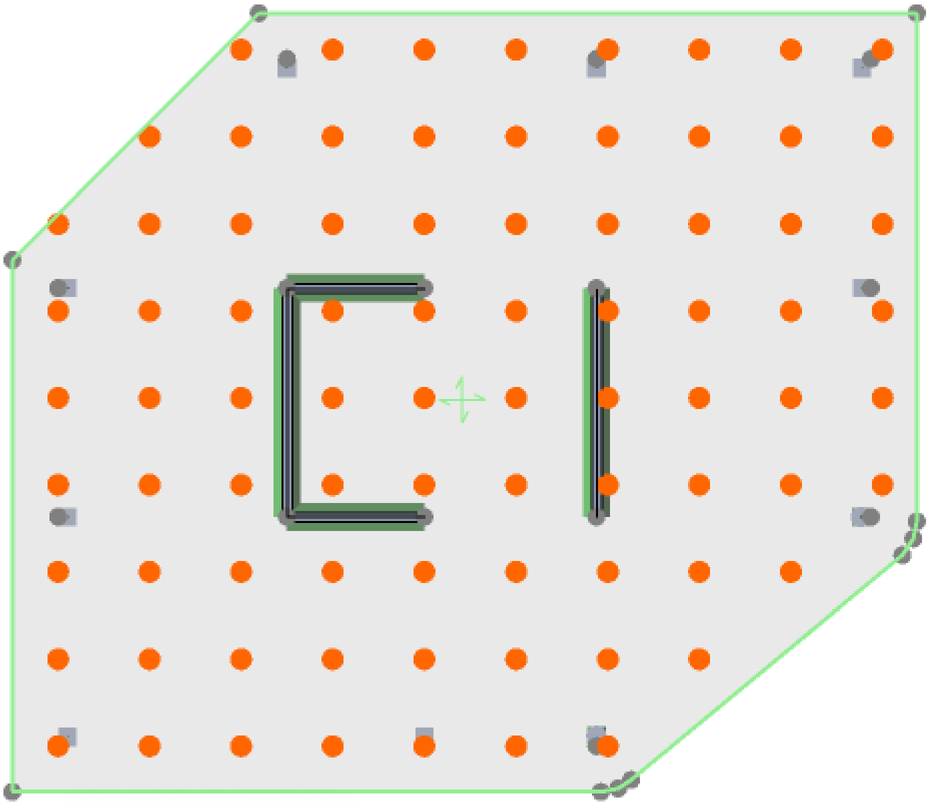
NOTE In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

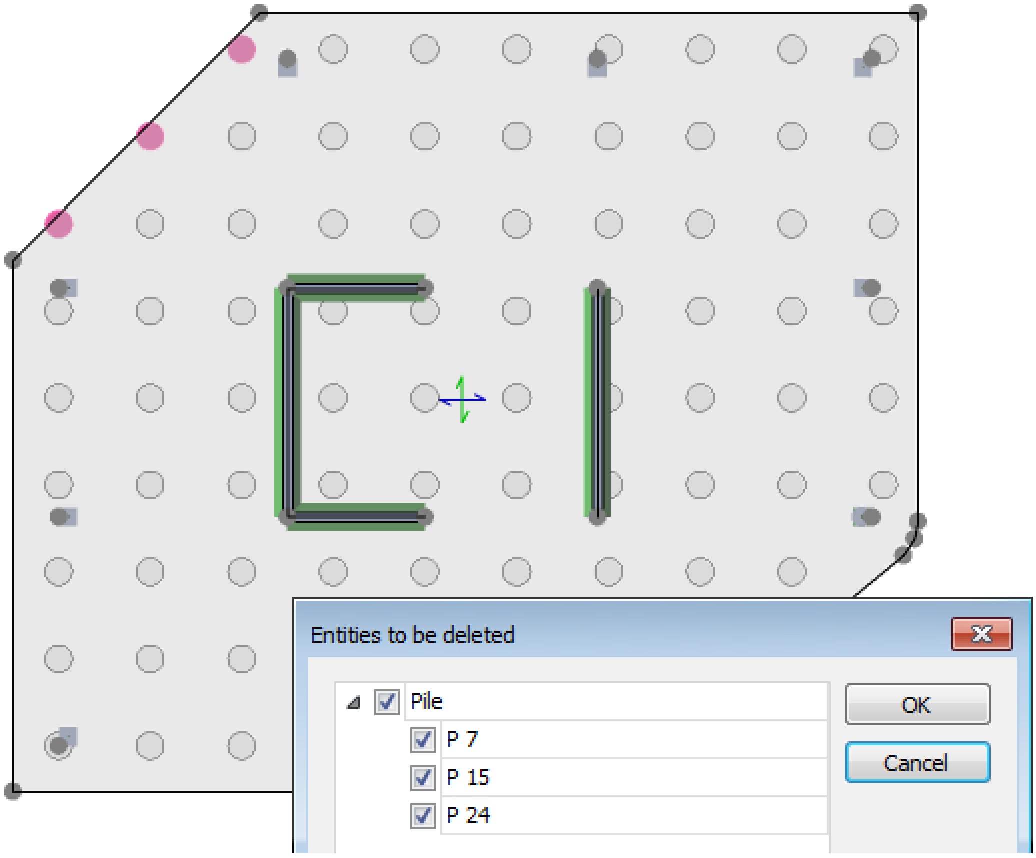
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

When not supported by a mat, columns and walls typically have supports at their bases.

When a piled mat is introduced these supports must be removed - as the mat now supports the whole building on pile springs. Consequently adding a piled mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

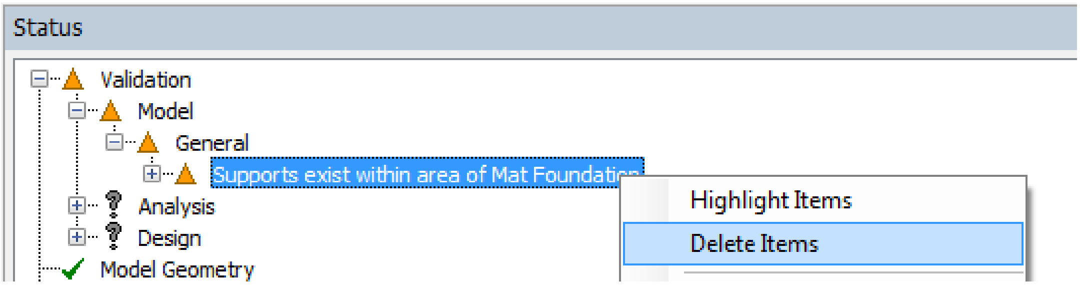
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

Perform the pile design

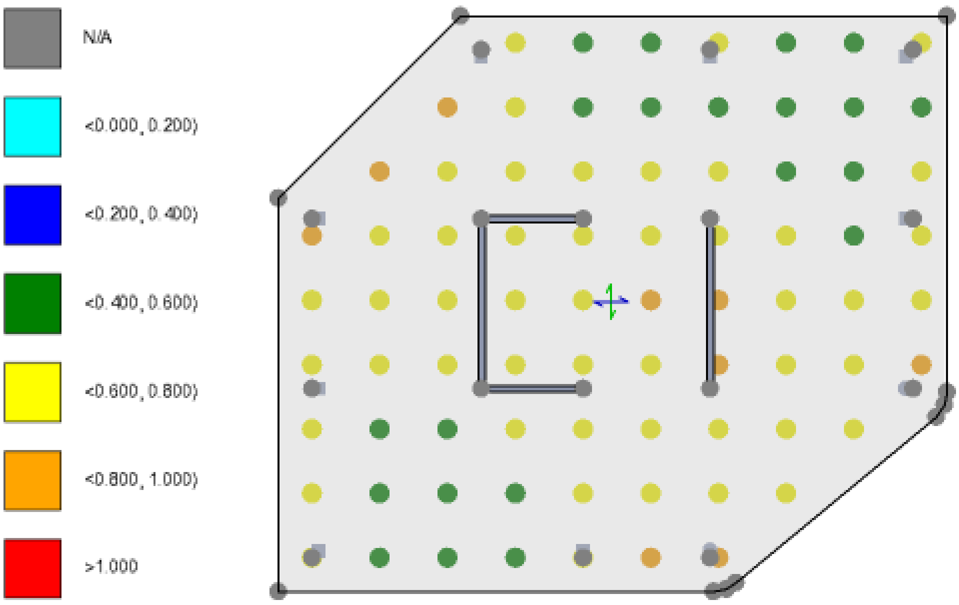
The piles are checked (and the mat is designed) by running Design Mats from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

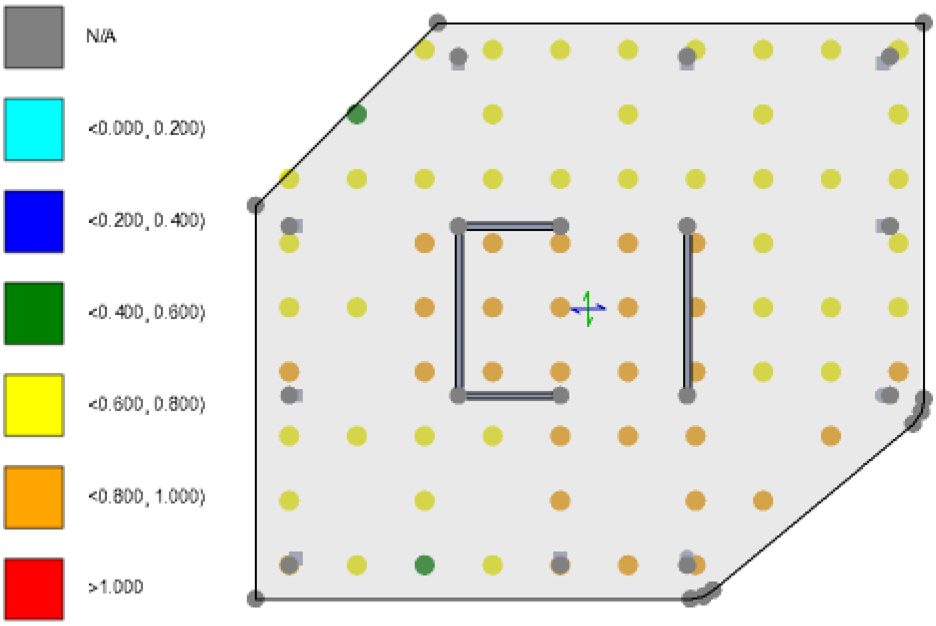
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyse All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

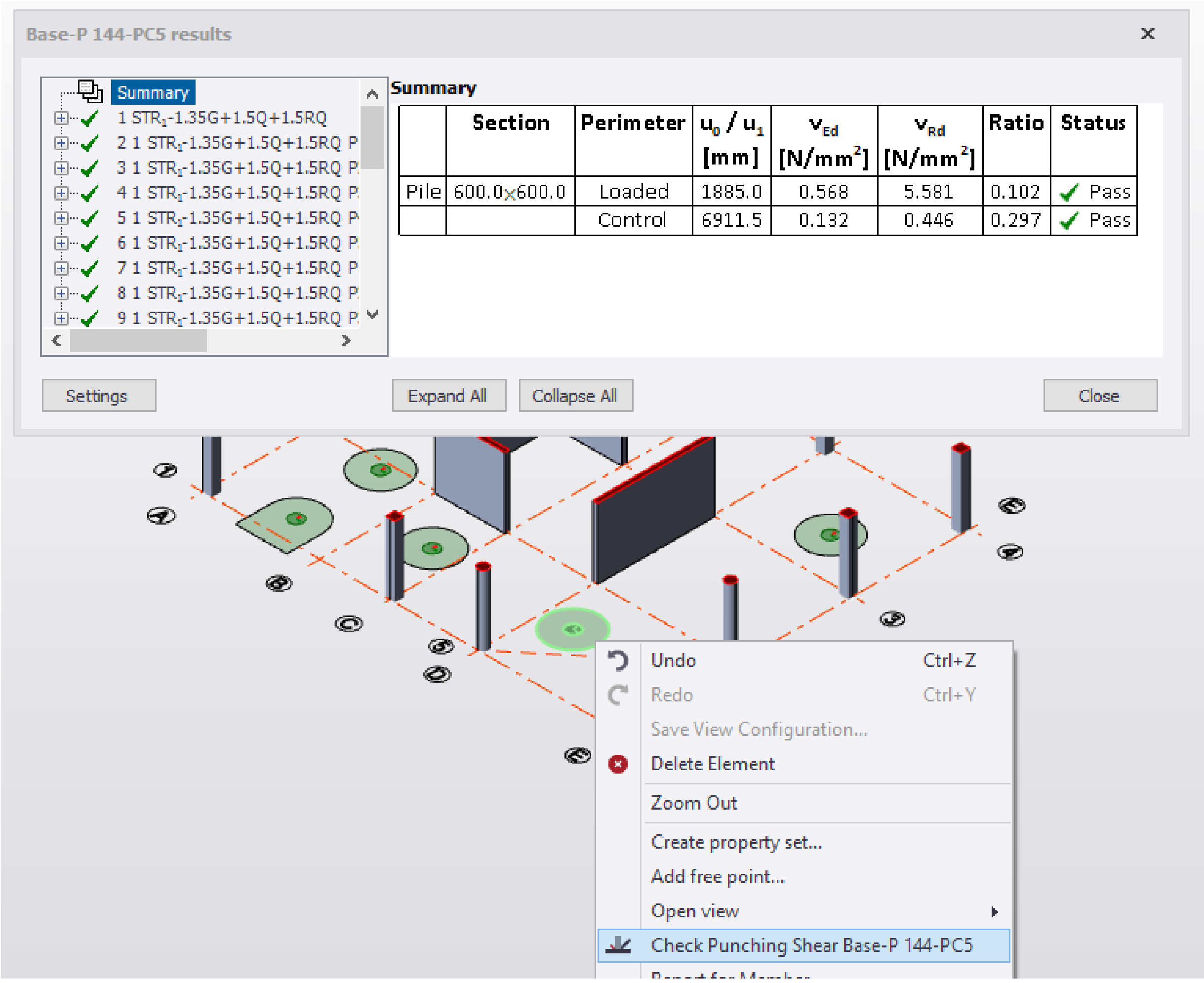
Add and run pile punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

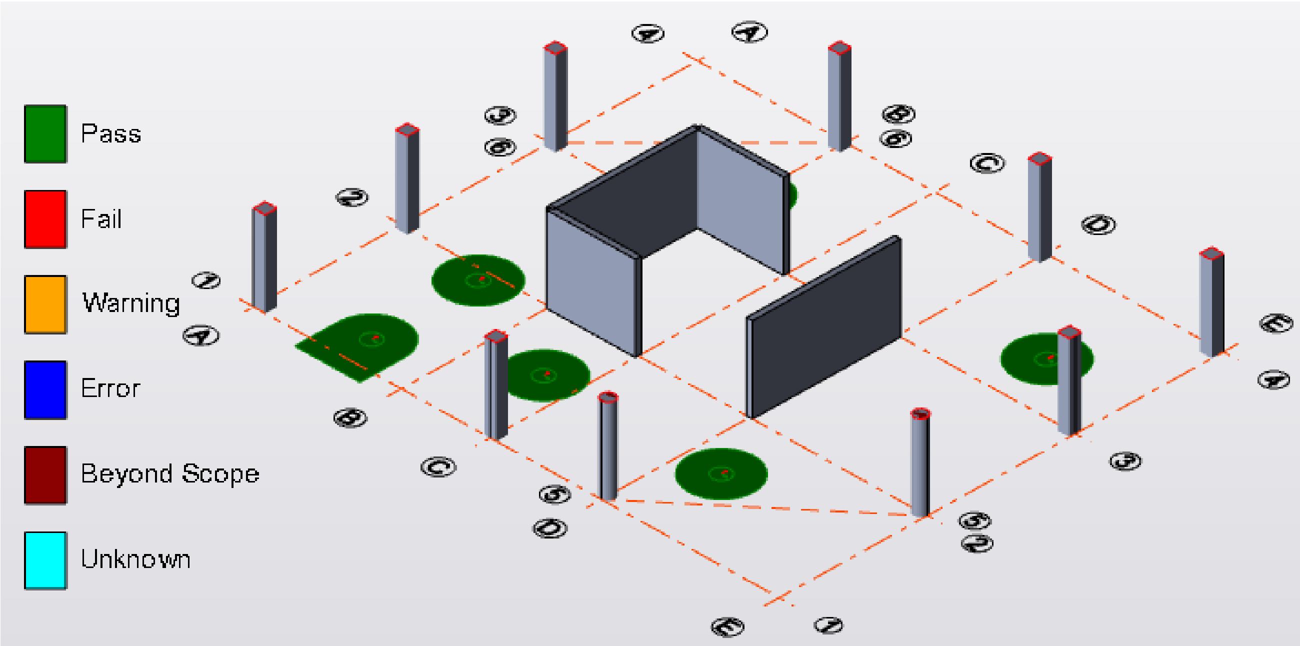
Punching checks can be added individually, or by windowing over an area. See: Create punching shear checks (page 801)

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See: Design and check punching shear (page 803)



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).

- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (metric units) (page 1594) topic.

13.11 Sustainability and Tekla Structural Designer

Click the link below to find out about the carbon impact of structures and how Tekla Structural Designer helps engineers to assess this for their projects:

- Measuring the carbon impact of a structure (page 1643)

See also

Export to One Click LCA (page 341)

Measuring the carbon impact of a structure

Global impact of construction industry

According to The Institution of Structural Engineers (IStructE):

- Buildings and construction account for about **40%** of energy related CO2 emissions
- Manufacture and disposal of materials used in structures accounts for about **11%** of all greenhouse gas emissions
- Analysis suggests that inefficiency in material use of up to **50%** is common

Typical emissions at each stage of the structure's life

In order to reduce carbon emissions it is first necessary to measure them. This can be done in different ways - in the UK for example by using BS EN 15978[1] the life cycle of the structure can be split into four stages as shown below, and the emissions assessed at each stage.

A1 - A3 **Product Stage**	**A4 - A5** **Construction Stage**	**B1 - B7** **Use Stage**	**C1 - C4** **End-of-Life Stage**
A1: Raw material supply A2: Transport A3: Manufacturing	A4: Transport A5: Construction	B1: Use B2: Maintenance B3: Repair B4: Replacement B5: Refurbishment B6: Operational energy use B7: Operational water use	C1: De-construction C2: Transport C3: Waste processing C4: Disposal

The distribution of emissions at each stage breaks down in approximately in these proportions:

Stage	Typical distribution of emissions
Product Stage	50%
Construction Stage	5%
Use Stage	43%
End-of-Life Stage	2%

It can be seen that the largest portion of emissions occurs at the Product Stage, and this is the stage that can be targeted by Tekla Structural Designer to make savings.

1. BS EN 15978:2011: Sustainability of construction works. Assessment of environmental performance of buildings. Calculation method. London: BSI, 2011

Measuring Product Stage Carbon

In the UK, the Product Stage is split into modules A1-A3, which are:

- Raw material extraction
- Transport of raw materials to manufacturing facilities
- Manufacture

The carbon impact of materials used in manufacture can then be measured by using an Embodied Carbon Factor (ECF).

This factor is specific to each material and is multiplied by the quantity of that material in order to give a quantity of CO_2 emitted in the Product Stage.

Many construction materials have standard ECF values that the engineer can use, and in addition manufacturers may also provide third-party verified Environmental Product Declarations (EPDs) which can be used when the engineer knows exactly which manufacturer the material will be sourced from.

Reporting and export of embodied carbon data

Tekla Structural Designer enables model material data to be exported to One Click LCA (page 341), which:

- is a widely used cloud-based software for life-cycle assessment and the calculation of embodied carbon.
- complies with a large number of standards including BREEAM and LEED.
- includes a huge database of generic and manufacturer-specific Environmental Product Declarations (EPDs)

The data can either be exported to a spreadsheet, or directly to the One Click LCA service, from where the environmental impact of the design can be reviewed and reports can be created.

NOTE The One Click LCA service can only be used if you have a One Click LCA account.

This feature enables the Engineer to quickly and easily determine the embodied carbon for the part of a project they are responsible for. This also helps them to develop and compare different scheme options. For a selected scheme they can then track progress as they make refinements to drive down the carbon impact.

See also

Export to One Click LCA (page 341)

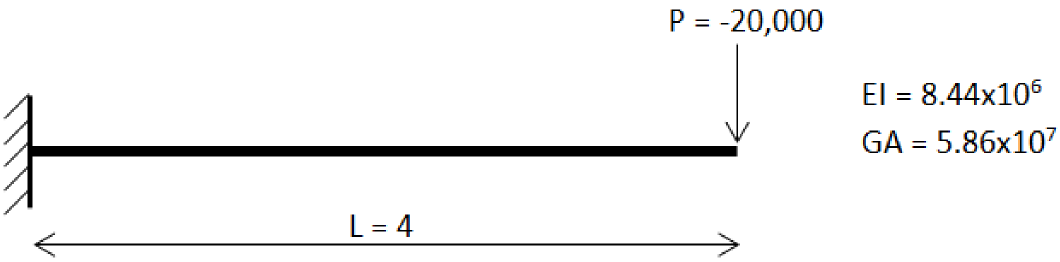
13.12 Analysis verification examples

A small number of verification examples are included in this section. Our full automatic test suite for the Solver contains many hundreds of examples which are run and verified every time the Solver is enhanced. These verification examples use SI units unless otherwise stated.

1st order linear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0%
Support Moment	PL	-80,000	-80,000	0%
Tip Deflection	$\frac{PL^3}{3EI}+\frac{PL}{GA}$	-0.0519	-0.0519	0%

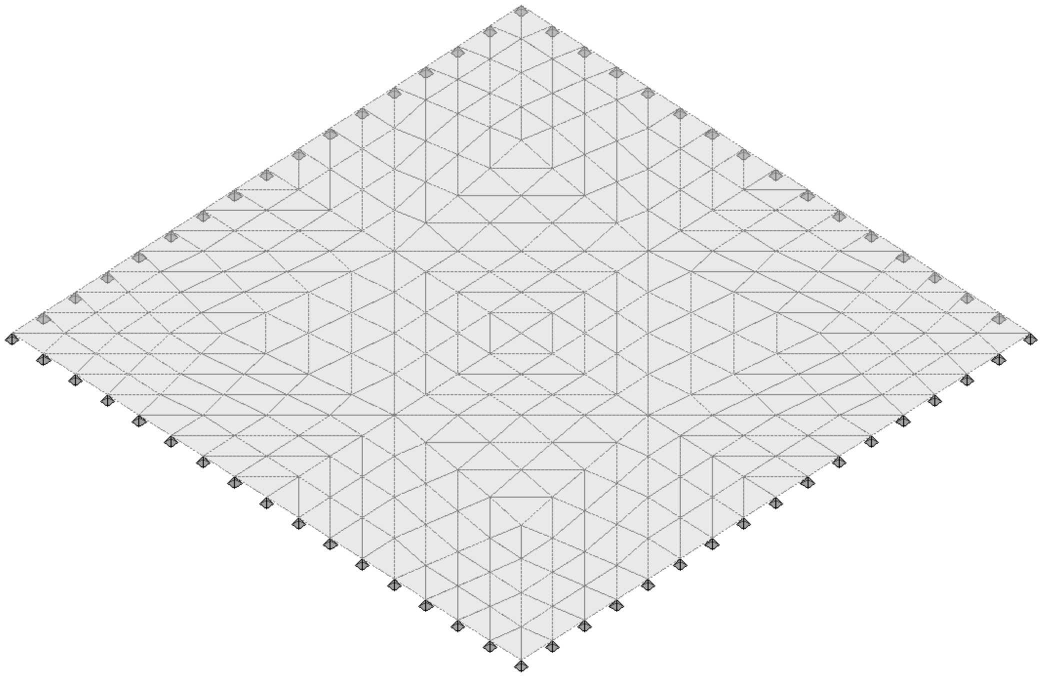
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order linear - Simply supported square slab

Problem definition

Calculate the mid span deflection of an 8x8 simply supported slab of 0.1 thickness under self-weight only. Take material properties E=2x10^{11}, G=7.7x10^{10} and ρ=7849.



Assumptions

A regular triangular finite element mesh is used with sufficient subdivision. Flexural and shear deformation is included, and the material is assumed to be isotropic.

Key results

The mid-span deformation is calculated using Navier's Method.

$$w = \frac{16q_0}{\pi^6 D} \sum_{m=1}^{9} \sum_{n=1}^{9} \frac{\sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

$$M_x = M_y = \frac{16q_0}{a^2\pi^4} \sum_{m=1}^{9} \sum_{n=1}^{9} \frac{m^2 \sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2} + \nu \frac{16q_0}{b^2\pi^4} \sum_{m=1}^{9} \sum_{n=1}^{9} \frac{n^2 \sin\frac{m\pi x}{a} \sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

Result	Theoretical value	Comparison 1	Solver value	% error
Mid-span deflection	7.002×10^{-3}	6.990×10^{-3}	7.031×10^{-3}	0.43%
Mid-span Moment	23616	23708	23649	0.14%

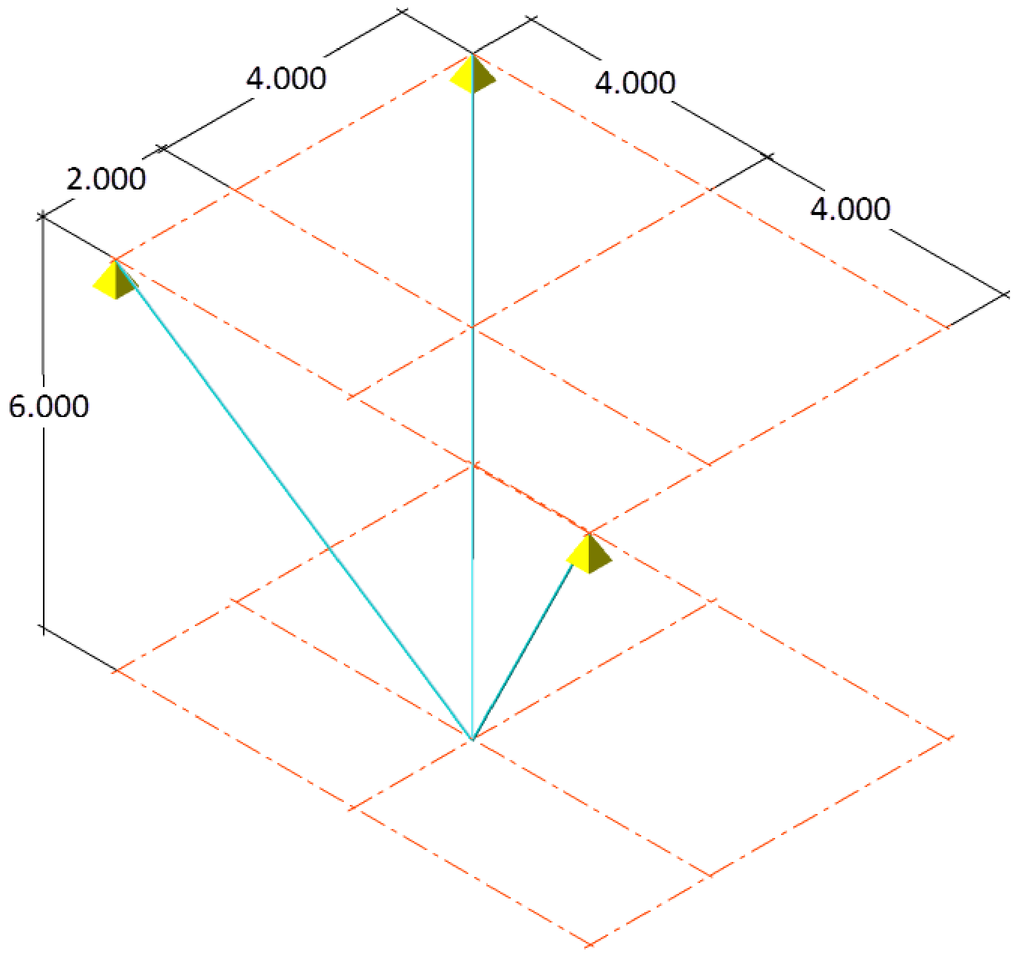
Conclusion

An acceptable match is observed between the theoretical values and the solver results. An acceptable match is also observed between the solver results and those obtained independently.

1st order linear - 3D truss

Problem definition

Three truss members with equal and uniform *EA* support an applied load of -50 applied at the coordinate (4, 2, 6). The start of each truss member is fixed and are located at (0, 0, 0), (8, 0, 0) and (0, 6, 0) respectively. Calculate the axial force in each element.



Key results

The results for this problem are compared against those published by Beer and Johnston, and against another independent analysis package.

Result	Beer and Johnston	Comparison 1	Solver value	% error
(0, 0, 0) - (4, 2, -6)	10.4	10.4	10.4	0.00 %
(8, 0, 0) - (4, 2, -6)	31.2	31.2	31.2	0.00 %
(0, 6, 0) - (4, 2, -6)	22.9	22.9	22.9	0.00 %

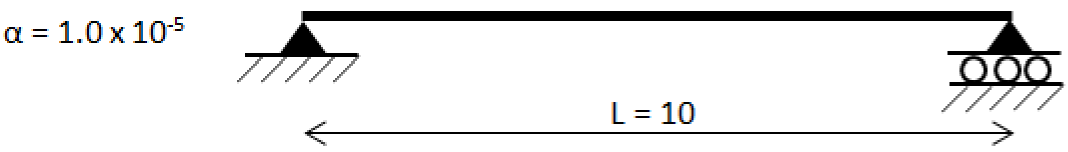
Conclusion

An exact match is observed between the values reported by the solver and those reported by Beer and Johnston.

1st order linear - Thermal load on simply supported beam

Problem definition

Determine the deflection, *U*, due to thermal expansion at the roller support due to a temperature increase of 5. The beam is made of a material with a thermal expansion coefficient of 1.0 x 10^{-5}.



Assumptions

The roller pin is assumed to be frictionless.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Translation at roller	$U = \Delta T \times \alpha \times L$	$5x10^{-4}$	$5x10^{-4}$	0%

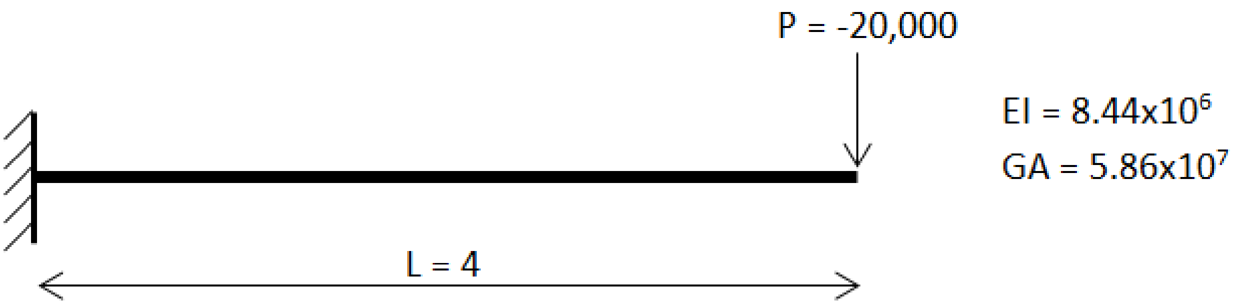
Conclusion

An exact match is shown between the theoretical result and the solver result.

1st order nonlinear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0 %
Support Moment	-PL	-80,000	-80,000	0 %
Tip Deflection		-0.0519	-0.0519	0 %

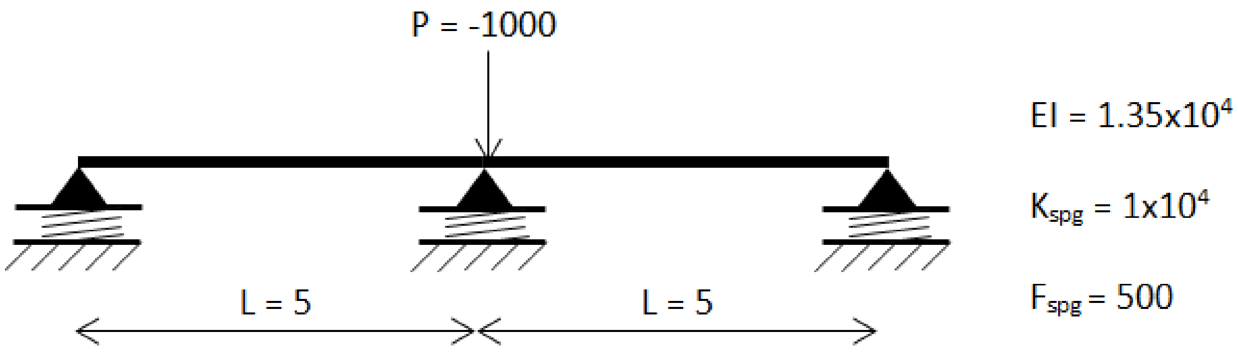
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order nonlinear - Nonlinear supports

Problem definition

A 10 long continuous beam is simply supported by three translational springs as shown. All springs have a maximum resistance force of 500. Calculate the reaction forces and deflection at each support.



Assumptions

Axial and shear deformations are ignored.

Key results

Result	Comparison 1	Solver value
LHS Reaction	250	250
Centre Reaction	500	500
RHS Reaction	250	250
LHS Displacement	-0.025	-0.025
Centre Displacement	-0.797	-0.797
RHS Displacement	-0.025	-0.025

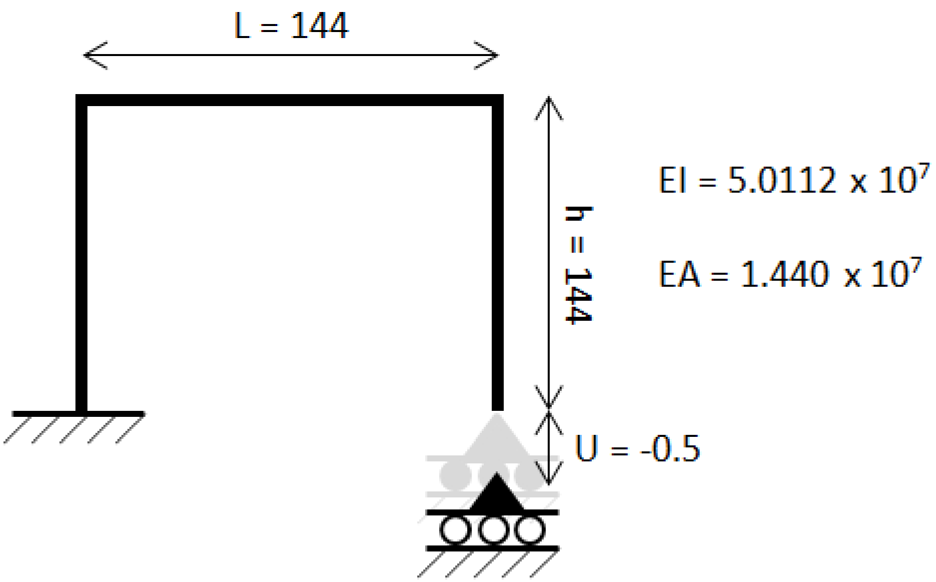
Conclusion

An exact match is shown between the solver and the independent analysis package.

1st order nonlinear - Displacement loading of a plane frame

Problem definition

Calculate the reaction forces of the plane moment frame shown below with the applied displacement *U*.



Assumptions

All elements are constant and equal *EI*. Axial and shear deformations are ignored; to achieve the former - analytically the cross sectional area was increased by a factor of 100,000 to make axial deformation negligible.

Key results

Results were compared with two other independent analysis packages.

Result	Comparison 1	Comparison 2	Solver value
LHS Vertical Reaction	6.293	6.293	6.293
LHS Moment Reaction	-906.250	-906.250	-906.250
RHS Vertical Reaction	-6.293	-6.293	-6.293

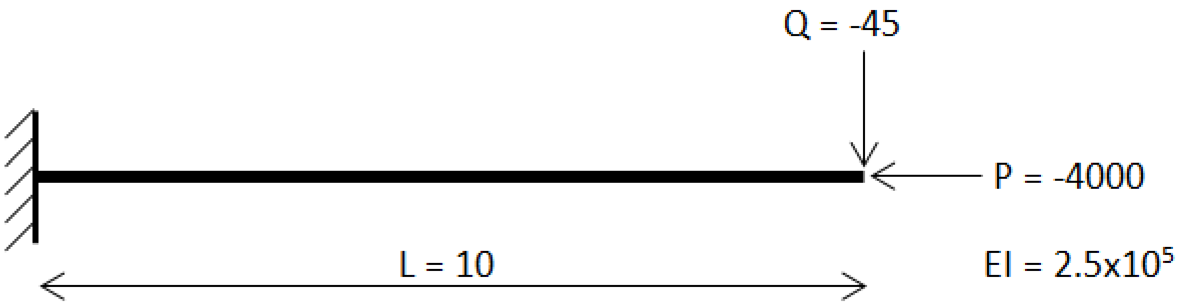
Conclusion

An exact match is shown between the solver and the two independent analysis packages.

2nd order linear - simple cantilever

Problem definition

A 10 long cantilever is subjected to a lateral tip load of 45 and an axial tip load of 4000.



Assumptions

Shear deformations are ignored. Results are independent of cross section area; therefore any reasonable value can be used. Second order effects from stress stiffening are included, but those caused by update of geometry are not. The beam is modelled with only one finite element, (if more elements had been used the result would converge on a more exact value).

Key results

Results were compared with an independent analysis package.

Result	Comparison	Solver value
Tip Deflection	-0.1677	-0.1677
Base Moment Reaction	-1121	-1121

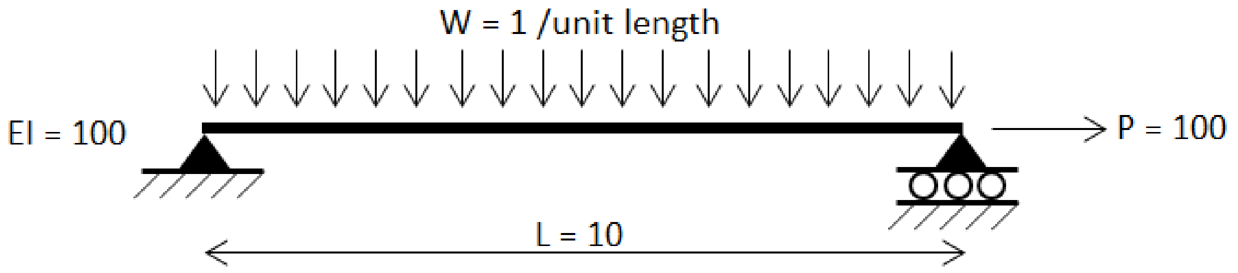
Conclusion

An exact match is observed between the values reported by the solver and the values reported in “Comparison”.

2nd order linear - Simply supported beam

Problem definition

Determine the mid-span deflection and moment of the simply supported beam under transverse and tensile axial load.



Assumptions

Shear deformations are excluded. Results are independent of cross section area; therefore any reasonable value can be used. The number of internal nodes varies from 0-9.

Key results

The theoretical value for deflection and moment are calculated as:

$$Y_{max} = -0.115 = \frac{5wL^4}{384EI} \times \frac{\frac{1}{\cosh U} - 1 + \frac{U^2}{2}}{\frac{5}{24}U^4}$$

$$M_{max} = -0.987 = \frac{wL^2}{8} \times \frac{2(\cosh U - 1)}{U^2 \cosh U}$$

Where U is a variable calculated:

$$U = 5 = \sqrt{\frac{PL^2}{4EI}}$$

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
1	-0.116	0.734 %	-0.901	8.631 %
3	-0.115	0.023 %	-0.984	0.266 %
5	-0.115	0.004 %	-0.986	0.042 %
7	-0.115	0.001 %	-0.986	0.013 %

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
9	-0.115	0 %	-0.986	0.005 %

Conclusion

As the element is subdivided the result converges to the correct theoretical value.

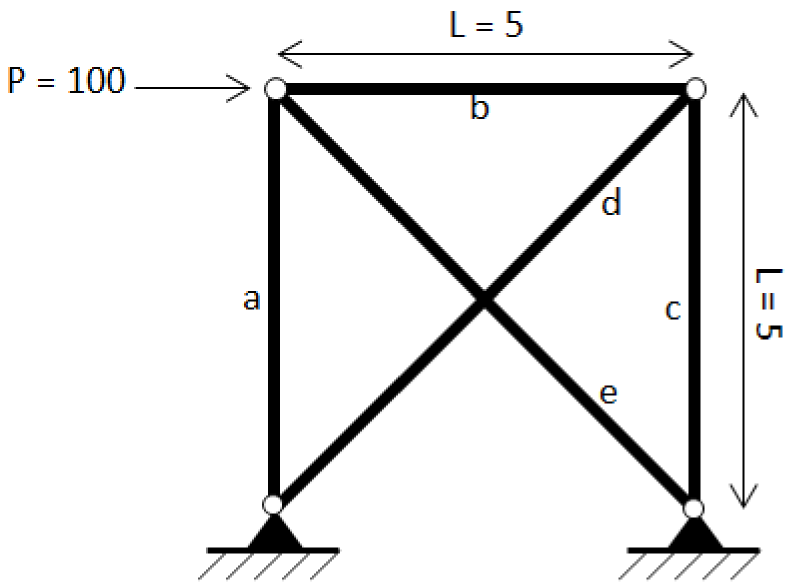
Reference

Timoshenko. S. 1956. *Strength of Materials, Part II, Advanced Theory and Problems*. 3rd Edition. D. Van Nostrand Co., Inc. New York, NY.

2nd order nonlinear - Tension only cross brace

Problem definition

Calculate the axial forces of the elements a-e shown in the 5x5 pin jointed plane frame shown below. Elements d and e can resist tensile forces only.



Assumptions

All elements are constant and equal EA. A smaller value of EA will increase the influence of second order effects, whereas a larger value will decrease the influence.

Key results

Under the applied loading element *e* becomes inactive. The theoretical formulas presented below are obtained using basic statics. Note that a positive value indicates tension. These results assume no 2^{nd} order effects; this requires the value of *EA* to be sufficiently large to make the 2^{nd} order effect negligible.

Result	Theoretical formula	Theoretical value	Solver value	% Error
a	0	0	0	0
b	-P	-100	-100	0
c	-P	-100	-100	0
d	$P\sqrt{2}$	141.42	141.42	0
e	0	0	0	0

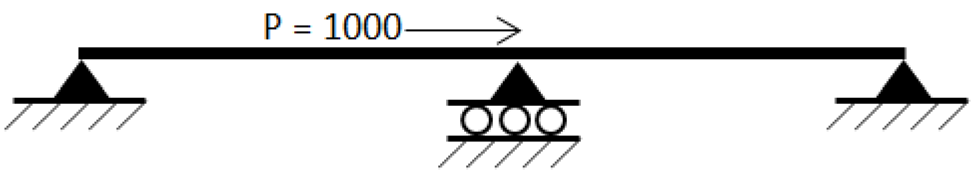
Conclusion

An exact match is observed between the values reported by the solver and the values predicted using statics. A 1st order nonlinear analysis can be used, with any section sizes, to confirm this result without second order effects.

2nd order nonlinear - Compression only element

Problem definition

Calculate the reaction forces for the compression only structure shown below.



Assumptions

All elements are constant and equal *EA*, and can resist only compressive forces

Key results

Under the applied loading the element on the left becomes inactive, therefore all applied loading is resisted by the support on the right.

Result	Theoretical formula	Theoretical value	Solver value
LHS Reaction	0	0	0
RHS Reaction	-P	-1000	-1000

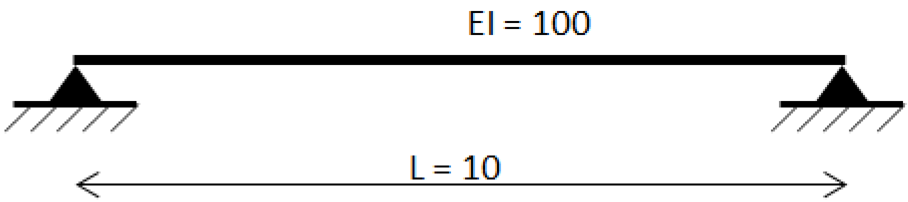
Conclusion

An exact match is observed between the values reported by the solver and the theoretical values.

1st order modal - Simply supported beam

Problem definition

Determine the fundamental frequency of a 10 long simply supported beam with uniform *EI* and mass per unit length equal to 1.0.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5. Consistent mass is assumed.

Key results

The theoretical value for the fundamental frequency is calculated as:

$$\omega = 0.9870 = \sqrt{\left(\frac{\pi}{10}\right)^4 \frac{100}{1}} = \sqrt{\left(\frac{\pi}{L}\right)^4 \frac{EI}{m/L}}$$

Where *m* is the total mass of the beam.

No. internal nodes	Solver value	% error
0	1.0955	10.995 %
1	0.9909	0.395 %
2	0.9878	0.081 %
3	0.9872	0.026 %
4	0.9871	0.011 %
5	0.9870	0.005 %

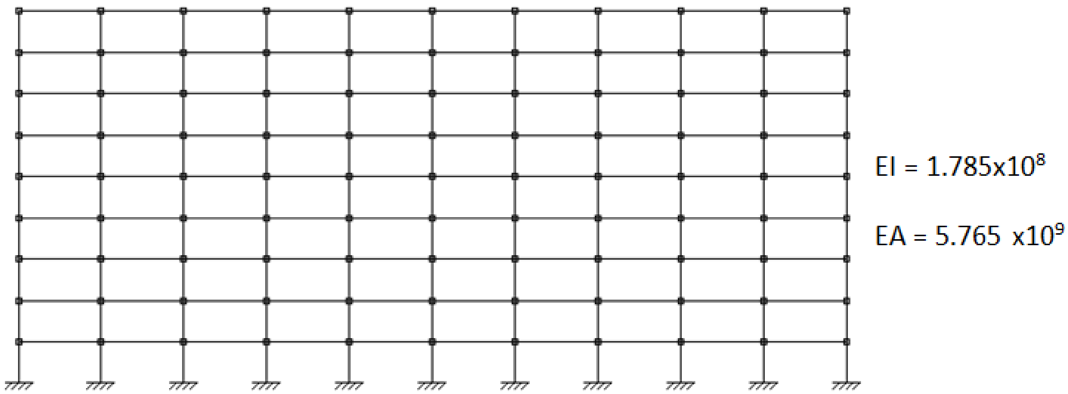
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

1st order modal - Bathe and Wilson eignenvalue problem

Problem definition

A 2D plane frame structure has 10 equal bays each measuring 6.096m wide and 9 stories 3.048m tall. The column bases are fully fixed. All beams and columns are the same section, which have a constant mass/unit length equal to 1.438. Calculate the first three natural frequencies (in Hz) of the structure under self-weight.



Assumptions

Shear deformations are excluded. Each beam/column is represented by one finite element. Consistent mass is assumed.

Key results

The results for this problem are compared with those published by Bathe and Wilson and against an independent analysis package.

Mode	Bathe and Wilson	Comparison	Solver value
1	0.122	0.122	0.122
2	0.374	0.374	0.375
3	0.648	0.648	0.652

Conclusion

The results show a good comparison with the original published results and against the other analysis packages.

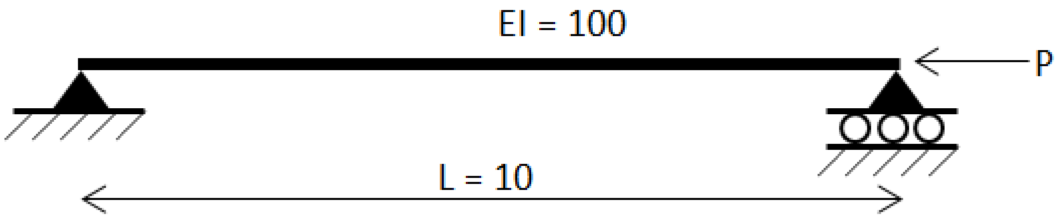
References

Bathe, K.J. and E.L. Wilson. 1972. *Large Eigen Values in Dynamic Analysis.* Journal of the Engineering Mechanics Division. ASCE Vol. 98, No. EM6. Proc. Paper 9433. December.

2nd order buckling - Euler strut buckling

Problem definition

A 10 long simply supported beam is subjected to an axial tip load of P.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5.

Key results

The theoretical value for the first buckling mode is calculated using the Euler strut buckling formula:

$$\lambda = 9.869 = \frac{\pi^2 EI}{L^2}$$

With P = -1.0 the following buckling factors are obtained

No. internal nodes	Solver value	% error
0	12.000	21.59 %
1	9.944	0.75 %
2	9.885	0.16 %
3	9.875	0.05 %
4	9.872	0.02 %
5	9.871	0.01 %

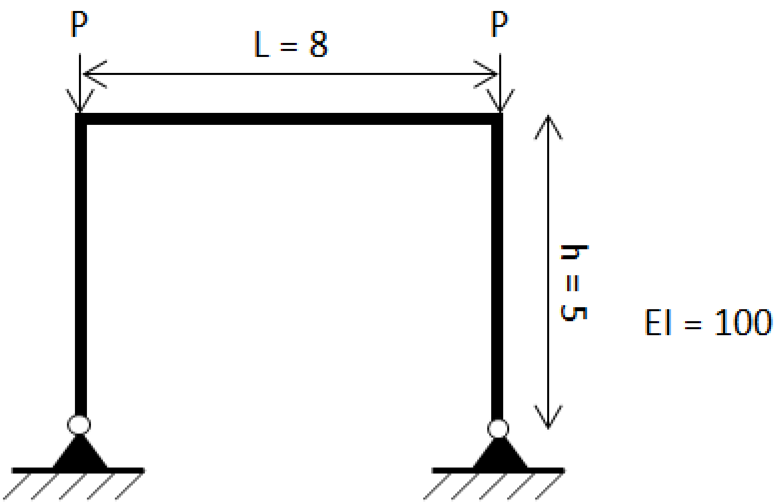
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

2nd order buckling - Plane frame

Problem definition

Calculate the buckling factor of the moment frame shown below.



Assumptions

All elements are constant and equal *EI*. Axial deformations are ignored; to achieve this the cross section area is set to 1000. The number of elements per member is varied between 0 and 5.

Key results

The theoretical buckling load is calculated by

$$P_{cr} = 6.242 = \frac{(kL)^2 EI}{h^2}$$

where

$$kL\tan(kL) = 1.249 = \frac{6h}{L}$$

Which can be solved using Newtons method and five iterations

$$kL_{n+1} = kL_n - \frac{f(kL_n)}{f'(kL_n)} = kL_n - \frac{kL_n\tan(kL_n) - \frac{6h}{L}}{kL_n\sec^2(kL_n) + \tan(kL_n)}$$

No. internal nodes/ members	Solver value	% error
0	6.243	0.17 %
1	6.243	0.01 %
2	6.242	0.00 %
3	6.242	0.00 %

No. internal nodes/ members	Solver value	% error
4	6.242	0.00 %
5	6.242	0.00 %

Conclusion

A good match is shown between the solver and theory. The discrepancy decreases as the level of discretization is increased.

References

Timoshenko, S. and J. M. Gere. 1961. *Theory of Elastic Stability*. 2nd Edition. McGraw-Hill Book Company.

14 Design codes reference

This section contains reference information relating to the action and resistance codes supported by Tekla Structural Designer.

14.1 US codes

Loading (ASCE7)

These topics provide a general overview of how loadcases and combinations are created in Tekla Structural Designer when the head code is set to United States(ACI/AISC). The ASCE7 Combination Generator is also described.

The following topics are covered:

Load cases (ASCE7)

Loadcase types (ASCE7)

The following load case types can be created:

Type	Calculated automatically	Include in the combination generator	Live load reductions	Pattern load
self weight (beams, columns and walls)	yes/no	yes/no	N/A	N/A
slab wet	yes/no	N/A	N/A	N/A
slab dry	yes/no	yes/no	N/A	N/A
dead	N/A	yes/no	N/A	N/A
live	N/A	yes/no	yes/no	yes/no
roof live	N/A	yes/no	yes/no	N/A
wind	N/A	yes/no	N/A	N/A
snow	N/A	yes/no	N/A	N/A
snow drift	N/A	yes/no	N/A	N/A
temperature	N/A	N/A	N/A	N/A
settlement	N/A	N/A	N/A	N/A
seismic	N/A	yes	N/A	N/A

As shown above, self weight loads can all be determined automatically. However, other gravity load cases need to be applied manually as you build the structure.

Self weight (ASCE7)

Self weight - excluding slabs loadcase

Tekla Structural Designer automatically calculates the self weight of the structural beams/columns for you. The **Self weight - excluding slabs** loadcase is pre-defined for this purpose. It's loadcase type is fixed as “Selfweight”. It cannot be edited and by default it is added to each new load combination.

Self weight of concrete slabs

Tekla Structural Designer expects the wet and dry weight of concrete slab to be defined in separate loadcases. This is required to ensure that members are designed for the correct loads at construction stage and post construction stage.

The **Slab self weight** loadcase is pre-defined for the dry weight of concrete post construction stage, its loadcase type is fixed as “Slab Dry”.

There is no pre-defined loadcase for the wet weight of concrete slab at construction stage, but if you require it for the design of any composite beams in the model the loadcase type should be set to “Slab Wet”.

Tekla Structural Designercan automatically calculate the above weights for you taking into account the slab thickness, the shape of the deck profile and wet/dry concrete densities. It does not explicitly take account of the weight of any reinforcement but will include the weight of decking. Simply click the **Calc Automatically** check box when you create each loadcase. When calculated in this way you can’t add extra loads of your own into the loadcase.

If you normally make an allowance for ponding in your slab weight calculations, Tekla Structural Designer can also do this for you. After selecting the composite slabs, you are able to review the slab item properties - you will find two ways to add an allowance for ponding (under the slab parameters heading). These are:

- as a value, by specifying the average increased thickness of slab
- or, as a percentage of total volume.

Using either of these methods the additional load is added as a uniform load over the whole area of slab.

Live and roof live loads (ASCE7)

Live load reductions

Reductions can be applied to roof live and live loads to take account of the unlikelihood of the whole building being loaded with its full design live load. The reduction is calculated based on total floor area supported by the design member. Roof live and live load types each have their own reductions applied in accordance with either Section 4.8 and 4.9 of ASCE 7-05, or Section 4.7 and 4.8 of ASCE 7-10 as appropriate.

Due to the complications associated with live load reduction when considering beams at an angle to the vertical or horizontal, reductions are only applied to:

- Horizontal steel beams with vertical webs (major axis horizontal) which are set to be “gravity only” pin ended only
- Columns of any material
- Concrete walls, mid-pier or meshed

Live load reduction factor

The live load reduction factor, R is calculated as follows:

R	=	(0.25 + 15 / Sqrt(K_{LL}*A_T)) - where R <= 1.0	US-units
R	=	(0.25 + 4.57 / Sqrt(K_{LL}*A_T))	metric-units

K_{LL} comes from Table 4-2 in ASCE7-05/ASCE7-10. Essentially:

Interior and exterior cols (no cantilever slabs)	K_{LL}= 4
Edge and interior beams (no cantilever slabs)	K_{LL}= 2
Interior beams (with cantilever slabs)	K_{LL}= 2
Cantilever beams	K_{LL}= 1
Edge cols (with cantilever slabs)	K_{LL}= 3
Corner cols (with cantilever slabs)	K_{LL}= 2
Edge beams (with cantilever slabs)	K_{LL}= 1
For all beams and column stacks supporting one floor	R ≥ 0.5
For all column stacks supporting two or more floors	R ≥ 0.4

NOTE As it is not possible to automatically assess where cantilever slabs are and what they are attached to - the K_{LL} factor can be manually specified for individual columns, wall stacks and beam spans.

Roof live load reduction factor

The roof live load reduction factor is calculated as follows:

R	=	R_1 * R_2	
where			
R_1	=	1.2 - 0.001 * A_T , where 1.0 ≥ R_1 ≥ 0.6	US-units
	=	1.2 - 0.011 * A_T	metric-units
R_2	=	1.0 (conservatively assumes roofs < 18 degs)	

Wind loads (ASCE7)

The ASCE7 Wind wizard...

NOTE The **Wind Wizard** is fully described in the Wind Modeling Engineer's Handbook.

The **Wind Wizard** assesses wind loading on your building structure via a choice of methods:

- Directional Procedure Part 1 - Rigid Buildings of All Heights (Chapter 27)
- Envelope Procedure Part 1 - Low-Rise Buildings (Chapter 28)

Wind load cases can then be generated and combined with other actions due to dead and imposed loads in accordance with Section 2.3.2 of ASCE7-10

In order to run the **Wind Wizard** the following assumptions/limitations exist:

- The shape of the building meets the limitations allowed for in the code.
- It must be a rigid structure.
- The structure must be either enclosed or partially enclosed.
- Parapets and roof overhangs are not explicitly dealt with.

Simple wind loading

If use of the **Wind Wizard** is not appropriate for your structure then wind loads can be applied via element or structure loads instead.

Patterning of live loads (ASCE7)

ASCE7 pattern loading for LRFD combinations is as follows:

Code class	Load combination	Loaded spans	Unloaded spans
LRFD	1.2D + 1.6L + 0.5Lr	1.2D + 1.6L + 0.5Lr	1.2D + 0.5Lr

Combinations (ASCE7)

Once your load cases have been generated as required, you then combine them into load combinations; these can either be created manually, by clicking **Add...** - or with the assistance of the Combinations Generator, by clicking **Generate...**

Application of notional loads in combinations (ASCE7)

Notional loads are applied to the structure in the building directions 1 and 2 as follows:

- NL Dir1+
- NL Dir1-
- NL Dir2+
- NL Dir2-

When you run the the Combinations Generator you are required to select the NL directions to add, and the factors to be applied as part of the process. Alternatively, you can set up the combinations manually and apply notional loads and factors to each as required.

Combination generator (ASCE7)

Accessed via the **Generate...** command, this automatically sets up combinations for both strength and serviceability.

NOTE Temperature and settlement load case types are not included in the **Generate...** command - these need to be added manually.

Combination generator - Combinations

The first page of the generator lists suggested ASD and LRFD combinations (with appropriate factors).

The “Generate” check boxes are used to select those combinations to be considered.

Combination generator - Service

This page indicates which combinations are to be checked for serviceability and the service factors applied.

Combination generator - NL

The last page is used to set up the notional loads. You can specify NL’s and factors in each of four directions. For each direction selected a separate NL combination will be generated.

Any combination with wind in is automatically greyed.

Click **Finish** to see the list of generated combinations.

Combination classes (ASCE7)

Having created your combinations you classify them as: Construction Stage, Gravity, Lateral, Seismic or Modal Mass.

NOTE If generated via the Combinations generator they are classified for you automatically.

Then (where applicable) you indicate whether they are to be checked for strength or service conditions, or both. You also have the option to make any of the combinations inactive.

Construction stage combination (ASCE7)

A Construction Stage load combination is only required for the purpose of designing any composite beams within the model. It is distinguished from other combinations by setting its “Class” to Construction Stage. Typically this combination would include a loadcase of type “Slab Wet”, (not “Slab Dry”), other loadcases being included in the combination as required.

NOTE The Slab Wet loadcase type should not be included in any other combination.

Gravity combination (ASCE7)

These combinations are considered in both the Gravity Sizing and Full Design processes.

They are used in the Gravity Sizing processes as follows:

- Design Concrete (Gravity) - concrete members in the structure are automatically sized (or checked) for the gravity combinations
- Design Steel (Gravity) - steel members in the structure are automatically sized (or checked) for the gravity combinations.
- Design All (Gravity) - all members in the structure are automatically sized (or checked) for the gravity combinations.

They are also used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the gravity combinations.
- Design Steel (All) - steel members in the structure are automatically sized (or checked) for the gravity combinations.
- Design All (All) - all members in the structure are automatically sized (or checked) for the gravity combinations.

Lateral combinations (ASCE7)

These combinations are **not** used in the Gravity Sizing processes.

They are used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the lateral combinations.

- Design Steel (All) - steel members in the structure which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.
- Design All (All) - all concrete members and all steel members which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.

Seismic combinations (ACSE7)

These combinations are only considered during the Full Design process. They are not used in the Gravity Sizing process.

Modal mass combinations (ASCE7)

For modal analysis, you are required to set up specific “modal mass” combinations. Provided these combinations are active they are always run through the modal analysis.

NOTE It is always assumed that all loads in the load cases in the combination are converted to mass for modal analysis. You are permitted to add lumped mass directly to the model.

Steel design to AISC 360 ASD and LRFD

Tekla Structural Designer designs steel and composite members to a range of international codes. This reference guide specifically describes the design methods applied when the AISC 360 ASD or AISC 360 LRFD resistance codes are selected.

General

Seismic design (AISC 360)

All “Gravity Only Design” members are designed as per the normal AISC Specification rules for the seismic load combinations.

Additional design rules are required for seismic combinations. These are as per the AISC Seismic Provisions (AISC 341-05) (Ref. 9), (AISC 341-10) (Ref. 10) or (AISC 341-16) (Ref. 11). These additional design rules ONLY apply to members in Seismic Load Resisting Systems. These rules are applied as follows:

- If SDC = A - no additional requirements
- If SDC = D, E or F, apply rules for AISC 341

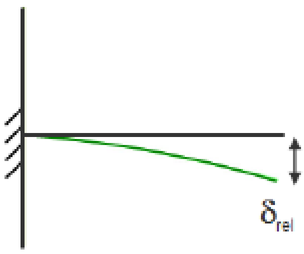
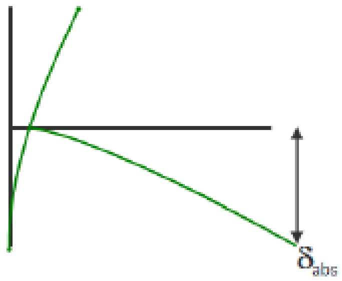
For each of X and Y directions:

- If SDC = B or C and R <=3 - no additional requirements
- If SDC = B or C and R > 3, apply rules for AISC 341

Deflection checks (AISC 360)

Relative and Absolute Deflections

Tekla Structural Designer calculates both ***relative*** and ***absolute*** deflections. Relative deflections measure the internal displacement occurring within the length of the member and take no account of the support settlements or rotations, whereas absolute deflections are concerned with deflection of the structure as a whole. The absolute deflections are the ones displayed in the structure deflection graphics. The difference between ***relative*** and ***absolute*** deflections is illustrated in the cantilever beam example below.

	
Relative Deflection	Absolute Deflection

Relative deflections are given in the member analysis results graphics and are the ones used in the member design.

Steel grade (AISC 360)
The steel grade can be chosen from the standard range for the USA or from an international range. User defined grades can also be added.

WARNING For composite beams, the upper limit for the steel grade is defined in the AISC Specification as 75 ksi (525 MPa) - see I1.2 (360-05) or I1.3 (360-10, 360-16). If you add a grade higher than this and apply to a composite beam all the design checks will be flagged as beyond scope.

WARNING For non-composite beams, the upper limit for the steel grade is defined in the AISC 360 Commentary A3.1a as 100 ksi (690 MPa). If you add a grade higher than this and apply to a non-composite member (rolled or built-up) all the design checks will be flagged as beyond scope.

The elastic modulus of steel for use in design is defined in the AISC Specification as E = 29,000 ksi

Steel beam design to AISC 360

Design method (Beams: AISC 360)

Either a load and resistance factor design (LRFD) or an allowable strength design (ASD) can be performed to determine the adequacy of the section for each condition.

The design method employed is consistent with the design parameters specified in the relevant chapters of the AISC Specification and associated 'Commentary', unless specifically noted otherwise. As the 2005 (Ref.1), 2010 (Ref. 2) and 2016 (Ref. 3) versions are all supported, where clauses are specific to a particular version these are indicated as (360-05), (360-10), or (360-16) as appropriate.

A basic knowledge of the design methods for beams in accordance with the specification is assumed.

Steel beam limitations and assumptions (Beams: AISC 360)

The following general limitations apply:

- Continuous beams (more than one span) must be co-linear in the plane of the web within a small tolerance (sloping in elevation is allowed),

- Rolled doubly symmetric prismatic sections, doubly symmetric hollow sections, channel sections are fully designed, plated beams are also fully designed
- Single angles, double angles and tees are designed, but additional limitations apply, (see Angle and tee limitations)
- Design of beams with web openings is beyond scope.

The following additional limitations apply for plated beams:

- Double and single symmetric I-sections allowed
- Single or multi-span allowed, including cantilever spans
- Design for axial force (tension or compression) or flexure (major or minor) or any combination of these
- Non-composite only
- Flanges and web all have same grade steel
- No design of curved beams (plan or elevation)
- No auto design
- No torsion design
- No seismic design

The following assumptions apply:

- All supports are considered to provide torsional restraint, that is lateral restraint to both flanges. This cannot be changed. It is assumed that a beam that is continuous through the web of a supporting beam or column together with its substantial moment resisting end plate connections is able to provide such restraint.
- If, at the support, the beam over sails the supporting beam or column then the detail is assumed to be such that the bottom flange of the beam is well connected to the supporting member and, as a minimum, has torsional stiffeners provided at the support.
- In the Tekla Structural Designer model, when not at supports, coincident restraints to both flanges are assumed when one or more members frame into the web of the beam at a particular position and the cardinal point of the centre-line model of the beam lies in the web. Otherwise, only a top flange or bottom flange restraint is assumed. Should you judge the actual restraint provided by the in-coming members to be different from to what has been assumed, you have the flexibility to edit the restraints as required.
- Intermediate lateral restraints to the top or bottom flange are assumed to be capable of transferring the restraining forces back to an appropriate system of bracing or suitably rigid part of the structure.
- It is assumed that you will make a rational and “correct” choice for the effective lengths between restraints for compression buckling. **The default**

value for the effective length factor of 1.0 may be neither correct nor safe.

Section classification (Beams: AISC 360)

Cross-section classification is determined using Table B4.1 (360-05), or Tables B4.1a+B4.1b (360-10).

At every cross section there are two classifications for each element in the section (flange or web) - one for axial compression and one for bending (flexure).

If axial compression does not exist (0kip or tension), the axial classification is not applicable. If bending is not present about both axes then the flexure classification is not applicable.

For axial compression the web and flanges are classified as either Compact or Slender and the worst of the two is the resultant axial classification.

For bending both the web and flange are classified as Compact, Non compact or Slender and the worst of the two is the resultant flexural classification.

The classification of the section must normally be Compact or Non compact, however sections which are classified as Slender will be allowed if they are subject to axial load only.

Classification for plated beams

Since built-up (plated) beams allow for asymmetric sections, the general approach in flexure classification for all built-up beams is:

- under major bending the compression flange is classified (both flanges are classified if double curvature exists, and the worst case is reported)
- under minor bending both flanges are classified
- under biaxial bending, major and minor bending are considered independently and the worst case is reported

D2. Axial tension (Beams: AISC 360)

If axial tension exists, tensile yielding and rupture checks are performed at the point of maximum tension in accordance with Equations D2.1 and D2.2.

NOTE In the rupture check the net area *Ae* is assumed to equal the gross area *Ag*.

A warning is issued if the slenderness ratio L/r exceeds 300.

E. Axial compression (Beams: AISC 360)

If axial compression exists, the member is assessed for Flexural Buckling and for Torsional and Flexural Torsional buckling.

The compressive strength is determined in accordance with Equations E3.1 and E4.1. For double angles these equations are subject to the modifications of Section E6.

The member length or member sub lengths between braces are checked for:

- Flexural buckling about major axis - for each unbraced length between adjacent points of major axis lateral brace and or torsional brace.
- Flexural buckling about minor axis - for each unbraced length between adjacent points of minor axis lateral brace and or torsional brace.
- Torsional and flexural torsional buckling - for each unbraced length between adjacent points of torsional brace (this check is not applied to hollow sections.)

For any unbraced length, the required compressive force P_r is taken as the maximum compressive force in the relevant length.

A warning is issued if the slenderness ratio KL/r exceeds 200.

G2. Shear strength (Beams: AISC 360)

Shear checks are performed at the point of maximum shear in accordance with Section G2.

Plated beams only

Since built-up (plated) beams allow for asymmetric sections, under minor shear the web shear coefficient, C_v is calculated for each flange separately and Equation G2-1 taken as:

$V_n = 0.6 * F_y * (A_{w,top} * C_{v,top} + A_{w,btm} * C_{v,btm})$

F2. Flexure (Beams: AISC 360)

The member is assessed for Flexure in accordance with Section F2 to F10 (as appropriate).

The following checks are potentially required:

About the x axis - within the LTB braced length

- Yielding
- Compression flange local buckling
- Web local buckling
- Local buckling
- Lateral torsional buckling (only required for I and C sections)

About the y axis in the LTB braced length

- Yielding
- Flange local buckling
- Web local buckling
- Local buckling

You can switch off the lateral torsional buckling checks for any unbraced length by indicating the length is continuously braced. If you use this option you must be able to provide justification that the unbraced length is adequately braced against lateral torsional buckling.

When the checks are required Tekla Structural Designer assumes a top flange (but not bottom flange) brace is provided at the position of each incoming beam. You can add or remove these braces if they don't reflect the actual brace provided by the incoming section. Each unbraced length which is not defined as being continuously braced is then checked in accordance with Section F2.

Plated beams only

The following additional checks apply about the x axis:

- Compression flange yielding
- Tension flange yielding

The approach to evaluating the web plastification factors, R_{pc} and R_{pt} in Section F4.2 and F4.4 of 360-10, has been adopted for 360-05 also i.e. the ratio I_{yc}/I_y is considered as well as h_c/t_w but note the following:

- under 360-05, I_{yc} is taken as the minimum inertia about the y axis of top and bottom flange (regardless which flange is in compression)
- under 360-10, I_{yc} is taken as the inertia about the y axis of the compression flange being considered

For flange local buckling about the y axis Equation F6-2 is used for both double and single symmetric sections, but in the latter case the more slender of the two flanges is assessed i.e. the higher λ value will be used.

H1. Combined forces (Beams: AISC 360)

Members subject to axial tension or compression and flexure about one or both axes are assessed in accordance with Section H1.

Plated beams only

For built-up (plated) beams a Proportioning Limit check applies. In AISC 360-05 and 360-10 this is detailed within the chapter on design for flexure (section F13.2). Load combinations which result in major axis bending on a built-up (plated) beam cause this check to be made. Any load combination which fails the Proportioning Limit check is considered as Beyond Scope for Combined Forces (regardless of whether any axial force is compressive or tensile).

DG9. Torsion (Beams: AISC 360)

Torsion design is carried out according to AISC design guide 9 (DG9), AISC 360-05 and AISC 360-10 for single span, pin ended steel beams with open and closed section types as follows:

Open sections (I- symmetric rolled)

- A torsion design and an angle rotation check can be carried out for applied torsion forces only

Closed sections (HSS only)

- •An angle of rotation check can be carried out for applied forces only

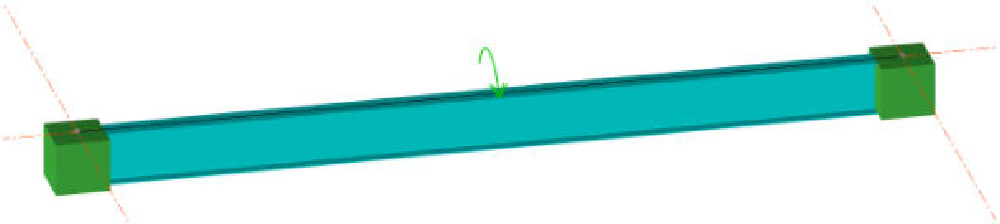
Torsion design - loading (Beams: AISC 360)

For design of open sections (i.e. rolled I sections in the current release) torsion design is carried out for “applied torsion loading” only and in accordance with those cases in Appendix B of DG9 with torsion fixed and warping free member ends (i.e. cases 3, 4 and 5 of DG9, with some extension for partial UDL and VDL).

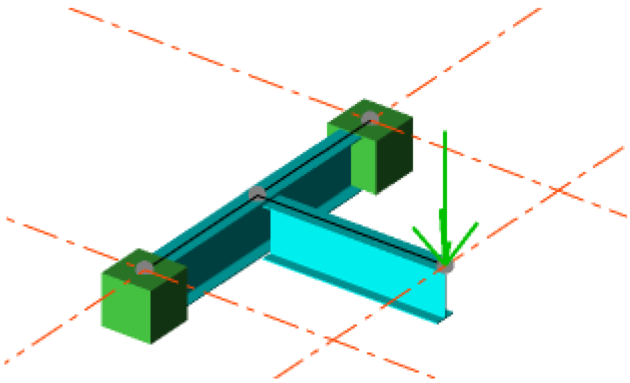
Applied torsion loading

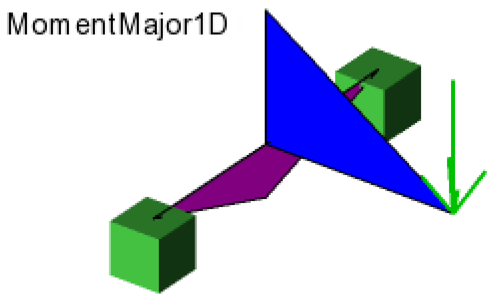
Tekla Structural Designer defines “applied torsion loading” as:

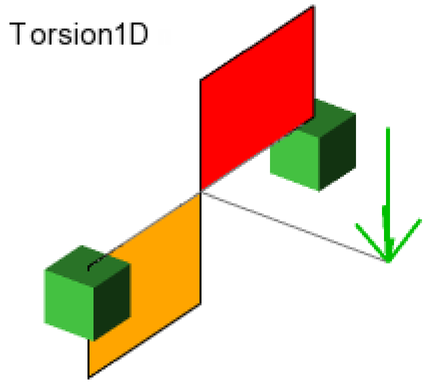
- A force that is manually applied by the User using the Member Loading in the Load ribbon, (as shown below)



- Or a force that is induced from a moment connection between primary and secondary beams, or a cantilever beam (as shown below, with bending and torsion moment diagrams).







Angle of rotation check (Beams: AISC 360)

I symmetric & HSS section

A torsion rotation check is optionally carried out based on the applied torsion loading only.

The check is applied by selecting “Apply rotation limit” (located in the steel beam properties under the Torsion heading). The default limit is also set in the steel beam properties as 2° but can be adjusted to suit.

Torsion design general checks (Beams: AISC 360)

1. Auto design is not carried out in the current release, only check design (and check design is only carried out if the “check for torsion” flag is set to on in the Edit dialog or Properties Window)
2. Lateral restraint amplification factor (in accordance with section 4.7.3 of DG9):
    a. It is assumed lateral displacement and twist are not restrained at any load point. Therefore, in accordance with section 4.7.3 of DG9, σ_{by} and σ_w will always be amplified in the presence of torsion.
    b. To avoid a negative value, Tekla Structural Designer applies a lower limit of 0.001ksi OR N/mm^2 to the denominator of the amplification factor,

(ϕF_{cr}^{e} - σ_{bx})

c. $F_{cr}^{e} = F_{cr,bx}$

d. Amplification factor = 1.0 when σ_{bx} = 0.0

3. Both major and minor axis shear buckling are checked if loaded in the relevant axis. A warning is issued if the buckling limit defined in AISC Sect G2 is exceeded. Torsion design will, however, be continued - the engineer is expected to deem if the shear buckling condition is safe.

4. Torsion shear stresses:

   a. A cross-section check is carried out at points of interest taken from the load analysis diagram.

5. Combined forces and torsion:

   a. HSS - sections

      1. A cross-section check is carried out at points of interest taken from the load analysis diagrams as well as 10th positions along the member. In cases where the final utilization ratio approaches 1.0 we strongly recommend the engineer considers other locations, where a more critical location than that chosen in Tekla Structural Designer may exist.

   b. I symmetric

      1. We take the most critical axial stress value across all axial strut lengths to determine $F_{cr,a}$

      2. We take the most critical major bending stress value across all LTB lengths to determine $F_{cr,bx}$

      3. In ASD design checks, the value used for $F_{cr,a}$ and $F_{cr,bx}$ is F_{cr} / 1.67 since F_a and F_b used in DG9 relate to the 1989 ASD Specification where this factor was effectively already taken into account.

Web openings (Beams: AISC 360)

Circular openings as an equivalent rectangle

Each circular opening is replaced by equivalent rectangular opening, the dimensions of this equivalent rectangle for use in all subsequent calculations are:

- d_o' = 0.9 * opening diameter
- l_o = 0.45 * opening diameter

Web opening design checks

Common design checks for both composite and non-composite beams

The following design checks are carried out at each opening for both composite and steel beams:

- **Section and opening dimension limit check** including the spacing of multiple openings if applicable.
- **Classification check**. Non-compact sections are beyond scope.
- **Moment-shear interaction check**. First the maximum pure flexural and shear strength is calculated following the guidelines of the Design Guide for the currently selected edition of the headcode. Then the direct formulas (3-5a and 3-5b) are used to calculate design shear and bending strength.
- **Deflection calculation**. As deflection calculations are headcode independent, for simplicity a single approach is used irrespective of the headcode selected.

Additional design checks for non-composite beams or composite beams at construction stage

The following additional design checks are carried out at each opening only for non-composite beams or composite beams at construction stage.

- **Lateral torsional buckling**. The 'standard' lateral torsional buckling check is run but the torsional constant is multiplied by a reduction factor according to the design guide. Strength over the openings should not be the governing UR.
- **Buckling of tee-shaped compression zone**. The tee which is in compression is investigated as an axially loaded column following the procedures of selected headcode. For unreinforced members this is not required when the aspect ratio of the tee is less than or equal to 4. For reinforced openings, this check is only required for large openings in regions of high moment.

Additional design checks for composite beams at composite stage

The following additional design checks are carried out at each opening only for composite beams at composite stage.

- **Slab reinforcement check**. The check of minimum transverse and longitudinal slab reinforcement ratio to prevent cracking of the slab in the vicinity of the web opening.
- **Number of shear connectors above the opening**. To limit the effect of bridging a minimum of two studs per foot is applied to the total number of studs. If this criterion is already satisfied by normal stud requirements, additional studs are not needed. A warning is shown when this criteria is not met.

Deflections

The simplified rules in DG2 are for limited cases and therefore have not been implemented. Instead Tekla Structural Designer uses a first principles approach as per Eurocodes.

The deflection of a beam with web openings will be greater than that of the same beam without openings. This is due to two effects:

- the reduction in the beam inertia at the positions of openings due to primary bending of the beam,
- the local deformations at the openings due to Vierendeel effects. This has two components - that due to shear deformation and that due to local bending of the upper and lower tee sections at the opening.

The primary bending deflection is established by 'discretising' the member and using a numerical integration technique based on 'Engineer's Bending Theory' - M/I = E/R = σ/y. In this way the discrete elements that incorporate all or part of an opening will contribute more to the total deflection.

The component of deflection due to the local deformations around the opening is established using a similar process to that used for cellular beams which is in turn based on the method for castellated beams given in the SCI publication, “Design of castellated beams. For use with BS 5950 and BS 449".

The method works by applying a 'unit point load' at the position where the deflection is required and using a 'virtual work technique to estimate the deflection at that position.

For each opening, the deflection due to shear deformation, δ_s, and that due to local bending, δ_{bt}, is calculated for the upper and lower tee sections at the opening. These are summed for all openings and added to the result at the desired position from the numerical integration of primary bending deflection.

Note that in the original source document on castellated sections, there are two additional components to the deflection. These are due to bending and shear deformation of the web post. For castellated beams and cellular beams where the openings are very close together these effects are important and can be significant. For normal beams the openings are likely to be placed a reasonable distance apart. Thus in many cases these two effects will not be significant. They are not calculated for such beams but in the event that the openings are placed close together a warning is given.

Seismic design rules (Beams: AISC 360)

Additional design rules are required for seismic combinations. These are as per the AISC Seismic Provisions (AISC 341-05) (Ref. 9). These additional design rules ONLY apply to members in Seismic Load Resisting Systems.

See Assumptions & limitations of the seismic provisions (AISC 341-05) for a list of the assumptions and limitations that apply with respect to the application of these rules to Tekla Structural Designer models.

The rules applied depend upon the seismic load resisting system as defined in the AISC Seismic Provisions and are listed below:

For a moment resisting frame

9. Special Moment Frame (SMF)

- 9.4a. Classification
- 9.8. Max spacing of bracing

10. Intermediate Moment Frame (IMF)

- 10.4a. Classification
- 10.8. Max spacing of bracing

11. Ordinary Moment Frame (OMF)

- 11.4. Classification

Moment resisting frame with a truss component

12. Special Truss Moment Frame (STMF) 1- Beyond Scope

For a braced frame

13. Special Concentrically Braced Frames (SCBF)

- 13.2d. Classification
- 13.4a.(2). Max lat brace spacing
- 13.4a. V and inverted V type

14. Ordinary Concentrically Braced Frames (OCBF)

- 14.2. Classification
- 14.3. Beams with V and inverted V type
- Beams (not columns with no K braces)
    - 14.2.(2). Max lat brace spacing

15. Eccentrically Braced Frames (EBF) - Beyond Scope

Buckling resistant braced frame

16. Buckling Restrained Braced Frames (BRBF) - Beyond Scope

Frames containing composite beams

- Composite Special Concentrically Braced Frames (C-SCBF) - Beyond Scope
- Composite Ordinary Braced Frames (C-OBF) - Beyond Scope
- Composite Eccentrically Braced Frames (C-EBF) - Beyond Scope

Composite beam design to AISC 360

- Design method (Composite beams: AISC 360) (page 1682)
- Serviceability limit state (SLS) (Composite beams: AISC 360) (page 1682)
- Construction stage (Composite beams: AISC 360) (page 1684)

• Composite stage (Composite beams: AISC 360) (page 1685)

Design method (Composite beams: AISC 360)

Either a load and resistance factor design (LRFD) or an allowable strength design (ASD) can be performed to determine the adequacy of the section for each condition.

The design method employed is consistent with the design parameters for simple composite beams as specified in Chapter I of the AISC Specification and associated 'Commentary', unless specifically noted otherwise. As the 2005 (Ref. 1), 2010 (Ref. 2) and 2016 (Ref. 3) versions are all supported, where clauses are specific to a particular version these are indicated as (360-05), (360-10), or (360-16) as appropriate.

A basic knowledge of the design methods for composite beams in accordance with the specification is assumed.

Serviceability limit state (SLS) (Composite beams: AISC 360)

Section properties (SLS)

In the calculation of the gross moment of inertia of the composite section the steel deck is ignored as is any concrete in tension. The concrete is converted into an equivalent steel section using an effective modular ratio based on the proportions of long and short term loads which are relevant to the particular calculation. Two alternative approaches are given - see p.16.1-308 in the 2005 Commentary, or p.16.1-353 in the 2010 Commentary for obtaining these properties.

One (the 'traditional method') calculates the gross uncracked inertia of the transformed section but uses 75% of the resulting value in the determination of deflections. The other uses a given formula to determine a 'lower-bound' inertia. While studies have shown that the simple application of a reduction factor (0.75) is more onerous than the lower-bound solution, the simpler 'traditional method' is the approach adopted in Tekla Structural Designer.

Tekla Structural Designer therefore calculates the deflection for the beam based on the properties as tabulated below.

Loadcase type	Properties used
self-weight	bare beam
Slab Dry	bare beam
Dead	composite properties calculated using the modular ratio for long term loads[1]
Live, Roof Live	composite properties calculated using the effective modular ratio[2]

Loadcase type	Properties used
	appropriate to the long term load percentage for each load.
Wind, Snow, Earthquake	composite properties calculated using the modular ratio for short term loads
Total loads	these are calculated from the individual loadcase loads as detailed above.

[1]The long term modulus is taken as the short term value divided by a factor (for shrinkage and creep), entered in the Slab properties.

n_S = the short term modular ratio = E_S/E_c

n_L = the long term modular ratio = $(E_S/E_c) * k_n$

[2]The effective modular ratio, n_E is based on the percentage of load which is considered long term. These calculations are repeated for each individual load in a loadcase. The effective modular ratio is given by,

$n_E = n_S + \rho_L * (n_L – n_S)$

ρ_L = the proportion of the load which is long term

The calculated Slab Dry, Live and Total load deflections (where necessary adjusted for the effect of partial interaction) are checked against the limits you specify.

NOTE All the beam deflections calculated above are “relative” deflections. For an illustration of the difference between relative and absolute deflection see Deflection checks.

Stress checks (SLS)

The Commentary (Section I3.1, paragraph 2 of the 2005 version, Section I3.2, of the 2010 version) suggests that where deflection controls the size of the beam then either it should be ensured that the beam is elastic at serviceability loading or that the inelastic deformations are taken into account. Tekla Structural Designer adopts the former approach. This is confirmed by checking that yield in the beam and crushing in the concrete do not occur at serviceability loading i.e. a service stress check. If they are found to fail, suggesting inelasticity at serviceability loading, then a warning will appear on the deflections page and the service stress results are available to view.

Tekla Structural Designer calculates the worst stresses in the extreme fibres of the steel and the concrete at serviceability limit state for each load taking into account the proportion which is long term and that which is short term. These stresses are then summed algebraically. The partial safety factors for loads are taken as those provided by you for the service condition on the Design

Combinations page. The stress checks assume that full interaction exists between the steel and the concrete at serviceability state.

Natural frequency checks (SLS)

The calculation of the natural frequency of a composite beam can be complex and is dependent upon the support conditions, the load profile and the properties of the composite section. In reality the vibration of a composite beam is never in isolation – the whole floor system (including the slabs and other adjacent beams) will vibrate in various modes and at various frequencies.

A simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behavior i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer will have the option to perform a Floor vibration analysis within the Tekla Structural Designer application.

Simplified approach

The natural frequency is determined from,

$NF = 0.18 * \sqrt{(g/\Delta_{NF})}$

Where:

- Δ_{NF} = the maximum static instantaneous deflection (in inches) that would occur under the effects of Slab Dry loading, and the proportion of dead loads and live loads specified by the user (as specified on the Natural Frequency page of the Design Wizard). It is based upon the composite inertia but not modified for the effects of partial interaction.
- g = the acceleration due to gravity (386.4 in/s^2)
- Factor of increased dynamic stiffness of concrete flange (default 1.35)

This is not given in the AISC Specification but is taken from Chapter 3 of Steel Design Guide Series 11. Floor Vibrations due to Human Activity. (Ref. 6) Its formulation is derived from the first mode of vibration of a simply supported beam subject to a udl.

Construction stage (Composite beams: AISC 360)

At construction stage the beam is acting alone before composite action is achieved and is unshored.

When you design or check a beam for construction stage loading the following checks are carried out in accordance with the relevant chapters of the AISC Specification, consistent with the approach (i.e. LRFD or ASD) used at the composite stage.

Section classification (Composite beams: AISC 360)

Cross-section classification is determined using Table B4.1 (360-05), or Tables B4.1a+B4.1b (360-10) and must be compact or non compact. Sections which are classified as slender are beyond the scope of Tekla Structural Designer.

Shear strength - I3.1b (360-05), I4.2 (360-10) (Composite beams: AISC 360)

Shear checks are performed at the point of maximum shear based upon the properties of the steel section alone in accordance with Section G2.

Strength during construction - I3.1c (360-05), I3.1b (360-10) (Composite beams: AISC 360)

Flexure

Checks are performed at the point of maximum moment along the beam based upon the properties of the steel section alone in accordance with Section F2.

Lateral torsional buckling checks

When the forms are attached to the top flange then full lateral restraint can be assumed, irrespective of the angle of the deck. In this case you should indicate the beam is continuously braced.

In other cases any incoming beams will be automatically identified.

Each sub-length which is not defined as being continuously braced is checked in accordance with Section F2.

Deflection checks (Composite beams: AISC 360)

Relative deflections are used in the composite beam design. (See: Deflection checks (page 1670))

The following deflections are calculated for the loads specified in the construction stage load combination:

- the Dead load deflections i.e. those due to the beam self weight, the Slab Wet loads and any other included dead loads,
- the live load deflections i.e. those due to construction live loads,
- the Total load deflection i.e. the sum of the previous items.

The loads are taken as acting on the steel beam alone.

The “Service Factor” (default 1.0), specified against each load case in the construction combination is applied when calculating the above deflections.

If requested by the user, the total load deflection is compared with either a span-over limit or an absolute value. The initial default limit is span/200, (as per CC.1.1 of ASCE 7-05 (Ref. 7) or ASCE 7-10 (Ref. 8)).

Composite stage (Composite beams: AISC 360)

Tekla Structural Designer performs all checks for the composite stage condition in accordance with Section I3 unless specifically noted otherwise.

Equivalent steel section (Composite beams: AISC 360)

An equivalent steel section is determined for use in the composite stage calculations by removing the fillet while maintaining the full area of the section. This approach reduces the number of change points in the calculations while maintaining optimum section properties.

Shear strength - I3.1b (360-05), I4.2 (360-10) (Composite beams: AISC 360)

Shear checks are performed at the point of maximum shear in accordance with Section G2 for the maximum required shear strength, V_r, at the composite stage. The shear check is performed on the bare beam alone at the composite stage ignoring any contribution from the concrete slab.

Strength of composite beams with shear connectors - I3.2 (Composite beams: AISC 360)

Section classification

For section classification purposes the true section is used. Tekla Structural Designer classifies the section in accordance with Section I3.2a. Only the web of the section is classified - the bottom flange is in tension and so cannot buckle locally and it is assumed that the top flange is sufficiently braced by the composite slab.

The classification of the web must be compact so that plastic stress blocks can be used.

Flexure

Checks are performed at the point of maximum moment and the position of application of each point load as well as all other points of interest along the beam. Flexure is calculated in accordance with Section I3.2 (360-05/-10). Since the flexural strength at all point loads is checked then this will inherently satisfy Section I3.2d (6) (360-05) or Section I8.2c (360-10) which require that “the number of shear connectors placed between any concentrated load and the nearest point of zero moment shall be sufficient to develop the maximum required flexural strength at the concentrated load point”.

During the selection process, in auto design mode point loads are taken to be “significant” if they provide more than 10% of the total shear on the beam. For the final configuration and for check mode all point loads are checked for flexure.

Shear connectors (Composite beams: AISC 360)

Tekla Structural Designer checks shear connectors to Section I1-3 (360-05), or Section I8 (360-10).

The nominal strength of headed stud shear connectors in a solid slab or a composite slab is determined in accordance with Section I3.2d (360-05), or Section I3.2d with shear connector strength from I8.2a (360-10).

Ribs perpendicular

The reduction factor R_p is taken as,

- 0.6 for any number of studs and $e_{mid\text{-}ht} < 2$ in
- 0.75 for any number of studs and $e_{mid\text{-}ht} >= 2$ in

In Tekla Structural Designer you are therefore not required to input the actual value of $e_{mid\text{-}ht}$ instead you simply indicate if it is less than 2 in.

Ribs parallel

$R_p = 0.75$ in all cases

Ribs at other angles

Where the ribs are at an angle θ_r to the beam there is no guidance in the AISC Specification. The approach adopted by Tekla Structural Designer is to apply a geometric adjustment of the reduction factors R_g and R_p which for the purposes of this adjustment are combined into one “k” factor. The combined reduction factor is calculated for perpendicular and parallel separately and then adjusted as shown below.

$k_s = k_1 * \sin^2\theta_r + k_2 * \cos^2\theta_r$

Where:

k_s = the adjusted value of the combined reduction factor $R_g * R_p$

k_1 = the value of the combined reduction factor $R_g * R_p$ for ribs perpendicular

k_2 = the value of the combined reduction factor $R_g * R_p$ for ribs parallel

Degree of shear connection

For efficient design the number of studs should be minimized. If the number provided has an overall capacity greater than the capacity of the concrete flange or steel beam (whichever is the lesser) then this is full shear connection. Anything less than this, is partial shear connection. There are, however, limits on the amount of partial interaction that are recommended by the AISC Specification – see note “3” (p.16.1-311 of the 2005 Commentary, or p.16.1-356 of the 2010 Commentary).

For all beams, the number of connectors required for full shear connection is,

$N_s = (\min(T_s, (C_{c1} + C_{c2})))/Q_n$ rounded up to the next group size above

Where:

T_s = the tensile yield strength of the steel section

C_{c1} = the strength of the concrete flange above the ribs

C_{c2} = strength of the concrete in the ribs (zero for perpendicular decks)

Q_n = the nominal strength of an individual shear connector

The degree of partial shear connection is given by,

$I_{nt} = N_a * Q_n / (\min((C_{c1} + C_{c2}), T_s))$

Where:

N_a = the number of shear connectors provided from the nearer point of support to the position under consideration

The degree of partial shear connection is checked at the point of maximum bending moment or the position of a point load if at that position the maximum utilization ratio occurs.

To determine the status of the check Tekla Structural Designer applies the following rules:

- If the partial interaction ratio at the position of maximum moment is less than the absolute minimum interaction ratio (default 25%), then this generates a FAIL status,
- If the partial interaction ratio at the position of maximum utilization ratio when this is at a different position to the maximum moment, is less than the absolute minimum interaction ratio, then this generates a WARNING status,
- If the partial interaction ratio at the position of maximum moment, or maximum utilization ratio if this is different, is greater than the absolute minimum interaction ratio, then this generates a PASS status,
- If the partial interaction ratio at any point load position that is not the maximum utilization ratio is less than the absolute minimum interaction ratio, then this does not affect the status in any way.
- If the partial interaction ratio at any position is less than the advisory minimum interaction ratio (default 50%) then this is given for information only and does not affect the status in any way.

Dimensional requirements

The dimensional limits given below are either recommendations or code limits:

- the nominal rib height of the profiled deck, h_r should be not greater than 3 in
- the mean width of the ribs of the profiled sheet, w_r should be not less than 2 in (for re-entrant decks the “mean” is taken as the minimum opening at the top of the rib)

- the nominal diameter of stud connectors, d_{sc} should be not greater than ¾ in
- the height of the stud after welding, H_s should be at least 1½ in greater than the nominal rib height of the profiled deck – see Section I3.2c(b) (360-05), or Section I3.2c(2) (360-10).
- the total depth of the composite slab, d_{cs} should not be less than 3¾ in
- the thickness of concrete above the main flat surface of the top of the ribs of the sheeting, d_{cs} - h_r should not be less than 2 in
- concrete cover, d_{cs} – H_s over the connector should not be less than ½ in – see Section I3.2c(b) (360-05), or Section I3.2c(2) (360-10).
- the longitudinal spacing should not exceed the lesser of 36 in or 8 * the slab depth, d_{cs} (see Section 6.2.6.2 of Structural Steel Designer's Handbook. Second Edition (Ref. 5))
- where studs are spaced at greater than 18 in centers puddle welds or other appropriate means are required to ensure anchorage of deck – see Section I3.2c (360-05), or Section I3.2c(4) (360-10).
- the clear distance between the edge of a connector and the edge of the steel beam flange should be not less than ¾ in (as universal good practice).
- Section I8.2d of the AISC Specification (360-10) requires that the minimum edge distance from the center of an anchor to a free edge in the direction of the shear force shall be 8 in for normal weight concrete and 10 in for lightweight concrete. This requirement will apply only in a limited number of configurations and therefore is not checked.
- the spacing of connectors in the direction of shear i.e. along the beam should be not less than, 6 * the stud diameter
- the spacing of connectors transverse to direction of shear i.e. across the beam should be not less than 4 * the stud diameter except for the condition given in the next item
- where rows of studs are staggered, the minimum transverse spacing between longitudinal lines of studs should be not less than 3 * the stud diameter with the amount of stagger such that the diagonal distance between studs on adjacent longitudinal lines is not less than 4 * the stud diameter
- the stud connector diameter should not exceed 2.5 times the flange thickness unless located directly over the web.

NOTE You should confirm that the chosen configuration of decking and studs meet those dimensional requirements that you deem appropriate.

Steel column design to AISC 360

Design method (Columns: AISC 360)

Either a load and resistance factor design (LRFD) or an allowable strength design (ASD) can be performed to determine the adequacy of the section for each condition.

The design method employed is consistent with the design parameters specified in the relevant chapters of the AISC Specification and associated 'Commentary', unless specifically noted otherwise. As the 2005 (Ref. 1), 2010 (Ref. 2) and 2016 (Ref. 3) versions are all supported, where clauses are specific to a particular version these are indicated as (360-05), (360-10), or (360-16) as appropriate. A basic knowledge of the design methods for columns in accordance with the specification is assumed.

Section classification (Columns: AISC 360)

Cross-section classification is determined using Table B4.1 (360-05), or Tables B4.1a+B4.1b (360-10).

At every cross section there are two classifications for each element in the section (flange or web) - one for axial compression and one for bending (flexure).

If axial compression does not exist (0kip or tension), the axial classification is NA. If bending is not present about both axes then the flexure classification is NA.

For axial compression the web and flanges are classified as either Compact or Slender and the worst of the two is the resultant axial classification.

For bending both the web and flange are classified as Compact, Non compact or Slender and the worst of the two is the resultant flexural classification.

The classification of the section must normally be Compact or Non compact, however sections which are classified as Slender will be allowed if they are subject to axial load only.

All unacceptable classifications are either failed in check mode or rejected in design mode.

D2. Axial tension (Columns: AISC 360)

If axial tension exists, tensile yielding and rupture checks are performed at the point of maximum tension in accordance with Equations D2.1 and D2.2.

NOTE In the rupture check the net area A_e is assumed to equal the gross area A_g.

A warning is also issued if the slenderness ratio L/r exceeds 300.

E. Axial compression (Columns: AISC 360)

If axial compression exists, the member is assessed for Flexural buckling and for Torsional and flexural torsional buckling. The compressive strength is determined in accordance with Equations E3.1 and E4.1. For double angles these equations are subject to the modifications of Section E6.

The member length or member sub lengths between braces are checked for:

- Flexural buckling about major axis - for each sub-length between adjacent points of major axis lateral bracing and or torsional bracing.
- Flexural buckling about minor axis - for each sub-length between adjacent points of minor axis lateral bracing and or torsional bracing.
- Torsional and flexural torsional buckling - for each sub-length between adjacent points of torsional bracing (this check is not applied to hollow sections.)

For any sub-length, the required compressive force P_r is taken as the maximum compressive force in the relevant sub-length.

A warning is also issued if the slenderness ratio KL/r exceeds 200.

G2. Shear strength (Columns: AISC 360)

Shear checks are performed for the absolute value of shear force normal to the x-x axis and normal to the y-y axis, F_{vx} and F_{vy}, at the point under consideration in accordance with Section G2.

F2. Flexure (Columns: AISC 360)

The member is assessed for Flexure in accordance with section F2. The following checks are potentially required:

About the x axis - within the LTB sub-length

- Yielding
- Compression flange local buckling
- Web local buckling
- Local buckling
- Lateral Torsional Buckling (only required for I and C sections)

About the y axis in the LTB sub-length

- Yielding
- Compression flange local buckling
- Web local buckling
- Local buckling

The lateral torsional buckling checks can be switched off for any sub-length by indicating the length is continuously braced. If you use this option you must be able to provide justification that the sub-length is adequately braced against lateral torsional buckling.

When the checks are required you can set the effective length of each sub-beam (the portion of the beam between one brace and the next) either by giving factors to apply to the physical length of the beam, or by entering the effective length that you want to use.

H1. Combined forces (Columns: AISC 360)

Members subject to axial tension or compression and flexure about one or both axes are assessed in accordance with section H1.

Seismic design rules (Columns: AISC 360)

Additional design rules are required for seismic combinations. These are as per the AISC Seismic Provisions (AISC 341-05) (Ref. 9). These additional design rules ONLY apply to members in Seismic Load Resisting Systems.

See Assumptions & limitations of the seismic provisions (AISC 341-05) for a list of the assumptions and limitations that apply with respect to the application of these rules to Tekla Structural Designer models.

The rules applied depend upon the seismic load resisting system as defined in the AISC Seismic Provisions and are listed below:

For a moment resisting frame

9. Special Moment Frame (SMF)

- 9.4a. Classification

- 9.4. Column strength check
- 9.6. Column/beam moment ratio

10. Intermediate Moment Frame (IMF) • 10.4a. Classification

- 10.4. Column strength

11. Ordinary Moment Frame (OMF)

- 11.4. Classification
- 11.4. Column strength check

Moment resisting frame with a truss component

12. Special Truss Moment Frame (STMF) 1- Beyond Scope

For a braced frame

13. Special Concentrically Braced Frames (SCBF)

- 13.2d. Classification
- 13.2b. Column strength check

14. Ordinary Concentrically Braced Frames (OCBF)

- 14.2. Classification
- 14.2. Column strength check

15. Eccentrically Braced Frames (EBF) - Beyond Scope

Buckling resistant braced frame

16. Buckling Restrained Braced Frames (BRBF) - Beyond Scope

Frames containing composite beams

Composite Special Concentrically Braced Frames (C-SCBF) - Beyond Scope

Composite Ordinary Braced Frames (C-OBF) - Beyond Scope

Composite Eccentrically Braced Frames (C-EBF) - Beyond Scope

Steel brace design to AISC 360

Design method (Braces: AISC 360)

Tekla Structural Designer allows you to analyze and design a member with pinned end connections for axial compression, tension and seismic design forces.

Either a load and resistance factor design (LRFD) or an allowable strength design (ASD) can be performed to determine the adequacy of the section for each condition.

The design method employed is consistent with the design parameters specified in the relevant chapters of the AISC Specification and associated 'Commentary', unless specifically noted otherwise. As the 2005 (Ref. 1), 2010 (Ref. 2) and 2016 (Ref. 3) versions are all supported, where clauses are specific to a particular version these are indicated as (360-05), (360-10), or (360-16) as appropriate.

A basic knowledge of the design methods for braces in accordance with the specification is assumed.

Section classification (Braces: AISC 360)

Cross-section classification is determined using Table B4.1 (360-05), or Tables B4.1a+B4.1b (360-10).

D2. Axial tension (Braces: AISC 360)

If axial tension exists, tensile yielding and rupture checks are performed at the point of maximum tension in accordance with Eqns D2.1 and D2.2.

A warning is also issued if the slenderness ratio L/r exceeds 300.

E. Axial compression (Braces: AISC 360)

If axial compression exists, the member is assessed for Flexural Buckling and for Torsional and Flexural Torsional buckling. The compressive strength is determined in accordance with Eqns E3.1 and E4.1. For double angles these equations are subject to the modifications of Section E6.

The member length or member sub lengths between braces are checked for:

- Flexural buckling about major axis - for each braced length between adjacent points of major axis lateral bracing and or torsional bracing.
- Flexural buckling about minor axis - for each braced length between adjacent points of minor axis lateral bracing and or torsional bracing.
- Torsional and flexural torsional buckling - for each braced length between adjacent points of torsional bracing (this check is not applied to hollow sections.)

For any braced length, the required compressive force P_r is taken as the maximum compressive force in the relevant length.

A warning is also issued if the slenderness ratio KL/r exceeds 200.

Seismic design rules (Braces: AISC 360)

Additional design rules are required for seismic combinations. These are as per the AISC Seismic Provisions (AISC 341-05) (Ref. 9). These additional design rules ONLY apply to members in Seismic Load Resisting Systems.

See Assumptions & limitations of the seismic provisions (AISC 341-05) for a list of the assumptions and limitations that apply with respect to the application of these rules to Tekla Structural Designer models.

The rules applied depend upon the seismic load resisting system as defined in the AISC Seismic Provisions and are listed below:

For a braced frame

13. Special Concentrically Braced Frames (SCBF)

- 13.2d. Classification
- 13.2b. brace required strength
- 13.2a. brace slenderness limit
- 13.2e. built up members - double angles

14. Ordinary Concentrically Braced Frames (OCBF)

- 14.2. Classification
- 14.2. Bracing members, V or A braces

15. Eccentrically Braced Frames (EBF) - Beyond Scope

Buckling resistant braced frame

16. Buckling Restrained Braced Frames (BRBF) - Beyond Scope

Frames containing composite beams

Composite Special Concentrically Braced Frames (C-SCBF) - Beyond Scope

Composite Ordinary Braced Frames (C-OBF) - Beyond Scope

Composite Eccentrically Braced Frames (C-EBF) - Beyond Scope

Truss member design to AISC 360

- Design method (Trusses: AISC 360) (page 1695)
- Design checks (Trusses: AISC 360) (page 1696)

Design method (Trusses: AISC 360)

Unless explicitly stated all truss calculations will adopt either a load and resistance factor design (LRFD) or an allowable strength design (ASD) as consistent with the design parameters as specified in the AISC Specification and associated Commentary.

Design checks (Trusses: AISC 360)

Truss members can either be defined manually, or the process can be automated using the **Truss Wizard**. Irrespective of the method used, the resulting Truss members will be one of four types:

- Internal
- Side
- Bottom
- Top

Depending on the type, different design procedures are adopted.

Internal and side truss members

The design checks for internal and side truss members are the same as those for braces. With the exception that seismic forces are not designed for. See: Steel brace design to AISC 360 (page 1693)

Top and bottom truss members

The design checks for top and bottom truss members are the same as those for beams. With the exception that seismic forces are not designed for. See: Steel beam design to AISC 360 (page 1671)

Steel single, double angle and tee section design to AISC 360

Design method (Angles and tees: AISC 360)

Either a load and resistance factor design (LRFD) or an allowable strength design (ASD) can be performed to determine the adequacy of the section for each condition. The design method adopted is dictated by the member characteristic type:

Beam, Truss member top, or Truss member bottom:

- Member is designed for axial tension, compression, shear, bending and combined forces
- This is consistent with the method detailed in Steel beam design to AISC 360

Brace, Truss internal, or Truss member side:

- Member is designed for axial tension, compression and compression buckling only
- This is consistent with the method detailed in Steel brace design to AISC 360

A basic knowledge of the design method for angles and tees in accordance with the specification is assumed.

NOTE For tees, single angles, and double angles - specific additional Angle and tee limitations apply to the above design methods.

NOTE When modelling a double angle, the Compression property 'a' (distance between connectors), has to be manually set (either in Compression section of the **Properties** window or on the Compression tab of the **Properties** dialog box). Refer to AISC 360 section E6 for further information.

Angle and tee limitations (AISC 360)

- All sections and in particular single angles are assumed to be effectively loaded through the shear center such that no additional torsion moments are developed. In addition no direct allowance is made for 'destabilizing loads'.
- Design excludes bending of the outstand leg of single and double angles loaded eccentrically e.g. supporting masonry.

Section axes (Angles and tees: AISC 360)

For all sections:

- x-x is the axis parallel to the flanges
- y-y is the axis perpendicular to the flanges
    - for Single Angles and Double Angles
        - y-y parallel to long side (leg) - single angles

- y-y parallel to long side (leg) - double angles with long leg back to back
- y-y parallel to short side (leg) - double angles with short leg back to back

- w-w is the major principal axis for single angles
- z-z is the minor principal axis for single angles

Design procedure for single angles (Angles and tees: AISC 360)

Single angles with continuous lateral-torsional restraint along the length are permitted to be designed on the basis of **geometric axis (x, y) bending**.

Single angles without continuous lateral-torsional restraint along the length are designed using the provision for **principal axis (w, z) bending** except where the provision for bending about geometric axis is permitted.

Geometric axis bending permitted:

- If single angles without continuous lateral torsional restraint and legs of angles are equal and there is no axial compression and bending about one of the geometric axis only

---

NOTE Design on the basis of geometric axis bending should also be permitted if single angles without continuous lateral torsional restraint but with lateral torsional restraint at the point of maximum moment only and legs of angles are equal and there is no axial compression and bending about one of the geometric axis only. However this is beyond scope of the current release of the program.

Geometric axis design

1. Nominal flexural strength Mnx – about X axis (major geometric axis)
2. Nominal flexural strength Mny – about Y axis (minor geometric axis)

Check:

IF LRFD

a. $M_{rx} \leq \varphi_b * M_{nx}$, where $\varphi_b = 0.9$

b. $M_{ry} \leq \varphi_b * M_{ny}$, where $\varphi_b = 0.9$

IF ASD

$M_{rx} \leq M_{nx} /\Omega_b$, where $\Omega_b = 1.67$

$M_{ry} \leq M_{ny} /\Omega_b$, where $\Omega_b = 1.67$

Principal axis design

1. Required flexural strength M_{rw} – about W axis
2. Required flexural strength M_{rz} – about Z axis

3. Nominal flexural strength M_{nw} – about W axis (major principal bending axis)
4. Nominal flexural strength M_{nz} – about Z axis (minor principal bending axis)

Check:

IF LRFD

a. $M_{rw} \leq \varphi_b * M_{nw}$, where $\varphi_b = 0.9$

b. $M_{rz} \leq \varphi_b * M_{nz}$, where $\varphi_b = 0.9$

IF ASD

a. $M_{rw} \leq M_{nw} / \Omega_b$, where $\Omega_b = 1.67$

b. $M_{rz} \leq M_{nz} / \Omega_b$, where $\Omega_b = 1.67$

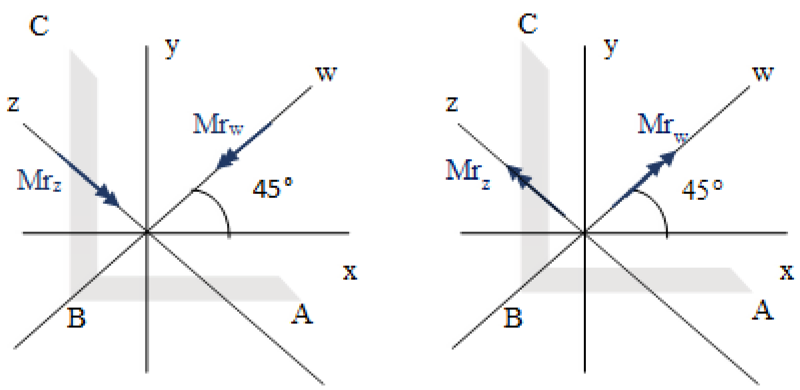
The principal axes moments are calculated from the following formulas for both LRFD and ASD:

$M_{rw} = M_{rx} \cos\alpha + M_{ry} \sin\alpha$

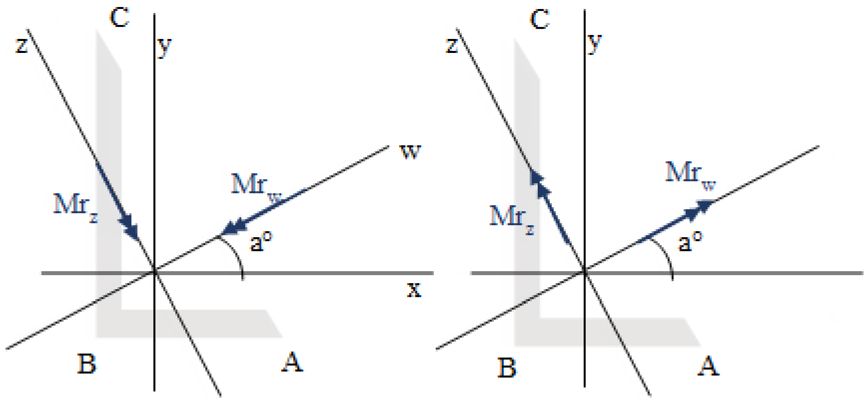
$M_{rz} = -M_{rx} \sin\alpha + M_{ry} \cos\alpha$

In the case of biaxial bending, or bending and axial force the combined stress ratio must be checked using the provisions of AISC, section H2.

For the **three points of the angle A, B, C** the combined ratio check should be performed.



Single Equal Angles - Sign of Moments



Single Unequal Angles - Sign of Moments

If the interaction of stresses at each point is seen to be less than 1.0 the member is adequate to carry the required load.

Check:[1]

Abs (f_{ra} / F_{ca} + f_{rbw} / F_{cbw} + f_{rbz} / F_{cbz}) $\leq$ 1.0

NOTE In axial tension when the sum of the moment ratios about the major and minor axis bending is greater or equal to 0 then the axial stress ratio is taken as 0.0 in order to give conservative results and the axial stress ratio is renamed “effective”.

Design procedure for tee sections (Angles and tees: AISC 360)

The nominal flexural strength M_n is the lowest value obtained according to the limit states of yielding (plastic moment), lateral-torsional buckling and leg local buckling.

$M_{nx} = M_{in} \{M_{nx,Yield}, M_{nx,LTB}, M_{nx,LLB}\}$

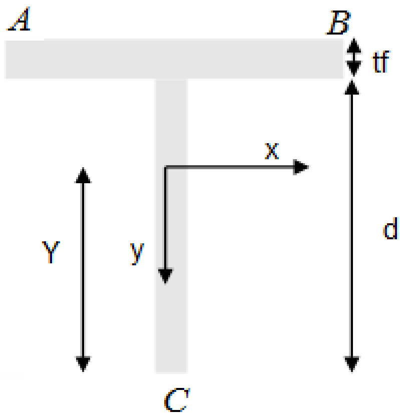
In the case of biaxial bending, or bending and axial force the combined stress ratio must be checked using the provisions of AISC, section H2.

The applied loads are

- P_r Axial
- M_{rx} Bending in x axis
- M_{ry} Bending in y axis

Check:[1]

Abs (f_{ra} / F_{ca} + f_{rbzx} / F_{cbx} + f_{rby} / F_{cby}) $\leq$ 1.0



Tees - Critical Points A, B & C

Design procedure for double angles (Angles and tees: AISC 360)

The nominal flexural strength M_n is the lowest value obtained according to the limit states of yielding (plastic moment), lateral-torsional buckling and leg local buckling.

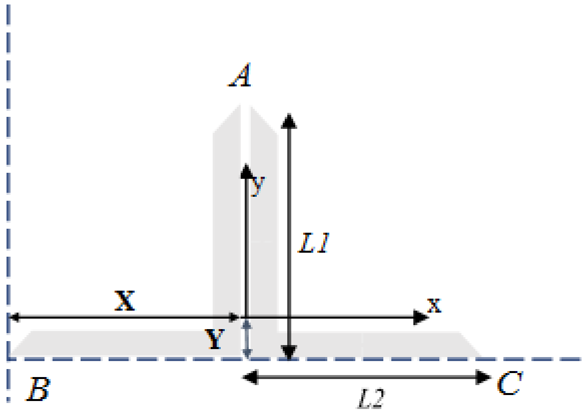
$M_{nx} = \text{Min}\ \{M_{nx,Yield}, M_{nx,LTB}, M_{nx,LLB}\}$

NOTE For the local buckling check of double angles the provisions of the 2010 code are used. In the 05 code, section F9.3 states Flange local Buckling of Tees and does not refer to double angles.

In the case of biaxial bending, or bending and axial force the combined stress ratio must be checked using the provisions of AISC, section H2. The applied loads are

- P_r Axial
- M_{rx} Bending in x axis
- M_{ry} Bending in y axis

Check:[1] $\text{Abs}\ (f_{ra} / F_{ca} + f_{rbw} / F_{cbx} + f_{rby} / F_{cby}) \leq 1.0$



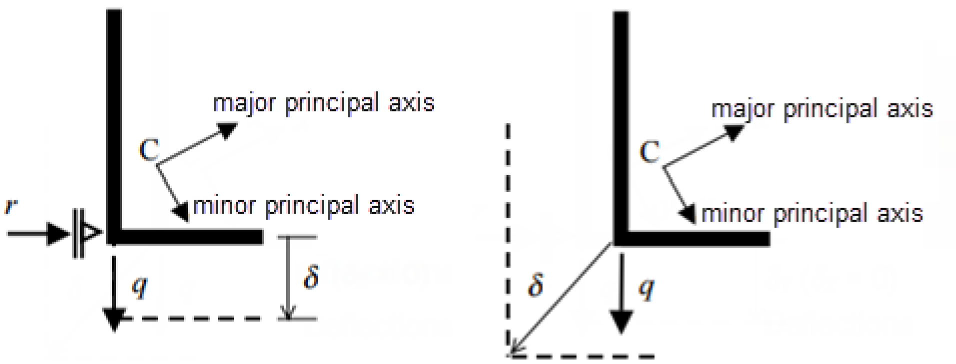
Double Angles - Critical Points A, B & C

Deflection of single angles (Angles and tees: AISC 360)

If a single angle is continuously restrained the major geometric moment and major geometric section properties are used in the general equation governing the beam deflection.

However, because single angle geometric axes are not coincident with the principal axes; a different procedure is required if the angle is not continuously restrained, the procedure being as follows:

1. External loads are transposed from the geometric axes to the principal axes.
2. The deflection equations are used to calculate deflections in the principal axes.
3. These principal axis deflections are then transposed to geometric axes again.



Single Angle Deflections (continuously restrained, unrestrained)

References (AISC 360)

1. **American Institute of Steel Construction**. ANSI/AISC 360-05 Specification for structural steel buildings. **AISC, 2005**.
2. **American Institute of Steel Construction**. ANSI/AISC 360-10 Specification for structural steel buildings. **AISC, 2010**.
3. **American Institute of Steel Construction**. ANSI/AISC 360-16 Specification for structural steel buildings. **AISC, 2016**.
4. **American Concrete Institute**. Building Code Requirements for Structural Concrete and Commentary. ACI 318-08. **ACI, 2008**.
5. **Brockenbrough, R. L. & Merritt, F. S**. Structural Steel Designer's Handbook. Second Edition. **McGraw-Hill 1994 USA**.
6. **American Institute of Steel Construction**. Steel Design Guide Series 11. Floor Vibrations due to Human Activity. **AISC, 1997**.
7. **American Society of Civil Engineers**. Minimum Design Loads for Buildings and Other Structures. ASCE/SEI 7-05. **ASCE, 2006**.
8. **American Society of Civil Engineers**. Minimum Design Loads for Buildings and Other Structures. ASCE/SEI 7-10. **ASCE, 2010**.
9. **American Institute of Steel Construction**. ANSI/AISC 341-05 Seismic Provisions for Structural Steel Buildings. **AISC, 2006**.
10. **American Institute of Steel Construction**. ANSI/AISC 341-10 Seismic Provisions for Structural Steel Buildings. **AISC, 2010**.
11. **American Institute of Steel Construction**. ANSI/AISC 341-16 Seismic Provisions for Structural Steel Buildings. **AISC, 2016**.

Steel seismic design to AISC 341

Additional seismic provisions are required to be applied to members that are part of the seismic force resisting system (SFRS) of a structure. These provisions are applied in addition to any standard requirements for structural steel buildings as per AISC 360. The seismic provisions are contained in AISC 341. Tekla Structural Designer covers non-seismic steel design to AISC 360-05, AISC 360-10 and AISC 360-016. The purpose of this guide is to describe the matching seismic design requirements contained in AISC 341-05, AISC 341-10 and AISC 341-16.

Criteria assumed to be met (Seismic: AISC 341)

Seismic design in the current release of Tekla Structural Designer covers only those checks detailed in later sections and presupposes certain criteria are met e.g. that lateral braces to beams are sufficiently strong. These presuppositions are noted below.

Common

AISC 341-16 and AISC 341-10

- Column bases are assumed to comply with the requirements of D2.6.
- Steel material grades used in particular members and SFRS type are assumed to comply with A3.1.

AISC 341-05

- Column bases are assumed to comply with the requirements of 8.5.
- Steel material grades used in particular members and SFRS type are assumed to comply with 6.1.

OMF

AISC 341-16 and AISC 341-10

- Beam to column connections used in the SFRS are assumed to satisfy the requirements of E1.6.

AISC 341-05

- Beam to column connections used in the SFRS are assumed to satisfy the requirements of 11.2.
- Continuity plates are assumed to comply with the requirements of 11.5.
- As per 11.9 column splices are assumed to comply with the requirements of 8.4a.

IMF

AISC 341-16 and AISC 341-10

- The lateral braces themselves will not be designed to meet the additional criteria of D1.2a & c - it is assumed that the user will check this independently.

- The position of lateral braces will not be checked for the location of points of concentrated force or positions of plastic hinge per D1.2c.
- The protected zone is assumed to comply with E2.5c.
- Connections used in the SFRS are assumed to satisfy the requirements of E2.6.

AISC 341-05

- Beam to column connections used in the SFRS are assumed to satisfy the requirements of 10.2.
- Panel zones in beam to column connections used in the SFRS are assumed to satisfy the requirements of 10.3.
- Continuity plates are assumed to comply with the requirements of 10.5.
- The lateral braces themselves will not be designed to meet the additional criteria of 10.8 - it is assumed that the user will check this independently.
- The position of lateral braces will not be checked for the location of points of concentrated force or positions of plastic hinge per 10.8.
- As per 10.9 column splices are assumed to comply with the requirements of 8.4a.

SMF

AISC 341-16 and AISC 341-10

- The lateral braces themselves will not be designed to meet the additional criteria of D1.2b & c - it is assumed that the user will check this independently.
- The position of lateral braces will not be checked for the location of points of concentrated force or positions of plastic hinge per D1.2c.
- Beam column connections are always assumed braced as per E3.4c(1).
- The protected zone is assumed to comply with E3.5c.
- Connections used in the SFRS are assumed to satisfy the requirements of E3.6.

AISC 341-05

- Beam to column connections used in the SFRS are assumed to satisfy the requirements of 9.2.
- Panel zones in beam to column connections used in the SFRS are assumed to satisfy the requirements of 9.3.
- Continuity plates are assumed to comply with the requirements of 9.5.
- Beam column connections are always assumed braced as per 9.7a.
- The lateral braces themselves will not be designed to meet the additional criteria of 9.8 - it is assumed that the user will check this independently.

- The position of lateral braces will not be checked for the location of points of concentrated force or positions of plastic hinge per 9.8.
- Column splices are assumed to comply with the requirements of 9.9 whilst those not part of the SFRS are assumed to comply with 8.4b

OCBF

AISC 341-16 and AISC 341-10

- Column splices are assumed to comply with D2.5.
- It is assumed that the beams in OCBF are continuous between columns in accordance with F1.4a.
- It is assumed that the user will ensure that the ends of V and A braces are vertically released so that they provide no support for dead and live loads as per F1.4a (1).
- It is assumed that the user will apply the relevant lateral restraint at position of V/A braces or establish that the beam has sufficient out of plane strength and stiffness to ensure stability in order to comply with F1.4a (2).
- K braces are not permitted for OCBF in accordance with F1.4b.
- Coincident V and A braces giving X type braced frames are out of scope for additional beam checks required by AISC 341-10 F1.4a.
- Bracing connections in the SFRS are assumed to satisfy the requirements of F1.6.
- OCBF above seismic isolation systems are currently beyond scope

AISC 341-05

- Column splices are assumed to comply with 8.4a. Column splices in columns not part of the SFRS are assumed to comply with 8.4b.
- It is assumed that the beams in OCBF are continuous between columns in accordance with 14.3.
- It is assumed that the user will ensure that the ends of V and A braces are vertically released so that they provide no support for dead and live loads as per 14.3 (1).
- Lateral braces are not designed to meet the additional criteria in 14.3 (2) - it is assumed that the user will check this independently.
- It is assumed that the user will apply the relevant lateral restraint at position of V/A braces or establish that the beam has sufficient out of plane strength and stiffness to ensure stability in order to comply with 14.3 (2).
- K braces are currently beyond scope for OCBF.
- Coincident V and A braces giving X type braced frames are out of scope for additional beam checks required by AISC 341-05 14.3.

- Bracing connections in the SFRS are assumed to satisfy the requirements of 14.4.
- OCBF above seismic isolation systems are currently beyond scope

SCBF

AISC 341-16 and AISC 341-10

- Column splices are assumed to comply with D2.5.
- Coincident V and A braces giving X type braced frames are out of scope for additional beam checks required by AISC 341-10 F2.3a.
- It is assumed that the force resisted by tension braces is between 30% and 70% of the total horizontal force along the line of braces as per F2.4a.
- It is assumed that the beams in SCBF are continuous between columns in accordance with F2.4b (1).
- It is assumed that the user will apply the relevant lateral restraint at position of V/A braces or establish that the beam has sufficient out of plane strength and stiffness to ensure stability in order to comply with F2.4b (2).
- K braces are not permitted for SCBF in accordance with F2.4c.
- Tension only braces are not permitted for SCBF in accordance with F2.4d.
- Bracing connections in the SFRS are assumed to satisfy the requirements of F2.5b.
- The protected zone is assumed to comply with F2.5c

AISC 341-05

- It is assumed that the force resisted by tension braces is between 30% and 70% of the total horizontal force along the line of braces as per 13.2c.
- Bracing connections in the SFRS are assumed to satisfy the requirements of 13.3.
- It is assumed that the user will ensure that the ends of V and A braces are vertically released so that they provide no support for dead and live loads as per 13.4a (1).
- It is assumed that the beams in SCBF are continuous between columns in accordance with 13.4a (2).
- Lateral braces are not designed to meet the additional criteria in 13.4a (2) - it is assumed that the user will check this independently.
- It is assumed that the user will apply the relevant lateral restraint at position of V/A braces or establish that the beam has sufficient out of plane strength and stiffness to ensure stability in order to comply with 13.4a.
- K braces are not permitted for SCBF in accordance with 13.4b.
- Coincident V and A braces giving X type braced frames are out of scope for additional beam checks required by AISC 341-05 13.4.

- Column splices are assumed to comply with 13.5. Column splices in columns not part of the SFRS are assumed to comply with 8.4b.
- The protected zone is assumed to comply with 13.6.

Design philosophy (Seismic: AISC 341)

All members are designed as per the normal AISC Specification rules for the seismic load combinations.

Additional design rules are required for seismic combinations. These are as per the AISC Seismic Provisions (AISC 341-05) (Ref. 3) or (AISC 341-10) (Ref. 4). These rules are applied as follows:

- If SDC = A - no additional requirements
- If SDC = D, E or F, apply rules for AISC 341

For each of Direction 1 and Direction 2:

- If SDC = B or C and R ≤ 3 - no additional requirements
- If SDC = B or C and R > 3, apply rules for AISC 341

Where requirements are necessary then they apply only to the members of the SFRS and are only checked for the seismic combinations.

Changes introduced in AISC 341-16 (Seismic AISC 341)

Design is available to AISC 341-05, AISC 341-10 and AISC 341-16. The changes between the 10 and 16 versions, as implemented in Tekla Structural Designer are listed below.

Column required strength (D1.4a) (applies to all SFRS)

- Only for 'overstrength seismic load'
- Previously called 'amplified seismic load'
- Capacity analysis' now omitted

OCBF and V&A braces (F1.4a)

- Tension yield strength/amplified seismic + post buckling strength has been replaced by overstrength seismic load + post buckling strength

OCBF and beam design (F1.5c)

- New requirement, beams in SFRS are now designed for overstrength seismic load

SMF Moment ratio (E3.4a) - strong column weak beam

- Enhanced plastic moment capacity of beams 1.1 R_y F_y Z has been replaced by maximum probable moment, Cpr/α_s R_y F_y Z. Cpr from AISC 358-16

Common seismic requirements (Seismic: AISC 341)

Required strength

The required strength (including overstrength effects) for a member should be determined from:

The expected yield stress $R_y \times F_y$

The expected tensile strength $R_t \times F_u$

Grade	F_y	R_y	R_t
A36	36	1.5	1.2
A53B	35	1.6	1.2
A500B	42	1.4	1.3
A500B	46	1.4	1.3
A500C	46	1.4	1.3
A500C	50	1.4	1.3
A501	36	1.4	1.3
A529	50	1.2	1.2
A529	55	1.1	1.2
A572	42	1.3	1.0[1]
A572	50	1.1	1.1
A572	55	1.1	1.1
A913	50	1.1	1.1
A913	60	1.1	1.1
A913	65	1.1	1.1
A992	50	1.1	1.1

[1]This value is 1.1 in AISC 341-05.

AISC 341-16 and AISC 341-10 seismic classification - all members

When required by the seismic checks, the classification of elements of the cross section for various member types is as follows.

Compiled from Table D1.1 of AISC 341-10

Section	Element	Width thickness ratio	Application	λ_{hd} – highly ductile	λ_{md} – moderately ductile
I (rolled)	Flange	$b_f/(2 * t_f)$	Beams, Columns, Braces	$0.30 * \sqrt{(E/F_y)}$	$0.38 * \sqrt{(E/F_y)}$
	Web	h/t_w	Braces	$1.49 * \sqrt{(E/F_y)}$	$1.49 * \sqrt{(E/F_y)}$
	Web	h/t_w	Columns, Beams	LRFD - $C_a = P_u/(\phi_c * F_yA_g)$ $\phi c = 0.9$ ASD - $C_a = c * P_a/(F_y * A_g)$ $\Omega_c = 1.67$ $C_a \leq 0.125$ $2.45 * \sqrt{(E/F_y)} * (1 - 0.93 * C_a)$ but for SMF only $\leq 2.45 * \sqrt{(E/F_y)}$ $C_a > 0.125$ $0.77 * \sqrt{(E/F_y)} \times (2.93 - C_a)$ but $\geq 1.49 * \sqrt{(E/F_y)}$	LRFD - $C_a = P_u/(\phi_c * F_yA_g)$ $\phi_c = 0.9$ ASD - $C_a = c * P_a/(F_y * A_g)$ $\Omega_c = 1.67$ $C_a \leq 0.125$ $3.76 * \sqrt{(E/F_y)} * (1 - 2.75 * C_a)$ but for IMF only $\leq 3.76 * \sqrt{(E/F_y)}$ $C_a > 0.125$ $1.12 * \sqrt{(E/F_y)} \times (2.33 - C_a)$ but $\geq 1.49 * \sqrt{(E/F_y)}$
RHS and SHS	Walls	$(b_f - 3 * t)/t$ and $(d - 3 * t)/t$	Braces	$0.55 * \sqrt{(E/F_y)}$	$0.64 * \sqrt{(E/F_y)}$
RHS and SHS	Walls	$(b_f - 3 * t)/t$ and $(d - 3 * t)/t$	Columns	$0.55 * \sqrt{(E/F_y)}$	$1.12 * \sqrt{(E/F_y)}$

CHS		D/t	Braces	0.038 * E/F_y	0.044 * E/F_y
CHS		D/t	Columns	0.038 * E/F_y	0.07 * E/F_y
C (rolled)	Flange	b_f/t_f	Braces	0.30 * √ (E/F_y)	0.38 * √ (E/F_y)
	Web	h/t_w	Braces	1.49 * √ (E/F_y)	1.49 * √ (E/F_y)
Tees	Flange	$b_f/(2 * t_f)$	Braces	0.30 * √ (E/F_y)	0.38 * √ (E/F_y)
	Stem	d/t_w	Braces	0.30 * √ (E/Fy)	0.38 * √ (E/Fy)
Angles	Both legs	L_1/t and L_2/t	Braces	0.30 * √ (E/F_y)	0.38 * √ (E/F_y)
Double angles	Outstand leg – legs in continuous contact	L_1/t (long leg B to B) or L_2/t (short leg B to B)	Braces	0.30 * √ (E/F_y)	0.38 * √ (E/F_y)
Double angles	Both legs – legs separated	L_1/t and L_2/t	Braces	0.30 * √ (E/F_y)	0.38 * √ (E/F_y)

In the above table the terms have their usual meaning as follows:

- b_f = width of flange and for RHS width of shorter side
- t_f = thickness of flange of I/H, channel or Tee
- h = height of web inside flanges (d – 2 * t_f) of I/H or channel
- t_w = thickness of web
- d = depth of SHS and for RHS depth of longer side
- t = thickness of hollow section RHS, SHS, CHS
- D = diameter of CHS
- L_1 = Short leg (from root to toe) of single angle
- L_2 = Long leg (from root to toe) of single angle
- E = modulus of elasticity of steel – 29000 ksi
- F_y = minimum yield stress
- P_u = required axial strength using LRFD (seismic) combinations
- P_a = required axial strength using ASD (seismic) combinations
- A_g = gross area of section

AISC 341-05 seismic classification - all members

When required by the seismic checks, the classification of elements of the cross section for various member types is as follows.

Compiled from I-8-1 of AISC 341-05 and Table B4.1 of AISC 360-05.

Section	Element	Width thickness ratio	Application	λ_{ps} – seismically compact	λ_{ps} – conventionally compact
I (rolled)	Flange	$b_f/(2 * t_f)$	Beams, Columns[1], Braces	$0.30 * \sqrt{(E/F_y)}$	$0.38 * \sqrt{(E/F_y)}$
	Web	h/t_w	Columns[2], Beams, Braces	LRFD - $C_a = P_u/(\phi_c * F_yA_g)$ $\phi_c = 0.9$ ASD - $C_a = c * P_a/(F_y * A_g)$ $\Omega_c = 1.67$ $C_a \leq 0.125$ $3.14 * \sqrt{(E/F_y)} * (1 - 1.54 * C_a)$ but for SMF only $\leq 2.45 * \sqrt{(E/F_y)}$ $C_a > 0.125$ $1.12 * \sqrt{(E/F_y)} * (2.33 - C_a)$ but $\geq 1.49 * \sqrt{(E/F_y)}$	$3.76 * \sqrt{(E/F_y)}$
RHS and SHS	Walls	$(b_f-3t)/t$ and $(d-3t)/t$	Columns, Braces	$0.64 * \sqrt{(E/F_y)}$	$1.12 * \sqrt{(E/F_y)}$
CHS		D/t	Columns, Braces	$0.044 * E/F_y$	$0.070 * E/F_y$
C (rolled)	Flange	b_f/t_f	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A

	Web	h/t_w	Braces	LRFD - $C_a = P_u/(\phi_c * F_yA_g)$ $\phi_c = 0.9$ ASD - $C_a = c * P_a/(F_y * A_g)$ $\Omega_c = 1.67$ $C_a \leq 0.125$ $3.14 * \sqrt{(E/F_y)} * (1 - 1.54 * C_a)$ $C_a > 0.125$ $1.12 * \sqrt{(E/F_y)} * (2.33 - C_a)$ but $\geq 1.49 * \sqrt{(E/F_y)}$	N/A
Tees	Flange	$b_f/(2 * t_f)$	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A
	Stem	d/t_w	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A
Angles	Both legs	L_1/t and L_2/t	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A
Double angles	Outstand leg – legs in continuous contact	L_1/t (long leg B to B) or L_2/t (short leg B to B)	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A
Double angles	Both legs – legs separated	L_1/t and L_2/t	Braces	$0.30 * \sqrt{(E/F_y)}$	N/A

Note 1: The relaxation on the compactness limit for columns in SMF as per note “b” to Table I-8-1 is not taken into account.

Note 2: These limits are not modified by Note [j] to Table I-8-1 i.e. this dispensation is not taken into account.

In the above table the terms have their usual meaning as follows:

- b_f = width of flange and for RHS width of shorter side
- t_f = thickness of flange of I/H, channel or Tee

- h = height of web inside flanges (d – 2 * t_f) of I/H or channel
- t_w = thickness of web
- d = depth of SHS and for RHS depth of longer side
- t = thickness of hollow section RHS, SHS, CHS
- D = diameter of CHS
- L_1 = Short leg (from root to toe) of single angle
- L_2 = Long leg (from root to toe) of single angle
- E = modulus of elasticity of steel – 29000 ksi
- F_y = minimum yield stress
- P_u = required axial strength using LRFD (seismic) combinations
- P_a = required axial strength using ASD (seismic) combinations
- A_g = gross area of section

Seismic checks - Beams (Seismic: AISC 341)

Classification
In all cases if the given "width to thickness ratio" is less than or equal to the given limit, then the seismic classification is satisfied.

AISC 341-16 and AISC 341-10

Beams in OMF and OCBF – No additional requirements.

Beams in IMF and SCBF – Beams must satisfy the requirements of clause D1.1b for "moderately ductile" members.

Beams in SMF – Beams must satisfy the requirements of clause D1.1b for "highly ductile" members.

The loading conditions affect the seismic classification in the following way,

- Axial tension only – no classification required.
- Any other loading condition – the appropriate rules in the section classification table are applied.

See: AISC 341-16 and AISC 341-10 seismic classification - all members (page 1709)

AISC 341-05

Beams in OMF, SCBF and OCBF – No additional requirements.

Beams in IMF – Beams must satisfy clause 8.2a i.e. the requirements for Compact sections to AISC 360-05 in Table B4.1.

Beams in SMF – Beams must satisfy clause 8.2b for "seismically compact" sections.

The loading conditions affect the seismic classification in the following way,

- Axial tension – no classification required.
- Major axis bending only – applies to beams in SMF only.
- Any other loading condition – apply appropriate rules in the section classification table.

See: AISC 341-05 seismic classification - all members (page 1711)

Stability bracing

AISC 341-16 and AISC 341-10

Beams in certain SFRS frame types must be provided with “stability bracing” to restrain lateral torsional buckling. There are two “levels” of requirement; one for ‘Moderately ductile members and one for “Highly ductile members”. The use of these depends upon the SFRS frame type as defined below.

Beams in OMF and OCBF – No additional requirements.

Beams in IMF and SCBF – Beams must satisfy clause D1.2a for “moderately ductile” members. For SCBF this only applies in the presence of V or A braces.

Beams in SMF – Beams must satisfy clause D1.2b for “highly ductile” members

Moderately Ductile

Beams shall be braced per D1.2a for moderately ductile members, i.e. maximum spacing per D1.2a(3)

$L_{pd} = 0.17 \times (E/F_y) \times r_y$

The design condition is,

$L_b \leq L_{pd}$

L_b = the laterally unbraced length of the compression flange taken as the beam length between locations where both the top flange and bottom flange are restrained for LTB.

Highly Ductile

Beams shall be braced per D1.2b for highly ductile members, i.e. maximum spacing,

$L_{pd} = 0.086 \times (E/F_y) \times r_y$

The design condition is,

$L_b \leq L_{pd}$

NOTE In both cases:

- The position of lateral braces is not checked for the location of points of concentrated force or positions of plastic hinge

- Lateral braces are not designed to meet the additional criteria for strength and stiffness.

AISC 341-05

OCBF and SCBF

Reference AISC 341-05 14.3 (2) and 13.4a (2) respectively.

For doubly symmetric I sections,

$L_{pd} = (0.12 + 0.076 \times (M_1/M_2)) \times (E/F_y) \times r_y$

Where

M_1 = the smaller moment at end of unbraced length – min(abs(M_a), abs(M_b))

M_2 = the larger moment at end of unbraced length – max(abs(M_a), abs(M_b))

NOTE Lateral braces are assumed to meet the strength and stiffness requirements of Equations A-6-7 and A-6-8 of AISC 360-05, Appendix 6.

NOTE Lateral braces are assumed to be provided at the intersection of the V/A brace and the beam or the beam has sufficient stiffness to satisfy the criteria in the “User Note” to AISC 341-05, 13.4a. In both cases the program assumes this intersection to be a braced point (with both flanges braced).

OMF

Reference AISC 341-05 11.8 – no additional requirements.

IMF

Reference AISC 341-05 10.8.

Max spacing of coincident restraints along top and bottom flange, continuous bracing is included while determining unbraced length for each flange,

$L_{pd} = 0.17 \times (E/F_y) \times r_y$

The design condition is,

$L_b \le L_{pd}$

L_b = the laterally unbraced length of the compression flange taken as the beam length between locations where both the top flange and bottom flange are restrained for LTB.

NOTE The position of lateral braces are not checked for the location of points of concentrated force or positions of plastic hinge.

NOTE Lateral braces are not designed to meet the additional criteria for strength and stiffness.

SMF

Reference AISC 341-05 9.8.

Max spacing of coincident restraints along the top and bottom flange, continuous bracing is included while determining the unbraced length for each flange,

$L_{pd} = 0.086 \times (E/F_y) \times r_y$

The design condition is,

$L_b \leq L_{pd}$

L_b = the laterally unbraced length of the compression flange taken as the beam length between locations where both the top flange and bottom flange are restrained for LTB.

NOTE The position of lateral braces are not checked for the location of points of concentrated force or positions of plastic hinge.

NOTE Lateral braces are not designed to meet the additional criteria for strength and stiffness.

Design for brace forces in SCBF and OCBF

In both variants of the code the beams in SCBF and OCBF that are configured with “V” or “A” braces are required to resist a “push-pull” force generated by the brace pair. In general terms the compression brace is assumed to retain a percentage of its resistance post-buckling and the tension brace is assumed to have a level of defined “overstrength”.

NOTE In meeting the requirements of F1.4 and F2.3, the design of beams is separated into two distinct approaches – those beams with V&A braces at mid-span and those with diagonal braces at their ends. Clearly, frames that are braced using V&A braces have to meet both requirements e.g. in chevron systems but each is checked individually and this is believed to be conservative.

AISC 341-16 and AISC 341-10, V and A braces

OCBF

Reference AISC 341-10 F1.4a.

NOTE The tension yield strength/amplified seismic + post buckling strength in AISC 341-10 is replaced by overstrength seismic load + post buckling strength in AISC 341-16.

NOTE The lower bound on the force in the tension brace, “The maximum force that can be developed by the system”, according to F1.4a (1) (i) (c) is not applied.

SCBF

Reference AISC 341-10 F2.3.

AISC 341-16 and AISC 341-10, all other braces

OCBF

No requirements.

SCBF

Reference AISC 341-10 F2.3.

The expected tension strength, the expected compression strength and the expected post buckling strength are determined in the same way as for V and A braces.

AISC 341-05, V and A braces

OCBF and SCBF

Reference AISC 341-05 14.3 and 13.4a respectively.

Seismic checks - Columns (Seismic: AISC 341)

Classification

In all cases if the given “width to thickness ratio” is less than or equal to the given limit, then the seismic classification is satisfied.

AISC 341-16 and AISC 341-10

Columns in OMF and OCBF – No additional requirements.

Columns in IMF – Columns must satisfy the requirements of clause D1.1b for “moderately ductile” members.

Columns in SMF and SCBF – Columns must satisfy the requirements of clause D1.1b for “highly ductile” members.

The loading conditions affect the seismic classification in the following way,

- Axial tension only – no classification required.

- Any other loading condition – the appropriate rules in the section classification table are applied.

See: AISC 341-16 and AISC 341-10 seismic classification - all members (page 1709)

AISC 341-05

Columns in OMF and OCBF – No additional requirements.

Columns in IMF – Columns must satisfy clause 8.2a i.e. the requirements for Compact sections to AISC 360-05 in Table B4.1.

Columns in SMF and SCBF – Columns must satisfy Clause 8.2b for “seismically compact” sections.

The loading conditions affect the seismic classification in the following way,

- Axial tension – no classification required.
- Any other loading condition – the appropriate rules in the section classification table are applied.

See: AISC 341-05 seismic classification - all members (page 1711)

AISC 341-16, D1.4a Required strength

This check consists of two analyzes of the column - one that uses the 'amplified seismic load' and a second that uses a 'capacity analysis' as an upper bound. The moments and shears in the column are ignored and the column is designed for axial load only.

NOTE While applying the column strength requirements of D1.4a, it is assumed that there are no loads applied to the column between locations of lateral support. Therefore applied moments are ignored and only the axial strength is considered as permitted in AISC 341-10 D1.4a(2).

NOTE While applying the column strength requirements of D1.4a (2), the upper limit on the required strength with respect to overturning uplift as per D1.4a (2) (b) is not applied.

Overstrength seismic load

P_{amp}, the required axial strength (either tension or compression) including the “overstrength seismic load” is given by,

$P_{amp} = P_r + f_E/\rho * P_E * (\Omega_o - \rho)$

Where;

P_r = the axial force (-ve for tension and +ve for compression) determined from the analysis of the seismic load combination (LRFD or ASD). This may be the result of a first or second order analysis.

P_E = the axial force (-ve for tension and +ve for compression) determined from the analysis of the seismic load case(s) associated with the seismic load combination.

f_E = the strength load factor associated with the seismic load in the seismic load combination (base combination factor x ρ) (for example, 0.683 in the ASD combination, D + 0.75 L + 0.75 Lr + 0.683 E).

ρ = the redundancy factor, ρ_1 when the column is assigned to Direction 1 and r2 when the column is assigned to Direction 2 (from the Seismic Wizard).

Ω_o = the overstrength factor, Ω_{o1} when the column is assigned to Direction 1 and Ω_{o2} when the column is assigned to Direction 2 (from the **Seismic Wizard...**).

NOTE The axial force from the load combination including the amplified seismic loads is calculated by swapping out the component due to the seismic loadcase $E_h = \rho\ Q_E$ and replacing it with the amplified seismic load, $E_{mh} = \Omega_E\ Q_E$.

Design condition

For each stack, the required axial strength, P_r is compared with the nominal axial strength, P_n, i.e. the design condition is,

$P_r \leq \phi \times P_n$ (LRFD) or P_n/Ω (ASD)

Where

P_n = the nominal axial strength in tension or compression as appropriate to the sign of P_r

ϕ = the resistance factor for tension or compression as appropriate

Ω = the safety factor for tension or compression as appropriate

AISC 341-10, D1.4a Required strength

This check consists of two analyzes of the column - one that uses the 'amplified seismic load' and a second that uses a 'capacity analysis' as an upper bound. The moments and shears in the column are ignored and the column is designed for axial load only.

NOTE While applying the column strength requirements of D1.4a, it is assumed that there are no loads applied to the column between locations of lateral support. Therefore applied moments are ignored and only the axial strength is considered as permitted in AISC 341-10 D1.4a(2).

NOTE While applying the column strength requirements of D1.4a (2), the upper limit on the required strength with respect to overturning uplift as per D1.4a (2) (b) is not applied.

Amplified seismic load

P_{amp}, the required axial strength (either tension or compression) including the "amplified seismic load" is given by,

$P_{amp} = P_r + f_E/\rho * P_E * (\Omega_o - \rho)$

Where;

P_r = the axial force (-ve for tension and +ve for compression) determined from the analysis of the seismic load combination (LRFD or ASD). This may be the result of a first or second order analysis.

P_E = the axial force (-ve for tension and +ve for compression) determined from the analysis of the seismic load case(s) associated with the seismic load combination.

f_E = the strength load factor associated with the seismic load in the seismic load combination (base combination factor x ρ) (for example, 0.683 in the ASD combination, D + 0.75 L + 0.75 Lr + 0.683 E).

ρ = the redundancy factor, ρ_1 when the column is assigned to Direction 1 and r2 when the column is assigned to Direction 2 (from the Seismic Wizard).

Ω_o = the overstrength factor, Ω_{o1} when the column is assigned to Direction 1 and Ω_{o2} when the column is assigned to Direction 2 (from the **Seismic Wizard...**).

NOTE The axial force from the load combination including the amplified seismic loads is calculated by swapping out the component due to the seismic loadcase $E_h = \rho\ Q_E$ and replacing it with the amplified seismic load, $E_{mh} = \Omega_E\ Q_E$.

Capacity analysis

At any level on a column there can be SFRS members and non-SFRS members. The principle of this check is that the former might be operating at their "capacity" in an earthquake and so they are likely to apply more force to the column than the global analysis would indicate. "Capacity" in this context also includes the possibility that the material is stronger than its specified yield (typical).

The capacity calculation involves establishing the capacities of the incoming SFRS members at each node (level) in the column – these might be zero if there are only non-SFRS members at that level. The capacities so determined are then resolved into the local x-axis of the column. The capacities are calculated for beams and braces only, not the columns themselves.

The end result of the capacity analysis is that for each stack there is an axial force, P_{cap} which can be compression or tension. This will be included in the results and used in this design check to D1.4a but will only govern if smaller than that from the "Amplified seismic load" analysis.

Design condition

For each stack, the required axial strength, P_r (the smaller of P_{cap} or P_{amp}) is compared with the nominal axial strength, P_n, i.e. the design condition is,

$P_r \leq \phi \times P_n$ (LRFD) or P_n/Ω (ASD)

Where

P_n = the nominal axial strength in tension or compression as appropriate to the sign of P_r

ϕ = the resistance factor for tension or compression as appropriate

Ω = the safety factor for tension or compression as appropriate

AISC 341-16 and -10, E3.4a Moment ratio

At each level in a column where an SMF beam connects into the strong axis of the column (i.e. into the flange), a check is performed for each seismic combination to ensure that the plastic moment capacity of the column is greater than the plastic moment capacity of the incoming beams.

The design condition is,

$\sum M_{pcol} / \sum M_{pbeam} > 1.0$

NOTE The exceptions in E3.4a (a) and (b) are ignored and the check is performed for all SMFs.

NOTE Beam column connections are always assumed braced as per E3.4c (1)

NOTE All beams with pinned connections are excluded in this calculation. Any beam with a moment connection into the web of the column is ignored even if they are assigned to a SMF. On the other hand, any beam with a moment connection to the column flange is included in the calculation even if they are not assigned to a SMF.

NOTE The additional moment due to shear amplification from the location of the plastic hinge to the column centre line (Muv and Mav) is calculated from two components,

1. the shear inferred by the moment at the plastic hinge position based on the expected flexural strength of the beam,

2. the shear force in the beam at the plastic hinge position from the factored gravity loads in the current seismic combination.

No account of angle of incoming members is taken into account in this calculation.)

AISC 341-16 and -10, F2.3 Analysis

This check applies to SCBF type frames only, the procedure being similar to that for 'Column Strength' to D1.4a.

NOTE The approach taken to the "capacity analysis" per F2.3 assumes that the SCBF is reasonably isolated. That is, the influence of the remainder of the structure due to the braces operating at their capacity does not adversely affect the required strength.

Design condition

Moments in the column are permitted to be ignored as per F2.3 (1) and so only the axial check (compression or tension) is required.

For each stack, the required axial strength, P_r, is compared with the nominal axial strength, P_n, i.e. the design condition is,

$P_r \leq \phi \times P_n$ (LRFD) or P_n/Ω (ASD)

Where P_r = the required axial strength, P_{cap}

P_n = the nominal axial strength in tension or compression as appropriate to the sign of P_r

ϕ = the resistance factor for tension or compression as appropriate

Ω = the safety factor for tension or compression as appropriate

NOTE While applying the column strength requirements of F2.3 (i) and (ii), it is assumed that there are no loads applied to the column between locations of lateral support. Therefore applied moments are ignored and only the axial strength is considered as permitted in AISC 341-10 F2.3 (1).

NOTE While applying the column strength requirements of F2.3 (i) and (ii), the upper limits on the required strength per F2.3 (2) (a), (b) and (c) are not applied.

NOTE It is assumed that braces do not carry significant gravity forces and therefore a separate analysis with braces omitted, in order to enhance the column gravity forces, is not carried out. [Ref. NEHRP Seismic Design Technical Brief No. 8].

AISC 341-05, 8.3 Required strength

The calculations for this check are exactly the same as those for the AISC 341-10, D1.4a Required Strength check except that they are only performed when the required axial force exceeds a certain limit as described below.

$P_r > 0.4 \times \phi_c \times P_n$ (LRFD)

$P_r > 0.4 \times P_n / \Omega_c$ (ASD)

Where

$\phi_c = 0.90$

$\Omega_c = 1.67$

P_n = the nominal axial strength of the stack in compression or tension as appropriate to the sign of P_r

Either,

$P_r = P_u$

= the maximum axial force in the stack from the current (LRFD) seismic combination

Or,

$P_r = P_a$

= the maximum axial force in the stack from the current (ASD) seismic combination

NOTE While applying the column strength requirements of 8.3, the upper limit on the required strength with respect to overturning uplift as per 8.3 (2) (b) is not applied.

AISC 341-05, 9.6 Moment ratio

At each level in a column where an SMF beam connects into the strong axis of the column (i.e. into the flange), a check is made to ensure that for each seismic combination the plastic moment capacity of the column is greater than the plastic moment capacity of the incoming beams. The calculations for this check are exactly the same as those for the AISC 341-10, E3.4a Moment Ratio check.

NOTE The exceptions in 9.6 (a) and (b) are ignored and the check is performed for all SMFs.

NOTE All beams with pinned connections are excluded in this calculation. Any beam with a moment connection into the web of the column is ignored even if they are assigned to a SMF. On the other hand, any beam with a moment connection to the column flange is included in the calculation even if they are not assigned to a SMF.

NOTE The additional moment due to shear amplification from the location of the plastic hinge to the column centre line (Muv and Mav) is calculated from two components,

(i) the shear inferred by the moment at the plastic hinge position based on the expected flexural strength of the beam,

(ii) the shear force in the beam at the plastic hinge position from the factored gravity loads in the current seismic combination. No account of angle of incoming members is taken into account in this calculation.

Seismic checks - Braces (Seismic: AISC 341)

Classification

In all cases if the given “width to thickness ratio” is less than or equal to the given limit, then the seismic classification is satisfied.

AISC 341-16 and AISC 341-10

Braces in OCBF – As per Clause F1.5a, braces must satisfy the requirements of clause D1.1b for “moderately ductile” members.

Braces in SCBF – As per Clause F2.5a, braces must satisfy the requirements of clause D1.1b for “highly ductile” members.

See: AISC 341-16 and AISC 341-10 seismic classification - all members (page 1709)

AISC 341-05

Braces in OCBF – As per Clause 14.2, braces must satisfy the requirements of clause 8.2b for 'seismically compact' members.

Braces in SCBF – As per Clause 13.2d, braces must satisfy clause 8.2b for “seismically compact” sections.

See: AISC 341-05 seismic classification - all members (page 1711)

Slenderness

AISC 341-16 and AISC 341-10

OCBF

In OCBF for V and A braces only, the design condition is checked for both major and minor axis as per F1.5b,

$KL/r \leq 4 * SQRT[E/F_y]$

Where

K = the effective length factor for the relevant axis

L = the system length of the brace

r = the radius of gyration of the brace for the relevant direction

E = modulus of elasticity of steel – 29000 ksi

F_y = minimum yield stress.

SCBF

For all braces in SCBF the design condition for both minor and major axis is checked as per F2.5b (1),

$KL/r \le 200$

Where

K = the effective length factor for the relevant axis

L = the system length of the brace

r = the radius of gyration of the brace for the relevant direction.

For built-up braces i.e. double angles the requirements for interconnection are checked as per F2.5b (2). The minimum number of connectors required by this clause is two and thus the maximum interconnection slenderness of the individual angles is based on a buckling length of one third of the system length, (which is conservative).

Thus,

$a/r_i \le 0.4 * MAX[KL/r]$

Where

a = the sub-length of the member between interconnections = taken as L/3

r_i = the minimum radius of gyration of the individual angle, taken as r_z

---

NOTE While checking the minimum slenderness of individual elements in built-up members to F2.5b (2), It is assumed the minimum number of shear connectors is provided i.e. two. The shear strength of the connectors is NOT checked against the tensile strength of each element.

NOTE The brace net area is NOT checked against the brace gross area as per F2.5b (3) and where this might be an issue suitable reinforcement is assumed to be provided.

AISC 341-05

OCBF

For V and A braces in OCBF the design condition for both minor and major axis is checked as per 14.2,

$KL/r \le 4 * SQRT[E/F_y]$

Where

K = the effective length factor for the relevant axis

L = the system length of the brace

r = the radius of gyration of the brace for the relevant direction.

SCBF

For all braces in SCBF there is a three stage design condition and both minor and major axis are checked as per 13.2a,

KL/r ≤ 4 * SQRT[E/F_y] PASS

KL/r > 200 FAIL

ELSE WARNING

“Brace slenderness satisfies, 4√(E/F_y) < KL/r ≤ 200. The available strength of the associated column is NOT checked as per 13.2a."

Where all variables are as given above.

For built-up braces i.e. double angles the requirements for interconnection are checked as per 13.2e. The minimum number of connectors required by this clause is two and thus the maximum interconnection slenderness of the individual angles is based on a buckling length of one third of the system length, (this will be conservative). Thus,

a/r_i ≤ 0.4 * MAX[KL/r]

Where

a = the sub-length of the member between interconnections = taken as L/3

r_i = the minimum radius of gyration of the individual angle, taken as r_z

NOTE While checking the minimum slenderness of individual elements in built-up members to 13.2e, it is assumed the minimum number of shear connectors is provided i.e. two. The shear strength of the connectors is NOT checked against the tensile strength of each element.

NOTE The brace net area is NOT checked against the brace gross area and where this might be an issue suitable reinforcement is assumed to be provided.

Brace strength

AISC 341-16 and AISC 341-10

OCBF

No additional requirements.

SCBF

Where the effective net area is less than the gross area the provisions of F2.5b (3) apply. This is more aimed at gusset plate connections where the cross section of the brace is reduced. The effective net area is specified by the user as a percentage or actual area.

The design condition should be (!),

$\phi_t * F_u * A_e \geq R_y * F_y * A_g$ LRFD

$F_u * A_e / \Omega_t \geq R_y * F_y * A_g / 1.5$ ASD

Where,

ϕ_t = resistance factor for tension

Ω_t = safety factor for tension

F_u = specified minimum tensile strength of steel

F_y = specified minimum yields stress of steel

A_e = effective area of brace (user input)

A_g = gross area of brace

R_y = the overstrength factor – see Section.

Note that for 50 ksi steel this will always fail but providing there is no reduction in area the brace is expected to yield. The Commentary in AISC 341 Comm. F2.5b indicates:

“Where there is no reduction in the section, or where the section is reinforced so that the effective net section is at least as great as the brace gross section, this requirement does not apply. The purpose of the requirement is to prevent net section fracture prior to significant ductility; having no reduction in the section is deemed sufficient to ensure this behavior.”

Consequently the design condition in Tekla Structural Designer is presented as follows, and considers the effective net area provided, $A_{e.prov}$, and the effective net area required, $A_{e.reqd}$, to satisfy F2.5b (3),

$A_{e.reqd} = MAX[A_g, (R_y * F_y * A_g/(F_u * \phi_t)]$ LRFD

$A_{e.reqd} = MAX[A_g, (R_y * F_y A_g * \Omega_t/(F_u * 1.5)]$ ASD

The design condition then becomes,

$A_{e.reqd} \leq A_{e.prov}$

AISC 341-05

OCBF

No additional requirements.

SCBF

The calculations for this check are exactly the same as those for the AISC 341-10 check.

NOTE The brace required strength to 13.2b is NOT limited to the "maximum load effect" as per 13.2b (b).

References (AISC 341)

1. **American Institute of Steel Construction**. ANSI/AISC 360-05 Specification for structural steel buildings. **AISC, 2005**.
2. **American Institute of Steel Construction**. ANSI/AISC 360-10 Specification for structural steel buildings. **AISC, 2010**.
3. **American Institute of Steel Construction**. ANSI/AISC 360-16 Specification for structural steel buildings. **AISC, 2016**.
4. **American Institute of Steel Construction**. ANSI/AISC 341-05 Seismic Provisions for Structural Steel Buildings. **AISC, 2006**.
5. **American Institute of Steel Construction**. ANSI/AISC 341-10 Seismic Provisions for Structural Steel Buildings. **AISC, 2010**.
6. **American Institute of Steel Construction**. ANSI/AISC 341-16 Seismic Provisions for Structural Steel Buildings. **AISC, 2016**.

Concrete design to ACI 318

When the Tekla Structural Designer head code is set to United States (ACI/AISC), you have the option to specify the Concrete Design Resistance Code as ACI 318 and the year as 2008, 2011 or 2014. If you are using US Customary Units the design is then performed in accordance with either ACI 318-08 (Ref. 1) (page 1843), ACI 318-11 (Ref. 3) (page 1843), or ACI 318-14 (Ref. 5) (page 1843). Design can also be performed for metric units in accordance with ACI 318M-08 (Ref. 2) (page 1843), ACI 318M-11 (Ref. 4) (page 1843), or ACI 318M-14 (Ref. 6) (page 1843)

Unless explicitly noted otherwise, all clauses, figures and tables used in the Reference Guides are from ACI 318-11, these have not yet been updated to reflect the new clause numbering in ACI 318-14.

The following topics are covered:

- Limitations (concrete members: ACI 318) (page 1730)
- Concrete beam design to ACI 318 (page 1731)
- Concrete column design to ACI 318 (page 1772)
- Concrete wall design to ACI 318 (page 1803)
- Concrete slab design to ACI 318 (page 1827)

- Pad and strip base design to ACI 318 (page 1827)
- Pile cap design to ACI 318 (page 1838)
- Seismic Design to ACI 318 (page 1840)

Limitations (concrete members: ACI 318)

The following general exclusions apply.

the current release will not:

- design members in lightweight concrete
- design members with coated reinforcement
- design members with stainless steel
- design prestressed concrete
- Consider fire resistance [you are however given full control of the minimum cover dimension to the reinforcement and are therefore able to take due account of fire resistance requirements],
- design structures subject to very aggressive exposure
- design watertight structures
- design multi-stack reinforcement lifts for columns/walls
- design beams as “deep beams” - beams classified as “deep” are designed as if they are regular beams and a warning is displayed.

NOTE Deep beams according to ACI 318 are:

(a) Members with clear spans equal to or less than 4 times overall member depth

(b) Members with concentrated loads within twice the member depth from the support

---

material limitations for concrete:

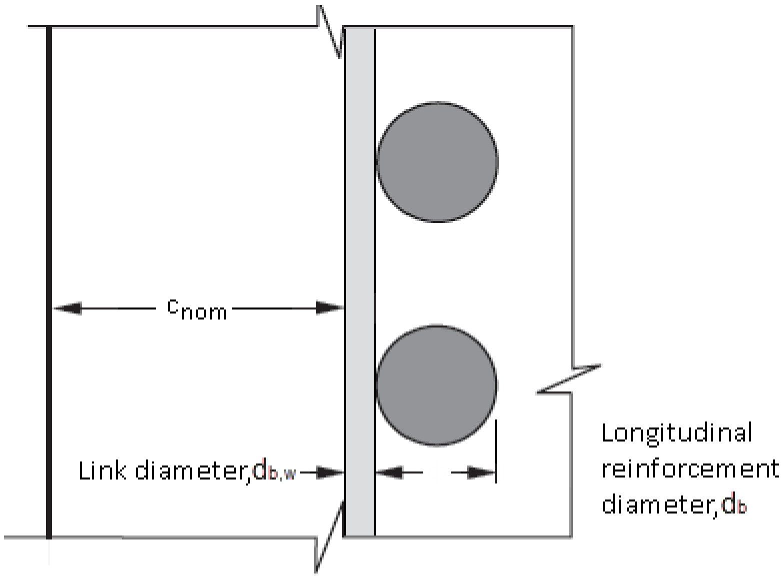
- for structural concrete compressive strength of concrete f_c' shall not be less than 17 MPa (2500psi)
- durability requirements are not implemented

material limitations for reinforcement:

- the values of specified yield strength of reinforcement; f_y and f_{yt} used in calculations shall not exceed 550 MPa (80000psi)
- specified yield strength of non-prestressed reinforcement; f_y and f_{yt} shall not exceed 420 MPa (60 000 psi) in design of shear or torsion reinforcement
- wire reinforcement design is not implemented

Cover to Reinforcement (ACI 318)

The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including ties and surface reinforcement where relevant) and the nearest concrete surface.



You are required to set a minimum value for the nominal cover, $c_{nom,\ u}$ for each member in the member properties.

These values are then checked against the nominal limiting cover, $c_{nom,lim}$

If $c_{nom,u} < c_{nom,lim}$ then a warning is displayed in the calculations.

Concrete beam design to ACI 318

The following topics are covered:

See also

Limitations (concrete members: ACI 318) (page 1730)

Seismic Design to ACI 318 (page 1840)

Limitations (concrete beam: ACI 318)

The following general exclusions apply.

the current release will not:

- design beams as “deep beams” - beams classified as “deep” are designed as if they are regular beams and a warning is displayed.

> **NOTE** Deep beams according to ACI 318 are:
>
> (a) Members with clear spans equal to or less than 4 times overall member depth
>
> (b) Members with concentrated loads within twice the member depth from the support

- design beams in lightweight concrete
- design beams with coated reinforcement
- design beams with stainless steel
- design prestressed concrete
- design structures subject to very aggressive exposure
- design watertight structures

material limitations for concrete:

- for structural concrete compressive strength of concrete f_c' shall not be less than 17 MPa (2500psi)
- durability requirements are not implemented

material limitations for reinforcement:

- the values of specified yield strength of reinforcement; f_y and f_{yt} used in calculations shall not exceed 550 MPa (80000psi)

- specified yield strength of non-prestressed reinforcement; f_y and f_{yt} shall not exceed 420 MPa (60 000 psi) in design of shear or torsion reinforcement
- wire reinforcement design is not implemented

Slender beams (ACI 318)

Spacing of lateral supports for a beam shall not exceed 50*b[1]

In the program the lateral supports are taken as the distance between the faces of the supports, and for simplification, b is taken as the web width b_w

If the above check fails then a 'Slender span' warning is displayed.

Effects of lateral eccentricity of load are considered in determining spacing of lateral supports.

Design parameters for longitudinal bars (ACI 318)

For each of these parameters, any user defined limits (as specified on the appropriate Reinforcement Settings page within Design Options) are considered in addition to any ACI code recommendations.

Minimum and maximum diameter of reinforcement

IF torsional reinforcement is required, there shall be at least one longitudinal bar in every corner of the stirrups. Longitudinal bars shall have a diameter at least 0.042 times the stirrup spacing, but not less than 9 mm (3/8 in).

The maximum diameters of reinforcement to be used in the various locations is set by the user.

Standard hooks for stirrups and ties are limited to No.8 bars, d_b=25mm (1.0in.) and smaller.

And the 90-degree hook with $6d_b$ extension is further limited to No. 5, d_b=16mm (0.625in.) bars and smaller.

For primary reinforcement there is no limit on bar size.

Minimum distance between bars

The minimum clear horizontal distance between individual parallel bars, $s_{cl,min}$, is given by;

$s_{cl,min} \geq$ MAX [d_b, 25 mm] metric-units

$s_{cl,min} \geq$ MAX [d_b, 1in] US-units

[1] ACI 318-08 and ACI 318-11 Section 10.4

IF the above check fails then a Warning is displayed.

Where parallel reinforcement is placed in two or more layers, bars in the upper layers shall be placed directly above the bars in the bottom layer with clear distance between layers not less than 25mm (1in.).

Maximum spacing of tension bars

The spacing of reinforcement closest to the tension face, s is given by;[1]

s	≤	MIN[380mm*280 MPa/fs-2.5*c_c, 300mm*(280MPa/ f_s)]	metric-units
s	≤	MIN[15in*40000psi/f_s-2.5*c_c, 12in*(40000psi/ f_s)]	US-units

where

c_c	=	the least distance from surface of reinforcement to the tension face
f_s	=	calculated stress in reinforcement at service load; it shall be permitted to take
	=	(2/3)*f_y*($A_{s,reqd}$/$A_{s,prov}$)

IF the above check fails then a Warning is displayed

IF torsional reinforcement is required:

the longitudinal reinforcement required for torsion shall be distributed around the perimeter of the closed stirrups with maximum spacing of 300mm (12 in.)[2]

Minimum area of beam reinforcement

The minimum area of longitudinal tension reinforcement, $A_{s,min}$, is given by;[3]

[1] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.6.4

[2] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.5.6.2

$A_{s,min} \geq MAX[(f'_c{}^{0.5}/(4*f_y))*b_w*d, 1.4MPa*b_w*d/ f_y]$ metric-units

$A_{s,min} \geq MAX[(3* f'_c{}^{0.5}/(f_y)*b_w*d, 200Psi*b_w*d/ f_y]$ US-units

where

f'_c		=	specified compressive strength of concrete
f_y		=	specified yield strength of reinforcement
b_w		=	web width; for statically determinate members with a flange in tension $b_w = MIN(2*b_w, b_{eff})$ [1]
d		=	distance from extreme compression fiber to centroid of longitudinal compression reinforcement

[1]: Assumption; the member is statically determinate in design

The above equation is used wherever reinforcement is needed, except where such reinforcement is at least one-third greater than that required by analysis, in which case $A_{s,min}$ not required.

Minimum area of slab reinforcement

For ACI 318-08 and ACI 318-11[5]

For structural slabs of uniform thickness the minimum area of tensile reinforcement in the direction of the span is:

For US-units:

IF Grade 40 to 50 deformed bars are used

$A_{s,min,reqd} \geq b*h*0.0020$

IF Grade 50 to 60 deformed bars or welted wire reinforcement are used

$A_{s,min,reqd} \geq b*h*0.0018$

For metric units:

IF Grade 280 to 350 deformed bars are used

$A_{s,min,reqd} \geq b*h*0.0020$

IF Grade 350 to 420 deformed bars or welted wire reinforcement are used

[3] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.5.1

[5] ACI 318-08 and ACI 318-11 and ACI 318M-11 Section 7.12.2.1 and 10.5.4

$A_{s,min,reqd} \geq b*h*0.0018$

IF yield stress exceeding 420 MPa

$A_{s,min,reqd} \geq b*h*[MAX(0.0014 , 0.0018*420/f_y)]$

Maximum area of reinforcement

Net tensile strain in extreme layer of longitudinal tension steel, ε_t should not be less than 0.004;

ε_t		0.004 $\geq$
$A_{s,max}$		$0.85*(f'_c / f_y)*\beta_1*b_w*d*[0.003/(0.003+0.004)]$ [1] $\leq$
		$0.85*(f'_c / f_y)*\beta_1*b_w*d*(3/7)$ $\leq$

[1]: Notes on ACI 318-08 Chap. 6 Section 10.3.5

where

A_g	=	the gross area of the concrete section	
β_1	=	stress block depth factor [1]	
metric units			
	=	0.85	for $f`_c \leq 28MPa$
	=	$0.85 - 0.05 *[(f'_c - 28MPa)/7MPa]$	for $28MPa < f`_c < 55MPa$
	=	0.65	for $f`_c \geq 55MPa$
US-units			
	=	0.85	for $f`_c \leq 4000$ psi

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.7.3

	=	0.85 - 0.05 *[(f'$_c$ - 4ksi)/1ksi]	for 4000 psi < f`$_c$ < 8000psi
	=	0.65	for f`$_c$ ≥ 8000 psi

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.7.3

Side skin reinforcement in beams (ACI 318)

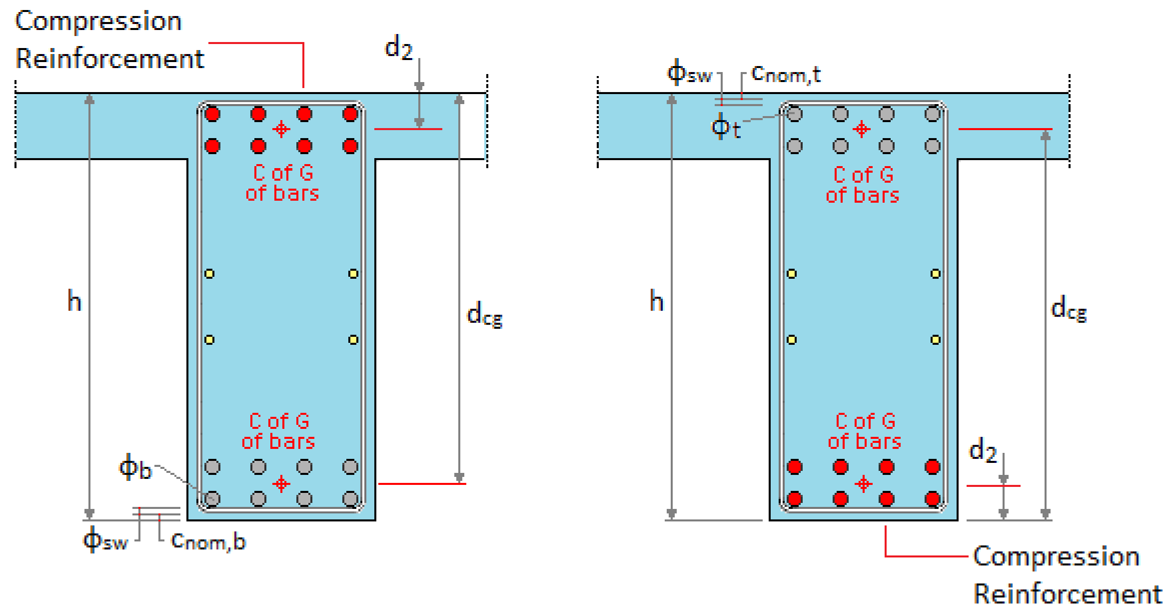
Where h of a beam or joist exceeds 900mm (36 in.), longitudinal skin (side) reinforcement is uniformly distributed along both side faces of the member.

The code requires that skin reinforcement shall extend for a distance h/2 from the tension face. Regardless of this in the current release of Tekla Structural Designer the skin reinforcement is provided to the full height of the beam.

Effective depth of section (concrete beam: ACI 318)

For the design of the longitudinal tension reinforcement, the effective depth of a section, d is defined as the distance from the extreme concrete fiber in compression to the center of gravity of the longitudinal tension reinforcement.

For the design of the longitudinal compression reinforcement, the effective depth in compression, d_2 is defined as the distance from the extreme fiber in compression to the center of gravity of the longitudinal compression reinforcement.



Tension Reinforcement in Bottom of Beam   Tension Reinforcement in Top of Beam

Design for bending for rectangular sections (beams and slabs: ACI 318)

Determine if compression reinforcement is needed

Nominal strength coefficient of resistance is given;[1]

R_n	=	M_u / (φ*b*d_2)
where		
M_u	=	factored moment at section
d	=	depth to tension reinforcement
b	=	width of the compression face of the member
φ	=	strength reduction factor[1]
	=	0.9 (corresponds to the tension-controlled limit)

[1]: ACI 318-08 and ACI 318-11 Section 9.3

[1] Notes on ACI 318-08 Chap 7.

IF R_n	$\leq$	R_{nt}	THEN compression reinforcement is not required.
IF R_n	$>$	R_{nt}	THEN compression reinforcement is required.

where			
R_{nt}	=	Limit value for tension controlled sections without compression reinforcement for different concrete strength classes[1]	
	=	ωt *(1-0.59 ωt)* $f`_c$	
$f`_c$	=	compressive strength of concrete	
ω_t	=	0.319*b1	
β_1	=	stress block depth factor [2]	
metric units			
	=	0.85	for $f`_c \leq 28MPa$
	=	0.85 - 0.05 *[(f'_c - 28MPa)/7MPa]	for 28MPa < $f`_c$ < 55MPa
	=	0.65	for $f`_c \geq 55MPa$
US-units			
	=	0.85	for $f`_c \leq$ 4000 psi

[1]: Notes on ACI 318-08 Section 10.3.4
[2]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.7.3

	=	0.85 - 0.05 *[(f'_c - 4ksi)/1ksi]	for 4000 psi < $f`_c$ < 8000psi
	=	0.65	for $f`_c \geq$ 8000 psi

[1]: Notes on ACI 318-08 Section 10.3.4
[2]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.7.3

Compression reinforcement is not required

The tension reinforcement ratio is given by;

$\rho = 0.85*f`_c/f_y * [1-(1- 2*R_n/\sqrt{}(0.85*f`_c))] \leq \rho t = 0.319*b_1*f`_c / f_y$

where

f_y = yield strength of reinforcement

The area of tension reinforcement required is then given by;

$A_s = \rho*b*d$

The area of compression reinforcement required is then given by;

$A_s` = M_n`/[(d-d`)* f_s`]$

where

$M_n` = M_n-M_{nt}$

$= (M_u/\varphi)-M_{nt}$

M_{nt} = nominal moment resisted by the concrete section[5]

$= R_n*b*d_2$

The area of tension reinforcement required is then given by[6];

$A_s = A_s`* f_s`/f_y + \rho*b*d$

where

$f_s` = MIN[E_s*(\varepsilon_u*(c-d`)/c) , f_y]$

$\rho = \rho_t*(d_t/d)$

$\rho_t = 0.319*b_1*f`_c / f_y$

$\varepsilon_u = 0.003$[7]

$c = 0.375*d_t$

[5] Notes on ACI 318-08 Section 10.3.4

[6] Notes on ACI 318-08 Chap. 7

[7] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.3

Design for bending for flanged sections (beams: ACI 318)

IF $h_f < 0.5*b_w$ THEN treat the beam as rectangular[1]

where

b_w = web width

Depth of the equivalent stress block is given [2];

$a = \rho*d*f_y/ (0.85*f'_c)$ (= $1.18*\omega*d$)

where

$\rho = 0.85*f'_c/f_y * [1-(1- 2*R_n/\sqrt{}(0.85*f'_c)]$

$R_n = (M_u/\varphi) / (b_{eff}*d_2)$ assumption $\varphi = 0.9$

IF $a \le h_f$ THEN the rectangular compression block is wholly in the depth of the flange and the section can be designed as a rectangular section with tension reinforcement only by setting $b = b_{eff}$ and checking the φ-factor as followed;

IF $(a/\beta_1)/d < 0.375$ THEN $\varphi=0.9$ (section tension controlled)

IF $0.375 > (a/\beta_1)/d < 0.600$ THEN $\varphi=0.7 + (\varepsilon_t -0.002 *(200/3)$

IF $(a/\beta_1)/d > 0.6$ THEN $\varphi=0.65$ (section comp. controlled)

where

$\varepsilon_t = [(d*\beta_1)/a-1]*0.003$

IF $a > h_f$ THEN the rectangular compression block extends into the rib of the flanged section and the following design method is to be used;

Required reinforcement is given;

$A_{sf} = 0.85* f`_c*(b_{eff}-b)*h_f / f_y$

Nominal moment strength of **flange**;

$M_{nf} = [A_{sf}*f_y(d-h_f /2)]$

Required nominal moment strength to be carried by the beam **web** is given;

$M_{nw} = M_u - M_{nf}$

Can be written as;

$M_{nw} = M_u - [(0.85* f`_c*(b_{eff}-b)*h_f / f_y)*f_y(d-h_f /2)] = M_u - [(0.85* f`_c*(b_{eff}-b)*h_f)*(d-h_f /2)]$

Reinforcement A_{sw} required to develop the moment strength to be carried by the web;

$A_{sw} = \omega_w* f`_c*b*d / f_y$

---

[1] ACI 318-08 and ACI 318-11 Section 8.12.4

[2] Notes on ACI 318-08 Section 7 (1)

where

$\omega_w = \rho * f_y / f`_c = 0.85 * f`_c / f_y * [1-(1- 2*(M_{nw} / (b*d_2))/\sqrt{(0.85*f`_c)}] * f_y / fc`$

Can be written as;

$A_{sw} = b*d*0.85*f`_c/f_y*[1-(1- 2*(M_{nw} / (b*d_2))/\sqrt{(0.85*f`_c)}]$

Total required reinforcement is given;

$As = A_{sf} + A_{sw}$

Check to see if the section is tension-controlled;

IF

$\rho_w \le \rho_t$ section is tension- controlled (φ=0.9)

ELSE add compression reinforcement where

$\rho_w = \omega_w f`_c / f_y$ $\rho_t = 0.319 * \beta_1 * f`_c / f_y$

Can be simplified as;

$\omega_w \le 0.319 * \beta_1$ section is tension-controlled (φ=0.9)

ELSE add compression reinforcement

Shear strength (beams: ACI 318)

Determine shear strength provided by the concrete[1];

Members subject to axial compression not applied at this stage

φV_c	=	$\varphi*0.17*\lambda*\sqrt{(f`_c)}*b_w*d$	metric-units
	=	$\varphi*2*\lambda*\sqrt{(f`_c)}*b_w*d$	US-units
where			
φ	=	0.75 for shear[1]	

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 9.3.2.3
[2]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 8.6.1
[3]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.1.2.1

[1] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.2.1.1

λ		1.0 for normal weight concrete[2] =
$f`_c^{0.5}$		square root of specified compressive strength of concrete[3] =

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 9.3.2.3
[2]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 8.6.1
[3]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.1.2.1

NOTE If the structure is defined as a joist construction V_c shall be permitted to be 10% more than that specified in above[5].

IF

$V_u - \varphi V_c$		$\varphi*0.66*\sqrt{(f`_c)}*b_w*d$[1] ≤	metric-units
		$\varphi*8*\sqrt{(f`_c)}*b_w*d$ =	US-units

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.7.9

where

V_u		the maximum design shear force acting anywhere on the beam =

THEN the shear design process can proceed.

ELSE the shear design process FAILS since the section size or strength of the concrete is inadequate for shear. No further shear calculations are carried out in the region under consideration and the user is warned accordingly.

The design shear capacity of the minimum area of shear links actually provided, $V_{s,min}$ is given by[7];

[5] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.1.2.1

[7] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.7.2

$V_{s,min} = (A_{v,min}/s)*\varphi*d*f_{yt}$

where

$A_{v,min}$ is the area of shear reinforcement provided to meet the minimum requirements.

For each beam determine the following;

$V_{u,maxL}$ = the maximum vertical shear force at the face of the left hand support

$V_{u,dL}$ = the vertical shear force at a distance *dL* from the face of the left hand support

$V_{u,maxR}$ = the maximum vertical shear force at the face of the right hand support

$V_{u,dR}$ = the vertical shear force at a distance *dR* from the face of the right hand support

$V_{u,S2L}$ = the maximum vertical shear force at the extreme left of region S2

$V_{u,S2R}$ = the maximum vertical shear force at the extreme right of region S2

where

dL = the minimum effective depth of the beam in regions T1 and B1

dR = the minimum effective depth of the beam in regions T5 and B3

In any region, *i*;

IF

$V_{u,i} \leq V_{s,min} + \varphi V_c$

where

$V_{u,i}$ = the maximum shear in region *i* from the above routines

OR

The structure is defined as a joist construction[8].

THEN

Minimum shear reinforcement shall be used;

And the nominal shear strength is given;

$\varphi V_n = \varphi V_c + V_{s,min}$ [9]

ELSE

$V_{u,i} > V_{s,min} + \varphi V_c$

THEN shear links are required in the region.

The area of shear reinforcement required is then given[10];

[8] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.6.1 - Terms (d) and (e) not applied.

[9] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.1

metric-units;

$(A_v /s)_{Si}$ = MAX[$(V_u$- $φ*V_c)$/ $(φ*f_{yt}*d)$, 0.062* $f`_c0.5$ $*b_w /f_{yt}$, 0.35Pa$*b_w/f_{yt}$]

US-units;

((A_v /s)$_{Si}$ = MAX[$(V_u$- $φ*V_c)$/ $(φ*f_{yt}*d)$, 0.75* $f`_c0.5$ $*b_w /f_{yt}$, 50psi$*b_w/f_{yt}$]

V_s = ((A_v/s)*φ*d*f_{yt}

IF

Vs ≤ $0.66*f'_c{}^{0.5}*b_w*d$[11] (metric -units) $8*f'_c{}^{0.5}*b_w*d$ (US-units)

THEN the shear design process passes.

And the nominal shear strength is given;

$φV_n = φV_c + V_s$

ELSE the shear design process FAILS since the section size or strength of the concrete is inadequate for shear.

Minimum area of shear reinforcement (beams: ACI 318)

The minimum area of shear reinforcement required, Av,min is given by[1];

	$A_{v,min}$		MAX (0.062* $f`_c{}^{0.5}$ $*b_w*s/ f_{yt}$, 0.35MPa$*b_w*s/ f_{yt}$, $A_{v,min,u}$) =	metric-units
			MAX (0.75* $f`_c{}^{0.5}$ $*b_w*s/ f_{yt}$, 50psi$*b_w*s/ f_{yt}$, $A_{v,min,u}$) =	US-units
	where			
	s		the spacing of the shear reinforcement along the longitudinal axis of the beam	
	f_{yt}		yield strength of transverse reinforcement =	
	$A_{v,min,u}$		the total minimum area of the shear reinforcement calculated from data supplied by the user i.e. maximum	

[10] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.6.3

[11] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.7.9

			spacing across the beam, minimum link diameter and number of legs. i.e. maximum spacing across the beam, minimum link diameter and number of legs	

Spacing of shear reinforcement (beams: ACI 318)

For Longitudinal spacing, *s* between the legs of shear reinforcement is given by[1];

IF

V_u - φ*V_c ≤ φ*0.33* $f`_c^{0.5}$ *b_w*d metric-units

φ*4* $f`_c^{0.5}$ *b_w*d US-units

THEN

$s_{min,u}$ ≤ s ≤ MIN[0.5*d, 600mm (24in.), $s_{max,u}$]

ELSE

$s_{min,u}$ ≤ s ≤ MIN[0.25*d, 300mm (12in.), $s_{max,u}$]

where

$s_{max,u}$ = the maximum longitudinal spacing specified by the user

$s_{min,u}$ = the minimum longitudinal spacing specified by the user

Moreover IF **compression reinforcement** is required the compression reinforcement shall be enclosed by ties[2]. This is an additional limit, not an alternative.

Vertical spacing of ties is then given by[3];

s ≤ MIN(16*d_b, 48*$d_{b,w}$, b_w, h)

where

d_b = the nominal diameter of the bar

$d_{b,w}$ = the nominal diameter of the link reinforcement

NOTE Unlike other design codes, ACI 318 does not specify a limit for maximum spacing of link legs across a beam. However, attention is drawn to an ACI Structural Journal Technical Paper - “Shear Reinforcement Spacing in Wide Members”, which suggests a limit of around "d".

[1] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.6.3

[1] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 11.4.5

[2] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 7.11.1

[3] ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 7.10.5.2

Deflection check (beams: ACI 318)

Deflection checks are divided between two deflection types: Immediate short-term deflections and long-term deflections which are resulting from creep and shrinkage of flexural members.

Two methods are given for controlling deflections:

1. By limiting span to depth ratio

For beams provision of a minimum overall thickness (min. total depth) as required by the following table satisfies the requirements of the code for members not supporting or attached to partitions or other construction likely be damaged by large deflections.

Support Conditions	Minimum thickness, h_t
Simply Supported	$l_n/16$
One end continuous	$l_n/18.5$
Both ends continuous	$l_n/2126$
Cantilever	$l_n/8$

If h		h_{min}	the design passes and no further calculations are required≥	

where

h		overall height of member =	
h_{min}		$h_t * f_{y,mod}$ =	
h_t		minimum thickness from above table =	
l_n		clear span length =	

$f_{y,mod}$		$0.4 + f_y$ / 700 MPa =	metric-units	

		0.4 + f_y / 100000 psi =	US-units	

If the deflection check fails the rigorous method below is used.

2. By calculating deflections using the rigorous method

For beams that do not meet minimum thickness requirements above, or that support or are attached to partitions or other constructions likely be damaged by large deflections, deflections are calculated by following method.

1. Firstly, the beam's cracked section moment of inertia, Icr is calculated.
2. Then the cracking moment Mcr is calculated.
3. The Long Term Deflection Period is read from the user specified value in Design settings. - 3 months to 5 years (default value: 5 years).
4. The Time at which brittle finishes are introduced is read from the user specified value in Design settings. - 1 month to 6 month (default value: 1 month).
5. For each loadcase with type = "dead", the % of load applied prior to sensitive finishes is read from the user specified value in the loadcase dialog (default value: 50%).
6. For each loadcase with type = "imposed", the % of load which is long term is read from the user specified value in the loadcase dialog (default value: 33%).
7. For each span in the element the critical gravity combination is determined from the analysis. The combination reporting the max relative deflection is the one considered in the deflection check.
8. The maximum deflections for the different situations below can then be determined:
    - Dead load deflection $(\Delta_i)_d$
    - Dead and live load deflection $(\Delta_i)_{dlive}$
    - Live load deflection $(\Delta_i)_{live}$
    - Sustained load deflection $(\Delta_i)_{sus}$
    - Total load deflection$(\Delta_i)_{tot}$
    - Deflection affecting sensitive finishes $(\Delta_i)_{af}$

The check passes if the calculated deflections are less than the deflection limits specified in the beam properties.

Seismic design and detailing (beams: ACI 318)

For overall limitations and assumptions, see:

- Limitations (beams seismic: ACI 318) (page 1749)

For beam design in moment resisting frames, see:

- General requirements (beams seismic: ACI 318) (page 1761)
- Flexural requirements (beams seismic: ACI 318) (page 1763)
- Transverse reinforcement (beams seismic: ACI 318) (page 1766)

For beams not part of a SFRS, see:

- Requirements when in SDC D - F (beams seismic: ACI 318) (page 1770)
- Seismic cantilevers (beams seismic: ACI 318) (page 1770)

For seismic detailing, see:

- Flexural reinforcement (beams seismic: ACI 318) (page 1771)
- Confinement reinforcement for ductility (beams seismic: ACI 318) (page 1771)

See also

Seismic Design to ACI 318 (page 1840)

Limitations (beams seismic: ACI 318)

The following limitations and assumptions apply.

- Seismic design is only performed for beams marked as part of a Seismic Force Resisting System and for seismic cantilevers.
- Requirements for beams particularly in the case of members not part of any SFRS when in Seismic Design Categories D through F are not considered in the current release.
- The design and detailing requirements of members part of Special Moment Frames is beyond scope (some checks are implemented but only due to their existence in lower toughness systems).

  NOTE A full list of the code checks that have and have not been implemented is provided in the table below.

- Seismic design checks are mostly based on capacity design obtained from the main reinforcement provided. This can lead to an over-design of structural members if the designer does not take steps to minimize excess capacity.
- Beam seismic design and detailing in the current release is based on the beam rectangular section and takes under consideration the beam reinforcement only. In particular cases allowances for the slab presence and reinforcement might be required on top of the current design.

- Seismic design and detailing requirements for structural diaphragms according to ACI318-11 sections 21.3.6 and 21.11 are not considered in the current release.

ACI 318 Seismic Code Checks for beams that have been implemented in Tekla Structural Designer

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.1.4.2	Minimum required compressive strength of concrete	SMF	-	-	-	✓
21.1.4.3	Maximum allowed compressive strength of light-weight concrete	SMF	-	-	-	✓
21.1.5.2	Maximum allowed steel characteristic yield strength of longitudinal reinforcement	SMF	-	-	-	✓
21.1.5.5	Maximum allowed longitudinal reinforcement yield strength used in the	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	calculation of transverse reinforcement					
21.1.6.2	Mechanical Splices within twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.1.6.2	Mechanical Splices outside twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.1.7.1	Welded Splices within twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.1.7.2	Welding of stirrups	SMF	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	or other elements to longitudinal reinf. required by design					
21.2.2	Minimum number of bars at top/ bottom faces continuous throughout	OMF	-	✓	-	-
21.2.2	Minimum number of bars at top/ bottom faces continuous throughout	IMF	-	-	✓	-
21.3.2	Maximum allowed factored axial force	IMF	-	-	✓	-
21.3.3.1	Minimum Design shear force	IMF	-	-	✓	-
21.3.4.1	Minimum +ve requirement (moment strength/ steel) at a joint face	IMF	-	-	✓	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.3.4.1	Minimum moment strength anywhere on a beam	IMF	-	-	✓	-
21.3.4.2	Type of transverse reinforcement in confinement regions (hook/ extension)	IMF	-	-	✓	-
21.3.4.2	Length of support regions measured from the face of the joint	IMF	-	-	✓	-
21.3.4.2	Maximum hoop spacing in support regions	IMF	-	-	✓	-
21.3.4.2	Maximum distance between first hoop and joint face in support regions	IMF	-	-	✓	-
21.3.4.3	Maximum hoop spacing outside confinem	IMF	-	-	✓	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	ent regions					
21.5	Beams of Special Moment Frames will frame into columns of SMF	SMF	-	-	✓	-
21.5.1.1	Maximum allowed factored axial force	SMF	-	-	✓	-
21.5.1.2	Maximum allowed effective depth	SMF	-	-	-	✓
21.5.1.3	Maximum allowed width	SMF	-	-	-	✓
21.5.1.4	Maximum allowed width	SMF	-	-	-	✓
21.5.2.1	Minimum number of bars at top/ bottom faces continuous throughout	SMF	-	-	-	✓
21.5.2.1	Minimum allowed area of reinforcement at	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	top/ bottom face throughout					
21.5.2.1	Maximum allowed area of reinforcement: Max steel ratio at top/ bottom/ side face of the beam, ρmax	SMF	-	-	-	✓
21.5.2.2	Minimum +ve requirement (moment strength / steel) at a joint face	SMF	-	-	-	✓
21.5.2.2	Minimum moment strength anywhere on a beam	SMF	-	-	-	✓
21.5.2.3	Lap splice location restrictions	SMF	-	-	-	✓
21.5.2.3	Lap Splice transverse reinforcement type	SMF	-	-	-	✓
21.5.2.3	Maximum	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	allowed hoop spacing at lap splices					
21.5.3.1	Length of support regions measured from the face of the joint	SMF	-	-	-	✓
21.5.3.1	Non-reversing plastic hinges: Flexural Yield region size (centered)	SMF	-	-	-	✗
21.5.3.2	Maximum hoop spacing in support regions	SMF	-	-	-	✓
21.5.3.2	Maximum distance between first hoop and joint face in support regions	SMF	-	-	-	✓
21.5.3.2	Non-reversing plastic hinges: Maximum horizonta	SMF	-	-	-	✗

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	l center spacing					
21.5.3.3	Maximum allowed spacing of flexural reinforcing bars	SMF	-	-	-	✓
21.5.3.3	Maximum allowed lateral link leg spacing in confinement regions	SMF	-	-	-	✓
21.5.3.4	Maximum hoop spacing outside confinement regions	SMF	-	-	-	✓
21.5.3.6	Type of transverse reinforcement in confinement regions (hook/ extension)	SMF	-	-	-	✗
21.5.3.6	Type of transverse reinforcement in beam sections that	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	extend laterally beyond the column core (hook/ extension)					
21.5.4.1	Minimum Design shear force	SMF	-	-	-	✓
21.5.4.2	Unreinforced concrete shear resistance at confinement regions	SMF	-	-	-	✓
21.7.2.1	Stress in the beam flexural tensile reinforcement at joints face for joint shear calculation	SMF	-	-	-	✓
21.7.2.2	Tension anchorage length of beam long. Reinf. at external joints (beyond column	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	inner face)					
21.7.2.3	Maximum longitudinal reinforcement bar size	SMF	-	-	-	✓
-	Minimum beam depth at a joint where it contributes to joint shear	SMF	-	-	-	✓
21.7.3.3	Spacing of confinement reinf. for longitudinal bars of beams outside the column core	SMF	-	-	-	✗
21.7.3.3	Maximum distance between link legs in beam sections that extend laterally beyond the column core	SMF	-	-	-	✗
21.7.5.1	Development	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	length for beam longitudinal bars with a standard 90° hook					
21.7.5.2	Development length for beam longitudinal straight bars	SMF	-	-	-	✓
21.7.5.4	Development length for epoxy-coated or zinc and epoxy dual-coated beam longitudinal bars	SMF	-	-	-	✓
21.8.2	Minimum distance from joint face for beam reinf. mechanical splices in ductile connections	SMF	-	-	-	✗
21.8.2	Minimum nominal shear strength of ductile connections	SMF	-	-	-	✗

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.8.3	Minimum nominal strength of the strong connection	SMF	-	-	-	✖
ASCE7/10 12.4.4	Extra design loads for horizontal cantilevers	-	-	-	-	✓

NOTE

- For further details of the checks that have been implemented, see: General requirements (beams seismic: ACI 318) (page 1761), Flexural reinforcement (beams seismic: ACI 318) (page 1771), Transverse reinforcement (beams seismic: ACI 318) (page 1766) or consult the respective clause reference in the code.
- Most of the requirements will be fulfilled through automatic design. In some cases specific design options will need to be set by the user.
- Additional requirements may apply to members that are not part of the SFRS when in SDC's D, E or F
- Confinement regions: - support regions; - Probable flexural yield regions; - Lap splice regions.

General requirements (beams seismic: ACI 318)

Maximum allowed factored axial force

Flexural elements with high axial loading values under any load combination are handled in the seismic design and detailing as compressive members.

If SFRS Type = Ordinary Moment Frame, then no axial compression limit applies.

If SFRS Type = Intermediate Moment Frame or SFRS Type = Special Moment Frame

	P_{max}	=	$A_g * f'_c / 10$
	where		

	P_{max}	=	Maximum allowed compression value on the member
	A_g	=	Gross area of the concrete section
	f'_c	=	specified compressive strength of concrete

The check passes if;

	P_u	≥	P_{max}
	where		
	P_u	=	Maximum factored compressive axial force anywhere in the span considering all load combinations

Maximum allowed effective depth

The maximum allowed effective depth of a beam part of a Special Moment Frame is proportional to its clear span length to limit the overloading of the adjacent joints and columns.

If SFRS Type = Special Moment Frame

	d		d_{max} ≤
	where		
	d		= Distance between the extreme compression fiber and the longitudinal tension reinforcement centroid
	d_{max}		= Maximum allowed distance between the extreme compression fiber and the longitudinal tension reinforcement centroid
	d_{max}		$0.25 * l_n$

			=
	where		
	l_n		Length of the clear span measured from face-to-face of supports =

Minimum allowed width

Beams part of Special Moment Frames in buildings subjected to earthquake effects have a minimum width limit for their web.

If SFRS Type = Special Moment Frame

	b_w		$b_{w,min}$ ≥	
	where			
	b_w		Beam web width =	
	$b_{w,min}$		Minimum allowed beam web width =	
			MAX(0.3 * h, 250 mm) =	Metric-units
			MAX(0.3 * h, 10 in) =	US-units
	where			
	h		Overall depth of the concrete section =	

Maximum allowed width

Despite not being advisable, beams in Special Moment Frames are allowed to be wider than the supporting columns up to a fixed limit.

NOTE The maximum lateral extension of a beam on each side of the joining column is beyond scope in the current release of Tekla Structural Designer.

Flexural requirements (beams seismic: ACI 318)

Minimum number of bars

The minimum allowed number of bars continuous along the beam span is required to be checked in the layers closest to the top and bottom faces of any beam in the SFRS.

The number of bars should be ≥ 2.

Maximum allowed bar size

This applies to end regions of beams where the beam reinforcement extends into the column core. The required development length of reinforcement bars extending into the column core restricts the minimum size of the column and vice-versa.

If SFRS Type = Special Moment Frame

“Anchorage requirements at the joint of special moment frames limit the maximum bar size at each end of the beam”

Minimum flexural strength

The minimum area of top and bottom steel required at any section of a beam part of a Moment Resisting Frame needs to comply with flexural strength requirements when considering earthquake effects.

Note that no seismic design requirements apply to beams that are part of Ordinary Moment Frames. All other Moment Resisting Frame types have minimum longitudinal moment requirements.

Minimum allowed area of reinforcement

The minimum allowed area of steel throughout the bottom and top faces of a beam part of a Special Moment Frame is limited as per ACI 318-11 equation (10-3).

Maximum allowed area of reinforcement

For the purpose of increasing the ductility response of beams in Special Moment Frames the area of reinforcement both at the top and bottom faces is limited.

If SFRS Type = Special Moment Frame

	(A_s- and As +)		$A_{s,max} = 0.025 * b_w * d$ ≤
	where		
	A_s		Area of non-prestressed longitudinal tension reinforcement

	b_w		Beam web width =
	d		Distance from extreme compression fiber to centroid of longitudinal tension reinforcement

No other Moment Resisting Frame type has a maximum area of steel requirement.

Maximum allowed center spacing of longitudinal bars

Limitations on the longitudinal bar spacing apply to beams part of Special Moment Frames.

If SFRS Type = Special Moment Frame, then the maximum allowed center spacing is checked for confinement regions as follows:

	$s_{cr,max}$		350 mm =	Metric-units
	A_s		14 in. =	US-units
	where			
	$s_{cr,max}$		maximum allowed center spacing =	

Non-reversing plastic hinges

Non-reversing plastic hinges are regions along the span of the beam where flexural yielding is likely to occur.

NOTE Non-reversing plastic hinges are beyond scope in the current release of Tekla Structural Designer.

Splices

Restrictions apply to the locations of reinforcement lap splices along the span of a beam part of Special Moment Frames.

Strength design of mechanical splices and restrictions to the use of welded splices as required by ACI318-11 apply to Special Moment Frames.

NOTE	These restrictions are not implemented in the current release of Tekla Structural Designer.

Transverse reinforcement (beams seismic: ACI 318)

Seismic requirements relating to transverse reinforcement take into account properties, strengths and outcomes which are shear related ignoring any reinforcement intended to deal with torsional effects.

Design shear force

The design shear force for members subjected to earthquake effects is obtained by consideration of the minimum required shear strength of the member. The required nominal shear strength of a flexural member part of a Moment Resisting Frame is checked considering the sum of shears resultant from the moment strengths due to reverse curvature bending acting at each end of the beam and from the tributary factored gravity loads.

Beams are checked for shear in three regions:

- Left region, S1;
- Central region, S2;
- Right region, S3.

Shear design is performed considering the Major axis shear force only. Shear Force in the minor axis is checked against the ignorable threshold.

If SFRS Type = Ordinary Moment Frame, then no shear seismic check applies.

If SFRS Type = Intermediate Moment Frame

	V_e		$\phi MIN(V_{e,Mn}+V_{e,gravity},V_{e,2E})$ =
	where		
	ϕ		Strength reduction factor = 1.0 =
	V_e		Minimum design shear force for load combinations including earthquake effects
	$V_{e,gravity}$		Shear due to factored gravity loads from seismic combinations (including vertical earthquake effects) retaining the sign from analysis
	$V_{e,2E}$		Maximum shear resultant from seismic combinations, withdoubled earthquake effect [i.e.: $V_{e.non\text{-}seismic} + V_{e.E} \times 2$]

			=
	$V_{e,Mn}$		Maximum shear associated with the development of reversed curvature bending due to nominal resisting moments at both ends of the member, considering both the clockwise and counter-clockwise cases

If SFRS Type = Special Moment Frame

	V_e		$\phi(V_{e,Mpr}+V_{e,gravity})$ =
	where		
	ϕ		Strength reduction factor = 1.0 =
	V_e		Minimum design shear force for load combinations including earthquake effects
	$V_{e,gravity}$		Shear due to factored gravity loads from seismic combinations (including vertical earthquake effects) retaining the sign from analysis
	$V_{e,Mpr}$		Maximum shear associated with the development of reversed curvature bending due to the probable flexural moment strength for both the clockwise and counter-clockwise situations, at both ends of the member

Maximum hoop spacing

The maximum allowed horizontal center spacing of hoops in confinement regions of beams is limited by ACI 318 depending on the type of Seismic Force Resisting System considered.

NOTE This check is performed for support regions only, it is beyond scope in the current release of Tekla Structural Designer for other confinement regions.

NOTE Non-reversing plastic hinge regions along the span have the same requirements as support regions, but these are beyond scope in the current release of Tekla Structural Designer.

For Support Regions:

If SFRS Type = Special Moment Frame

The maximum allowed center hoop spacing in support regions, $s_{cr,max,sup}$ is calculated as follows:

According to ACI318-08[1]:

	$s_{cr,max,sup}$	=	MIN(d/4, 8 * $d_{b,smallest}$, 24 * $d_{b,w}$, 300mm)	Metric-units
	$s_{cr,max,sup}$	=	MIN(d/4, 8 * $d_{b,smallest}$, 24 * $d_{b,w}$, 12 in.)	US-units

According to ACI318-11[1]:

	$s_{cr,max,sup}$	=	MIN(d/4, 6* $d_{b,smallest}$, 150 mm)	Metric-units
	$s_{cr,max,sup}$	=	MIN(d/4, 6 * $d_{b,smallest}$, 6 in.)	US-units

	where		
	d	=	Distance from extreme compression fiber to centroid of longitudinal tension reinforcement
	$d_{b,smallest}$	=	Smallest longitudinal reinforcement bar diameter
	$d_{b,w}$	=	Link (hoop) diameter

If SFRS Type = Intermediate Moment Frame

The maximum allowed center hoop spacing in support regions, $s_{cr,max,sup}$ is calculated as follows:

	$s_{cr,max,sup}$	=	MIN(d/4, 8* $d_{b,smallest}$, 24 * $d_{b,w}$, 300mm)	Metric-units

[1] ACI318-08 Section 21.5.3.2. This requirement has changed from ACI318-08 to ACI318-11

	$s_{cr,max,sup}$		MIN(d/4, 8 * $d_{b,smallest}$, 24 * $d_{b,w}$, 12 in.) =	US-units

	where		
	d		Distance from extreme compression fiber to centroid of longitudinal tension reinforcement
	$d_{b,smallest}$		Smallest longitudinal reinforcement bar diameter =
	$d_{b,w}$		Link (hoop) diameter =

For Span Regions:

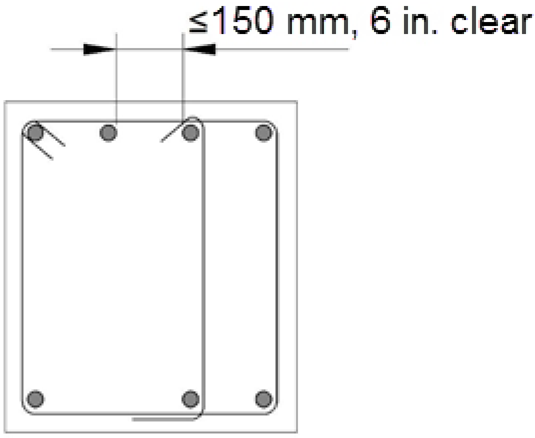
If SFRS Type = Special Moment Frame, or Intermediate Moment Frame

The maximum allowed hoops spacing outside confinement regions, $s_{cr,max,span}$ is calculated as follows:

	$s_{cr,max,span}$		d/2 =

Maximum allowed lateral link leg spacing

The clear spacing between link legs at right angles to the span is limited in confinement reinforcement regions of members part of Special Moment Frames only.



NOTE This check is performed for support regions only, it is beyond scope in the current release of Tekla Structural Designer for other confinement regions.

Requirements when in SDC D - F (beams seismic: ACI 318)

When designing members for earthquake effects, beams not part of the SFRS when in Seismic Design Categories D through F are required to be designed with seismic provisions all the same.

NOTE With the exception of seismic cantilevers, the design of these members for seismic provisions is beyond scope in the current release of Tekla Structural Designer.

Seismic cantilevers (beams seismic: ACI 318)

Horizontal cantilever structural members in structures assigned to Seismic Design Category D, E or F are required to be designed to the applicable load combinations plus an isolated minimum net upward force of 0.2* times the dead load.

If a cantilever beam has been marked as a seismic cantilever, then provided the seismic design category = D, E or F the minimum design moment at the restrained end is checked as follows:

Calculate minimum positive design moment:

	Mmin +		0.2 * $M_{e,dead}$- =
	where		
	M_{min}+		Minimum positive design bending moment at the restrained end. =

	$M_{e,dead}$-		Critical negative bending moment at the restrained end due to dead loads only obtained for the considered seismic combination.

Check the minimum design moment at the restrained end:

	M_u+		M_{min}+ ≥
	where		
	M_u+		Critical negative bending moment at the restrained end due to dead loads only obtained for the considered seismic combination.

If the check fails the region is designed for M_u+ = M_{min}+

Flexural reinforcement (beams seismic: ACI 318)

Longitudinal reinforcement at the top and bottom faces of a beam in any Moment Resisting Frame are required to have at least two continuous steel bars along the span for structural integrity and constructability purposes.

Anchorage

Longitudinal reinforcement terminated at a column in beams that are part of Special Moment Frames shall be anchored within the element confined core for a length measured from the critical section at the element's face.

Tekla Structural Designer performs the calculation steps for the required development length at the supports of bars in tension for both the case where straight bars are used and where hooks are provided.

Lap Splices

Specific seismic requirements apply only to lap splices in flexural members that are part of Special Moment Frames.

For beams that are part of Special Moment Frames lap splices are not allowed to be located:

- Within joints or within a distance of twice the members depth from the face of a joint
- Within regions where flexural yielding is likely to occur.

NOTE The latter requirement to avoid lap splices in regions where flexural yielding is likely to occur is beyond scope in the current release of Tekla Structural Designer

Confinement reinforcement for ductility (beams seismic: ACI 318)

Reinforcement Type

Confinement reinforcement should consist of hoops, i.e. closed or continuously wound ties with a seismic hook at each end.

Detailing Regions

Confinement reinforcement is required to be provided over the following confinement regions in beams that are part of Intermediate and Special Moment Frames. This is not a requirement for beams that are part of Ordinary Moment Frames

- Support regions: These are probable flexural yielding regions flexural yielding regions next to beam-column joints

  NOTE Beam-wall moment frames are beyond scope in the current release of Tekla Structural Designer.

- Non-reversing plastic hinge regions: These are probable flexural yielding regions outside support regions.

  NOTE Non-reversing plastic hinge regions are not identified in the current release of Tekla Structural Designer.

- Lap splices: Over the full length of lap splices in members that are part of Special Moment Frames

Concrete column design to ACI 318

The following topics are covered:

See also

Limitations (concrete members: ACI 318) (page 1730)
Seismic Design to ACI 318 (page 1840)

Design parameters for longitudinal bars (concrete column: ACI 318)

For some of the longitudinal reinforcement design parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Column > Reinforcement Layout.

Minimum Longitudinal Bar Spacing

For design to ACI ;

minimum clear distance		MAX (1.5 * longitudinal db , 1.5in., 1.33*hagg) $\geq$	US units
minimum clear distance		MAX (1.5 * longitudinal db , 40mm, 1.33*hagg) $\geq$	metric units

where

d_b	=	bar diameter
h_{agg}	=	aggregate size

Maximum Longitudinal Bar Spacing

You are given control over this value by specifying an upper limit in Design Options > Column > Reinforcement Layout.

Minimum Longitudinal Total Steel Area

For design to ACI;

1% * column area

Maximum Longitudinal Total Steel Area

For design to ACI;

8% * column area

Ultimate Axial Load Limit (column and wall:ACI 318)

The strength of a column under truly concentric axial load is

P_{no}	=	$0.85*f'_c*(A_g - A_{st}) + f_y*A_{st}$

For nonprestressed compression members with tie reinforcement,

φPnmax	=	$0.80*φ*[0.85*f'_c*(A_g - A_{st}) + f_y*A_{st}]$

Effective length calculations (column and wall:ACI 318)

Unsupported Length

The unsupported length, l_u, of a column is the clear distance between lateral supports.

If, at an end of the compression member (stack), no effective beams, flat slab or slab on beams to include is found, then the clear height includes the (compression member) stack beyond this restraint, and the same rules apply for finding the end of the clear height at the end of the next stack (and so on). If there is no stack beyond this restraint (i.e. this is the end of the column), the clear height ends at the end of the column.

Effective Length

The effective length, l_e is calculated automatically from ACI R10.10.1. You have the ability to override the calculated value.

Tekla Structural Designer will impose the following limits for stacks that are designated as braced:

$0.5 \leq l_e / l_u \leq 1$

When both ends of an unbraced compression member are hinged (pinned), a “Beyond Scope” warning is displayed.

The effective length of the stack (compression member) is given by:

$l_e = k * l_u$

The program uses the bottom end of the stack (compression member) as end 1 and the top as end 2.

Any beams framing into the end of the compression member (stack) within 45 degrees of the axis being considered are said to be restraining beams for the stack in that direction. No adjustment is made to the restraint provided by the beam for the angle (i.e. the full value of “E * I / l” is used for all beams within 45 degrees of the axis).

A beam is to be considered as a restraining beam in the direction considered if:

$-45° < β \leq 45°$

where

β is the angle from the axis in the direction considered to the beam, measured anti-clockwise when viewed from above (i.e. back along the length of the stack from the end towards the start).

Fixed Column Base

Since in practical structures there is no such thing as a truly fixed end, Tekla Structural Designer limits ψ ≥ 0.20. This being the practical lower limit suggested in “RC Mechanics” by McGregor and Wright.

Pinned Column End

In any situation where the end of a column anywhere in the structure is pinned, ψ = 1000. (This being the upper limit on ψ that is imposed by Tekla Structural Designer.)

No Effective Beams Found

If no effective beams are found to restrain the end of the stack in the direction in question, then the program will consider whether there is a flat slab restraining the stack at this end. If a flat slab is found it will either be considered as a restraint, or not, in each direction at each end of the stack - this is controlled by checking the option Use slab for calculation... located as a Stiffness setting in the column properties.

If there are no effective beams and there is no flat slab (or any flat slab is not to be considered), then the program will look for a slab on beams. If a slab on beams is found, this acts as a restraint at the position. Slabs on beams will only be considered if the “Use slab for calculation...” option is selected, as is the case for flat slabs.

If no beams and no flat slab or slab on beams is found, then the program will look for the far end of the stack on the other side of the joint, and look at the restraints there, and so on until a restraint with an effective beam, flat slab or slab on beams to be considered is found.

If the stack is restrained by a flat slab, then the slab will be considered to act as a beam in this direction - note that it is one beam in the direction and NOT a beam on each side of the column. The beam's length is taken as four times its width.

If the stack is restrained by a slab on beams, this will have a zero contribution to the stiffness. This thoretically has the effect of setting ψ = infinity, though it is limited to 1000 in Tekla Structural Designer before being used in the calculations.

If the stack is an end stack and there are no supports, beams, flat slabs or slabs on beams considered to restrain the stack at this end in the direction, the end is therefore free in this direction and ψ = 1000.

Column stack and wall panel classification (column and wall:ACI 318)

Slenderness ratio

For columns: The slenderness ratio, k lu/r, of the restrained length (note: not necessarily the stack length – it will be longer if there is no restraint at either end of the stack) about each axis is calculated as follows:

$(k\ lu/r)_y = k * lu_y / (\sqrt{(I_y / A_g)})$

$(k\ lu/r)_z = k * lu_z / (\sqrt{(I_z / A_g)})$

where

slenderness ratio = k*lu/r

k is an effective length factor

lu_y is the unsupported column length in respect of major axis (y axis)

lu_z is the unsupported column length in respect of minor axis (z axis)

r_y is the radius of gyration of the column in the y-direction

r_z is the radius of gyration of the column in the z-direction

I_y is the second moment of area of the stack section about the major axis (y axis)

I_z is the second moment of area of the stack section about the major axis (z axis)

A_g is the cross-sectional area of the stack section

For unbraced columns

IF $(k\ lu/r)_y \leq 22$

THEN slenderness can be neglected and column can be designed as short column

ELSE, column is considered as slender

IF $(k\ lu/r)_z \leq 22$

THEN slenderness can be neglected and column can be designed as short column

ELSE, column is considered as slender

For braced columns

IF $(k\ lu/r)_y \leq$ MIN((34-12*M1/M2), 40)

THEN slenderness can be neglected and column can be designed as short column

ELSE, column is considered as slender

IF $(k\ lu/r)_z \leq$ MIN((34-12*M1/M2), 40)

THEN slenderness can be neglected and column can be designed as short column

ELSE, column is considered as slender

where

M1 = the smaller factored end moment on the column, to be taken as positive if member is bent in single curvature and negative if bent in double curvature

= MIN [ABS(M_{top}), ABS(M_{bot})]

M2 = the larger factored end moment on the column always taken as positive

= MAX [ABS(M_{top}), ABS(M_{bot})]

Design Moment Calculations (column and wall: ACI 318)

For each combination and for each analysis model (Building Analysis, Grillage Analysis, FE Analysis) the end moments about the two local member axes, 'major' and 'minor' are established. From these and the local load profile, the moments and axial force at any position in the member can be established. These moments will be from a first-order or second-order analysis at user choice - (in making the choice, the value of the 'stability index', Q should be taken into account).

Note that M2 and M1 are the end moments with M2 being the larger numeric value.

Step 1, minimum moment

Calculate the minimum moment due to non-concentric axial force in each of the two directions from,

M_{min}	=	Pu*(0.6+0.03*h)] in	(US units)
	=	Pu*(15+0.03*h)] mm	(metric units)

where

h	=	The major dimension of the column in the direction under consideration
P_u	=	The max compression force at any design position in the stack under consideration. If stack is in tension set to zero

Step 2 - member slenderness

It is determined whether the member is slender or not. Note that in the determination for braced columns M1 and M2 are always the end moments even if lateral loading is present.

Step 3 - non-slender column

Calculate the design moment at the top, bottom and mid-fifth of the column in each direction taking into account if lateral loads are “significant”, or “not significant”.

As the column is non-slender no further calculations are required to establish design moments.

Step 4 - slender member amplifier

Calculate the “amplifier” due to buckling about each of the major and minor axes excluding the uniform moment factor which is dealt with separately,

k_{ns}	=	$1/(1 - (P_u / (0.75*Pc))) \geq$ zero

where

P_c	=	The critical buckling load	
	=	$\pi^2*(EI)/(k*lu)^2$	

where EI can be computed by Eq. (10-14) or Eq (10-15)

Step 5 - uniform moment factor

For lateral loads that are “not significant”,

	C_m		0.6 + 0.4 *(M1 / M2)	

retaining moment signs

Else

	C_m		1.0 =

Step 6 - moment magnifier

Calculate the moment magnifier from Equ. 10.12 as,

d_{ns}	=	MAX [C_m * k_{ns}, 1.0]

Step 7 - amplified minimum moment

Calculate the amplified minimum moment as,

M_{min_amp}	=	M_{min} * k_{ns}

Step 8 - design moments

Calculate the design moment at the top, bottom and mid-fifth of the column in each direction taking into account if lateral loads are "significant", or "not significant".

Design for combined axial and bending (column and wall:ACI 318)

Tekla Structural Designer designs the column for an applied axial force and applied bending about one or both axes of the section. In the case of bi-axial bending, a resultant moment is created for the combination of the applied moments.

$$\sqrt{(M^2_{major} + M^2_{minor})} \;/\; \sqrt{(M^2_{major,res} + M^2_{minor,res})} \le 1.0$$

Where

M_{major}	=	Moment about the major axis
M_{minor}	=	Moment about the minor axis
$M_{major,res}$	=	Moment of resistance about the major axis
$M_{minor,res}$	=	Moment of resistance about the minor axis

Design parameters for shear (column and wall:ACI 318)

For some of the shear design parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Column > Reinforcement Layout.

Minimum Shear Link Diameter

For Ties, minimum shear reinforcement size

IF maximum longitudinal bar ≤ 1.27 in. (32.3mm)

shear reinforcement diameter = 0.375 in. (9.5mm)

Minimum shear reinforcement diameter = 0.50 in. (12.7mm)

Maximum Span Region Shear Link Spacing

Controlled by seismic detailing requirements.

Maximum Support Region Shear Link Spacing

Controlled by seismic detailing requirements.

Column confinement (column and wall:ACI 318)

The ACI requirement is that every alternate longitudinal bar should be restrained by a link corner or bar tie, and no bar should have more than 6" (150 mm) clear distance from a restrained bar.

Seismic design and detailing (columns: ACI 318)

For overall limitations and assumptions, see:

- Limitations and assumptions (columns seismic: ACI 318) (page 1781)

For column design in moment resiting frames, see:

- General requirements (columns seismic: ACI 318) (page 1794)
- Flexural requirements (columns seismic: ACI 318) (page 1796)
- Transverse reinforcement (columns seismic: ACI 318) (page 1799)

For seismic detailing, see:

- Flexural reinforcement (columns seismic: ACI 318) (page 1802)
- Confinement reinforcement for ductility (columns seismic: ACI 318) (page 1803)

See also

Seismic Design to ACI 318 (page 1840)

Limitations and assumptions (columns seismic: ACI 318)

The following limitations and assumptions apply.

- Seismic design is only performed for columns marked as part of a Seismic Force Resisting System.
- Requirements for columns particularly in the case of members not part of any SFRS when in Seismic Design Categories D through F are not considered in the current release.
- The design and detailing requirements of members part of Special Moment Frames is beyond scope (some checks are implemented but only due to their existence in lower toughness systems).

  NOTE A full list of the code checks that have and have not been implemented is provided in the table below.

- No height limitations apply to Seismic Force Resisting Systems in the form of Moment Resisting Frames according to ASCE7-10, Table 12.2-1.
- The use of spiral reinforcement as well as all seismic design checks and related assumptions are not considered due to the fact that this type of reinforcement is not currently available in Tekla Structural Designer.
- The Seismic Force Resisting System set by the user in each direction through the Seismic Wizard for analysis purposes is not checked for applicability against the allowed types of the resultant Seismic Design Category. This is a user responsibility.

- Seismic design checks are mostly based on capacity design obtained from the main reinforcement provided. This can lead to an over-design of structural members if the designer is not careful enough to minimize excess capacity, especially in columns considering the weak beam - strong column philosophy.

ACI 318 Seismic Code Checks for columns that have been implemented in Tekla Structural Designer

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.1.4.2	Minimum required compressive strength of concrete	SMF	-	-	-	✓
21.1.4.3	Maximum allowed compressive strength of light-weight concrete	SMF	-	-	-	✓
21.1.5.2	Maximum allowed steel characteristic yield strength of longitudinal reinforcement	SMF	-	-	-	✓
21.1.5.4	Maximum yield strength of transverse reinforcement in confinem	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	ent regions of columns					
21.1.5.5	Maximum allowed longitudinal reinforcement yield strength used in the calculation of transverse reinforcement	SMF	-	-	-	✓
21.1.5.5	Maximum allowed characteristic yield strength of shear reinforcement	SMF	-	-	-	✓
21.2.3	Design shear force (page 1799)	OMF	-	✓	-	-
21.3.2	Minimum factored axial force (page 1794)	IMF	-	-	✓	-
21.3.3.2	Design shear force (page 1799)	IMF	-	-	✓	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.3.5.2	Minimum Support Region size (confinement reinforcement applies)	IMF	-	-	✓	-
21.3.5.2	Maximum allowed center hoop spacing (page 1799) in confinement regions	IMF	-	-	✓	-
21.3.5.3	First hoop placing distance from face of joint in support regions	IMF	-	-	✓	-
21.3.5.3	First hoop placing distance from face of joint in support regions	SMF	-	-	-	✓
21.3.5.5	Minimum depth of column transverse reinforcement into the joint	IMF	-	-	✓	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.3.5.5	Minimum area of column rectangular transverse reinforcement	IMF	-	-	✓	-
21.3.5.5	Minimum depth of column transverse reinforcement into the joint	SMF	-	-	-	✓
21.3.5.5	Minimum volumetric ratio / area of column spiral or circular transverse reinforcement	IMF	-	-	✓	-
21.3.5.6	Length of confinement region when inside footings, mats or pile caps	IMF	-	-	✓	-
21.3.5.6	Length of confinement region for columns supporting/above discontin	IMF	-	-	✓	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	uous stiff members (walls)					
21.6.1	Minimum factored axial force (page 1794)	SMF	-	-	-	✓
21.6.1.1	Minimum overall dimension	SMF	-	-	-	✓
21.6.1.2	Minimum shortest dimension to perpendicular dimension ratio	SMF	-	-	-	✓
21.6.2.2/21.6.2.3	Minimum flexural strength (page 1796)	SMF	-	-	-	✓
21.6.3.1	Minimum allowed area of reinforcement	SMF	-	-	-	✓
21.6.3.1	Maximum allowed area of reinforcement (page 1802)	SMF	-	-	-	✓
21.6.3.2	Minimum allowed number of bars in columns	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	with circular hoops					
21.6.3.3	Lap splice allowed locations	SMF	-	-	-	✓
21.6.3.3	Mechanical Splices within twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.6.3.3	Mechanical Splices outside twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.6.3.3	Welded Splices within twice the member depth from column/ beam face or yielding regions	SMF	-	-	-	✖
21.6.3.3	Welding of stirrups	SMF	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	or other elements to longitudinal reinf. required by design					
21.6.3.3	Minimum lap splice length	SMF	-	-	-	✓
21.6.4.1	Minimum Support Region size (confinement reinforcement applies)	SMF	-	-	-	✓
21.6.4.1	Minimum length of confinement region at other flexural yielding sections	SMF	-	-	-	✗
21.6.4.2	Type of confinement reinforcement (hook/ extension)	SMF	-	-	-	✓
21.6.4.2	Maximum allowed cross section center link leg spacing in confinem	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	ent regions					
21.6.4.3	Maximum allowed center hoop spacing (page 1799) in confinement regions	SMF	-	-	-	✓
21.6.4.3	Maximum hoop spacing at lap Splices (page 1796)	SMF	-	-	-	✗
21.6.4.4a)	Minimum volumetric ratio / area of spiral or circular confinement reinforcement	SMF	-	-	-	✓
21.6.4.4b)	Minimum area of rectangular confinement reinforcement	SMF	-	-	-	✓
21.6.4.5	Maximum center link/ stirrup spacing in non special	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	transverse reinf. regions					
21.6.4.6	Length of confinement region when inside footings, mats or pile caps	SMF	-	-	-	✓
21.6.4.6	Length of confinement region for columns supporting/above discontinuous stiff members (walls)	SMF	-	-	-	✓
21.6.4.7	Spacing of transverse reinforcement for non-structural extensions	SMF	-	-	-	✗
21.6.5.1	Design shear force (page 1799)	SMF	-	-	-	✓
21.6.5.2	Unreinforced Shear resistance at confinement regions	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.7.3.1	Minimum volumetric ratio / area of column spiral or circular transverse reinforcement	SMF	-	-	-	✓
21.7.3.1	Minimum area of column rectangular transverse reinforcement	SMF	-	-	-	✓
21.7.3.1	Type of column transverse reinforcement (hook/ extension)	SMF	-	-	-	✓
21.7.3.1	Maximum allowed column cross section center link leg spacing	SMF	-	-	-	✓
21.7.3.1	Maximum allowed center hoop spacing	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	(page 1799)					
21.7.3.1	Spacing of column transverse reinforcement for non-structural extensions	SMF	-	-	-	✖
21.7.3.2	Minimum column transverse reinf.with beams in all directions ≥ 3/4 the column's width	SMF	-	-	-	✖
21.7.4.1	Maximum nominal shear strength for joints confined by beams on all four faces	SMF	-	-	-	✓
21.7.4.1	Maximum nominal shear strength for joints confined by beams on three faces or two	SMF	-	-	-	✓

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	opposite faces					
21.7.4.1	Maximum nominal shear strength for joints not confined by beams	SMF	-	-	-	✓
21.8.3	Minimum nominal strength of the strong connection for column-to-column connections	SMF	-	-	-	✗
21.8.3	Minimum nominal moment strength of the strong connection for column-to-column connections	SMF	-	-	-	✗
21.8.3	Minimum nominal shear strength of the strong connection for column-	SMF	-	-	-	✗

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	to-column connections					

NOTE

- For further details of the checks that have been implemented, see: General requirements (columns seismic: ACI 318) (page 1794), Flexural reinforcement (columns seismic: ACI 318) (page 1802), Transverse reinforcement (columns seismic: ACI 318) (page 1799), or consult the respective clause reference in the code.
- Most of the requirements will be fulfilled through automatic design. In some cases specific design options will need to be set by the user.
- Additional requirements may apply to members that are not part of the SFRS when in SDC's D, E or F
- Confinement regions: - support regions; - Probable flexural yield regions; - Lap splice regions.

General requirements (columns seismic: ACI 318)

End Fixity

Reinforced concrete columns assigned to Moment Resisting Frames have their end fixities at the base of the building limited to:

- Fixed base;
- Pinned base;
- Spring base (foundation flexibility).

Minimum factored axial force

Members experiencing axial compression forces higher than the minimum threshold in the code from any of the load combinations are required to be checked for flexural strength and to consider flexural detailing within the strong column - weak beam design philosophy according to their assigned SFRS type.

If SFRS Type = Ordinary Moment Frame, then no No axial compression load requirement applies.

If SFRS Type = Intermediate Moment Frame, or Special Moment Frame

	P_{min}		$A_g * f'_c / 10$ =

	where		
	P_{min}		Minimum required axial compression =
	A_g		Gross area of the concrete section =
	f'_c		Specified compressive strength of concrete =

The check passes and the member is designed for seismic provisions as a compressive member if Pu > Pmin

	where		
	P_u		Largest factored compressive axial force at the top of the stack from any load combination. =

NOTE This check is no longer required in ACI 318-14.

Maximum recommended axial force

ACI 318 allows for the maximum design axial load to be as high as $0.8*\phi*P_{n,max}$, where $P_{n,max}$ is the maximum compression resistance of the section composed of concrete and steel. However in the event of a severe earthquake a full height beam yielding mechanism could occur inducing higher compression strain on the columns than the one predicted by elastic design.

Good practice recommends that the maximum compressive strain in a column part of a Special Moment Frame should remain below the balanced value.

NOTE The current release of Tekla Structural Designer does not check if the compressive strain is below the balanced point.

Minimum cross-section dimension

ACI 318 limits the shortest cross-sectional dimension of a column that is part of a Special Moment Frame to a lower fixed value in any direction measured in a straight line passing through the section centroid and also to a fraction of the length of the perpendicular dimension.

If designing to ACI 318-14, there is an additional limit of half the height of the deepest beam connecting at the joint.

These minimum dimension restrictions are calculated and applied accordingly when the column that is part of a Special Moment Frame.

Flexural requirements (columns seismic: ACI 318)

Minimum flexural strength

Columns that are part of Special Moment Frames are required by ACI 318 to have a minimum amount of flexural strength depending on the connecting beams flexural capacities so as to promote the formation of beam yielding mechanisms in the case of an earthquake. This is done by establishing a ratio between the beam and column moment strengths in the moment resisting frame direction.

The design of the main reinforcement in a column is done for the top, middle and bottom region of the stack and moment capacity is calculated for the factored axial force in each region for the Major and Minor directions.

If SFRS Type = Special Moment Frame then flexural strength checks are performed at the joints:

	$\Sigma M_{nc,l}$		$M_{nc,l,bot} + M_{nc,l,top}$ =
	$\Sigma M_{nc,r}$		$M_{nc,r,bot} + M_{nc,r,top}$ =
	where		
	$\Sigma M_{nc,l}$, $\Sigma M_{nc,r}$		Sum of the Nominal Flexural Strengths of the columns framing into the joint in the relevant direction.
	$M_{nc,l,bot}$, $M_{nc,r,bot}$		Nominal Moment Strength of the stack below the joint obtained for the axial force value consistent with the minimum Nominal Moment Strength respectively for the left and right sway cases.
	$M_{nc,l,top}$, $M_{nc,r,top}$		Nominal Moment Strength of the stack above converging on the same joint for the axial force value consistent with the minimum Nominal Moment Strength respectively for the left and right sway cases.

The sum of the beam strengths are obtained as follows:

	$\Sigma M_{nb,l}$	=	$M_{nb,l}$- + $M_{nb,l}$+
	$\Sigma M_{nb,r}$	=	$M_{nb,r}$- + $M_{nb,r}$+
	where		
	$\Sigma M_{nb,l}$, $\Sigma M_{nb,r}$	=	Sum of the Nominal Flexural Strengths of the beams framing into the joint in the relevant direction.
	$M_{nb,l}$- , $M_{nb,r}$-	=	Nominal Moment Strength at the joint from the beam on the left from current reinforcement arrangement respectively for the left and right sway cases.
	$M_{nb,l}$+ , Mnb,r+	=	Nominal Moment Strength at the joint from the beam on the right from current reinforcement arrangement respectively for the left and right sway cases.

The minimum strength ratio between columns and beams in both left and right sway cases is checked as follows:

	$\Sigma M_{nc,l}$	≥	6/5 * $\Sigma M_{nb,l}$
	$\Sigma M_{nc,r}$	≥	6/5 * $\Sigma M_{nb,r}$

Maximum allowed area of reinforcement

The maximum area of longitudinal reinforcement in columns part of Special Moment Frames is limited as follows.

If SFRS Type = Special Moment Frame

Then calculate maximum area of steel, $A_{s,max}$ as follows:

	$A_{s,max}$		$0.06 * A_g$ =	
	where			
	$A_{s,max}$		Maximum allowed area of reinforcement =	
	A_g		Gross area of the concrete section. =	

Maximum allowed longitudinal bar center spacing

Limitations on the longitudinal bar spacing emerge from the code requirement for maximum allowed cross-section center link spacing of the confinement reinforcement due to the method of link leg distribution across the column section.

If SFRS Type = Special Moment Frame

Then check maximum longitudinal reinforcement bar distance, scr,max as follows:

	$s_{cr,max}$		350 mm =	Metric-units
	$s_{cr,max}$		14 in =	US-units

NOTE In ACI 318-14, scr,max is limited to 200mm for non-circular columns with, Pu > 0.3*Ag*fc OR fc > 70 MPa (10,000 psi).

Non-reversing plastic hinges

Non-reversing plastic hinges are regions along the stack of the column where flexural yielding is likely to occur.

NOTE Non-reversing plastic hinges are beyond scope in the current release of Tekla Structural Designer.

Splices

Columns that are part of Special Moment Frames have restrictions on the allowed locations of lap splices.

Strength design of mechanical splices and restrictions to the use of welded splices as required by ACI 318-11 apply to Special Moment Frames

NOTE These restrictions are not implemented in the current release of Tekla Structural Designer.

Transverse reinforcement (columns seismic: ACI 318)

Joint shear strength

The calculation of joint shear strength is a requirement of ACI 318 for joints of Special Moment Frames and it is obtained by considering both the free body diagram of the column and of the joint. The stress in the beam's tensile reinforcement at the joint's face is assumed as at least η *fy by considering the probable moment strengths of framing beams.

The calculation is performed on the following basis:

- Probable Moment Strengths are obtained from beams in the same direction as the column's considered direction
- Whether beams in the column SFRS Direction are included in the SFRS or not
- At the top region of stack only
- For both sway right and sway left cases
- Beams with pinned connection are ignored
- Axial stress in beams is assumed to be zero
- Contribution of the slab longitudinal reinforcement in the beam effective flange width is recommended to be considered, but remains beyond scope in the current release.

Design shear force

The Design Shear Force of a member subjected to flexure as well as axial loading part of a Moment Resisting Frame is checked taking into consideration the shear from the moment strengths of the connected flexural members due to reverse curvature bending.

- Shear design for columns is done for the entire stack as a single region and checked for the minimum requirements from the design code.
- The design is performed independently for both the orthogonal directions.

Minimum area of transverse reinforcement

For columns that are part of Special Moment Frames, the minimum area of transverse reinforcement required in confinement regions of a column is obtained as below:

NOTE For Special Moment Frames the amount of confinement reinforcement in joints with beams on all 4 sides wider than ¾ of the column width is allowed to be reduced to half and the spacing to be relaxed within the depth of the shallowest member - this is beyond scope in the current release of Tekla Structural Designer.

NOTE Non-reversing plastic hinge regions along the span have the same requirements as support regions - Non-reversing plastic hinge regions are beyond scope in the current release of Tekla Structural Designer.

For circular columns:

ρ_s		MAX[0.12 * (f'_c / f_{yt}) , 0.45 * [(A_g / A_{ch})-1] * (f'_c / f_{yt})] =
where		
ρ_s		ratio of volume of circular reinforcement to total volume of confined concrete core.
f'_c		Specified compressive strength of concrete. =
f_{yt}		Specified yield strength of transverse reinforcement. =
A_g		Gross area of concrete section. =
A_{ch}		Area of concrete member section measured to the outside of the transverse reinforcement.

For other supported column geometries

ACI 318-11		

A_{sh}		MAX[0.3 * s * b_c * (f'_c / f_{yt}) * [(A_g / A_{ch})-1] , 0.09 * s * b_c * (f'_c / f_{yt})]
ACI 318-14		
A_{sh}		MAX[0.3 * s * b_c * (f'_c / f_{yt}) * [(A_g / A_{ch})-1] , 0.09 * s * b_c * (f'_c / f_{yt}), 0.2 * k_f * k_n * (P_u /(MIN[f_{yt} , 700MPa] * A_{ch})]
where		
A_{sh}		total cross-section of transverse reinforcement, including cross-ties, within spacing s and perpendicular to dimension bc.
s		Center to center spacing of transverse reinforcement along the region's height.
b_c		Cross section dimension of the member core measured to the outside of the transverse reinforcement and in the direction perpendicular to the considered reinforcement link legs.
A_g		Gross area of concrete section.
A_{ch}		Area of concrete member section measured to the outside of the transverse reinforcement.

Support regions of columns belonging to any other Moment Resisting Frame type have the minimum area of transverse reinforcement as per conventional design requirements.

Maximum allowed center hoop spacing

The maximum allowed horizontal center spacing of hoops in confinement regions of columns part of Moment Resisting Frames is limited as below.

If SFRS Type = Ordinary Moment Frame:

• No spacing requirement applies to support regions when designing for seismic provisions.

If SFRS Type = Special Moment Frame and the region is a support region:

$s_{cr,max,sup}$		MIN[6*$d_{b,smallest}$, 0.25*MIN(c_1, c_2), 100 mm ≤ 100+((350 – h_x)/3) ≤ 150 mm]	metric units

$s_{cr,max,sup}$		MIN[6*db,smallest, 0.25*MIN(c_1, c_2), 4 in. ≤ 4+((14 - h_x)/3) ≤ 6 in.]	US units
where			
$d_{b,smallest}$		Smallest longitudinal reinforcement bar diameter	
c_1		Rectangular or equivalent rectangular column dimension in the direction of the span for which moments are being considered	
c_2		Dimension of the column perpendicular to c_1	
h_x		Maximum center-to-center horizontal spacing of crossties at any face of the column	

If SFRS Type = Special Moment Frame and the region is not a support region:

$s_{cr,max,span}$		MIN[6 * $d_{b,smallest}$, 150 mm]	metric units
$s_{cr,max,span}$		MIN[6 * $d_{b,smallest}$, 6 in.]	US units

If SFRS Type = Intermediate Moment Frame and the region is a support region:

$s_{cr,max,sup}$		MIN[8*$d_{b,smallest}$, 24 *$d_{b,w}$, 1/2 * MIN(c_1, c_2), 300 mm]	metric units
$s_{cr,max,sup}$		MIN[8*$d_{b,smallest}$, 24 **$d_{b,w}$, 1/2 * MIN(c_1, c_2), 12 in.]	US units
where			
$d_{b,w}$		= Link (hoop) diameter	

If SFRS Type = Intermediate Moment Frame and the region is not a support region:

- No spacing requirement applies beyond the conventional design requirements.

Flexural reinforcement (columns seismic: ACI 318)

Development Length at the Foundation

Columns that are part of Special Moment Frames shall have their longitudinal reinforcement extended into supporting footings, foundation mats or pile caps for a length not less than the full development length in tension.

Lap Splices

Specific seismic requirements apply only to lap splices in compressive members that are part of Special Moment Frames.

For columns that are part of Special Moment Frames:

- lap splices are only allowed at the center half of the column
- lap splices regions should be properly confined - hoop spacing should not exceed the maximum allowed hoop spacing

NOTE Both of these requirements are beyond scope in the current release of Tekla Structural Designer.

Confinement reinforcement for ductility (columns seismic: ACI 318)

Reinforcement Type

Confinement reinforcement in columns at regions where provided should consist of hoops, i.e. closed or continuously wound ties with a seismic hook at each end.

NOTE Confinement reinforcement in the form of spiral reinforcement is beyond scope in the current release of Tekla Structural Designer.

Detailing Regions

Confinement reinforcement is required to be provided over three types of regions along reinforced concrete columns that are part of Intermediate Moment Frames and Special Moment Frames:

- Support regions: These are probable flexural yielding regions at the top and bottom of the stack next to column-beam joints;
- Non-reversing plastic hinge regions: These are probable flexural yielding regions outside support regions.

  NOTE Non-reversing plastic hinge regions are not identified in the current release of Tekla Structural Designer.

- Lap splices: Confinement reinforcement in the form of hoops is required to be provided over the length of lap splices in reinforced concrete columns part of Special Moment Frames

NOTE The requirement for hoop spacing not to exceed the maximum allowed hoop spacing at lap splices is beyond scope in the current release of Tekla Structural Designer.

Concrete wall design to ACI 318

The following topics are covered:

See also

Limitations (concrete members: ACI 318) (page 1730)

Seismic Design to ACI 318 (page 1840)

Design parameters for vertical bars (concrete wall: ACI 318)

For some of the longitudinal reinforcement design parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Wall > Reinforcement Layout.

Minimum and Maximum Vertical Bar Diameter

There are no code provisions, but user defined limits can be applied to the minimum and maximum bar diameters - specified in Design Options > Wall > Reinforcement Layout

Minimum and Maximum Vertical Loose Bar Spacing

Limiting minimum horizontal spacing of the vertical bars, $s_{v,lim,min}$ is controlled by the diameters of the 2 adjacent bars and aggregate size[1].

$s_{v,lim,min}$	=	$0.5*(d_{bv,i} + d_{bv,(i+1)}) + c_{gap}$

[1] Clause 7.6.3

where

c_{gap}		min. clear distance bet. bars =	
		max (1.5*$d_{bv,i}$, 1.5*$d_{bv,(i+1)}$, 1.33* hagg, 1.5 in.)=	US units
		max (1.5*$d_{bv,i}$, 1.5*$d_{bv,(i+1)}$, 1.33* h_{agg}, 38mm) =	metric units
where			
$d_{bv,i}$ and $d_{bv,(i+1)}$		the diameters of the two adjacent vertical bars =	
h_{agg}		aggregate size =	

Limiting maximum horizontal spacing of the vertical bars, sv,lim,max is controlled by the wall thickness.

$s_{v,lim,max}$		min (3*hw, 18 in.) =	US units
		min (3*hw, 450mm) =	metric units

You are given control over these values by specifying minimum and maximum spacing limits in Design Options > Wall > Reinforcement Layout.

Minimum and Maximum Reinforcement Area

The code provisions which control the vertical reinforcement area are,

- Limiting minimum ratio of vertical reinforcement area to gross concrete area, *r*v,lim,min
- Limiting maximum ratio of vertical reinforcement area to gross concrete area, *r*v,lim,max

The controlling values are:

IF d_{bv} ≤ No. 5 (No. 16) with f_y ≥ 60,000 psi (420 MPa) OR WWR ≤ W31 or D31 then $\rho_{v,lim,min}$ = 0.0012 else 0.0015 for all other deformed bars

Total minimum area of vertical reinforcement, $A_{s,min} = \rho_{v,lim,min}*A_{cg}$

Total maximum area of vertical reinforcement, $A_{s,max} = \rho_{v,lim,max}*A_{cg} = 0.08*A_{cg}$

where A_{cg} = Gross area of the concrete wall.

Where 2 layers are specified, this should be distributed equally to each face.

You are given further control over the minimum and maximum reinforcement ratio values via user limits in Design Options > Wall > Reinforcement Layout. These will be used if they are more onerous than the code limits.

Design parameters for horizontal bars (concrete wall: ACI 318)

For some of the horizontal bar parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Wall > Reinforcement Layout

Minimum and Maximum Reinforcement Area

The code provisions which control the horizontal reinforcement area are,

- Limiting minimum ratio of horizontal reinforcement area to gross concrete area, *r*h,lim,min
- Limiting maximum ratio of horizontal reinforcement area to gross concrete area, *r*h,lim,max

The controlling values are:

IF d_{bv} ≤ No. 5 (No. 16) with f_y ≥ 60,000 psi (420 MPa) OR WWR ≤ W31 or D31

THEN $\rho_{h,lim,min}$ = 0.002 ELSE 0.0025 for all other deformed bars

Total minimum area of horizontal reinforcement, As,min = $\rho_{h,lim,min}*A_{cg}$

Total maximum area of vertical reinforcement, As,max = $\rho_{h,lim,max}*A_{cg}$ = $0.08*A_{cg}$

where A_{cg}= Gross area of the concrete wall.

You can select a minimum ratio which will be the start point for the design in Design Options > Wall > Reinforcement Layout.

Minimum and Maximum Horizontal Bar Spacing

This is identical in principle to min vertical bar spacing.

Minimum and Maximum Confinement Bar Spacing

There are Code provisions that control the maximum spacing:

The recommended values are,

Limiting maximum transverse spacing, $s_{w,lim,max}$ = min ($16*d_{bv}$, $48*d_{bw}$, h_w)

Design for in plane shear (walls:ACI 318)

In the plane of the wall the factored shear must be equal to or less than the design shear strength of the wall

V_u	$\leq$	ΦV_n

The design shear strength of the wall is equal to the design shear strength of the concrete plus that of the shear reinforcing

V_u	$\leq$	$\Phi V_c + \Phi V_s$

The shear strength, V_n, may not be taken greater than $10 * \sqrt{f'_c} * h * d$.

V_n	$\leq$	$10 * \sqrt{f'_c} * h * d$	US units
V_n	$\leq$	$0.83 * \sqrt{f'_c} * h * d$	metric units

where

h	=	wall thickness
d	=	$0.8 * l_w$
l_w	=	length of the wall

Out of plane the shear design calculations are the same whether the design element is a column or a wall - see: Design parameters for shear (column and wall:ACI 318) (page 1780)

Seismic design (walls: ACI 318)

Direction dependant seismic checks are performed in the in plane direction only as this is the only direction in which shear walls are considered to act as Seismic Force Resisting Systems.

Walls included in the SFRS as Ordinary Reinforced Concrete Structural Walls (ORCSW) have no specific seismic design provisions according to ACI318-11.

For overall limitations and assumptions, see:

- Limitations and assumptions (walls seismic: ACI 318) (page 1808)

For seismic design, see:

- General requirements (walls seismic: ACI 318) (page 1824)

See also

Seismic Design to ACI 318 (page 1840)

Limitations and assumptions (walls seismic: ACI 318)

The following limitations and assumptions apply.

- For design purposes Tekla Structural Designer recognizes walls as isolated elements and as such the influence of flanges from adjacent walls are not to be considered when fulfilling seismic design requirements in those elements. Requirements of ACI318-11 section 21.9.5.2 and section 21.9.6.4(b) do not apply.
- Additional requirements for wall piers not part of any SFRS when in Seismic Design Categories D through F are not considered in the current release.
- Design and/or detailing requirements of Special Reinforced Concrete Structural Walls is beyond scope (some checks are implemented but only due to their existence in lower toughness systems).
- The use of spiral reinforcement as well as all seismic design checks and related assumptions are not considered due to the fact that this type of reinforcement is not currently available in Tekla Structural Designer.
- Special boundary elements in walls are directly linked with wall end-zones in Tekla Structural Designer. The current settings do not allow for end-zones of width different from the width of the wall itself.
- The Seismic Force Resisting System set by the user in each direction through the Seismic Wizard for analysis purposes is not checked for applicability against the allowed types from the resultant Seismic Design Category. This is user responsibility.
- Seismic design checks are mostly based on capacity design obtained from the main reinforcement provided. This can lead to an over-design of structural members if the designer is not careful enough to minimize excess capacity.
- The seismic design requirements for end-zone reinforcement are beyond scope in the current release.
- Construction joints are beyond scope in the current release.
- Seismic design provisions specific for beam-wall frames are beyond scope in the current release.
- Seismic design of walls with openings is beyond scope in the current release.
- Seismic design of wall piers is beyond scope in the current release.
- Seismic design of coupling beams is beyond scope in the current release.
- Seismic detailing requirements apply to Special Reinforced Concrete Structural Walls these are beyond scope in the current release.

- **NOTE** A full list of the code checks that have and have not been implemented is provided in the table below.

ACI 318 Seismic Code Checks for walls that have been implemented in Tekla Structural Designer

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.1.4.2	Minimum required compressive strength of concrete	SRCSW	-	-	-	✓
21.1.4.3	Maximum allowed compressive strength of light-weight concrete	SRCSW	-	-	-	✓
21.1.5.2	Maximum allowed steel characteristic yield strength of longitudinal reinforcement	SRCSW	-	-	-	✓
21.1.5.4	Maximum yield strength of transverse reinforcement in confinement	SRCSW	-	-	-	✗

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	regions of columns					
21.1.5.5	Maximum allowed longitudinal reinforcement yield strength used in the calculation of transverse reinforcement	SRCSW	-	-	-	✓
21.1.6.1a)	Mechanical Splices outside twice the member depth from column/ beam face or yielding regions	SRCSW	-	-	-	✗
21.1.6.1b)	Mechanical Splices within twice the member depth from column/ beam face or yielding regions	SRCSW	-	-	-	✗
21.1.7.1	Welded Splices	SRCSW	-	-	-	✗

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	within twice the member depth from column/ beam face or yielding regions					
21.1.7.2	Welding of stirrups or other elements to longitudinal reinf. required by design	SRCSW	-	-	-	✗
21.12.2.3	Confinement reinf.: Length inside footing when Special Boundary Element is within half the foundation depth from the footing edge	SRCSW	-	-	-	✗
21.4.3	Minimum yield strength of connection not designed to yield	IPCSW	-	-	✗	-

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.4.4/21.9.8	Design of wall piers as columns ($l_w/b_w \leq 2.5$)	SRCSW	-	-	-	✖
21.9.2.1	Minimum reinforcement ratio in each of the wall plane orthogonal directions	SRCSW	-	-	-	✓
21.9.2.1	Maximum allowed center spacing	SRCSW	-	-	-	✓
21.9.2.2	Minimum number of reinforcement layers (page 1824)	SRCSW	-	-	-	✓
21.9.2.3a)	Minimum discontinuous vertical bar extension	SRCSW	-	-	-	✖
21.9.2.3a)	Development length at locations where flexural yielding is likely to occur	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.9.2.3c)	Minimum yield strength for development length and lap splices	SRCSW	-	-	-	✖
21.9.3	Factored shear force at any section from lateral load analysis	SRCSW	-	-	-	✓
21.9.4.1	Maximum nominal shear strength	SRCSW	-	-	-	✓
21.9.4.3	Reinforcement in wall plane provided in both directions	SRCSW	-	-	-	✓
21.9.4.3	Minimum in plane reinforcement ratios (page 1824)	SRCSW	-	-	-	✓
21.9.4.4	Maximum wall segment combined nominal shear strength	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.9.4.4/21.9.4.5	Maximum individual vertical or horizontal wall segment or coupling beam shear strength	SRCSW	-	-	-	✖
21.9.5.2	Effective width of flanged wall sections	SRCSW	-	-	-	✖
21.9.6.3	Minimum extreme fiber compressive stress to require Special Boundary Elements	SRCSW	-	-	-	✖
21.9.6.3	Minimum stress to discontinue Special Boundary Element	SRCSW	-	-	-	✖
21.9.6.4(a)	Minimum length of end-zone towards the center of the cross-section when Special Boundary Elements	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	are required					
21.9.6.4(b)	Minimum length of end-zone towards the center of the cross-section in flanged sections with Special Boundary Elements	SRCSW	-	-	-	✖
-	Minimum width of end-zone with special boundary elements	SRCSW	-	-	-	✖
21.9.6.4(c)	Special confining reinforcement type (hook/ extension)	SRCSW	-	-	-	✖
21.9.6.4c)	Confining reinf.: Maximum spacing allowed between cross ties	SRCSW	-	-	-	✖
21.9.6.4c)	Confining reinf.: Maximum allowed longitudi	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	nal center link spacing					
21.9.6.4c)	Confinement reinf.: Minimum volumetric ratio / area of spiral or circular reinforcement	SRCSW	-	-	-	✖
21.9.6.4c)	Confinement Reinf.: Minimum area of rectangular transverse reinforcement	SRCSW	-	-	-	✖
21.9.6.4d)	Confining reinf.: Length of region inside footings, mats or pile caps, l_d	SRCSW	-	-	-	✖
21.9.6.4d)	Confining reinf.: Length of region into support	SRCSW	-	-	-	✖
21.9.6.4e)	Development / anchorage of horizontal	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	reinforcement with boundary elements					
21.9.6.5a)	Ordinary Boundary Confinement region length - towards the center of the cross-section - at an end where special boundary elements are not required	SRCSW	-	-	-	✖
21.9.6.5a)	Maximum spacing allowed between cross ties in high compression confinement reinf. at a boundary where special boundary elements are not required	SRCSW	-	-	-	✖
21.9.6.5a)	Maximum long. center spacing	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	of high compression confinement reinforcement at an end where special boundary elements are not required					
21.9.6.5a)	Maximum spacing allowed between cross ties or legs of hoops	SRCSW	-	-	-	✖
21.9.6.5b)	Design shear threshold for ignoring the need of engaging horizontal bars at the ends with standard hooks	SRCSW	-	-	-	✖
21.9.7.1	Minimum value of aspect ratio (l_n/h) to consider diagonal reinf.	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
21.9.7.2	Maximum shear allowed before considering diagonal reinf.	SRCSW	-	-	-	✖
21.9.7.4a)	Nominal Shear Strength of a coupling beam	SRCSW	-	-	-	✖
21.9.7.4b)	Minimum number of bars to be provided along each diagonal	SRCSW	-	-	-	✖
21.9.7.4b)	Minimum length of diagonal bars embedded into the wall	SRCSW	-	-	-	✖
21.9.7.4c)	Minimum breadth of the concrete core measured to the external face of the confining reinforcement	SRCSW	-	-	-	✖
21.9.7.4c)	Minimum dimension of the	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	concrete core in any direction than not the parallel to b_w, measured to the external face of the confining reinf.					
21.9.7.4c)	Special confining reinforcement (hook/ extension)	SRCSW	-	-	-	✖
21.9.7.4c)	Confining reinf.: Maximum allowed center link spacing	SRCSW	-	-	-	✖
21.9.7.4c)	Confining reinf.: Minimum volumetric ratio / area of spiral or circular reinf.	SRCSW	-	-	-	✖
21.9.7.4c)	Confining Reinf.: Minimum area of rectangular	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	transverse reinf.					
21.9.7.4c)	Minimum allowed total area of the additional longitudinal reinforcement	SRCSW	-	-	-	✖
21.9.7.4c)	Maximum allowed spacing between the additional longitudinal bars	SRCSW	-	-	-	✖
21.9.7.4c)	Minimum allowed area of the additional transverse reinforcement	SRCSW	-	-	-	✖
21.9.7.4c)	Maximum allowed spacing between the additional transverse bars	SRCSW	-	-	-	✖
21.9.8.1a)	Design shear force calculation for wall piers with l_w/b_w >	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	2.5 and not designed as columns					
21.9.8.1b)	Nominal shear strength and distributed shear reinforcement	SRCSW	-	-	-	✖
21.9.8.1c)	Reinforcement type requirement for wall piers with l_w/b_w > 2.5 and not designed as columns	SRCSW	-	-	-	✖
21.9.8.1d)	Maximum allowed vertical spacing of transverse reinforcement. Wall piers with l_w/b_w > 2.5 and not designed as columns	SRCSW	-	-	-	✖
21.9.8.1e)	Length of the	SRCSW	-	-	-	✖

Code Ref.	Requirement	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	transverse reinf. region above and below the wall pier. Wall piers with l_w/b_w > 2.5 and not designed as columns					
21.9.8.1f)	Consider boundary elements	SRCSW	-	-	-	✖
21.9.8.2	Horizontal reinf. In adjacent walls when wall pier is at the edge of a wall	SRCSW	-	-	-	✖
ASCE7/10 - 12.2.1	Limiting Height (page 1824)	SRCSW	-	-	-	✖
ASCE7/10 - 12.2.1	Limiting Height (page 1824)	IPCSW	-	-	-	✖
R21.9.1	Vertical Segment Classification (page 1824): Conditions for wall segments to require specific	SRCSW	-	-	-	✖

Code Ref.	Require ment	SFRS	SDC A	SDC A	SDC B	SDC D,E,F
	wall pier design					

NOTE
- For further details of the checks that have been implemented, see: General requirements (walls seismic: ACI 318) (page 1824), or consult the respective clause reference in the code.
- Most of the requirements will be fulfilled through automatic design. In some cases specific design options will need to be set by the user.
- Additional requirements may apply to members that are not part of the SFRS when in SDC's D, E or F
- Confinement regions: - support regions; - Probable flexural yield regions; - Lap splice regions.

General requirements (walls seismic: ACI 318)

Maximum recommended axial force

As to impose the wall ductile behaviour axial force values are recommended to be kept low in the design of Special Reinforced Concrete Structural Walls resisting earthquake effects. The maximum axial force value in walls is recommended to be kept below the balanced point. Compression controlled walls should be avoided.

NOTE The current release of Tekla Structural Designer does not check if the axial force is below the balanced point.

Limiting Height

Buildings in which Special Reinforced Concrete Structural walls compose the SFRS in any of the main directions should have their height limited. Where dual systems of SRCSW and Moment Frames resisting at least 25% of the total shear exist there are no height restrictions.

NOTE The current release of Tekla Structural Designer does not check the maximum allowed building height based on Seismic Design Category.

Vertical Segment Classification

Isolated vertical segments within a wall - wall with openings - can be classified as wall segments or wall piers. For Special Reinforced Structural Walls and depending on the classification of the segment the governing design provisions can be split into provisions for walls and provisions for wall piers.

NOTE This is beyond scope in the current release of Tekla Structural Designer.

Mid-zone Reinforcement

Vertical and horizontal bars in mid-zones of Special Reinforced Concrete Structural Walls are designed according to the requirements of the following sections.

Minimum number of reinforcement layers

The minimum number of reinforcement layers allowed to be used in a Special Reinforced Concrete Structural Wall is governed by the amount of in-plane shear sustained by the wall

If the wall thickness b_w is greater than 250mm (10 in.) then at least two curtains of reinforcement are required.

If the wall thickness b_w is less than 250mm (10 in.) and the SFRS Type = Special Reinforced Concrete Structural Wall, then the minimum number of layers is dependent on the maximum shear force in the panel (ACI 318-11); or the maximum shear force and wall geometry (ACI 318-14). This is checked accordingly.

Additionally, reinforcement is required to be provided in both of the orthogonal directions in the wall plane - This requirement is automatically met as Tekla Structural Designer does not design walls with reinforcement in only one of the orthogonal directions in wall plane.

Minimum in plane reinforcement ratios

The minimum reinforcement ratio required in each of the orthogonal direction in the wall plane is dependent on the maximum panel factored shear force from analysis for seismic combinations and calculated as follows:

NOTE Grouped bars reinforcing the edges of the walls (end-zones) are not considered for the purpose of calculating reinforcement ratios.

IF SFRS Type = Special Reinforced Concrete Structural Wall then the maximum factored shear force at the panel is checked:

	If V_u		$V_{u,lim}$ >
	where		
	V_u		Maximum factored shear force in the wall panel obtained from the analysis for seismic combinations.

	$V_{u,lim}$		Minimum factored shear force in the wall above which horizontal and vertical main reinforcement minimum ratios need to be checked.
	$M_{nc,l,top}$, $M_{nc,r,top}$		Nominal Moment Strength of the stack above converging on the same joint for the axial force value consistent with the minimum Nominal Moment Strength respectively for the left and right sway cases.

Then a check for minimum reinforcement ratio in each orthogonal direction on the wall plane is performed as follows:

	ρ_l		ρ_{min} $\geq$
	ρ_t		ρ_{min} $\geq$
	where		
	ρ_l, ρ_t		Respectively the ratio of area of distributed vertical and horizontal reinforcement to gross concrete area perpendicular to each of those reinforcements.
	ρ_{min}		Minimum allowed ratio of reinforcement in the wall plane = 0.0025. =

Else if V_u		$V_{u,lim}$ $\leq$

ρ_l and ρ_t are allowed to be taken as the wall design conventional values.

Depending on the overall dimensions of a wall the vertical reinforcement ratio, ρl in Special Reinforced Concrete Structural Walls is limited to be of the same value or larger than the horizontal reinforcement ratio, ρ_t

End-zone Reinforcement

NOTE The seismic design requirements for end zone reinforcement are beyond scope in the current release of Tekla Structural Designer.

Shear Strength

Minimum shear strength

The basic design requirement for shear reinforcement in a wall is to have the reduced shear strength higher or the same as the maximum factored shear force at the considered section resultant from earthquake combinations. Some level of over-strength is expected when designing to multiple load combinations.

IF SFRS Type = Special Reinforced Concrete Structural Wall then the following check for shear strength is performed:

	ϕV_n		V_u ≥
	where		
	ϕ		Strength reduction factor. For purposes of checking the nominal shear strength = 0.6
	V_n		Maximum nominal shear strength at the considered panel. =
	V_u		Maximum factored shear force in the wall panel obtained from the analysis for seismic combinations.

Concrete slab design to ACI 318

The following topics are covered:

- Cover to Reinforcement (ACI 318) (page 1731)
- Design parameters for longitudinal bars (ACI 318) (page 1733)
- Design for bending for rectangular sections (beams and slabs: ACI 318) (page 1738)

Pad and strip base design to ACI 318

The following topics are covered:

- Checks performed (pad and strip base:ACI 318) (page 1828)
- Foundation Bearing Capacity (pad and strip base:ACI 318) (page 1828)
- Design for shear (pad and strip base:ACI 318) (page 1832)
- Check for transfer forces at column base (pad and strip base:ACI 318) (page 1832)

- Check for transfer of horizontal forces by shear friction (pad and strip base:ACI 318) (page 1834)
- Design for bending for rectangular sections (beams and slabs: ACI 318) (page 1738)
- Check for overturning forces (pad and strip base:ACI 318) (page 1834)
- Check for sliding (pad and strip base:ACI 318) (page 1834)
- Check for uplift (pad and strip base:ACI 318) (page 1835)
- Checks for limiting parameters (pad and strip base: ACI 318) (page 1836)

Checks performed (pad and strip base:ACI 318)

The checks performed for both directions are:

- Max soil bearing pressure must not exceed allowable bearing pressure.
- Provided steel must be greater than As(min) for both vertical directions.
- Provided bar spacing must be inside the limiting spacing
- Provided bar size must be inside the limiting sizes
- Check for bending moment capacity
- Check for shear capacity - wide beam action at 'd' from column face
- Punching check at 'd/2' from column face - two-way action
- Check for transfer of forces at column base
- Check for transfer of horizontal forces by shear friction theory
- Check for overturning forces - **not in the current release**
- Check for sliding
- Check for uplift

Foundation Bearing Capacity (pad and strip base:ACI 318)

Check for Pad Base Bearing Capacity

Bearing capacity calculations are done using service (soil) -combinations.

Total base reaction:

T	=	$F_{swt} + F_{soil} + F_{dl,sur} + F_{ll,sur} - P$

Moment about X axis:

$M_{x,c}$	=	$M_{x,sup} - P * e_y - t_{ftg} * F_{y,sup}$

Moment about Y axis:

$M_{y,c}$	=	$M_{y,sup} + P * e_x + t_{ftg} * F_{x,sup}$

Where:

L_x	=	Length of foundation in X-direction	
L_y	=	Length of foundation in Y-direction	
A_f	=	$L_x * L_y$ = Foundation area	
t_{ftg}	=	Depth of foundation	
D_s	=	Depth of soil above the foundation	
l_x	=	Length of column/ wall in X-direction	
l_y	=	Length of column/ wall in Y-direction	
A_c	=	cross section of the column/wall segment	
e_x	=	eccentricity in X direction	
e_y	=	eccentricity in Y direction	
ρ_c	=	density of concrete	
ρ_s	=	density of soil	
F_{swt}	=	$A_f * t_{ftg} * \rho_c$ = foundation self-weight	

F_{soil}	=	$(A_f - A_c)*D_s* \rho_s$ = soil self-weight	
$F_{dl,sur}$	=	$(A_f - A_c)*sc_{dl}$ = Dead load from surcharge	
$F_{ll,sur}$	=	$(A_f - A_c)*sc_{ll}$ = Live load from surcharge	
sc_{dl}	=	Surcharge in dead load case	
sc_{dl}	=	Surcharge in live load case	
P	=	axial load acting on support in service combinations	
$M_{x,sup}$	=	Moment acting on support around X-axis in service comb.	
$M_{y,sup}$	=	Moment acting on support around Y-axis in service comb.	
A_c	=	cross section of the column/wall	
$F_{x,sup}$	=	Horizontal force acting on support X-direction in service comb.	
$F_{y,sup}$	=	Horizontal force acting on support Y-direction in service comb.	

Eccentricity of base reaction in X-direction:

e_{Tx}	=	$M_{y,c}$ / T

Eccentricity of base reaction in Y-direction:

e_{Ty}	=	$M_{x,c}$ / T

If $abs(e_{Tx}) / L_x + abs(e_{Ty}) / L_y \leq 0.167$

Then base reaction acts within kern distance - no loss of contact in X-direction, and:

Pad base pressures:

q1	=	$T/A_f – 6* M_{y,c} / (L_x*A_f) + 6* M_{x,c} / (L_y*A_f)$
q2	=	$T/A_f – 6* M_{y,c} / (L_x*A_f) - 6* M_{x,c} / (L_y*A_f)$
q3	=	$T/A_f + 6* M_{y,c} / (L_x*A_f + 6* M_{x,c} / (L_y*A_f)$
q4	=	$T/A_f + 6* M_{y,c} / (L_x*A_f - 6* M_{x,c} / (L_y*A_f)$

Max base pressure:

q_{max}		$max (q_1, q_2, q_3, q_4)$ =

Else base reaction acts outside kern distance - loss of contact.

In this case the pressure calculations are more complex - in Tekla Structural Designer these are done using sets of equations presented in an article by Kenneth E. Wilson published in the Journal of Bridge Engineering in 1997

Check for Strip Base Bearing Capacity

The principles used in the strip base bearing capacity calculations are similar to those for pad foundations. Only the direction X is checked (around Y-axis) using segment widths.

If $abs(e_{Tx}) / L_x \leq 0.167$

Then - no loss of contact, and:

max base pressures for segment:

q_{max}	=	$T/A_f + max[- 6* M_{y,c} / (Lx*A_f) , 6*M_{y,c} / (L_x*A_f)]$

Else - loss of contact and

max base pressures for segment:

q_{max}	=	$2*T/[3* L_y* (Lx /2 - abs(e_{Tx}))]$

where

L_y = segment width

Design for shear (pad and strip base:ACI 318)

Pad base shear design check

The nominal shear strength of the concrete in beam action, is given by[1]

v_n	=	2*λ*MIN(√(f`$_c$), 100psi)*d
	=	0.17*λ*MIN(√(f`$_c$) ,8.3MPa)*d

where

λ	=	modification factor related to the density of the concrete
λ	=	1.0 for normal weight concrete

If

v_u	≤	Φ_{shear}*v_n

Then the foundation thickness is adequate for shear -

Utilization ratio is then;

U-ratio	=	max [v_u / (Φ_{shear}*v_n)]

Else the check has failed, the foundation thickness is inadequate.

NOTE If the thickness is inadequate and the auto-design footing depth option is active then the foundation thickness gets increased.

Strip base shear design check

The principle of the strip base shear design check is similar to that for the pad base. Only the direction X is checked (around Y-axis) using segment widths.

Check for transfer forces at column base (pad and strip base:ACI 318)

This check applies when a concrete column is attached to the foundation.

Determinate the bearing strength of the column:

[1] ACI 318-08 Sections 11.1.2 and 11.2.1.1 Eqn (11-3)

$\Phi*P_{nb,c}$		$\Phi_{bearing}$ (0.85*f'$_c$ * A_c) =
If		
$\Phi*P_{nb,c}$		- P_u <
Then check fails		
Else determinate the bearing strength of the footing:		
$\Phi*P_{nb,f}$		min [√(A_2/A_c), 2] * Φbearing (0.85*f'$_c$ * A_c) =
where for rectangular columns:		
A_2		min {L_x , 2t_{ftg} + l_x+ min [(L_x-l_x)/2 - abs(e_x), 2t_{ftg}]} * min {L_y , 2t_{ftg} + l_y + min [(L_y-l_y)/2 - abs(e_y), 2t_{ftg}]}

NOTE Circular columns are treated as square members with the same area.

If

$\Phi*P_{nb,f}$		- Pu <
Then check fails		
Required min. area of dowel bars between column and footing is then:[1]		
$A_{s,min}$		0.005*A_c =
Currently dowel bars are not designed.		
The area of the provided column reinforcement $A_{s,prov,column}$ is the same as the provided reinforcement of starter/dowel bars.		
If		
$A_{s,min}$		$A_{s,prov,column}$ >
Then check fails		

[1]: ACI 318-08 Section 15.8.2.1

Check for transfer of horizontal forces by shear friction (pad and strip base:ACI 318)

This check applies when a concrete column is attached to the foundation.

Determinate if the shear-friction design method applicable[1]:

When surface is not intentionally roughened (conservative assumption)

If	$V_u \leq \Phi_{shear}$ * Ac min(0.2*f'$_c$, 800psi)	for US units
	$V_u \leq \Phi_{shear}$ * Ac min(0.2*f'$_c$, 5.5MPa)	for metric units
	where	
	V_u = max[abs(Fy,sup,u) , abs(Fx,sup,u)]	

Then maximum shear transfer is permitted at the base of the column.

Required area of dowel reinforcement:

A_{vf}	=	$V_u / (f_y * \Phi_{shear} * \mu)$

where

μ = 0.6 when concrete not intentionally roughened (assumption)

μ = 1.0 when concrete intentionally roughened

μ = 1.4 when concrete placed monolithically

Currently dowel bars are not designed.

The area of the provided column reinforcement As,prov,column is the same as the provided reinforcement of starter/dowel bars.

If

A_{vf}	>	$A_{s,prov,column}$

Then check fails

Check for overturning forces (pad and strip base:ACI 318)

NOTE Checks for overturning forces are beyond scope in the current release of Tekla Structural Designer.

[1] ACI 318-08 Section 15.8.2.1

Check for sliding (pad and strip base:ACI 318)

The check for sliding is carried out for pad foundations only.

If there is no horizontal force acting on foundation check for sliding is not required.

Resultant Force on foundation:

$H_d = \sqrt{[(F_{x,sup})^2 + (F_{y,sup})^2]}$

Resultant Force Angle α_{Hd} = tan-1 $[(F_{y,sup} / F_{x,sup})]$

where

$F_{x,sup}$ = horizontal force acting on support in X-dir. (from analysis)

$F_{y,sup}$ = horizontal force acting on support in Y-dir. (from analysis)

Resistance to sliding due to base friction:

$H_{friction} = [-P + F_{swt}] * \tan\delta$

where

δ = design base friction – user input

Passive pressure coefficient:

$K_p = (1+\sin\Phi') / (1-\sin\Phi')$

where

Φ' = design shear strength of soil – user input

Passive resistance of soil in X direction:

$H_{xpas} = 0.5*K_p*(h^2 +2*h*h_{soil})*L_x* \rho_{soil}$

Passive resistance of soil in Y direction:

$H_{ypas} = 0.5*K_p*(h^2 +2*h*h_{soil})*L_y* \rho_{soil}$

Resultant Passive Resistance:

$H_{res,pas} = abs(H_{xpas}*\cos \alpha_{Hd}) + abs(H_{ypas}*\sin \alpha_{Hd})$

Total resistance to sliding:

$R_{Hd} = (H_{friction} + H_{res,pas}) / 1.5$

If

$R_{Hd} \geq {}_{Hd}$

The check for stability against sliding passes

Check for uplift (pad and strip base:ACI 318)

For combinations producing tension at the support, the tension value is compared to the stabilizing loads and checked against a factor of safety (FOS).

Auto-design can automatically increment the base size to achieve a passing status.

The FOS considered for the uplift check is specified under Design Options > Concrete > Foundations > Isolated Foundations > General Parameters (default value = 1.50).

Checks for limiting parameters (pad and strip base: ACI 318)

Limiting reinforcement parameters are specified in Design Options > Foundations > Isolated Foundations > Reinforcement Layout

Limits on bar size and reinforcement quantities

For structural foundations of uniform thickness the minimum areaof tensile reinforcement shall be:

For metric units:

IF Grade 280 to 350 deformed bars are used

As,min,reqd	≥	b*h*0.0020

IF Grade 350 to 420 deformed bars or welted wire reinforcement are used

As,min,reqd	≥	b*h*0.0018

IF yield stress exceeds 420 MPa

As,min,reqd	≥	b*h* [MAX(0.0014 , 0.0018*420/fy)]

For US-units:

IF Grade 40 to 50 deformed bars are used

As,min,reqd	≥	b*h*0.0020

IF Grade 50 to 60 deformed bars or welted wire reinforcement are used

As,min,reqd	≥	b*h*0.0018

IF yield stress exceeds 60000 psi

As,min,reqd	≥	b*h*[MAX(0.0014 , 0.0018*60000/fy)]
where		

b	=	unit width

The maximum area of tensile reinforcement shall be:

As,max	≤	0.85*(f`c/fy)*β1*b*d*(3/7)
where		
Ag	=	the gross area of the concrete section
V	=	stress block depth factor[1]

[1]: ACI 318-08, ACI 318-11, ACI 318M-08 and ACI 318M-11 Section 10.2.7.3

metric-units

β1	=	0.85	for f`c ≤ 28Mpa
	=	0.85 – 0.05 *[(f'c – 28MPa)/7MPa]	for 28MPa < f`c < 55Mpa
	=	0.65	for f`c ≥ 55MPa

US-units

β1	=	min(max(0.85 - 0.05 ´ (f'c - 4 ksi) / 1ksi, 0.65), 0.85)	
	=	0.85	for f`c ≤ 4000 psi
	=	0.85 – 0.05 *[(f'c – 4ksi)/1ksi]	for 4000 psi < f`c < 8000psi

		0.65 =	for f`c ≥ 8000 psi

Pile cap design to ACI 318

The forces acting on a pile cap are applied to the foundation at the foundation level. The foundation can take axial load and bi-axial shear and moment.

Pile cap design is divided between pile design (pile capacity check) and structural design of the pile cap which includes bending, shear and punching shear design checks.

Pile capacity (ACI 318)

The pile capacity is compared to the axial service load acting on pile:

Pile capacity passes if:

R_c	≥	$P_n \geq - R_t$
Where:		
R_c	=	Pile compression capacity (service)
R_t	=	Pile tension capacity (service)
P_n	=	Pile load

Design for bending (pile cap:ACI 318)

The pile cap is treated as a beam in bending, where the critical bending moments for the design for the bottom reinforcement are taken at the face of the column.

The basic design method is identical to that for beams - see:Design for bending for rectangular sections (beams and slabs: ACI 318) (page 1738)

Shear design (pile cap: ACI 318)

Pile shear capacity passes if:

v_{su}	≤	$\Phi\ v_c$
and		
$v_{su,d}$	≤	$\Phi\ v_{c,d}$

for both sides and directions.

Refer to CRSI Design Handbook 2002 - Chapter 13, page 13-18...13-21

Checks for limiting parameters (pile cap: ACI 318)

Limiting reinforcement parameters are specified in Design Options > Foundations > Isolated Foundations > Reinforcement Layout

Check for distance of pile cap overhang

The check passes if:

min e_i		min (e_{min} , $e_{min,user}$) >		
where:				
e_{min}		MAX[230mm, 380mm - 0.5 * l_p] =	when $R_c \leq 534$ kN	Metric
e_{min}		MAX[9in, 15in - 0.5 * l_p] =	when $R_c \leq 120$ kips	US Customary
e_{min}		MAX[230mm, 530mm - 0.5 * l_p] =	when 534 kN < $R_c \leq$ 1068 kN	Metric
e_{min}		MAX[9in, 21in - 0.5 * l_p] =	when 120 kips < $R_c \leq$ 240 kips	US Customary
e_{min}		MAX[230mm, 685mm - 0.5 * l_p] =	when 1068 kN < $R_c \leq$ 1779 kN	Metric
e_{min}		MAX[9in, 27in - 0.5 * l_p] =	when 240 kips < $R_c \leq$ 400 kips	US Customary
e_{min}		MAX[230mm, 760mm - 0.5 * l_p] =	when R_c > 1779 kN	Metric

e_{min}		MAX[9in, 30in - 0.5 * l_p] =	when R_c > 400 kips	US Customary
l_p		least width/diameter of the pile =		

Check for minimum pile spacing

Check center to center spacing “s” between piles “i” and “j” in a pile group:

The check passes if:

If $s_{ij} > \min(s_{min}, s_{min,user})$

where

$s_{min,user}$ = user input

s_{min} = max (least width of the pile + 0.6m , 0.9m) for metric units

s_{min} = max (least width of the pile + 2ft. , 3 ft) for US customary units[1]

Check for maximum pile spacing

Check center to center maximum spacing “s” between piles “i” and “j” in a pile group:

The check passes if:

If s_{ij} < smax,user

$s_{max,user}$ = user input

Other checks

The remaining checks are identical to those for pad bases.

See: Pad Base and Strip Footing Design - Checks for limiting parameters (pad and strip base: ACI 318) (page 1836).

Seismic Design to ACI 318

Reinforced concrete structures in buildings subjected to earthquake effects are designed elastically to the strains and displacements both from static and dynamic forces which they are subjected to. It is recognised that during an earthquake the building and its structural elements are very likely to be exposed to displacements well into their inelastic range and special precautions need to be taken as to increase the strength of critical sections in

[1] CRSI – Design handbook page 13-18

members which contribute to the building's lateral resistance while also contributing to the ductile behaviour of the building in order to allow for the dissipation of induced stresses.

In the case of reinforced concrete structures, particular design and detailing requirements (provisions) need to be fulfilled beyond the conventional design of the elements as to provide them with ductile response capabilities. Such requirements are mainly addressed to structural elements part of structural systems built for the purpose of resisting seismic lateral forces - Seismic Force Resisting Systems [SFRS]. Seismic provisions also apply to reinforced concrete elements not part of the SFRS when the building is assigned to a higher Seismic Design Category - SDC D, E or F.

The seismic design checks and detailing requirements of reinforced concrete members in Tekla Structural Designer are based mainly on ACI318, Chapter 21 - Earthquake-Resisting Structures.

Seismic Force Resisting Systems

The level of design and detailing required of members that are part of a seismic resisting structural system can differ depending on the amount of toughness they are intended to provide to the building. ACI318-11 groups the main structural systems into "Ordinary", "Intermediate" and "Special" groups. Different types of structural systems have limitations to their application in each of the Seismic Design Categories.

Structural System	Allowed in SDC
Ordinary Moment Frames	A, B
Ordinary Cast in Place Structural Walls	A, B, C
Intermediate Moment Frames	A, B, C
Intermediate Precast Walls	A, B, C
Special Moment frames	A, B, C, D, E, F
Special Structural Walls (Precast / Cast in Place)	A, B, C, D, E, F

As the current release of Tekla Structural Designer does not fully include the design requirements for all the Seismic Force Resisting Systems, SFRS types have been classified as included or excluded from the member design.

Members in included SFRS types are fully covered for seismic design provisions while those in excluded types are covered to a limited extent only.

- Seismic Force Resisting Systems included in the design:
  - Intermediate Moment Frames
  - Ordinary Moment Frames
  - Ordinary Reinforced Concrete Structural Walls
- Seismic Resisting Systems excluded from the design:

- Special Moment Frames
- Special Reinforced Concrete Structural Walls
- Intermediate Precast Structural Walls

Consequently the current release of Tekla Structural Designer can be said to consider the design requirements for each of the Seismic Design Categories as follows:

Seismic Design Category	Seismic Requirements
SDC A	N/A
SDC B	Considered
SDC C	Considered
SDC D, E, F	Not Considered

Materials

Additional seismic material requirements apply to:

- concrete beams and columns assigned to a Special Moment Frame SFRS
- concrete walls assigned as Special Reinforced Concrete Structural Walls

Concrete Compressive Strength

The requirements for compressive strength of concrete are limited:

Minimum compressive strength of normal weight concrete	f'_c	=	21 MPa (3,000 psi)
Maximum compressive strength of lightweight concrete	f'_c	=	35 MPa (5,000 psi)

Reinforcement Steel

Reinforcement steel shall comply with ASTM 706(M), Grade 420 (60,000 psi).

$(f_y)_{actual} - (f_y)_{specified}$	$\leq$	125 MPa (18,000 psi)
$(f_u)_{actual} / (f_y)_{actual}$	$\geq$	1.25

where,

$(f_y)_{actual}$	=	Actual yielding strength of the reinforcement based on mill tests, MPa (psi)

$(f_y)_{specified}$	=	Specified yield strength of reinforcement, MPa (psi)
$(f_u)_{actual}$	=	Actual ultimate tensile strength of the reinforcement, MPa (psi)

Reinforcement Characteristic Yield Strength

Requirements for the characteristic yield strength of the reinforcement steel are:

- Maximum allowed characteristic yield strength of longitudinal reinforcement: f_y = 420 MPa (Grade 60 - 60,000 psi);
- Maximum allowed characteristic yield strength of shear reinforcement: f_{yt} = 420 MPa (Grade 60 - 60,000 psi);

References ACI 318

1. American Concrete Institute. Building Code Requirements for Structural Concrete (ACI 318-08) and Commentary. ACI 2008.
2. American Concrete Institute. Building Code Requirements for Structural Concrete (ACI 318M-08) and Commentary. ACI 2008.
3. American Concrete Institute. Building Code Requirements for Structural Concrete (ACI 318-11) and Commentary. ACI 2011.
4. American Concrete Institute. Metric Building Code Requirements for Structural Concrete (ACI 318M-11) and Commentary. ACI 2011.
5. American Concrete Institute. Building Code Requirements for Structural Concrete (ACI 318-14) and Commentary. ACI 2014.
6. American Concrete Institute. Metric Building Code Requirements for Structural Concrete (ACI 318M-14) and Commentary. ACI 2014.

Vibration of floors to DG11

These topics describe the DG11 floor vibration calculations that can be performed in Tekla Structural Designer.

The following topics are covered:

Introduction to DG11 floor vibration

This handbook describes the DG11 floor vibration calculations that can be performed in Tekla Structural Designer.

With the advent of long span floors, multiple openings in webs, minimum floor depth zones etc. the vibration response of floors in multi-storey buildings under normal occupancy has increasingly become of concern to clients and their Engineers and Architects.

Detailed guidance on the subject is available through the AISC Steel Design Guide Series 11: Vibrations of Steel-Framed Structural Systems Due to Human Activity Second Edition, otherwise referred to as DG11. (page 1863)

This handbook describes the method for the assessment of floor vibration in accordance with DG11 that has been adopted in Tekla Structural Designer. The method seeks to establish, with reasonable accuracy, the response of the floor to dynamic excitation expected in offices of normal occupancy. This excitation is almost solely based on occupants walking. With appropriate design criteria, the approach is likely to be equally applicable to sectors other than offices.

The existing solution to checking this type of criterion - a simple calculation of the natural frequency of an individual beam - is felt in many cases to be insufficiently accurate. More importantly, such calculations do not consider two important factors,

- the natural frequency is only the 'response side' of the equation. The 'action' side of the equation is also important i.e. the dynamic excitation - this is the activity that might cause an adverse response from the floor. Walking, dancing and machine vibration are all on the 'action' side of the equation and are all very different in their potential effect.
- the natural frequency of an isolated beam is exactly that and takes no account of the influence (good or bad) of the surrounding floor components. In particular, with composite floors, the slabs will force other beams to restrict or sympathize with the beam under consideration.

The culmination of the calculations carried out by Tekla Structural Designer is the calculation of the peak acceleration of the floor system due to walking excitation expressed as a fraction of the acceleration of gravity.

Scope of DG11 floor vibration

The reference upon which Tekla Structural Designer's floor vibration check is based is the main limiting factor with regard to scope. This publication is AISC Steel Design Guide Series 11: Vibrations of Steel-Framed Structural Systems

Due to Human Activity Second Edition, otherwise referred to as DG11. (page 1863)

You are able to define an area on a particular floor level that is to be subject to the vibration response analysis and design. The layout of beams in real multi-story buildings can be of almost any configuration. **The methodology adopted in DG11 is only applicable to regular structures which by and large have to be created from rectilinear grids.** It is your responsibility to make an appropriate selection of the beams etc. that are to be the basic components of the idealized case.

As you proceed through the input making your selections, Tekla Structural Designer will, where it is possible to do so, interrogate the underlying model and retrieve the appropriate data. Once all the data has been assembled, you are then able to perform the check, after which a detailed set of results will be available for review. If you are unhappy with the outcome of your choices you can close the results window and make alternative selections by editing the Floor Vibration Check item properties.

Limitations and Assumptions of DG11 floor vibration

The scope is primarily defined by the DG11 (page 1863) but the following additional limitations and assumptions should be noted.

- The design guidance is based on composite floors acting compositely with the steel beams. It is unclear whether the design approach is directly applicable to non-composite construction.
- In DG11, if the slab is attached to the supporting member, the construction can be classed as composite for the purpose of carrying out a vibration analysis even in the absence of shear connectors. Tekla Structural Designer does not define such construction as composite and therefore will only class truly composite construction as composite.
- For simplicity and to avoid the necessity of Tekla Structural Designer having to identify all the beams in the area selected for vibration assessment, the component of the unit mass from the self-weight of the beams is ignored. This will lead to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous). Note, however, that beam self-weight is included in the calculation of beam deflection but only when the self-weight loadcase is included in the load combination.
- Cantilever beams are excluded from the analysis.
- In DG11, if a beam or girder is moment connected to a column, its natural frequency can be enhanced because of the flexural restraint offered by the columns. Tekla Structural Designer does not incorporate this feature.
- Shear deflection in beams and girders is included in the analysis carried out by Tekla Structural Designer.
- Precast slabs are excluded.

Design philosophy of DG11 floor vibration

General

The Engineer ensures the safety of building occupants by satisfying all design criteria at the Ultimate Limit State. Similarly, the health of building occupants is partly taken care of when deflection limits at the Serviceability Limit State are satisfied (although this Limit State does have other purposes than simply the health of occupants).

However, for floors that are subject to cyclic or sudden loading, it is the human perception of motion that could cause the performance of a floor to be found unsatisfactory. Such perception is usually related to acceleration levels. In most practical building structures, the reaction of the occupants to floor acceleration varies between irritation and a feeling of insecurity. This is based on the instinctive human perception that motion in a 'solid' building indicates inadequacy or imminent failure.

The working environment also affects the perception of motion. For busy environments, where the occupant is surrounded by the activity that is producing the vibrations, the perception of motion is reduced. In contrast, for quieter environments (such as laboratories and residential dwellings), where the source of vibration is unseen, the perception of motion is significantly heightened.

The design philosophy to ensure that the potential for such human response is minimized, has a number of facets,

- the **dynamic excitation** causing the vibration i.e. the disturbing force profile, which is force and time dependent. For the sorts of building and occupancy considered here, this is the act of walking.
- the **acceptance criteria**. This depends upon the type of environment. As discussed above this, in turn, depends upon the involvement of the occupant in the generation of the vibration and also on the nature of the occupancy. The latter is important for laboratories carrying out delicate work, or operating theaters, for example.
- the **provided performance**. This is the "resonance response function" and is dependent on the system natural frequency and, more importantly, the participating mass. This function is expressed as a ratio of the floor acceleration to the acceleration of gravity.

Dynamic excitation

In a classical spring-mass system that includes a (viscous) damper, when a simple force is applied to the mass to extend (or contract) the spring, the mass moves up and down (oscillates). This movement is significant at first but eventually reduces to zero due to the resistance offered by the damper. In a floor system in a building,

- the mass is the self-weight of the floor and any other loading that is present for the majority of the time that the occupants could be exposed to vibration effects,
- the spring is the stiffness of the floor system, which will have a number of different component beams (joists and girders) and the floor slab,
- the damper is provided by a number of elements that are able to absorb energy from the free vibration of the system. There will be energy absorbed,
  - within connections, since they behave 'better' than the ideal that is assumed
  - from losses due to the unsymmetrical nature of real buildings e.g. grid layout, and dispersion of loads from furnishings and contents
  - from components such as partitions that are out-of-plane of the vibration and interfere with the 'mode'.

The determination of the contribution of each of these components as they affect real floor systems is given in detail in later sections. These describe the 'response' side of the floor system. In order to establish the required performance of the system the 'input' must also be defined i.e. that event, events or continuum that is the 'dynamic excitation'.

In the simple example described at the start of this section the 'input' was simply a force that caused a displacement to the system and was then released. This might be equivalent to a person jumping off a chair onto the floor. However, in the context of the concerns over the vibration of floors, it is not this sort of input that is of interest. The main concern is the excitation of the floor brought about by walking.

Unlike the simple example, walking produces loading that is cyclic. This loading can be idealized into a series of sine curves of load against time. Each curve is an exact multiple of the walking frequency called harmonics. When one of these harmonics of the cyclic loading coincides with the natural frequency of the floor system then resonance is set up. The consequence of resonance that is detected, and may disturb occupants, is the associated peak acceleration. The peak acceleration due to walking is estimated by selecting the lowest harmonic for which the forcing frequency can match a natural frequency of the floor and is dependent upon the applied force (a constant = 0.29kN [65lb] for floors), the mass of the system (the self-weight of the floor plate plus other loading that could be considered as permanent), and the amount of damping in the system (the damping ratio, β).

Hence, the dynamic excitation of a floor is dependent upon the forcing function due to walking and its relationship to the natural frequency of the floor system. It is the level of the peak acceleration that this generates that is particularly important in determining the performance of the floor.

Acceptance criteria for human comfort

The required performance of a floor system is very dependent upon the potential response of humans. Human response is a very complex subject since there is no such thing as a 'standard human'. The perception of vibration will differ from person to person, their body mass varies significantly and the body's reaction will depend upon age, gender etc. The human response has been studied and the acceptance criterion adopted by DG11 was developed using the acceleration limits as recommended by ISO 2631-2, 1989 adjusted for intended occupancy.

The accelerations acceptable for different use of buildings are described using the 'base' limits. Multiplying factors are used to increase the base acceleration limit according to the intended use of the building. The target acceleration ratio of the floor under consideration is given in DG11 guidance as,

- a_o/g = 0.5% for offices, residences, churches, schools and quiet areas
- a_o/g = 1.5% for shopping malls

You should choose a required acceleration ratio based on both engineering judgement and the advice given in DG11.

A separate acceleration ratio limit for high frequency floors (i.e. those floors with fundamental frequency, f_n, in the range $9 < f_n <= 15$ Hz) also needs to be defined by the Engineer, again based on engineering judgement and with reference to DG11 figure 2-1

Design for walking excitation

The start point is the calculation of the natural frequency of the floor system. The fundamental floor frequency, f_n, is evaluated using the Dunkerley relationship for the combined mode.

In accordance with the guidance given at Section 4.1 of DG11, an advisory Warning is displayed when the fundamental frequency of the floor system, f_n, is < 3.0 Hz.

Next the 'Equivalent Panel Weight' is required. This is dependent upon the physical size of the floor plate selected and an effective width and/or length.

The calculation requires the 'damping ratio' - this is a user input.

The damping ratio, the effective panel weight, the fundamental frequency of the floor and the constant excitation force are used to calculate the peak acceleration ratio, a_p/g.

The design condition is simply,

$a_p/g \leq a_o/g$

For floors with high fundamental frequencies, when the calculated value of f_n is in the range 9 Hz < $f_n \leq$ 15 Hz, then an equivalent sinusoidal peak acceleration, a_{ESPA}, is checked against the separate acceleration ratio limit for high frequency floors.

Design for walking excitation DG11

Floor slab

Slab Properties

For composite slabs the transformed moment of inertia per unit width of the slab, D_s, is calculated from,

D_s	$= d_e^3 /(12*n)$	mm^4/mm in metric units
D_s	$= 12*d_e^3/(12*n)$	ins^4/ft in US Customary units

Where

d_e	=	effective depth of slab taken as slab depth less one half depth of steel decking (mm or inches)
n	=	dynamic modular ratio
	=	$E_s /(1.35*E_c)$
E_s	=	the steel modulus (N/mm^2 or ksi)
E_c	=	the concrete modulus (N/mm^2 or ksi)

For generic slabs, the transformed moment of inertia per unit width is to be provided by the user.

Beam panel mode

Beam Panel Mode Deflection

The beam panel mode deflection, Δ_j, is the maximum simply supported deflection of the beam or joist calculated using,

$$\Delta_j = 5 * ((w*b)+w_{swt}) * L_j^4 / (384*E_s *I_{sj})$$

where

w	=	unit supported weight (psi or N/mm^2)
b	=	beam or joist spacing (in or mm)
w_{swt}	=	beam or joist self weight (zero unless self weight loadcase in combination) (pli or N/mm)
L_j	=	span of beam or joist (in or mm)
E_s	=	steel modulus (psi or N/mm^2)
I_{sj}	=	the inertia of the beam or joist from the database (in^4 or mm^4)

Beam Panel Mode Frequency

The beam panel mode fundamental frequency, f_j, is given by,

f_j	=	$0.18*\sqrt{(g/\Delta_j)}$ Hz

Where

g	=	the acceleration of gravity
Δ_j	=	the maximum simply supported deflection of the beam or joist calculated as above.

Beam Panel Mode Effective Weight

The effective panel weight for the beam panel mode, W_j, is given by,

W_j	=	$k_j *w*B_j *L_j$

Where

k_j	=	beam continuity factor
	=	1.0 generally but 1.5 where beams are continuous over supports and an adjacent span is > $0.7*L_j$
w	=	unit supported weight (see Data Derived from

		Tekla Structural Designer (page 1859))
B_j	=	the effective width of the beam panel
	=	C_j *(D_s /D_j)$^{0.25}$ *L_j but ≤ (2/3)*FW
L_j	=	the span of the beam

and

C_j	=	effective width coefficient for beam
	=	2.0 generally but 1.0 for beams parallel to interior edge
D_s	=	transformed moment of inertia of slab per unit width as above
D_j	=	transformed moment of inertia of beam per unit width
	=	I_j /S
S	=	beam spacing
FW	=	floor width
	=	n_g *L_g
n_g	=	number of bays in the direction of the girder span.
L_g	=	the span of the girder

Girder panel mode

Girder Panel Mode Deflection

The girder panel mode deflection, Δ_g, is the maximum girder deflection derived from the analysis model.

NOTE As Δ_g is taken directly from load analysis there is no need for any adjustment to the girder deflection, as suggested in DG11 section 3.1, when there is only one supported beam.

Girder Panel Mode Frequency

The girder panel mode fundamental frequency, f_g, is given by,

f_g	=	0.18*√(g/Δ_g) Hz

Where

g	=	the acceleration of gravity
Δ_g	=	the maximum simply supported deflection of the girder derived from the analysis model.

Girder Panel Mode Effective Weight

The effective panel weight for the girder panel mode, W_g, is given by,

W_g	=	k_g * w *B_g * L_g

Where

k_g	=	girder continuity factor
	=	1.0 generally but 1.5 where girders are continuous over supports and an adjacent span is > 0.7*L_g
w	=	unit supported weight (see Data Derived from Tekla Structural Designer (page 1859))
B_g	=	the effective width of the girder panel

For the general case,

B_g	=	C_g *$(D_j /D_g)^{0.25}$ *L_g but ≤ (2/3)*FL

If the girder is an interior edge girder as specified by the user,

B_g	=	L_j*(2/3)

Where

L_g	=	the span of the girder
C_g	=	effective width coefficient for beam
	=	1.8 generally but 1.6 for girders supporting joists

		connected to the girder flange (joist seats)
D_j	=	transformed moment of inertia of beam per unit width as above
D_g	=	transformed moment of inertia of girder per unit width
	=	I_g/L_j generally but
	=	$2*(I_g/L_j)$ for edge girders
FL	=	floor length
	=	$n_j *L_j$
n_j	=	number of bays in the direction of the beam span.

If the girder supports beams with unequal spans, say L_{j1} and L_{j2}, the average beam span length $L_{av} = (L_{j1}+L_{j2})/2$ should replace L_j in the above equation for D_g. The user should confirm the value to be used in such circumstances.

Combined panel mode

There are three possible conditions is to be checked for the combined mode.

If $L_g/B_j > 1.0$, the combined equivalent panel weight, W_{comb}, is given by,

W_{comb}	=	$(\Delta_j/(\Delta_j+\Delta_g))*W_j + (\Delta_g/(\Delta_j+\Delta_g))*W_g$

The floor fundamental frequency, f_{comb}, is given by

f_{comb}	=	$0.18*\sqrt{g/(\Delta_j+\Delta_g)}$

If $0.5 \le L_g/B_j \le 1.0$, the combined equivalent panel weight, W_{comb}, is given by,

W_{comb}	=	$(\Delta_j/(\Delta_j+\Delta_{gred}))*W_j + (\Delta_{gred}/(\Delta_j+\Delta_{gred}))*W_g$

The floor fundamental frequency, f_{comb}, is given by

f_{comb}	=	$0.18*\sqrt{g/(\Delta_j+\Delta_{gred})}$

Where

Δ_{gred}	=	$(L_g/B_j)^* \Delta_g$

If L_g/B_j < 0.5, the combined equivalent panel weight, W_{comb}, is given by,

W_{comb}	=	$(\Delta_j/(\Delta_j+\Delta_{gred}))^*W_j + (\Delta_{gred}/(\Delta_j+\Delta_{gred}))^*W_g$

The floor fundamental frequency, f_{comb}, is given by

f_{comb}	=	$0.18^*\sqrt{g/(\Delta_j+\Delta_{gred})}$

Where

Δ_{gred}	=	$0.5^* \Delta_g$

In addition, if $L_j/L_g < 0.5$, then the peak acceleration ratio is separately checked for the **beam panel mode** and for the **combined panel mode** as above.

Evaluation

The peak acceleration ratio, a_p/g, is evaluated for each f_n in turn (with its associated W), and is given by,

a_p/g	=	$MAX[100^*P_0^*e^{(-0.35^*fn)}/(\beta^*W)]$ %

where

f_n	=	f_j, f_g, f_{comb}
W	=	the value of W_j, W_g or W_{comb} appropriate to f_n
P_0	=	constant force equal to 0.290 kN [65lb]
β	=	damping ratio

The acceleration limit, a_o/g, is a user input and leads to the final design condition,

a_p/g	≤	a_o/g

NOTE The 'fundamental frequency of the floor' output in the results is the f_n associated with the MAX peak acceleration ratio (and not just the MIN f_n).

High Frequency Floors

For floor systems having a natural frequency greater than 9 Hz (but ≤ 15 Hz), the equivalent sinusoidal peak acceleration ratio, a_{ESPA}/g, is given by

a_{ESPA}/g	=	$MAX[100*(154 / W)*(f_{step}^{1.43}/ f_n^{0.3})*\{[1-e^{(-4*\pi*h*\beta)}]/ (h*\pi*\beta)\}^{0.5}]$ %	US-units
	=	$MAX[100*(686 / W)*(f_{step}^{1.43}/ f_n^{0.3})*\{[1-e^{(-4*\pi*h*\beta)}]/ (h*\pi*\beta)\}^{0.5}]$ %	metric-units

where

W	=	the value of W_j, W_g or W_{comb} appropriate to f_n
f_{step}	=	footstep frequency Hz
f_n	=	fundamental frequency of the floor
h	=	the harmonic matching f_n
	=	5 for 9 Hz < f_n ≤ 11 Hz
	=	6 for 11 Hz < f_n ≤ 13.2 Hz
	=	7 for 13.2 Hz < f_n ≤ 15 Hz
β	=	damping ratio

NOTE The a_{ESPA}/g equation assumes a bodyweight value of 168 lbs [0.75 kN] as indicated in DG11.

The acceleration limit for high frequency floors is a user input and leads to the final design condition,

a_{ESPA}/g	≤	acceleration limit for high frequency floors

Sensitive use analysis DG11

The vibration check calculations can be performed for sensitive equipment & occupancy if required.

These calculations make use of Chapter 6 of DG11 2nd Edition (2016) with revisions and errata of 27 July 2018. They only cover ⅓ octave spectral velocity and acceleration.

For these calculations the mode shape factors ϕ_E and ϕ_W are taken as 1.0. i.e. it is conservatively assumed that the walker and sensitive equipment or sensitive occupant are both at mid-bay.

Use Case - Sensitive Equipment

If walking speed = Very Slow ,

f_{step}	= 1.25 Hz	[Table 6-1]
$V_{1/3}$	= $(250 * 10^6) / (\beta * W) * (f_{step}^{2.43} / f_n^{1.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-3a]
$A_{1/3}/g$	= $(100 * 4.2) / (\beta * W) * (f_{step}^{2.43} / f_n^{0.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-8a]

Else if walking speed = Slow , or Moderate or Fast

f_{step} , f_L , f_U and γ as appropriate to selected walking speed	[Table 6-1]

If $f_n \leq f_L$

$V_{1/3}$	= $(175 * 10^6) / (\beta * W * f_n^{0.5}) * (e^{-\gamma*fn})$	[Eqn 6-3b][1]
$A_{1/3}/g$	= $(100 * 6.4) / (\beta * W) * (e^{-\gamma*fn})$	[Eqn 6-8b][1]

If $f_n \geq f_U$

$V_{1/3}$	= $(250 * 10^6) / (\beta * W) * (f_{step}^{2.43} / f_n^{1.8}) * (1 -$	[Eqn 6-3b][2]

	$e^{-2*\pi*\beta*fn / fstep})$	
$A_{1/3}/g$	$(100 * 4.2) / (\beta * \overline{W}) * (f_{step}^{2.43} / f_n^{0.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-8b][2]

Else if $f_L < f_n < f_U$

For $V_{1/3}$

1. evaluate [Eqn 6-3b][1] with $f_n = f_L$, then
2. evaluate [Eqn 6-3b][2] with $f_n = f_U$, then
3. linear interpolate for actual f_n

For $A_{1/3}/ g$

1. evaluate [Eqn 6-8b][1] with $f_n = f_L$, then
2. evaluate [Eqn 6-8b][2] with $f_n = f_U$, then
3. linear interpolate for actual f_n

Use Case - Sensitive Occupancy

NOTE DG11 2nd Edn does not provide explicit ⅓ octave spectral Acceleration equations for **Sensitive Occupancy**, so in Tekla Structural Designer these equations have been derived from the ⅓ octave spectral Acceleration equations for **Sensitive Equipment** by ‘modifying’ them by factors 200/250 (for very slow walking) and 120/175 (for other walking speeds).

These modification factors are derived from consideration of the differences between ⅓ octave spectral Velocity equations for Sensitive Equipment and Sensitive Occupancy, which are given explicitly in DG11 2nd Edn.

If walking speed = Very Slow ,

f_{step}	1.25 Hz	[Table 6-1]
$V_{1/3}$	$(200 * 10^6) / (\beta * \overline{W}) * (f_{step}^{2.43} / f_n^{1.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-9a]

$A_{1/3}/g$	$(200/250)*(100 * 4.2) / (\beta * W) * (f_{step}^{2.43} / f_n^{0.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-8a] modified

Else if walking speed = Slow , or Moderate or Fast

f_{step} , f_L , f_U and γ as appropriate to selected walking speed	[Table 6-1]

If $f_n \leq f_L$

$V_{1/3}$	$(120 * 10^6) / (\beta * W * f_n^{0.5}) * (e^{-\gamma*fn})$	[Eqn 6-9b][1]
$A_{1/3}/g$	$(120/175)*(100 * 6.4) / (\beta * W) * (e^{-\gamma*fn})$	[Eqn 6-8b][1] modified

If $f_n \geq f_U$

$V_{1/3}$	$(200 * 10^6) / (\beta * W) * (f_{step}^{2.43} / f_n^{1.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-9b][2]
$A_{1/3}/g$	$(200/250)*(100 * 4.2) / (\beta * W) * (f_{step}^{2.43} / f_n^{0.8}) * (1 - e^{-2*\pi*\beta*fn / fstep})$	[Eqn 6-8b][2] modified

Else if $f_L < f_n < f_U$

For $V_{1/3}$

1. evaluate [Eqn 6-9b][1] with $f_n = f_L$, then
2. evaluate [Eqn 6-9b][2] with $f_n = f_U$, then
3. linear interpolate for actual f_n

For $A_{1/3}$/ g

1. evaluate [Eqn 6-8b][1] modified with $f_n = f_L$, then
2. evaluate [Eqn 6-8b][2] modified with $_n = f_U$, then
3. linear interpolate for actual f_n

Input requirements for DG11 floor vibration

General

The simplified method for the analysis of the vibration of floors given in the AISC Publication DG11, on which on which the Tekla Structural Designer check is based, is only applicable to regular structures which, by and large, are created from rectilinear grids.

Of course the floor layouts of 'real' multi-storey buildings are rarely uniform and Tekla Structural Designer therefore provides you with the opportunity to select the more irregular floor areas to be assessed with grids that are other than rectilinear.

In so far as the selection of the beams and girders to be used in the analysis is concerned, only beams or girders with Non-Composite, Steel Joist or Composite attributes are valid for selection and, within these confines, the user should be able to:

- select a single beam or girder
- select a girder span as critical plus an adjoining span (in a two or three span configuration)

In all cases, and subject to the above restrictions, which beams and girders from the selected area of floor are chosen is entirely at your discretion and under your judgement, but it is expected that the beams and girders chosen will be those that are typical, common or the worst case. Irrespective,Tekla Structural Designer will take these beams as those that form the idealized floor layout. There is no validation on what the you select (although there is some validation on which beams and girders are selectable i.e. those which have no slab for part of their length, those from angle sections, those with no adjoining span when a 2-span configuration is chosen, and those with no adjoining span at both ends when a 3-span configuration is chosen will not be selectable).

Data Derived from Tekla Structural Designer

Note that, where appropriate, the derived data is for each design combination under SLS loads only.

Unit supported weight

The unit supported weight is used to establish the 'effective panel weight' - that is the weight of the floor and its permanent loading that has to be set in motion during vibration of the floor. It is taken as the slab self-weight (and to be accurate, the beam self-weight), other permanent 'Dead' loads and the proportion of the 'Live' loads (expressed as a percentage) that can be considered as permanent.

The unit supported weight is obtained by summing all the loads (or the appropriate percentage in the case of live loads) that act over or in the selected area. This includes any blanket, area, line and spot loads that are present within the selected area. The component of any of these load types that lie outside of the selected area are ignored. Nodal loads directly on columns are also ignored. The total load is then divided by the area selected.

The slab self-weight will usually be in the Slab Dry loadcase - note that in the case of composite slabs this includes the weight of decking. The beam self-weight is in a separate protected loadcase. For simplicity this component of the unit supported weight is ignored. This leads to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous).

Note that the use of live load reductions has no effect on the floor vibration check.

Slab data

If there are more than one set of slab attributes in the selected area then you have to choose which of these it is appropriate to use. From the designated slab attributes the following information/data is obtained,

- the transformed inertia in mm^4 per mm width [in^4/ft].
- the short-term modular ratio for normal or lightweight concrete as appropriate.

If the designated slab attributes are for a 'generic' slab, then you are asked for the transformed inertia and the dynamic modular ratio.

Beam data

When these are non-composite beams, the inertia is obtained from the sections database. When these beams are of composite construction the inertia is the gross, uncracked composite inertia based on the dynamic modular ratio that is required. Steel joist inertias from the database are assumed to be 'gross' inertias of the chords and are editable. Following guidance contained in AISC Steel Design Guide 11 (page 1863), section 3.5, the gross steel joist inertia is factored by quantity C_r and displayed as the 'effective' inertia in the results viewer. C_r is derived from the following formulas,

For joists with angle web members:

C_r	$0.90*(1 - e^{-0.28(L/D)})^{2.8}$	valid for $6 \le L/D \le$ 0.9

For joists with rod web members:

C_r	=	0.721 + 0.00725*(L/D)	valid for 10 ≤ L/D ≤ 0.9

Where

L	=	Joist span
D	=	Joist nominal depth

NOTE There is no guidance in DG11 what to do if L/D < 6 for angle web members or L/D < 10 for rod web members. Therefore, in these cases Tekla Structural Designer calculates C_r with L/D = 6 (angle) or L/D = 10 (rod).

The span of the critical/base beam and the adjoining beams is required.

The deflection of the critical beam under the permanent loads is required. To calculate this value, the deflection under the Dead loads and the appropriate percentage of the Live load deflection is summed.

Girder data

The same data is required as that for the beams.

Floor plate data

The dimensions of the floor plate in the idealized cases are defined in one direction by the number of beam bays and in the orthogonal direction by the number of girder bays. In practice, given that the idealized case may not attain, the floorplate dimensions are derived from the slab items you select as participating in the mass.

User Input Data

Secondary Beam Spacing

You must confirm the spacing of the secondary beams - an average value when the spacing is non-uniform.

Permanent Live Loads

You are required to specify the proportion of the permanent live loads that are to be used in the vibration analysis as a percentage of the live load.

Beam Continuity Factor

This has a value of either 1.0 or 1.5 and you are offered the choice when a two or three span beam configuration is selected. The default value is 1.0.

Girder Continuity Factor

This has a value of either 1.0 or 1.5 and you are offered the choice when a two or three span girder window is selected. The default value is 1.0.

This factor is not generally applicable for girders as they usually frame directly into columns and this potential increase in the effective panel weight does not apply under those circumstances.

Beam and Girder location

You are required to specify whether the beam and the girder under consideration is an "internal" location or an "internal edge" location. This information is required to set the constants that are used in calculating the effective widths of the beam and girder panel modes.

Number of bays used to establish effective panel weight

You are required to specify the number of bays in the direction of the beam span, nj, and the number of bays in the direction of the girder span, ng, that are to be used to establish the effective panel weight. The number of bays ranges from 1 to 4 for both directions.

Damping ratio

Floors do not vibrate as a free mass but have some damping i.e. dissipation of the energy in the system. Values of the damping ratio for individual components, β_i are recommended in DG11 as,

Recommended Component Damping Values	
Component	**Ratio of Actual Damping-to-Critical Damping, β_i**
Structural System	0.01
Ceiling and ductwork	0.01
Electronic office fit-out	0.005
Paper office fit out	0.01
Churches, schools and malls	0.0
Lightly furnished quiet spaces	0.005
Full-height dry wall partitions in bay	0.02 to 0.05*
*Depending on the number of partitions in the bay and their location; near the center of the bay provides more damping.	

For example, a floor with ceiling and ductwork supporting an electronic office area has $\beta = \Sigma\beta_i$ = 0.01 (floor) + 0.01 (ceiling and ductwork) + 0.005 (electronic office area) = 0.025, or 2.5% critical damping.

Since an even higher damping ratio might be justified for storage floors for example, a range of up to 10% is offered.

Acceleration limit

You must enter acceleration limits appropriate to the floor under consideration, one limit for floors with frequencies in the range 3-9 Hz and a separate limit for High Frequency floors (in the range 9-15 Hz). These limits will be based on your engineering judgement and the advice given in DG11 - which gives a range of values between 0.5% and 5.0% depending on structural form (for building floors see the table below).

Table 4-1. Recommended Tolerance Limits for Building Floors

Occupancy	Acceleration Limit a_o/g x 100%
Offices, residences, churches, schools and quiet areas	0.5%
Shopping Malls	1.5%

Footstep frequency

You should enter the footstep frequency to be used if a high frequency floor is detected. The range being 1.2 Hz to 2.2 Hz.

Sensitive use

You have to specify the Use case and walking speed if this analysis is performed. See: Sensitive use analysis DG11 (page 1855)

Vibration of floors to DG11 references

1. **AISC Steel Design Guide Series.**11: Vibrations of Steel-Framed Structural Systems Due to Human Activity Second Edition (2016) with revisions and errata of 27 July 2018.

14.2 Eurocodes

Loading (Eurocode)

This handbook provides a general overview of how loadcases and combinations are created in Tekla Structural Designer when the head code is set to the base Eurocode, or Eurocode with a specific National Annex applied. The Eurocode Combination generator is also described.

The following topics are covered:

- Nationally Determined Parameters (NDP's) (Eurocode) (page 1864)
- Load cases (Eurocode) (page 1871)
- Combinations (Eurocode) (page 1875)
- Minimum lateral load requirements of the Singapore National Annex (Eurocode) (page 1882)

Nationally Determined Parameters (NDP's) (Eurocode)

The Eurocode has differing NDP's for the Eurocode (Base) and for each of Eurocode (UK), Eurocode (Irish) etc. These are defined in the relevant country's National Annex.

Gamma (γ) factors and psi (Ψ) factors for each National Annex are listed below:

Combination gamma factors

Factor	EC Base Value	UK Value	Irish Value	Singapore Value	Malaysia Value	Finland Value	Norway Value	Sweden Value
EQU combs								
$\gamma_{Gj,sup}$	1.10	1.10	1.10	1.10	1.10	$1.10k_{FI}$	1.2	$1.1\gamma_d$
$\gamma_{Gj,inf}$	0.9	0.9	0.9	0.9	0.9	0.9	0.9	0.9
γ_Q (fav)	1.5	1.5	1.5	1.5	1.5	$1.5k_{FI}$	1.5	$1.5\gamma_d$
STR combs								
$\gamma_{Gj,sup}$	1.35	1.35	1.35	1.35	1.35	$1.35k_{FI}$	1.35	$1.35\gamma_d$
$\gamma_{Gj,inf}$	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0
γ_Q (fav)	1.5	1.5	1.5	1.5	1.5	$1.5k_{FI}$	1.5	$1.5\gamma_d$
ξ	0.85	0.925	0.85	0.925	0.925	0.85	0.89	0.89
GEO combs								
$\gamma_{Gj,sup}$	1.0	1.0	1.0	1.0	1.0	$1.0k_{FI}$	1.0	$1.1\gamma_d$
$\gamma_{Gj,inf}$	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0
γ_Q	1.3	1.3	1.3	1.3	1.3	$1.3k_{FI}$	1.3	$1.4\gamma_d$

The k_{FI} factor used in the Finnish National Annex is set by specifying an appropriate Consequence Class in the Structure Properties (accessed via the Project Workspace).

- CC3 (high consequence for loss of human life or economic social or environmental consequences very great)
- CC2 (medium consequence for loss of human life or economic social or environmental consequences considerable)
- CC1 (Low consequence for loss of human life, economic, social or environmental consequences small or negligible)

The γ_d factor used in the Swedish National Annex is set by specifying an appropriate Reliability Class in the Structure Properties (accessed via the Project Workspace).

- RC3 (major risk of serious personal injury)
- RC2 (some risk of serious personal injury)
- RC1 (minor risk of serious personal injury)

psi factors

UK, Ireland, Singapore Malaysia

Factor	EC Base value			UK value			Irish value			Singapore value			Malaysia value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
Category A - imposed domestic / residential	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3
Category B - imposed	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3

Factor	EC Base value			UK value			Irish value			Singapore value			Malaysia value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
office															
Category C - imposed congregation	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6
Category D- imposed shopping	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6
Category E- imposed storage	1.0	0.9	0.8	1.0	0.9	0.8	1.0	0.9	0.8	1.0	0.9	0.8	1.0	0.9	0.8
Category H- imposed roofs	0	0	0	0.7	0	0	0.6	0	0	0.7	0	0	0.7	0	0
Snow	0.5	0.2	0	0.5	0.2	0	0.5	0.2	0	0.5	0.2	0	0.5	0.2	0

Factor	EC Base value			UK value			Irish value			Singapore value			Malaysia value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
Loads < 1000m															
Snow Loads > 1000m Snow $s_k \geq 2.75$ kN/m^2 Snow ≥ 2 kN/m^2 ie loads (2 ≤ s_k < 3 kN/m^2)	0.7	0.5	0.2	0.5	0.2	0	0.5	0.2	0	0.5	0.2	0	0.5	0.2	0
Ice Loads	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
Wind Loads	0.6	0.2	0	0.5	0.2	0	0.6	0.2	0	0.5	0.2	0	0.5	0.2	0
Thermal	0.6	0.5	0	0.6	0.5	0	0.6	0.5	0	0.6	0.5	0	0.6	0.5	0

Factor	EC Base value			UK value			Irish value			Singapore value			Malaysia value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
Loads															

Nordic Countries

Factor	EC Base value			Finish value			Norwegian value			Swedish value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
Category A - imposed domestic / residential	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3
Category B - imposed office	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3	0.7	0.5	0.3
Category C - imposed congregation	0.7	0.7	0.6	0.7	0.7	0.3	0.7	0.7	0.6	0.7	0.7	0.6
Category D- imposed shopping	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6	0.7	0.7	0.6

Factor	EC Base value			Finish value			Norwegian value			Swedish value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
Category E- imposed storage	1.0	0.9	0.8	1.0	0.9	0.8	1.0	0.9	0.8	1.0	0.9	0.8
Category H- imposed roofs	0	0	0	0	0	0	0	0	0	0	0	0
Snow Loads (< 1000m) ($1 \le s_k < 2$ kN/m^2)	0.5	0.2	0	0.7	0.4	0.2	0.7	0.5	0.2	0.6	0.3	0.1
Snow Loads > 1000m Snow $s_k \ge 2.75$ kN/m^2 Snow ≥ 2	0.7	0.5	0.2	0.7	0.5	0.2	0	0	0	0.7	0.4	0.2

Factor	EC Base value			Finish value			Norwegian value			Swedish value		
	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2	Ψ_0	Ψ_1	Ψ_2
kN/m^2 ie loads ($2 \leq s_k < 3$ kN/m^2)												
Snow Loads ≥ 3kN/m^2	0	0	0	0	0	0	0	0	0	0.8	0.6	0.2
Ice Loads	0	0	0	0.7	0.3	0	0	0	0	0	0	0
Wind Loads	0.6	0.2	0	0.6	0.2	0	0.6	0.2	0	0.3	0.2	0
Thermal Loads	0.6	0.5	0	0.6	0.5	0	0.6	0.5	0	0.6	0.5	0

Seismic phi (Ψ) factors

Nordic countries

An additional phi (Ψ) factor for Seismic design - from BS EN 1998-1 Table 4.2 is required for creation of the Seismic Inertia Combination. This is defined with each new imposed or roof imposed loadcase - alongside the psi factors for each of the load types.

Category	Default for		Factor φ from BS EN 1998-1 Table 4.2			
			EC	No Finnish NA so use EC	Norwegian	No Swedish NA so use EC

Category	Default for		Factor φ from BS EN 1998-1 Table 4.2			
A - C	Roof Imposed	Roofs	1.0	1.0	1.0	1.0
A - C	Imposed = A, B, C	Stories with correlated occupancies	0.8	0.8	1.0	0.8
A - C		Stories independently occupied	0.5	0.5	1.0	0.5
D - F	Imposed = D, E	D-F and Archives	1.0	1.0	1.0	1.0

Load cases (Eurocode)

Loadcase types (Eurocode)

The following load case types can be created:

Loadcase yype	Calculated automatically	Include in the Combination Generator	Imposed load reductions	Pattern load
self weight (beams, columns and walls)	yes/no	yes/no	N/A	N/A
slab wet	yes/no	N/A	N/A	N/A
slab dry	yes/no	yes/no	N/A	N/A
dead	N/A	yes/no	N/A	N/A
imposed	N/A	yes/no	yes/no	yes/no

Loadcase yype	Calculated automatically	Include in the Combination Generator	Imposed load reductions	Pattern load
roof imposed	N/A	yes/no	N/A	N/A
wind	N/A	yes/no	N/A	N/A
snow	N/A	yes/no	N/A	N/A
snow drift	N/A	yes/no	N/A	N/A
temperature	N/A	N/A	N/A	N/A
settlement	N/A	N/A	N/A	N/A
seismic	N/A	yes	N/A	N/A

As shown above, self weight loads can all be determined automatically. However, other gravity load cases have to be applied manually as you build the structure.

Self weight (Eurocode)

Self weight - excluding slabs loadcase

Tekla Structural Designer automatically calculates the self weight of the structural beams/columns for you. The **Self weight - excluding slabs** loadcase is pre-defined for this purpose. Its loadcase type is fixed as “Selfweight”. It cannot be edited and by default it is added to each new load combination.

Self weight of concrete slabs

Tekla Structural Designer expects the wet and dry weight of concrete slab to be defined in separate loadcases. This is required to ensure that members are designed for the correct loads at construction stage and post construction stage.

The **Slab self weight** loadcase is pre-defined for the dry weight of concrete post construction stage, its loadcase type is fixed as “Slab Dry”.

There is no pre-defined loadcase for the wet weight of concrete slab at construction stage, but if you require it for the design of any composite beams in the model the loadcase type should be set to “Slab Wet”.

Tekla Structural Designercan automatically calculate the above weights for you taking into account the slab thickness, the shape of the deck profile and wet/dry concrete densities. It does not explicitly take account of the weight of any reinforcement but will include the weight of decking. Simply click the **Calc Automatically** check box when you create each loadcase. When calculated in this way you can’t add extra loads of your own into the loadcase.

If you normally make an allowance for ponding in your slab weight calculations, Tekla Structural Designer can also do this for you. After selecting the composite slabs, you are able to review the slab item properties - you will find two ways to add an allowance for ponding (under the slab parameters heading). These are:

- as a value, by specifying the average increased thickness of slab
- or, as a percentage of total volume.

Using either of these methods the additional load is added as a uniform load over the whole area of slab.

Imposed and roof imposed loads (Eurocode)

Definition of psi factors for imposed load cases

In the Loadcase dialog when an imposed loadcase is selected, you are able to select the Category of imposed load as follows - default Category B - office:

- Category A - domestic/residential
- Category B - office
- Category C - congregation
- Category D - shopping
- Category E - storage

The default values of Ψ_0, Ψ_1 and Ψ_2 vary depending on the category selected and also with the National Annex being worked to. The values can be edited if required. See “psi factors”

Definition of psi factors for roof imposed load cases

Roof imposed loads are not categorised so the default values of Ψ_0, Ψ_1 and Ψ_2 only vary depending on the National Annex being worked to. Again, the values can be edited if required.

Imposed load reduction (Eurocode)

Reductions can be applied to imposed loads to take account of the unlikelihood of the whole building being loaded with its full design imposed load. Reductions can not however be applied to roof imposed loads.

NOTE If the imposed load is considered as an accompanying action (i.e. a Ψ factor is applied to the imposed load case in a combination) then as stated in the Base Eurocode cl 3.3.2, the imposed load reduction should not be applied at the same time.

Imposed loads are only automatically reduced on:

- Columns of any material
- Concrete walls, mid-pier or meshed

The method used for determining the reductions is dependant on the National Annex:

- In the Base Eurocode a formula is given in cl 6.3.1.2(11), this is also used if the Irish, Finish, Norwegian, Swedish or Singaporean National Annex is selected.
- In the UK, and Malaysia the NA permits an alternative method of reduction using NA 2.6.

Although the code allows for imposed load reductions to be applied to floors, Tekla Structural Designer does not implement this automatically. For steel beams, concrete beams, slabs and mats it is however possible to define the level of imposed load reduction manually via the beam/slab item properties.

This is particularly relevant for the design of transfer beams/slabs:

- The imposed load reduction for beams, slabs and mats is intended to work with loads applied from columns acting on the beam or slab when the slab is acting in transfer or for a mat foundation supporting a column. (The theory being that if you want to design the columns for the reduced axial load, you should also design the supporting member for the reduced axial load applied by the column.)
- The engineer would need to work out the reduction of the axial load in the column and apply this as a the reduction percentage, i.e. if the raw axial load in the column is 100kN and the reduced load is 60kN, the reduction is 40%. You would than apply the 40% reduction to the transfer beam/slab or mat as well.
- The reduction is not applied to loads for analysis - it is a post-analysis process which does not affect the analysis results. It does not get applied solely to the imposed load applied directly to the beam or slab panel, but instead is applied to the design moment used in the beam/slab or mat design process.

Snow and snow drift loads (Eurocode)

Definition of psi factors for snow and snow drift load cases

In the Loadcase dialog when a snow, snow drift, or ice loadcase is selected, the default values of Ψ_0, Ψ_1 and Ψ_2 are displayed. These vary depending on the National Annex being worked to. The values can be edited if required. See “psi factors”

NOTE Snow drift loads are considered to be accidental load cases and are combined in the Accidental combinations.

Wind loads (Eurocode)

EC1-4 Wind wizard...

NOTE The **Wind Wizard...** used for automatic wind loadcase generation is fully explained in the Wind Modelling Engineer’s Handbook.

The Wind Wizard is run to create a series of static forces that are combined with other actions due to dead and imposed loads in accordance with BS EN 1990.

The following assumptions/limitations exist:

- The shape of the building meets the limitations allowed for in the code.
- It must be a rigid structure.
- The structure must be either enclosed or partially enclosed.
- Parapets and roof overhangs are not explicitly dealt with.

Simple wind loading

If use of the **Wind Wizard...** is not appropriate for your structure then wind loads can be applied via element or structure loads instead.

Definition of psi factors for wind load cases

In the Loadcase dialog when a wind loadcase is selected, the default values of Ψ_0, Ψ_1 and Ψ_2 are displayed. These vary depending on the National Annex being worked to. The values can be edited if required. See "psi factors".

Combinations (Eurocode)

Once your load cases have been generated as required, you then combine them into load combinations; these can either be created manually, by clicking **Add...** - or with the assistance of the Combinations Generator, by clicking **Generate...**

- Manually defined combinations (Eurocode) (page 1875)
- Nationally Determined Parameters (NDP's) (Eurocode) (page 1864)
- Equivalent horizontal forces (EHF) (Eurocode) (page 1876)
- Combination generator (Eurocode) (page 1877)
- Combination classes (Eurocode) (page 1880)

NOTE The Foreword to the Singapore National Annex to EN 1991-1-4 Wind Actions has a minimum horizontal load requirement (1.5% characteristic dead weight). Therefore if this National Annex has been applied, we are assuming that the wind load applied in manually defined combinations, or via the combination generator, satisfies this minimum horizontal load requirement. See: Minimum lateral load requirements of the Singapore National Annex (page 1882)

Manually defined combinations (Eurocode)

As you build up combinations manually, the combination factors are automatically adjusted as load cases are added and removed from the combination.

If you add/remove a load case type from a combination - the factors are defaulted as follows:

- 'Self weight' - default Strength factor = 1.35, default Service factor = 1.0
- 'Slab Dry' - default Strength factor = 1.35, default Service factor = 1.0
- 'Dead' - default Strength factor = 1.35, default Service factor = 1.0
- 'Imposed'- default Strength factor = 1.5, default Service factor = 1.0
- 'Roof Imposed'- default Strength factor = 1.5, default Service factor = 1.0
- With an Imposed load case
    - 'Wind' - default Strength factor = 0.75, default Service factor = 0.5
    - 'Snow' - default Strength factor = 0.75, default Service factor = 0.5
- With No Imposed load case
    - 'Wind' - default Strength factor = 1.5, default Service factor = 1.0
    - With Wind load case
        - 'Snow' - default Strength factor = 0.75, default Service factor = 0.5
    - With no Wind load case
        - 'Snow' - default Strength factor = 1.5, default Service factor = 1.0
- 'Snow drift'- default Strength factor = 1.0, default Service factor = 1.0
- 'Temperature'- default Strength factor = 1.0, default Service factor = 1.0
- ‘Settlement'- default Strength factor = 1.0, default Service factor = 1.0

Equivalent horizontal forces (EHF) (Eurocode)

EHFs are used to represent frame imperfections. The Eurocode requires they are applied to all combinations. (Lateral wind combinations therefore should also have EHFs applied).

EHFs are automatically derived from the factored load cases within the current combination. They are applied in the analysis as a horizontal force at each beam column intersection as a specified percentage of the vertical load in the column at the column/beam intersection.

Settings that control the EHF percentage can be adjusted from **Home** --> **Model Settings** --> **EHF** . (The default settings conservatively result in 0.5% EHF in both directions).

EHFs are applied to the structure in the building directions 1 and 2 as follows:

- EHF Dir1+

- EHF Dir1-
- EHF Dir2+
- EHF Dir2-

Combination generator (Eurocode)

The Combination generator is accessed via the **Generate...** button. This automatically sets up combinations for both strength and serviceability.

Combination generator - Initial Parameters

At the start of the generator, you need to define certain parameters so that the correct combinations are created - these are described below:

Combination for design of structural members (STR)

You can chose between:

- Table A1.2(B) - Eq 6.10, or
- Table A1.2(B) - Eq 6.10,a&b

Eq 6.10 is always equal to or more conservative than either 6.10a or 6.10b. The most economic combination of 6.10a or b will depend on if permanent actions are greater than 4.5 times the variable actions (except for storage loads).

Include GEO combinations - Table A1.2(C) - Eq 6.10

You should check this option in order to create the GEO combinations required for foundation design.

Include Accidental combinations - Table A2.5 Eq 6.11a&b

If you have defined an accidental load type such as Snow drift you should check this option for the correct load combinations to be generated.

NOTE The Combinations Generator refers to the relevant National Annex when determining the g factors to apply in the above combinations, as they may vary from the Base Eurocode values.

Include Seismic combinations - Table A2.5 Eq 6.12a&b

If you have defined seismic loads you should check this option for the correct load combinations to be generated.

NOTE Temperature and settlement load case types not included in the Generator at all - these have to be added manually.

Combination generator - Combinations

The second page of the generator lists the combinations applicable (with appropriate factors) for the selections made on the first page. Any factors in bold will be multiplied by the relevant psi factors for that load case.

The type of structure chosen on the previous page affects which combinations default to being generated.

The combination names are automatically generated as per the table below:

No.	BS EN 1990 State and eqn	Type	Load combination
1	**Str – 6.10**	Gravity	Str_1 - $\gamma_{GJ,sup}D + \gamma_Q I + \gamma_Q RI$
2	"	"	Str_2 - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q S$
3	"	Lateral (EHF)	$Str_{3.n}$ -$\gamma_{GJ,sup}D + \gamma_Q I + \gamma_Q RI + EHF$
4	"	"	$Str_{4.n}$ - $\gamma_{GJ,sup}D + \gamma_Q I + \gamma_Q \Psi_0 S + EHF$
5	"	"	$Str_{5.n}$ - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q S + EHF$
6	"	Lateral (Wind)	$Str_{6.n}$ - $\gamma_{GJ,sup}D + \gamma_Q I + \gamma_Q \Psi_0 S + \gamma_Q \Psi_0 W + EHF$
7	"	"	$Str_{7.n}$ - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q S + \gamma_Q \Psi_0 W + EHF$
8	"	"	$Str_{8.n}$ - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q \Psi_0 S + \gamma_Q W + EHF$
9	"	Uplift	$Str_{9.n}$ - $\gamma_{GJ,inf}D + \gamma_Q W + EHF$
1	**Str – 6.10a&b**	Gravity	Str_1 - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q \Psi_0 RI$
2	"	"	Str_2 - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q \Psi_0 S$
3	"	"	Str_3 - $\xi\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q RI$
4	"	"	Str_4 - $\xi\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q S$
5	"	Lateral (EHF)	$Str_{5.n}$ - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q \Psi_0 RI + EHF$
6	"	"	$Str_{6.n}$ - $\gamma_{GJ,sup}D + \gamma_Q \Psi_0 I + \gamma_Q \Psi_0 S + EHF$

No.	BS EN 1990 State and eqn	Type	Load combination
7	“	“	$Str_{7.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q RI$ + EHF
8	“	“	$Str_{8.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q \Psi_0 S$ + EHF
9	“	“	$Str_{9.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q S$ + EHF
10	“	Lateral (Wind)	$Str_{10.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q \Psi_0 S$ + $\gamma_Q \Psi_0 W$ + EHF
11	“	“	$Str_{11.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q \Psi_0 S$ + $\gamma_Q \Psi_0 W$ + EHF
12	“	“	$Str_{12.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q S$ + $\gamma_Q \Psi_0 W$ + EHF
13	“	“	$Str_{13.n}$ - $\xi \gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q \Psi_0 S$ + $\gamma_Q W$ + EHF
14	“	Uplift	$Str_{14.n}$ - $\gamma_{GJ,inf}D$ + $\gamma_Q W$ + EHF
1	**Geo - 6.10**	Lateral (EHF)	$Geo_{1.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q RI$ + EHF
2	“	“	$Geo_{2.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q \Psi_0 S$ + EHF
3	“	“	$Geo_{3.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q S$ + EHF
4	“	Lateral (Wind)	$Geo_{4.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q I$ + $\gamma_Q \Psi_0 W$ + $\gamma_Q \Psi_0 S$ + EHF
5	“	“	$Geo_{5.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q S$ + $\gamma_Q \Psi_0 W$ + EHF
6	“	“	$Geo_{6.n}$ - $\gamma_{GJ,sup}D$ + $\gamma_Q \Psi_0 I$ + $\gamma_Q \Psi_0 S$ + $\gamma_Q W$ + EHF

No.	BS EN 1990 State and eqn	Type	Load combination
7	“	Uplift	$Geo_{7.n}$ - $\gamma_{GJ,inf}D$ + $\gamma_{Q,1}W$ + EHF
1	**Acc 6.11**	Lateral (EHF)	$Acc_{1.n}$ - G + A_d + Ψ_1I + EHF
2	“	Lateral (Wind)	$Acc_{2.n}$ - G + A_d + Ψ_2I + Ψ_1W + EHF
	Seis 6.12	Seismic	$Seis_{.n}$ - G + A_{Ed} + Ψ_2RI + Ψ_2S + EHF
			$Seis_{.n}$ - G + A_{Ed} + EHF

NOTE If working to the Swedish NA (EK11), although shown in the above table, variable actions are no longer considered in Tekla Structural Designer combinations of actions based on equation 6.10a

Combination Generator - Service Factors

This page indicates which combinations are to be checked for serviceability and the factors applied.

Combination Generator - Wind/EHF Directions

This page is used to select which EHF direction goes with each combination containing a specific wind load case.

All wind load cases are listed vertically, and the four EHF options (+Dir1, -Dir1, +Dir2, -Dir2) are each displayed with a factor (default 1.000).

By default (on first entry), none of the directions are set for any wind load case. You are required to set at least one for every wind load case and can set two, three or all four if you wish- these are then used when generating the combinations.

Combination Generator - EHF

The last page is used to set up the equivalent horizontal forces. You can specify EHF’s and factors in each of four directions.

For each direction selected a separate EHF combination will be generated. Any combination with wind in is automatically greyed as all the required information has already been set via the previous page.

Click **Finish** to see the list of generated combinations.

Combination classes (Eurocode)

Having created your combinations you classify them as: Construction Stage, Gravity, Lateral, Seismic, or Modal Mass.

NOTE If generated via the Combinations generator they are classified for you automatically.

Then (where applicable) you indicate whether they are to be checked for strength or service conditions, or both. You also have the option to make any of the combinations inactive.

Construction stage combination (Eurocode)

A Construction Stage load combination is only required for the purpose of designing any composite beams within the model. It is distinguished from other combinations by setting its “Class” to Construction Stage.

Typically this combination would include a loadcase of type “Slab Wet”, (not “Slab Dry”), other loadcases being included in the combination as required.

If you add/remove a load case type from this combination - the factors are defaulted as follows:

- Self weight - default Strength factor = 1.35, default Service factor = 1.0
- Slab Wet - default Strength factor = 1.35, default Service factor = 1.0
- Dead - default Strength factor = 1.35, default Service factor = 1.0
- Imposed - default Strength factor = 1.5, default Service factor = 1.0

NOTE The Slab Wet loadcase type should not be included in any other combination.

Gravity combination (Eurocode)

These combinations are considered in both the Gravity Sizing and Full Design processes.

They are used in the Gravity Sizing processes as follows:

- Design Concrete (Gravity) - concrete members in the structure are automatically sized (or checked) for the gravity combinations
- Design Steel (Gravity) - steel members in the structure are automatically sized (or checked) for the gravity combinations.
- Design All (Gravity) - all members in the structure are automatically sized (or checked) for the gravity combinations.

They are also used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the gravity combinations.
- Design Steel (All) - steel members in the structure are automatically sized (or checked) for the gravity combinations.

- Design All (All) - all members in the structure are automatically sized (or checked) for the gravity combinations.

Quasi Permanent SLS Gravity Combination

In order to cater for the quasi-permanent SLS load combination, a gravity combination is permitted to have two SLS sets of factors. The quasi permanent combination is only used for the spacing of reinforcement calculation for RC beams (and nothing else).

Lateral combinations (Eurocodes)

These combinations are **not** used in the Gravity Sizing processes.

They are used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the lateral combinations.
- Design Steel (All) - steel members in the structure which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.
- Design All (All) - all concrete members and all steel members which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.

Seismic Combinations (Eurocode)

NOTE Although included in this documentation, these are only available for use in regions where seismic design is required.

These combinations are only considered during the Full Design process. They are not used in the Gravity Sizing process.

Modal mass combinations (Eurocode)

For modal analysis, you are required to set up specific “modal mass” combinations. Provided these combinations are active they are always run through the modal analysis.

NOTE It is always assumed that all loads in the load cases in the combination are converted to mass for modal analysis. You are permitted to add lumped mass directly to the model.

Minimum lateral load requirements of the Singapore National Annex (Eurocode)

The foreword to the “Singapore National Annex to EN 1991-1-4 Wind Actions” states:

- “For continuation of an established design philosophy, all buildings should be capable of resisting, as a minimum, a design ultimate horizontal load applied at each floor or roof level simultaneously equal to 1.5% of the characteristic dead weight of the structure between mid-height of the storey below and either mid-height of the storey above or roof surface. The design ultimate wind load should not be taken as less than this value when considering load combinations.”

In Tekla Structural Designer this requirement can be met by applying lateral loads at each floor level equal to 1.5% of the dead load at that level; these can then be designed for if they exceed the design ultimate wind load.

The checking procedure to be followed can be summarised as:

1. Establish the minimum lateral load to be resisted.
2. Compare the horizontal reaction this produces against that of the existing wind load.
3. If the minimum lateral load is greater than the wind load it must be designed for as necessary in up to four directions (+/- Dir 1, +/- Dir 2); if it is less, then no further action is required.

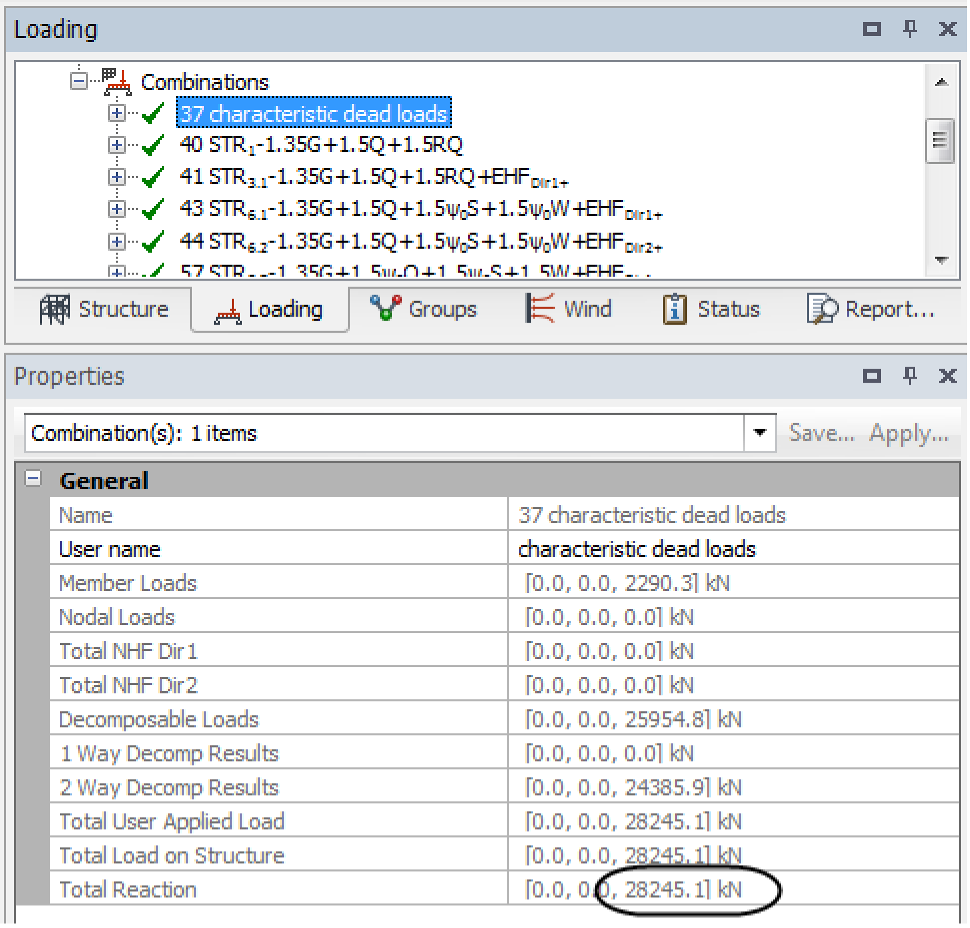
The checking procedure in detail

NOTE EN1994 requires all wind combinations to include EHFs - Settings that control the magnitude of EHFs can be adjusted from Home > Model Settings > EHF. (The default settings conservatively result in 0.5% EHF in both directions). It is recommended that these settings are reviewed prior to undertaking the procedure described below. (Otherwise, if the EHF settings are subsequently changed, both the wind load combinations, and the factors applied to the minimum lateral load combinations are affected and consequently steps 2 and 3 of the checking procedure would have to be repeated.)

Step 1. Establish the minimum lateral load to be resisted

This can be determined as follows:

1. Create a special “characteristic dead loads” combination which only comprises the total dead weight of the structure and no EHFs. Make this combination “Active”, but leave “Strength” and “Service” unchecked.
2. From the Analyse menu run **1st Order Linear** for all combinations.
3. From the Project Workspace Loading tab, select the “characteristic dead loads” combination and make a note of the total vertical reaction.



In the example shown above the vertical reaction = 28245 kN

4. Calculate 1.5% of this value – this is the minimum lateral load to be resisted.

   H_{min} = 0.015 * 28245 = 423.7kN

Step 2. Compare the minimum lateral load against the wind load

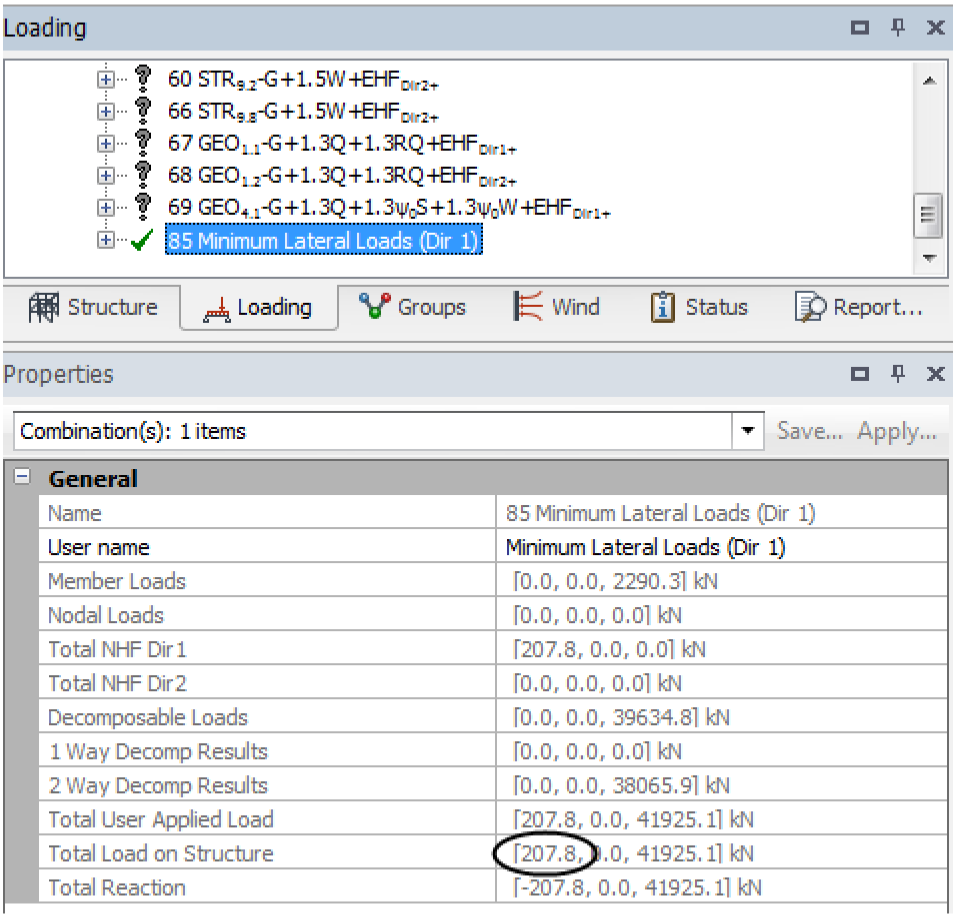
1. Still in the Project Workspace Loading tab, click on each of the wind load combinations and compare their lateral reactions against H_{min}
2. If H_{min} is greater than the maximum lateral reaction from all of the wind combinations, this indicates that the minimum lateral load governs and consequently you must ensure that the building is designed for this condition. (If it is not, then minimum lateral load does not govern and no further action is required.)

Step 3. Create (and design for) the minimum lateral load design combinations

Assuming that the above comparison has established that the minimum lateral load governs, you will have to create minimum lateral load combinations in each of four directions (+/- Dir 1, +/- Dir 2) as follows:

1. Copy the existing dead and imposed only combination to create a new combination named “Minimum Lateral Loads (Dir 1+)”
2. Ensure the new combination includes EHF in (Dir 1+) only
3. The EHF strength factor has to be adjusted to generate Hmin laterally - to do this:

   - From the Analyze menu run **1st Order Linear** for this new combination.
   - Record the total horizontal load on the structure, (H_1).



   In the example shown above the total horizontal load, H1 = 207.8kN

   - Calculate the EHF strength factor required as the ratio: H_{min}/H_1

     EHF strength factor = 423.7/ 207.8 = 2.04

4. Create a second minimum lateral load combination, “Minimum Lateral Loads (Dir 1-)”, this is similar to the first, using the same adjusted EHF strength factor, but with the EHF (Dir1+) loadcase replaced by EHF(Dir1-)
5. Repeat the above process to create similar minimum lateral load combinations in direction 2.
6. Run **Design All (Static)** to design the model for all combinations.

Steel design to EC3 and EC4 (Eurocode)

Tekla Structural Designer designs steel members and composite members to a range of international codes. This reference guide specifically describes the design methods applied in the software when the BS EN 1993-1-1:2005 (Ref.1) and BS EN 1994-1-1:2004 (Ref. 4) codes are selected.

Within the remainder of this handbook BS EN 1993-1-1:2005 and BS EN 1994-1-1:2004 are referred to as EC3 and EC4 respectively.

Unless explicitly noted otherwise, all clauses, figures and tables referred to are from EC3; apart from the Composite Beam section, within which references are to EC4 unless stated.

Basic principles (EC3 Eurocode)

This section covers definitions, convention for members axis and deflection checks.

Definitions (EC3 Eurocode)

The following terms are relevant when using Tekla Structural Designer to design to the Eurocodes.

National Annex (NA)

Safety factors in the Eurocodes are recommended values and may be altered by the national annex of each member state.

Tekla Structural Designer currently has the following EC3 national annex options available:

- EC3 Europe
- EC3 UK NA
- EC3 Ireland NA
- EC3 Ireland NA
- EC3 Malaysia NA
- EC3 Singapore NA

You can select the desired National Annex as appropriate, in which case the nationally determined parameters are automatically applied (see next section), or if you choose EC3 Europe, the Eurocode recommended values are applied.

Nationally Determined Parameters (NDP's)

NDP's are choices of values, classes or alternative methods contained in a National Annex that can be applied in place of the base Eurocode, EC3 Europe.

Partial Factors for Buildings

The partial factors γ_M for buildings as described in clause 6.1(1) Note 2B should be applied to the various characteristic values of resistance as follows:

- resistance of cross-sections irrespective of class: γ_{M0}
- resistance of members to instability assessed by member checks: γ_{M1}
- resistance of cross-sections in tension to fracture: γ_{M2}

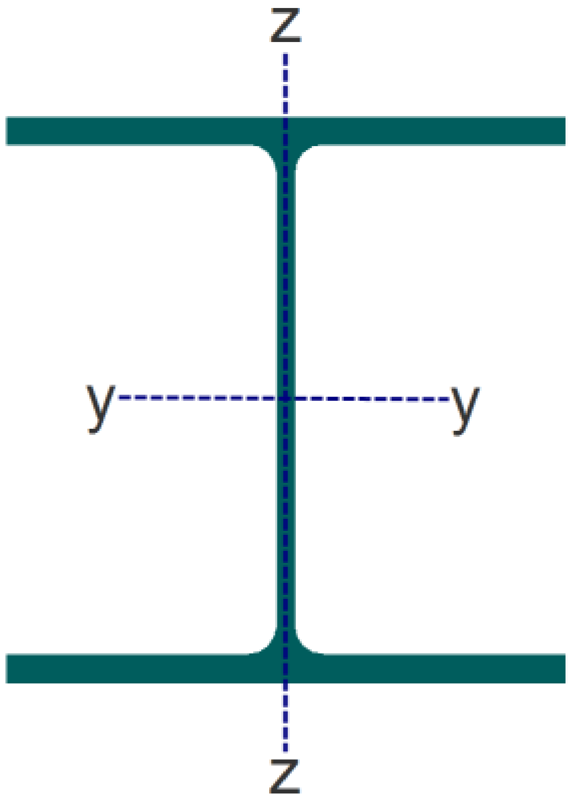
Depending on your choice of National Annex the above partial factors for buildings are set as follows:

Factor	EC3 Base value	UK	Ireland	Malaysia	Singapore
γ_{M0}	1.00	1.00	1.00	1.00	1.00
γ_{M1}	1.00	1.00	1.00	1.00	1.00
γ_{M2}	1.25	1.10*	1.25	1.20	1.10

NOTE - for connection design BS EN1991-1-8 - γ_{M2} = 1.25

Convention for member axes (EC3 Eurocode)

The sign convention for member axes when designing to Eurocodes is as shown below.

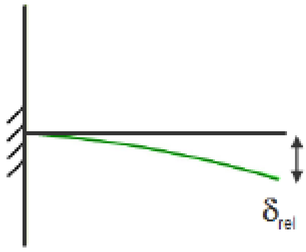
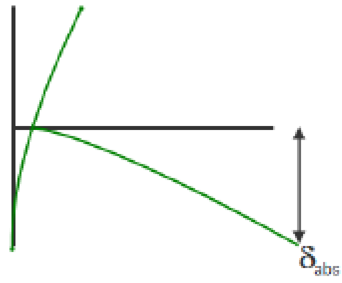


Section axes - (x is into the page along the centroidal axis of the member).

Deflection checks (EC3 Eurocode)

Relative and Absolute Deflections

Tekla Structural Designer calculates both ***relative*** and ***absolute*** deflections. Relative deflections measure the internal displacement occurring within the length of the member and take no account of the support settlements or rotations, whereas absolute deflections are concerned with deflection of the structure as a whole. The absolute deflections are the ones displayed in the structure deflection graphics. The difference between ***relative*** and ***absolute*** deflections is illustrated in the cantilever beam example below.

	
Relative Deflection	Absolute Deflection

Relative deflections are given in the member analysis results graphics and are the ones used in the member design.

Steel beam deflections

Deflections of steel beams in design are calculated from first order linear results since these are SLS deflections that are compared with standard limits such as span/360. This means that the effects of any non-linearity such as a continuous beam sitting on sinking supports i.e. non-linear springs are not taken into account in design. If these springs are linear this is not an issue.

Steel beam design to EC3 (Eurocode)

Design method (Beams: EC3 Eurocode)

Unless explicitly stated all calculations are in accordance with the relevant sections of EC3 (Ref. 1) and any associated National Annex. A basic knowledge of the design methods for beams in accordance with the code is assumed.

Steel beam limitations and assumptions (Beam: EC3 Eurocode)

The following limitations apply:

- Continuous beams (more than one span) must be co-linear in the plane of the web within a small tolerance (sloping in elevation is allowed),
- Rolled doubly symmetric prismatic sections (i.e. I- and H-sections), doubly symmetric hollow sections (i.e. SHS, RHS and CHS), and channel sections are fully designed,
- Single angles, double angles and tees are designed, but certain checks are beyond scope, (see Angle and Tee Limitations)
- Plated beams are fully designed provided the section type is either “Plated Beam” or “Plated Column”.

- All other plated section types ("Rolled I Sections with Plates", "Double Rolled I Sections" etc.) are analyzed only but not designed,
- Fabsec beams (with or without openings) are excluded.

The following assumptions apply:

- All supports are considered to provide torsional restraint, that is lateral restraint to both flanges. This cannot be changed. It is assumed that a beam that is continuous through the web of a supporting beam or column together with its substantial moment resisting end plate connections is able to provide such restraint.
- If, at the support, the beam oversails the supporting beam or column then the detail is assumed to be such that the bottom flange of the beam is well connected to the supporting member and, as a minimum, has torsional stiffeners provided at the support.
- In the Tekla Structural Designermodel, when not at supports, coincident restraints to both flanges are assumed when one or more members frame into the web of the beam at a particular position and the cardinal point of the centre-line model of the beam lies in the web. Otherwise, only a top flange or bottom flange restraint is assumed. Should you judge the actual restraint provided by the in-coming members to be different from to what has been assumed, you have the flexibility to edit the restraints as required.
- Intermediate lateral restraints to the top or bottom flange are assumed to be capable of transferring the restraining forces back to an appropriate system of bracing or suitably rigid part of the structure.
- It is assumed that you will make a rational and "correct" choice for the effective lengths between restraints for both LTB and compression buckling. **The default value for the effective length factor of 1.0 may be neither correct nor safe.**

Ultimate Limit State - Strength (Beams: EC3 Eurocode)

The strength checks relate to a particular point on the member and are carried out at regular intervals along the member and at "points of interest".

Classification (Beams: EC3 Eurocode)

General

The classification of the cross section is in accordance with EC3 Cl. 5.5 Table 5.2

A steel non-composite beam can be classified as:

- Plastic Class = 1
- Compact Class = 2
- Semi-compact Class = 3

- Slender Class = 4

Class 4 sections are unacceptable and are either failed in check mode or rejected in design mode.

Implementation of the below clauses is as follows:

- Classification is determined using clause 5.5.2 (6) and not 5.5.2 (7).
- Clause 5.5.2 (9) is not implemented as clause (10) asks for the full classification to be used for buckling resistance.
- Clause 5.5.2 (11) is not implemented.
- Clause 5.5.2 (12) is not implemented.

The note at the end of clause 5.5.2 is not implemented. A brief study by CSC (UK) Ltd of UK rolled UBs and UCs showed that flange induced buckling in normal rolled sections is not a concern. No study was undertaken for plated sections.

Hollow sections

The classification rules for SHS and RHS relate to “hot-finished hollow sections” and “cold-formed hollow sections”.

Shear capacity (Beams: EC3 Eurocode)

Major and minor axis shear

Checks are performed according to clause 6.2.6 (1) for the absolute value of shear force normal to each axis at the point under consideration.The following points should be noted:

• No account is taken of fastener holes in the flange or web - see 6.2.6 (7)

• Shear is not combined with torsion and thus the resistance is not reduced as per 6.2.6 (8)

Web Shear buckling

Shear web buckling design applies to rolled and plated I/H sections only.

National Annex dependency

Plates with unstiffened webs are checked for shear web buckling where:

$h_w/t_w > 72\ \varepsilon/\eta$

where

ε	=	√ (235 / f_w

η is NA dependant and is defined in the table below:

National Annex	η	Applicable to
Eurocode value	1.20	Up to and including S460, else use η = 1.00
UK	1.00	All steel grades
Irish	1.00	All steel grades
Malaysia	1.00	All steel grades
Singapore	1.00	All steel grades

Contribution from flanges

When the flange resistance is not fully utilized in resisting the bending moment (M_{Ed} <$M_{f,Rd}$), the contribution from the flanges is taken as:

$V_{bf,Rd} = (b_f\, t^2{}_f\, f_{yf}) / (c\, \gamma_{M1}) * (1-(M_{Ed}/M_{f,Rd})^2)$

where

- b_f and t_f are taken for the flange which provides the least axial resistance
- b_f is not taken as larger than $15\varepsilon t_f$ on each side of the web
- $M_{f,Rd}$ is the moment of resistance of the cross section consisting of the effective area of the flanges only
- $c = a\,(0.25 + 1.6 b_f\, t_f^2\, f_{yf} / t\, h_w^2\, f_y f_w)$
- a is the distance between stiffeners

As we are only designing for the case where no stiffeners are being used, a → ∞ therefore c → ∞ so $V_{bf,Rd} \rightarrow 0$.

Contribution from the web

The contribution from the web is taken as:

$V_{bw,Rd} = (x_w\, f_{yw}\, h_w\, t) / (\sqrt{3}\gamma_{M1})$

$x_w \leq \eta$

Design resistance

The design resistance for shear is taken as:

$V_{b,Rd} = V_{bw,Rd} + V_{bf,Rd} \leq \eta\, (f_{yw}\, h_w\, t) / (\sqrt{3}\gamma_{M1})$

Influence of shear

According to 7.1 of EN 1993-1-5 provided $V_{Ed} \leq 0.5\, V_{bw,Rd}$ the design resistance to bending moment and axial force does not need to be reduced to allow for shear force.

In Tekla Structural Designer V_{Ed} is restricted to 0.5 $V_{bw,Rd}$, values above this are deemed beyond scope.

This restriction is only applicable if $h_w/t_w > 72\, \varepsilon/\eta$

Assumptions

The following points should be noted:

- Non-rigid end post is a more conservative approach than a rigid end post.
- Physical support conditions can be taken as equivalent to "transverse stiffeners at supports only".
- It is assumed there is negligible contribution to the design shear force VEd from shear from torque
- All hole cut outs are small in accordance to section EN 1993-1-5:2006 2.3
- As the case being designed for is where no stiffeners are being used, a → ∞ therefore c → ∞ so $V_{bf,Rd}$→ 0.
- If a grade of steel is used other than S335, S355 and S460, η will be taken as 1.00 regardless of National Annex.

Moment capacity (Beams: EC3 Eurocode)

Major and minor axis bending checks are performed in accordance with Section 6.2.5.

Major axis bending

For the low shear case the calculation uses equation 6.13 for class 1 and 2 cross sections and equation 6.14 for class 3 cross sections. In the high shear case equation 6.29 is used for class 1 and 2 cross sections and equation 6.14 for class 3 cross sections. Where the high shear condition applies, the moment capacity calculation is made less complicated by conservatively adopting a simplified shear area.

Minor axis bending

For the low shear case the calculation uses equation 6.13 for class 1 and 2 cross sections and equation 6.14 for class 3 cross sections. High shear in the minor axis is beyond the current program scope.

NOTE Fastener holes in the flange or web are not accounted for in the calculations.

Axial capacity (Beam: EC3 Eurocode)

Axial Tension

Checks are performed according to equation 6.5

Implementation of the below clauses is as follows:

- Clause 6.2.3 (3) - is not considered

  Clause 6.2.3 (4) - is not considered
- Clause 6.2.3 (5) - is not considered
- Eqn 6.7 is not considered for steel non-composite beams.

Axial Compression

Checks are performed according to equation 6.9.

Combined bending and axial capacity (Beams: EC3 Eurocode)

The combined bending and axial capacity check covers the interaction of axial load and bending to Clause 6.2.9 appropriate to the type (for example - doubly symmetric) and classification of the section, and provided there is no high shear present.

If high shear is present only on the major axis (i.e. parallel to the web) of a doubly symmetric rolled or plated I section, in combination with uniaxial major bending and axial compression or tension, then the combined bending and axial capacity check will be carried out to the EC3 design rules described in a research paper by Goczek and Supel.[1]

All other high shear conditions, including the presence of shear web buckling or class 4 sections, are given a Beyond Scope status.

Class 1 and 2 cross sections (with low shear)

Equation 6.41 is applied. Note that in these calculations the combined effects of axial load and bending are assessed - clause 6.2.9 (4) is not considered.

Also note that the current "reduced plastic moduli" approach that is used in the published tables is adopted and not the approximate method given in clause 6.2.9.1(5). The latter is less conservative than the current approach at low levels of 'n'.

Class 3 cross sections (with low shear)

Equation 6.42 is applied.

[1] *'Resistance of steel cross-sections subjected to bending, shear and axial forces' by Jerzy Goczek and Lukasz Supel, Engineering Structures 70 (2014) 271-277, Elsevier.*

Ultimate limit state - Buckling (Beams: EC3 Eurocode)

NOTE Classification for buckling checks - For rolled I sections, RHS and SHS classification varies along the member length due to the section forces changing along the member length - for combined buckling, the worst classification of the whole member should be used. In theory it should be the worst classification in the “check length” considered for buckling. However, the “check lengths” for lateral torsional buckling, minor axis strut buckling and major axis strut buckling can all be different. It is simpler and conservative therefore to use the worst classification in the entire member length.

Compression buckling (Beams: EC3)

Beams must be checked to ensure adequate resistance to buckling about both the major and minor axes and they must also be checked in the torsional mode over an associated buckling length. Since the axial force can vary throughout the beam and the buckling lengths in the two planes do not necessarily coincide, all buckling modes must be checked. There may be circumstances where it would not be safe to assume that the combined buckling check will always govern (see below).

NOTE A warning message will be given in the compression buckling check results whenever (major or minor axis) high shear is present in a load combination. "*HIgh shear is assumed not to affect buckling design. This assumption should be verified by the Engineer.*"

Effective lengths

In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length in any way. Different values can apply in the major and minor axis. It is your responsibility to adjust the value from 1.0 where you believe it to be justified.

NOTE It is assumed that you will make a rational and "correct" choice for the effective lengths between restraints. The default value for the effective length factor of 1.0L may be neither correct nor safe.

Coincident restraint points in the major and minor axis define the 'check length' for torsional and torsional flexural buckling (which also has an effective length factor but is assumed to be 1.0L and cannot be changed).

All intermediate major and minor restraints in a cantilever span are ignored.

Any major or minor strut buckling 'check length' can take the type 'Continuous' to indicate that it is continuously restrained over that length. There is no facility for specifying torsional or torsional flexural buckling 'check lengths' as 'Continuous'.

There is no guidance in EC3 on the values to be used for effective length factors for beam-columns.

There is no guidance in EC3 on the values to be used for effective length factors for beam-columns.

Compression resistance

The relevant buckling resistances are calculated from Equation 6.47.

These consist of the flexural buckling resistance about both the major and minor axis i.e. $N_{b,y,Rd}$ and $N_{b,z,Rd}$ over the buckling lengths L_{yy} and L_{zz} and where required the buckling resistance in the torsional or flexural-torsional modes, $N_{b,x,Rd}$.

The elastic critical buckling load, N_{cr} for flexural buckling about major and minor axes is taken from standard texts. The elastic critical buckling loads for torsional, $N_{cr.T}$ and for torsional flexural buckling, $N_{cr.TF}$ are taken from the

NCCI “Critical axial load for torsional and torsional flexural buckling modes” available free to download at www.steel-ncci.co.uk.

All section types are checked for flexural buckling. It is only hollow sections that do not need to be checked for torsional and torsional-flexural buckling.

Lateral torsional buckling (Beams: EC3)

Lateral torsional buckling checks are required between supports, or LTB restraints on a flange which is in bending compression, for all lengths that are not continuously restrained.

NOTE A warning message will be given in the lateral torsional buckling check results whenever (major or minor axis) high shear is present in a load combination. “*HIgh shear is assumed not to affect buckling design. This assumption should be verified by the Engineer.*”

Note that **coincident** LTB restraints (i.e. top & bottom flange) are the equivalent of a support and will define one end of a 'check length' for both flanges regardless of whether a particular flange is in compression or tension at the coincident restraint position. However, note also that in a cantilever all intermediate restraints are ignored.

Tekla Structural Designer allows you to 'switch off' LTB checks for either or both flanges by specifying that the entire length between start and end of the beam span is continuously restrained against lateral torsional buckling. If you use this option you must be able to provide justification that the beam is adequately restrained against lateral torsional buckling.

All intermediate LTB restraints in a cantilever span are ignored.

When the checks are required you can set the effective LTB length of each 'check length' by giving factors to apply to the physical length of the beam. Any individual 'check length' less than the full span length can be continuously restrained in which case no LTB check will be carried out for that 'check length' provided all segments of the 'check length' have been marked as Continuous. Each 'check length' which is not defined as being continuously restrained for its **whole length** is checked in accordance with clause 6.3.2.3.

The formula for elastic critical buckling moment, M_{cr} is taken from standard texts. The moment factor C_1 that is part of the standard formula has been derived analytically.

LTB does not need to be checked for the following sections:

- circular and square hollow sections,
- equal and unequal flanged I/H sections loaded in the minor axis only.

Effective lengths

The value of effective length factor is entirely the choice of the engineer. The default value is 1.0. There is no specific factor for destabilizing loads - so you will have to adjust the 'normal' effective length factor to allow for such effects.

Combined buckling in Tekla Structural Designer is limited to doubly symmetric sections (I, H, CHS, SHS, RHS). In the context of combined buckling, beams are assumed to be dominated by moment with axial force.

NOTE A warning message will be given in the combined buckling check results whenever (major or minor axis) high shear is present in a load combination. "*HIgh shear is assumed not to affect buckling design. This assumption should be verified by the Engineer.*"

Restraints are treated as described previously and summarized as follows:

- Each span is assumed to be fully supported at its ends (i.e LTB, y-y and z-z restraint) - this cannot be changed.
- Tension flange LTB restraints are ignored unless they are coincident (see next point).
- Coincident top and bottom flange restraints are considered as 'torsional' restraints i.e. as good as the supports.
- All intermediate LTB and strut restraints in a cantilever span are ignored.

For each span of the beam, the design process is driven from the standpoint of the individual LTB lengths i.e. the LTB lengths and the y-y lengths that are associated with each LTB length and the z-z lengths associated with the y-y length. Thus a 'hierarchy' is formed - see the "Design Control (page 1898)" section below for details. Both Equation 6.61 and Equation 6.62 are evaluated recognizing that the combined buckling check is carried out for both the top flange and the bottom flange.

Effective lengths

In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length in any way. Different values can apply in the major and minor axis. It is your responsibility to adjust the value from 1.0 where you believe it to be justified.

NOTE It is assumed that you will make a rational and "correct" choice for the effective lengths between restraints. The default value for the effective length factor of 1.0L may be neither correct nor safe.

Combined buckling resistance

Equations 6.61 and 6.62 are used to determine the combined buckling resistance.

With regard to these equations the following should be noted:

- The "k" factors used in these equations are determined from Annex B only, and reported as follows:

- k_{yy} is reported as components k'_{yy} and Cmy where k'_{yy} is simply the Annex B term for k_{yy} with C_{my} **excluded**
- k_{yz} is reported as k'_{yz}, a multiple of k'_{zz} (see below)
- k_{zy} is reported as k'_{zy}, a multiple of k'_{yy} (see above), but only for members not susceptible to torsional deformations (i.e. SHS and CHS sections at all times, and I or H sections which have both flanges continuously restrained for LTB). For members which are susceptible to torsional deformations k_{zy} is reported per Table B.2 (i.e. with C_{mLT} **included**)
- $k_{zy,LT1}$ is a factor reported for columns only and is the Table B.2 term for k_{zy} with C_{mLT} set to 1.0
- k_{zz} is reported as components k'_{zz} and C_{mz} where k'_{zz} is simply the Annex B term for k_{zz} with C_{mz} excluded.
- The note to Table B.3 that C_m should be limited to 0.9 is not applied.

WARNING Danger. Equations 6.61 and 6.62 are limited to doubly symmetric sections and do not consider torsional or torsional flexural buckling. **Should either of these buckling modes govern the compression buckling check, you should consider very carefully whether the calculations provided by Tekla Structural Designer for combined buckling can be considered valid.**

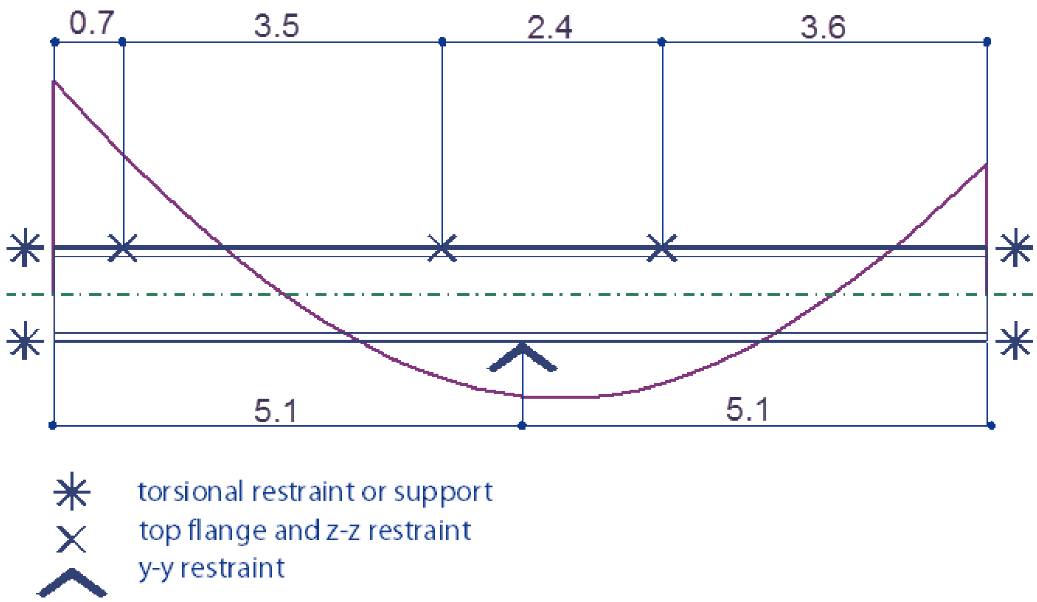
Design control (Beams: EC3 Eurocode)

Principles

There are multiple check lengths to deal with (LTB, y-y buckling and z-z buckling) all of which can be contained within or overlapped by their associated lengths. Consequently, a 'hierarchy' of checks is defined. In the approach taken the LTB segment length is taken as the driver and the other lengths whether overlapping or contained by this segment are mapped to it.

Design example

The following example illustrates how the checks are applied to I- and H-sections with equal flanges.



The beam (span) is 10.2 m long and has torsional restraints at each end. The top flange is restrained out-of-plane at 0.7m, 4.2m and 6.6 m – these provide restraint to the top flange for LTB and to the beam as a whole for out-of-plane strut buckling. The bottom flange has one restraint at mid-span and this restrains the bottom flange for LTB and the beam as a whole for in-plane strut buckling. (This is probably difficult to achieve in practice but is useful for illustration purposes.)

Note that the top flange LTB restraints and z-z restraints are coincident in this example but will not always be coincident.

Tekla Structural Designeridentifies the following lengths and checks. (in this example all the effective length factors are assumed to be 1.0 for simplicity.)

LTB Segment	**Equation**	**In-plane strut segment**	**Out-of-plane strut segment**
length (m)		**length (m)**	**length (m)**
Top flange 0 – 4.2 (first restraint ignored since top flange is in tension at this point)	6.61	0 – 5.1	0 – 0.7
	6.62	0 – 5.1	0 – 0.7
	6.61	0 – 5.1	0.7- 4.2
	6.62	0 – 5.1	0.7- 4.2
Top flange 4.2 – 6.6	6.61	0 – 5.1	4.2 – 6.6
	6.62	0 – 5.1	4.2 - 6.6
	6.61	5.1 - 10.2	4.2 - 6.6
	6.62	5.1 - 10.2	4.2 - 6.6

LTB Segment	Equation	In-plane strut segment	Out-of-plane strut segment
Top flange 6.6 - 10.2	6.61	5.1 - 10.2	6.6 - 10.2
	6.62	5.1 -10.2	6.6 - 10.2
Bottom flange 0 – 10.2	6.61	0 – 5.1	0 – 0.7
	6.62	0 – 5.1	0 – 0.7
	6.61	0 – 5.1	0.7- 4.2
	6.62	0 – 5.1	0.7- 4.2
	6.61	0 – 5.1	4.2 - 6.6
	6.62	0 – 5.1	4.2 - 6.6
	6.61	5.1 - 10.2	4.2 - 6.6
	6.62	5.1 - 10.2	4.2 - 6.6
	6.61	5.1 - 10.2	6.6 - 10.2
	6.62	5.1 - 10.2	6.6 - 10.2

Torsion (Beams: EC3 Eurocode)

Torsion design is carried out on request according to SCI P385, but only for single span, pin ended steel and cold formed beams with open and closed section types.

Open sections (I- symmetric rolled)

A torsion design and an angle rotation check can be carried out for applied torsion forces only.

The following should be noted with regard to the torsion design:

- Axial force is not taken into account
- It is assumed that load is applied at the shear center. The effect of stabilizing/destabilizing loads is not considered.

Closed sections (HSS only)

An angle of rotation check can be carried out for applied forces only.

Angle of rotation check

The angle of rotation check is optionally carried out based on the applied torsion loading only.

The check is applied by selecting “Apply rotation limit” (located in the steel beam properties under the Torsion heading). The default limit is also set in the steel beam properties as 2° but can be adjusted to suit.

Natural frequency checks (SLS) (Beams: EC4 Eurocode)

Tekla Structural Designer calculates the approximate natural frequency of the beam based on the simplified formula published in the Design Guide on the vibration of floors (Ref. 6) which states that Natural frequency = 18 / √ δ

In line with the calculation of natural frequency of 18 / √ δ for a pin ended beam with applied UDL, we calculate δ as the maximum static instantaneous deflection based upon the composite inertia (using the short term modular ratio) but not modified for the effects of partial interaction as:

$\delta = \%max\ \delta_{self+slab} + \%max\ \delta_{other\ dead} + \%max\ \delta_{live}$

The engineer can specify:

- Percentage self wt + slab deflection (default 100%)
- Percentage other dead deflection (default 100%)
- Percentage live load deflection (default 10%)
- Factor of increased dynamic stiffness of concrete flange (default 1.1)

Fire resistance check (Beams: EC3 Eurocode)

Scope

This check determines the mechanical resistance of a steel beam subjected to major axis bending in case of fire during the required time of exposure in accordance with EN 1993 & national annex for the UK, Ireland, Singapore, Malaysia, Sweden, Norway, Finland or the recommended Eurocode values.

The check can be applied to non-composite, simply supported rolled steel beams, which:

- may be unprotected (bare steel) or protected,
- may be exposed on 3 (under a slab) or 4 sides.

The check is applied for gravity load combinations only.

Temperature domain verification uses the **Critical temperature method** as described in EN 1993-1-2, Cl 4.2.4

Limitations and assumptions

The following limitations and assumptions apply:

- The calculation is limited to a restrained single-span beam in bending.
- As per EN 1993-1-2, Cl 4.1(3) simple calculation models are simplified design methods for individual members, which are based on conservative assumptions.
- According to EN 1991-1-2 Cl.3.1 (10) the standard temperature-time curve is used to calculate gas temperature –curve as described in EN 1991-1-2 Cl.3.2.1.
- The calculation considers that shear buckling resistance according to section 5 of EN1993-1 can be ignored so it doesn’t check it as per cl.6.2.6(6) EN 1993-1-1.

- A conservative approach has been taken regarding the fire moment resistance of the steel beam at time t so it’s limited to the ultimate moment capacity of the beam.
- The effects of indirect actions are not considered. These actions, such as internal forces and moments induced in the structure by deformations and restrained thermal expansion, do not need to be considered when the fire safety is based on the standard –temperature time curve.
- The shadow effect caused by local shielding is not taken into account, therefore the shadow effect factor is conservatively taken as 1.
- For Malaysia & Ireland check is as per UK NA.
- Fire protection properties (self weight of fire protection material) are not considered for loading and analysis.

Web openings (Beams: EC3 Eurocode)

Circular openings as an equivalent rectangle

Each circular opening is replaced by equivalent rectangular opening, the dimensions of this equivalent rectangle for use in all subsequent calculations are:

- d_o'= 0.9*opening diameter
- l_o = 0.45*opening diameter

Properties of tee sections

When web openings have been added, the properties of the tee sections above and below each opening are calculated in accordance with Section 3.3.1 of SCI P355 (Ref. 8) and Appendix B of the joint CIRIA/SCI Publication P068 (Ref. 9). The bending moment resistance is calculated separately for each of the four corners of each opening.

Design

The following calculations are performed where required for web openings:

- Axial resistance of tee sections
- Classification of section at opening
- Vertical shear resistance
- Vierendeel bending resistance
- Web post horizontal shear resistance
- Web post bending resistance
- Web post buckling resistance
- Lateral torsional buckling
- Deflections

Deflections

The deflection of a beam with web openings will be greater than that of the same beam without openings. This is due to two effects,

- the reduction in the beam inertia at the positions of openings due to primary bending of the beam,
- the local deformations at the openings due to Vierendeel effects. This has two components - that due to shear deformation and that due to local bending of the upper and lower tee sections at the opening.

The primary bending deflection is established by 'discretising' the member and using a numerical integration technique based on 'Engineer's Bending Theory' - $M/I = E/R = \sigma/y$. In this way the discrete elements that incorporate all or part of an opening will contribute more to the total deflection.

The component of deflection due to the local deformations around the opening is established using a similar process to that used for cellular beams which is in turn based on the method for castellated beams given in the SCI publication, “Design of castellated beams. For use with BS 5950 and BS 449".

The method works by applying a 'unit point load' at the position where the deflection is required and using a 'virtual work technique to estimate the deflection at that position.

For each opening, the deflection due to shear deformation, δ_s, and that due to local bending, δ_{bt}, is calculated for the upper and lower tee sections at the opening. These are summed for all openings and added to the result at the desired position from the numerical integration of primary bending deflection.

Note that in the original source document on castellated sections, there are two additional components to the deflection. These are due to bending and shear deformation of the web post. For castellated beams and cellular beams where the openings are very close together these effects are important and can be significant. For normal beams the openings are likely to be placed a reasonable distance apart. Thus in many cases these two effects will not be significant. They are not calculated for such beams but in the event that the openings are placed close together a warning is given.

Composite beam design to EC4 (Eurocode)

Design method (Composite beams: EC4 Eurocode)

The construction stage calculations are performed in accordance with the relevant sections of EC3 (Ref. 1) and the associated UK (Ref. 2) or Irish (Ref. 3) National Annex.

The composite stage design adopts a limit state approach consistent with the design parameters for simple and continuous composite beams as specified in EC4 (Ref. 4) and the associated UK (Ref. 5) or Irish National Annex.

Unless explicitly noted otherwise, all clauses, figures and tables referred to are from EC4.

A basic knowledge of EC3 and the design methods for composite beams in EC4 is assumed.

Overview (Composite beams: EC4 Eurocode)

Construction stage design checks (Composite beams: EC4)

When you design or check a beam for the construction stage (the beam is acting alone before composite action is achieved) the following conditions are examined in accordance with EC3:

- section classification (EC3 Table 5.2),
- major axis shear capacity (EC3 clause 6.2.6 (1)),
- web shear buckling (EC3 clause 6.2.6 (6)),
- moment capacity:
  - EC3 Equation 6.13 for the low shear condition,
  - EC3 Equation 6.29 for the high shear condition,
- lateral torsional buckling resistance (EC3 clause 6.3.2.3),

  NOTE This condition is only checked in those cases where the profile decking does not provide adequate restraint to the beam.

- construction stage total load deflection check.

Composite stage design checks

When you design or check a beam for the composite stage (the beam and concrete act together, with shear interaction being achieved by appropriate shear connectors) the following Ultimate limit state and Serviceability limit state conditions are examined in accordance with EC4, unless specifically noted otherwise.

Ultimate limit state checks

- section classification - the classification system defined in EC3 clause 5.5.2 applies to cross-sections of composite beams,
- vertical shear capacity in accordance with EC3 clause 6.2.6,
- longitudinal shear capacity allowing for the profiled metal decking, transverse reinforcement and other reinforcement which has been defined,
- number of shear connectors required (EC4 clause 6.6.1.3 (5)) between the point of maximum moment and the end of the beam, or from and between the positions of significant point loads,
- moment capacity,

- web openings.

Serviceability limit state checks

- service stresses - although there is no requirement to check these in EC4 for buildings (EC4 clause 7.2.2), concrete and steel top/bottom flange stresses are calculated but only reported if the stress limit is exceeded.
- deflections,
  - self-weight,
    - SLAB loadcase,
    - dead load,
    - imposed load,
    - total deflections,
- natural frequency check.

Profiled metal decking (Composite beams: EC4 Eurocode

You may define the profiled metal decking to span at any angle between 0° (parallel) and 90° (perpendicular) to the direction of span of the steel beam. You can also specify the attachment of the decking for parallel, perpendicular and angled conditions.

Where you specify that the direction of span of the profiled metal decking to that of the steel beam is ≥ 45°, thenTekla Structural Designer assumes it is not necessary to check the beam for lateral torsional buckling during construction stage.

Where you specify that the direction of span of the profiled metal decking to that of the steel beam is < 45°, then you are given the opportunity to check the steel beam for lateral torsional buckling at the construction stage.

NOTE This check is not mandatory in all instances. For a particular profile, gauge and fixing condition etc. you might be able to prove that the profiled metal decking is able to provide a sufficient restraining action to the steel beam until the concrete hardens. If this is so, then you can specify that the whole beam (or a part of it) is continuously restrained. Where you request to check the beam for lateral torsional buckling during construction then this is carried out in accordance with the requirements of EC3.

Where you specify that the direction of span of the profiled metal decking and that of the steel beam are parallel, you again have the same opportunity to either check the steel beam for lateral torsional buckling at the construction stage, or to set it as continuously restrained.

Concrete slab (Composite beams: EC4 Eurocode)

You can define concrete slabs in both normal and lightweight concrete.

Warnings are issued in the design if you do not comply with the following constraints:

- Normal weight concrete range C20/25 - C60/75 - See EN 1994-1-1:2004 Clause 3.1(2),
- Lightweight concrete range LC20/22 - LC60/66 - See EN 1994-1-1:2004 Clause 3.1(2),
- Minimum density for lightweight concrete 1750 kg/m3 - see EN 1994-1-1:2004 Clause 6.6.3.1(1).

Precast concrete planks (Composite beams: EC4 Eurocode)

The design of composite beams with precast concrete planks is carried out in accordance with the guidance given in SCI P401. The design basis in P401 is, in general, in accordance with Eurocode 4, supplemented by NCCI derived test data where applicable.

As the implications of applying NCCI PN002 or SCI P405 to composite beams with PC planks have not being considered in the first release, only pure EC design will be carried out regardless of whether **apply NCCI PN002** is selected or not.

Where a choice has been made the condition of the most common application has been taken: shop welded, hollow core unit with partial interaction.

Construction stage design (Composite beams: EC4 Eurocode)

All checks are performed for this condition in accordance with EC3.

Section classification (Composite beams: EC4 Eurocode)

Cross-section classification is determined using EC3 Table 5.2.

At construction stage the classification of the section must be Class 1, Class 2 or Class 3.

Sections which are classified as Class 4 are beyond scope.

NOTE Clause 5.5.2 (6) is implemented, not the alternative 5.5.2 (7).

Clause 5.5.2 (11) is not implemented.

Clause 5.5.2 (12) is not implemented.

Member strength checks (Composite beams: EC4 Eurocode)

Member strength checks are performed at the point of maximum moment, the point of maximum shear, the position of application of each point load, and at all other “points of interest” along the beam.

Shear capacity

Shear capacity is determined in accordance with EC3 clause 6.2.6 (1). Where the applied shear force exceeds 50% of the capacity of the section, the high shear condition applies to the bending moment capacity checks (see below).

The following points should be noted:

- No account is taken of fastener holes in the flange or web - see EC3 6.2.6 (7)
- Shear is not combined with torsion and thus the resistance is not reduced as per EC3 6.2.6(8)

Web Shear buckling

See: Steel Beam Design to EC3 - Web shear buckling

Bending moment capacity

For low shear this is calculated to EC3 Equation 6.13. In the high shear case Equation 6.29 is used. Where the high shear condition applies, the moment capacity calculation is made less complicated by conservatively adopting a simplified shear area.

Lateral torsional buckling checks (Composite beams: EC4 Eurocode)

You can switch off lateral torsional buckling checks by specifying that the entire length between the supports is continuously restrained.

If you use this option you must be able to provide justification that the beam is adequately restrained against lateral torsional buckling during construction.

When the checks are required you can position restraints at any point within the length of the main beam and can set the effective length of each sub-beam (the portion of the beam between one restraint and the next) either by giving factors to apply to the physical length of the beam, or by entering the effective length that you want to use. Each sub-beam which is not defined as being continuously restrained is checked in accordance with EC3 clause 6.3.2.3.

Deflection checks (Composite beams: EC4 Eurocode)

Tekla Structural Designer calculates relative deflections. (see: Deflection checks (page 1888))

The following deflections are calculated for the loads specified in the construction stage load combination:

- the dead load deflections i.e. those due to the beam self weight, the Slab Wet loads and any other included dead loads,
- the imposed load deflections i.e. those due to construction live loads,
- the total load deflection i.e. the sum of the previous items.

The loads are taken as acting on the steel beam alone.

The “Service Factor” (default 1.0), specified against each load case in the construction combination is applied when calculating the above deflections.

If requested by the user, the total load deflection is compared with either a span-over limit or an absolute value The initial default limit is span/200.

NOTE Adjustment to deflections. If web openings have been defined, the calculated deflections are adjusted accordingly. See: Web openings (page 1915)

Composite stage design (Composite beam: EC4 Eurocode)

Tekla Structural Designer performs all checks for the composite stage condition in accordance with EC4 unless specifically noted otherwise.

Equivalent steel section - Ultimate limit state (ULS) (Composite beams: EC4 Eurocode)

An equivalent steel section is determined for use in the composite stage calculations by removing the root radii whilst maintaining the full area of the section. This approach reduces the number of change points in the calculations while maintaining optimum section properties.

Section classification (ULS) (Composite beams: EC4 Eurocode)

Tekla Structural Designer classifies the section in accordance with the requirements of EC3, 5.5.2 except where specifically modified by those of EC4.

A composite section is classified according to the highest (least favorable) class of its steel elements in compression. The compression flange and the web are therefore both classified and the least favorable is taken as that for the whole section.

Flanges of any class that are fully attached to a concrete flange are assumed to be Class 1. The requirements for maximum stud spacing according to clause 6.6.5.5 (2) are checked and you are warned if these are not satisfied.

There are a small number of sections which fail to meet Class 2 at the composite stage. Although EC4 covers the design of such members they are not allowed in this release ofTekla Structural Designer.

Member strength checks (ULS) (Composite beams: EC4 Eurocode)

It is assumed that there are no loads or support conditions that require the web to be checked for transverse force. (clause 6.5)

Member strength checks are performed at the point of maximum moment, the point of maximum shear, the position of application of each point load, and at all other points of interest along the beam.

Shear capacity (Vertical)

The resistance to vertical shear, V_{Rd}, is taken as the resistance of the structural steel section, $V_{pl,a,Rd}$. The contribution of the concrete slab is neglected in this calculation.

The shear check is performed in accordance with EC3, 6.2.6.

Moment capacity

For full shear connection the plastic resistance moment is determined in accordance with clause 6.2.1.2. For the partial shear connection clause 6.2.1.3 is adopted.

In these calculations the steel section is idealized to one without a root radius so that the position of the plastic neutral axis of the composite section can be determined correctly as it moves from the flange into the web.

Where the vertical shear force, V_{Ed}, exceeds half the shear resistance, V_{Rd}, a (1-ρ) factor is applied to reduce the design strength of the web - as per clause 6.2.2.4.

Shear capacity (Longitudinal)

The design condition to be checked is: $v_{Ed} \leq v_{Rd}$ where:

v_{Ed} = design longitudinal shear stress

v_{Rd} = design longitudinal shear strength (resistance)

v_{Ed} is evaluated at all relevant locations along the beam and the maximum value adopted.

v_{Rd} is evaluated taking account of the deck continuity, its orientation and the provided reinforcement.

This approach uses the “truss analogy” from EC2. (See Figure 6.7 of EC2).

In these calculations, two planes are assumed for an internal beam, and one for an edge beam. Only the concrete above the deck is used in the calculations.

The values of v_{Rd} based on the concrete “strut” and the reinforcement “tie” are calculated. The final value of v_{Rd} adopted is then taken as the minimum of these two values.

The angle of the strut is minimised to minimise the required amount of reinforcement - this angle must lie between 26.5 and 45 degrees.

In the calculations of v_{Rd} the areas used for the reinforcement are as shown in the following table.

Decking angle	Reinforcement type	Area used
perpendicular	transverse	that of the single bars defined or for mesh the area of the main wires[1]

Decking angle	Reinforcement type	Area used
	other	that of the single bars defined or for mesh the area of the main wires[1]
parallel	transverse	that of the single bars defined or for mesh the area of the main wires[1]
	other	single bars have no contribution, for mesh the area of the minor wires[2]

[1]These are the bars that are referred to as longitudinal wires in BS 4483: 1998 Table 1.

[2]These are the bars that are referred to as transverse wires in BS 4483: 1998 Table 1.

If the decking spans at some intermediate angle (θ_r) between these two extremes then the program calculates:

- the longitudinal shear resistance as if the sheeting were perpendicular, $v_{Rd,perp}$,
- the longitudinal shear resistance as if the sheeting were parallel, $v_{Rd,par}$,
- then the modified longitudinal shear resistance is calculated from these using the relationship, $v_{Rd,perp}\sin^2(\theta_r) + v_{Rd,par}\cos^2(\theta_r)$.

Minimum area of transverse reinforcement (Composite beams: EC4 Eurocode)

The minimum area of transverse reinforcement is checked in accordance with clause 6.6.6.3.

Shear connectors (ULS) (Composite beams: EC4 Eurocode)

Dimensional requirements

Various limitations on the use of studs are given in the code.

The following conditions in particular are drawn to your attention:

Parameter	Rule	Clause/Comment
Spacing	Ductile connectors may be spaced uniformly over length between critical cross-sections if: - All critical cross-sections are Class 1 or 2	6.6.1.3(3) - not checked

Parameter	Rule	Clause/Comment
	- The degree of shear connection, h is within the range given by 6.6.1.2 and - the plastic resistance moment of the composite section does not exceed 2.5 times the plastic resistance moment of the steel member alone.	
Edge Distance	$e_D \geq 20$ mm	6.6.5.6(2) - not checked
	$e_D \leq 9 * t_f*sqrt(235/f_y)$	6.6.5.5(2) - applies if bare steel beam flange is Class 3 or 4 - not checked
Location	If it cannot be located in the center of trough, place alternately either side of the trough throughout the span	6.6.5.8(3) - not checked
Cover	The value from EC2 Table 4.4 less 5mm, or 20mm whichever is the greater.	6.6.5.2(2) - not checked

The program does not check that the calculated stud layout can be fitted in the rib of the deck.

Design resistance of the shear connectors

For ribs parallel to the beam the design resistance is determined in accordance with clause 6.6.4.1. The reduction factor, k_l is obtained from Equation 6.22. For ribs perpendicular to the beam, clause 6.6.4.2 is adopted.

The reduction factor, k_t is obtained from Equation 6.23.

The factor k_t should not be taken greater than the appropriate value of $k_{t,max}$ from the following table:

No of stud connectors per rib	Thickness of sheet, t mm	Studs with ≤ 20 mm and welded through profiled steel sheeting, $k_{t,max}$	Profiled sheeting with holes and studs with d = 19 or 22 mm, k_t,max
n_r = 1	≤ 1.0	0.85	0.75
	> 1.0	1.00	0.75
n_r = 2	≤ 1.0	0.70	0.60
	> 1.0	0.80	0.60

NOTE Only the first column of values of $k_{t.max}$ is used from the above table since the technique of leaving holes in the deck so that studs can be welded directly to the beam is not used.

For cases where the ribs run at an angle, θ_r the reduction factor is calculated as:

$k_t * \sin^2 \theta_r + k_l * \cos^2\theta_r$

Stud optimization is a useful facility since there is often some over conservatism in a design due to the discrete changes in the size of the section.

If you choose the option to optimize the shear studs, then Tekla Structural Designer will progressively reduce the number of studs either until the minimum number of studs to resist the applied moment is found, until the minimum allowable interaction ratio is reached or until the minimum spacing requirements are reached. This results in partial shear connection.

The program can also automatically layout groups of 1 or 2 studs with constraints that you specify.

The degree of shear connection is checked at the point of maximum bending moment or the position of a point load if at that position the maximum utilization ratio occurs.

NOTE During the selection process, in auto design mode point load positions are taken to be “significant” (i.e. considered as positions at which the maximum utilization could occur) if they provide more than 10% of the total shear on the beam. For the final configuration and for check mode all point load positions are checked.

To determine if the degree of shear connection is acceptable Tekla Structural Designer applies the following rules:

- If the degree of shear connection at the point of maximum moment is less than the minimum permissible shear connection, then this generates a FAIL status,
- If the point of maximum utilization ratio occurs at a point that is not the maximum moment position and the degree of shear connection is less

than the minimum permissible shear connection, then this generates a WARNING status,

- If the degree of shear connection at any other point load is less than the minimum permissible shear connection, then this does not affect the status in any way.

Lateral torsional buckling checks (ULS) (Composite beams: EC4 Eurocode)

The concrete slab is assumed to be laterally stable and hence there is no requirement to check lateral torsional buckling at the composite stage. (Clause 6.4.1).

Section properties - serviceability limit state (SLS) (Composite beams: EC4 Eurocode)

A value of the short term elastic (secant) modulus, E_{cm} is defaulted in Tekla Structural Designer for the selected grade of concrete. The long term elastic modulus is determined by dividing the short term value by a user defined factor - default 3.0. The elastic section properties of the composite section are then calculated using these values as appropriate (see the table below).

This approach is used as a substitute for the approach given in EC4 Equation 5.6 in which a knowledge of the creep coefficient, ϕ_t, and the creep multiplier, Ψ_L is required. It is envisaged that you will make use of EN 1992-1-1 (Ref. 6) when establishing the appropriate value for the factor.

EN 1994-1-1, clause 7.3.1.(8) states that the effect on deflection due to curvature imposed by restrained drying shrinkage may be neglected when the ratio of the span to the overall beam depth is not greater than 20. This relates to normal weight concrete. Tekla Structural Designer makes no specific allowance for shrinkage curvature but does provide you with a Warning when the span to overall depth exceeds 20 irrespective of whether the concrete is normal weight or lightweight. Where you consider allowance should be made, it is suggested that you include this as part of the 'factor' described above.

Tekla Structural Designer calculates the deflection for the beam based on the following properties:

Loadcase type	Properties used
self-weight	bare beam
Slab dry	bare beam
Dead	composite properties calculated using the long term elastic modulus
Live	composite properties calculated using the effective elastic modulus appropriate to the long term load percentage for each load. The deflections for all loads in the

Loadcase type	Properties used
	loadcase are calculated using the principle of superposition.
Wind	composite properties calculated using the short term elastic modulus
Total loads	these are calculated from the individual loadcase loads as detailed above again using the principle of superposition

Deflection checks (SLS) (Composite beams: EC4 Eurocode)

Tekla Structural Designercalculates relative deflections. (see: Deflection checks (page 1888)).

The composite stage deflections are calculated in one of two ways depending upon the previous and expected future load history:

- the deflections due to all loads in the Slab dry loadcase and the self-weight of the beam are calculated based on the inertia of the steel beam alone (these deflections are not modified for the effects of partial interaction).

  NOTE It is the Slab dry deflection alone which is compared with the limit, if any, specified for the Slab loadcase deflection.

- the deflections for all loads in the other loadcases of the Design combination will be based on the inertia of the composite section allowing for the proportions of the particular load that are long or short term (see above). When necessary these will be modified to include the effects of partial interaction.

  NOTE Tekla Structural Designerreports the deflection due to imposed loads alone (allowing for long and short term effects). It also reports the deflection for the SLAB loadcase, as this is useful for pre-cambering the beam. The beam Self-weight, Dead and Total deflections are also given to allow you to be sure that no component of the deflection is excessive.

  NOTE Adjustment to deflections. If web openings have been defined, the calculated deflections are adjusted accordingly. See: Web Openings (page 1915)

Stress checks (SLS) (Composite beams: EC4 Eurocode)

There is no requirement to check service stresses in EC4 for buildings (clause 7.2.2). However, since the deflection calculations are based on elastic analysis then at service loads it is logical to ensure that there is no plasticity at this load level.

Tekla Structural Designer calculates the worst stresses in the extreme fibers of the steel and the concrete at serviceability limit state for each load taking into account the proportion which is long term and that which is short term. These stresses are then summed algebraically. Factors of 1.00 are used on each loadcase in the design combination (you cannot amend these). The stress checks assume that full interaction exists between the steel and the concrete at serviceability state. The stresses are not reported unless the stress limit is exceeded, in which case a warning message is displayed.

Cracking of concrete (SLS) (Composite beams: EC4 Eurocode)

Clause 7.4.1(4) simply supported beams in unpropped construction, requires a minimum amount of longitudinal reinforcement over an internal support. This is not checked by Tekla Structural Designer as it is considered a detailing requirement.

Web openings (Composite beams: EC4 Eurocode)

Circular openings as an equivalent rectangle

Each circular opening is replaced by equivalent rectangular opening, the dimensions of this equivalent rectangle for use in all subsequent calculations are:

d_o'= 0.9 * opening diameter

l_o = 0.45 * opening diameter

Properties of tee sections

When web openings have been added, the properties of the tee sections above and below each opening are calculated in accordance with Section 3.3.1 of SCI P355 (Ref. 8) and Appendix B of the joint CIRIA/SCI Publication P068 (Ref. 9). The bending moment resistance is calculated separately for each of the four corners of each opening.

Design at construction stage

The following calculations are performed where required for web openings:

- Axial resistance of tee sections
- Classification of section at opening
- Vertical shear resistance
- Vierendeel bending resistance
- Web post horizontal shear resistance
- Web post bending resistance
- Web post buckling resistance

- Lateral torsional buckling
- Deflections

Design at composite stage

The following calculations are performed where required for web openings:

- Axial resistance of concrete flange
- Vertical shear resistance of the concrete flange
- Global bending action - axial load resistance
- Classification of section at opening
- Vertical shear resistance
- Moment transferred by local composite action
- Vierendeel bending resistance
- Web post horizontal shear resistance
- Web post bending resistance
- Web post buckling resistance
- Deflections

Deflections

The deflection of a beam with web openings will be greater than that of the same beam without openings. This is due to two effects,

- the reduction in the beam inertia at the positions of openings due to primary bending of the beam,
- the local deformations at the openings due to vierendeel effects. This has two components - that due to shear deformation and that due to local bending of the upper and lower tee sections at the opening.

The primary bending deflection is established by 'discretising' the member and using a numerical integration technique based on 'Engineer's Bending Theory' - M/I = E/R = σ/y. In this way the discrete elements that incorporate all or part of an opening will contribute more to the total deflection.

The component of deflection due to the local deformations around the opening is established using a similar process to that used for cellular beams which is in turn based on the method for castellated beams given in the SCI publication, “Design of castellated beams. For use with BS 5950 and BS 449".

The method works by applying a 'unit point load' at the position where the deflection is required and using a 'virtual work technique to estimate the deflection at that position.

For each opening, the deflection due to shear deformation, δ_s, and that due to local bending, δ_{bt}, is calculated for the upper and lower tee sections at the opening. These are summed for all openings and added to the result at the desired position from the numerical integration of primary bending deflection.

Note that in the original source document on castellated sections, there are two additional components to the deflection. These are due to bending and shear deformation of the web post. For castellated beams and cellular beams where the openings are very close together these effects are important and can be significant. For normal beams the openings are likely to be placed a reasonable distance apart. Thus in many cases these two effects will not be significant. They are not calculated for such beams but in the event that the openings are placed close together a warning is given.

Precast concrete planks

The effect of web openings on composite beams with PC planks is not within the scope of SCI P401. Web openings can be modeled but are ignored in both design at Construction stage and design at Composite stage when a PC plank is used. Design will be carried out treating the steel beam as one with no web openings.

Application of NCCI PN002 to Partial Shear Connection (Composite beams: EC4 Eurocode)

An **Apply NCCI PN002** check box is available on the Stud strength page of the Beam Properties. When this option is selected Tekla Structural Designer calculates partial shear limits described in PN002 for edge beams and SCI P405 for internal beams.

It should be noted that to obtain the benefits of this NCCI,

- for all deck types and orientation the design live load (γ_q q_k) is limited to 9 kN/m^2
- for all deck types and orientation the beam should be “unpropped” at the construction stage (this is a general assumption in Tekla Structural Designer for all composite beams).
- for perpendicular trapezoidal decks the studs should be placed on the “favorable” side or in the central position.
- for perpendicular trapezoidal decks the reinforcement is assumed to be above the head of the stud. Consequently, a reduction is made to the stud resistance in accordance with NCCI PN001.
- for limits of maximum longitudinal stud spacing the relevant NCCI must be satisfied.
- for slab the nominal total depth must not exceed 180mm (depth of concrete over the decking must not exceed 100mm)
- for all deck profiles the nominal height (to shoulder) must not exceed 80mm (applies to SCI P405 only)

It is the user's responsibility to ensure compliance with the above since the program makes no check on these items.

For perpendicular trapezoidal decks the reduction in stud resistance to which point 4 above refers, will be conservative if the reinforcement is placed in a more favorable (lower) position. Even though the NCCI is relevant to the UK this option is also available for all EC head-codes in Tekla Structural Designer.

More information is given in the PN001, PN002 and SCI P405 on www.steel-ncci.co.uk and on http://www.steelbiz.org/

Steel column design to EC3 (Eurocode)

Design method (Columns: EC3 Eurocode)

Unless explicitly stated all steel column calculations in Tekla Structural Designerare in accordance with the relevant sections of EC3 (Ref. 1) and the associated National Annex.

A full range of strength, buckling and serviceability checks are carried out.

NOTE A sway assessment is also performed. This can optionally be de-activated for those columns for which it would be inappropriate, by unchecking the Alpha Crit Check box on the Column Properties dialog.

Simple columns (Columns: EC3 Eurocode)

A general column could be designated as a “simple column” to indicate that it does not have any applied loading in its length. Simplified design rules exist for such columns as they are only subject to axial forces and moments due to eccentricity of beam reactions, (moments due to frame action or due to member loading are assumed not to occur).

NOTE The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as “beyond scope” when they are designed.

Ultimate limit state strength (Columns: EC3 Eurocode)

Strength checks relate to a particular point on the member and are carried out at 5th points and “points of interest”, (i.e. positions such as maximum moment, maximum axial etc.)

Classification (Columns: EC3 Eurocode)

The classification of the cross section is in accordance with Table 5.2. General columns can be classified as:

- Plastic Class = 1

- Compact Class = 2
- Semi-compact Class = 3
- Slender Class = 4

Class 4 sections are not allowed.

Implementation of the below clauses is as follows:

- Classification is determined using 5.5.2 (6) and not 5.5.2 (7).
- 5.5.2 (9) is not implemented as clause (10) asks for the full classification to be used for buckling resistance.
- 5.5.2 (11) is not implemented.
- 5.5.2 (12) is not implemented. A brief study of UK rolled UBs and UCs showed that flange induced buckling in normal rolled sections is not a concern.

Axial capacity (Columns: EC3 Eurocode)

The axial tension and compression capacity checks are performed according to clause 6.2.3 and clause 6.2.4 respectively.

The following points should be noted:

- Clause 6.2.3 (3) - is not considered
- Clause 6.2.3 (4) - is not considered
- Clause 6.2.3 (5) - is not considered

Shear capacity (Columns: EC3 Eurocode)

The shear check is performed at the point under consideration according to clause 6.2.6(1):

- for the absolute value of shear force normal to the y-y axis, $V_{y,Ed}$, and
- for the absolute value of shear force normal to the z-z axis, $V_{z,Ed}$

The following points should be noted:

- No account is taken of fastener holes in the flange or web - see 6.2.6 (7)
- Shear is not combined with torsion and thus the resistance is not reduced as per 6.2.6 (8)

Shear buckling

When the web slenderness exceeds 72ε shear buckling can occur in rolled sections. Tekla Structural Designer designs for shear web buckling with accordance to EN 1993-1-5:2006.

The following should however, be noted:

- The approach to design assumes a non-rigid end post, this is more conservative than the design that takes the approach assuming a rigid end post.
- Physical support conditions have been assumed to be equivalent to “transverse stiffeners at supports only”.
- All hole cut outs must be small in accordance to section EN 1993-1-5:2006 2.3
- If a grade of steel was to be used other than S335, S355 and S460 η will be taken as 1.00 regardless of National Annex.

As we are only designing for the case where no stiffeners are being used, a → ∞ therefore c → ∞ so $V_{bf,Rd} \rightarrow 0$, where $V_{bf,Rd}$ is the contribution from the flange - see 5.4(1)

The design assumes negligible contribution to the design shear force V_{Ed} from shear from torque, therefore V_{Ed} is restricted to 0.5 $V_{bw,Rd}$. Tekla Structural Designer will warn you if this limit is exceeded - see 7.1(1)

Moment capacity (Columns: EC3 Eurocode)

The moment capacity check is performed at the point under consideration according to clause 6.2.5(1):

- for the moment about the y-y axis, $M_{y,Ed}$, and
- for the moment about the z-z axis, $M_{z,Ed}$

The moment capacity can be influenced by the magnitude of the shear force (“low shear” and “high shear” conditions). Where the high shear condition applies, the moment capacity calculation is made less complicated by conservatively adopting a simplified shear area.

The maximum absolute shear to either side of a point of interest is used to determine the moment capacity for that direction.

High shear condition about y-y axis

The treatment of high shear is axis dependent. In this release for CHS, if high shear is present, the moment capacity check about the y-y axis is Beyond Scope.

High shear condition about z-z axis

For rolled sections in this release, if high shear is present normal to the z-z axis then the moment capacity check about the z-z axis is Beyond Scope.

For hollow sections, there is greater potential for the section to be used to resist the principal moments in its minor axis. Of course for CHS and SHS there is no major or minor axis and so preventing high shear arbitrarily on one of the two principal axes does not make sense. Nevertheless, if high shear is present normal to the z-z axis then in this release the moment capacity about the z-z axis is not calculated, the check is Beyond Scope.

If high shear is present in one axis or both axes and axial load is also present, the moment capacity check is given a Beyond Scope status.

If high shear and moment is present in both axes and there is no axial load ("biaxial bending") the moment capacity check is given a Beyond Scope status.

Combined bending and axial capacity (Columns: EC3 Eurocode)

The combined bending and axial capacity check covers the interaction of axial load and bending to clause 6.2.9 appropriate to the type (for example – doubly symmetric) and classification of the section.

If high shear is present in one axis or both axes and axial load is also present, the cross-section capacity check is given a Beyond Scope status.

If high shear and moment is present in both axes and there is no axial load ("biaxial bending") the cross-section capacity check is given a Beyond Scope status.

The following additional points should be noted:

the combined effects of axial load and bending are assessed and clause 6.2.9 (4) is not considered.

the current "reduced plastic moduli" approach in the published tables is used and not the approximate method given in 6.2.9.1(5). The latter is less conservative than the current approach at low levels of 'n'.

Ultimate limit state buckling (Columns: EC3 Eurocode)

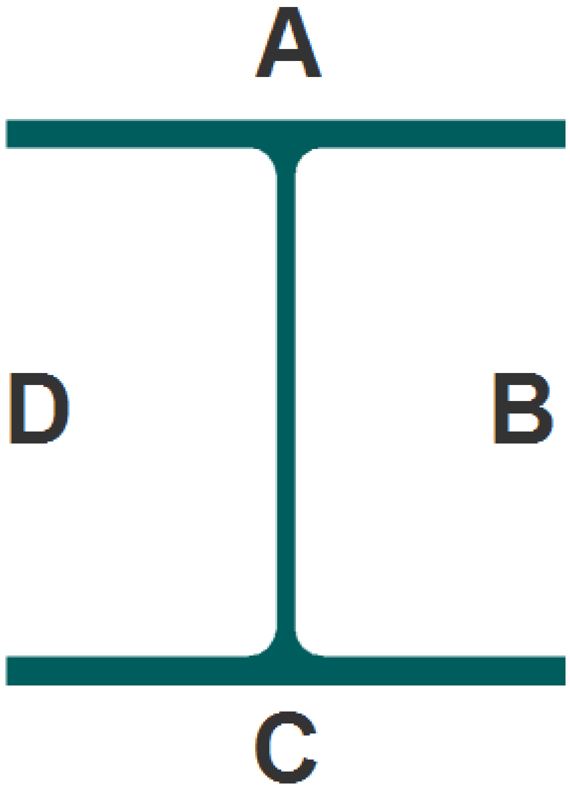
NOTE Classification for buckling checks - For rolled I sections, RHS and SHS classification varies along the member length due to the section forces changing along the member length - for combined buckling, the worst classification of the whole member (column stack) should be used. In theory it should be the worst classification in the segment length considered for buckling. However, the segment lengths for lateral torsional buckling, minor axis strut buckling and major axis strut buckling can all be different. It is simpler and conservative therefore to use the worst classification in the entire member length (column stack).

Compression buckling (Columns: EC3 Eurocode)

General columns must be checked to ensure adequate resistance to buckling about both the major and minor axes and they must also be checked in the torsional mode over an associated buckling length. Since the axial force can vary throughout the column and the buckling lengths in the two planes do not necessarily coincide, all buckling modes must be checked. There may be circumstances where it would not be safe to assume that the combined buckling check will always govern (see below).

Restraints

Restraints to strut buckling are determined from the incoming members described within Tekla Structural Designer. The buckling checks are based on these.



Restraining members framing into either Face A or C will provide restraint to major axis strut buckling. Members framing into either Face B or D will provide restraint to minor axis strut buckling. Tekla Structural Designer determines the strut buckling restraints but you can override these.

NOTE The program assumes that any member framing into the major or minor axis of the column provides restraint against strut buckling in the appropriate plane. If you believe that a certain restraint in a particular direction is not effective then you can either override the restraint or adjust the effective length to suit – to 2.0L for example.

Torsional and torsional flexural buckling restraint is only provided at points restrained coincidentally against major **and** minor axis strut buckling.

NOTE Provided a level is restrained coincidentally against major and minor axis strut buckling, the program assumes that any member framing into the appropriate faces provides restraint against torsional and torsional flexural buckling at that level. There are a number of practical conditions that could result in torsional restraint not being provided at floor levels. At construction levels this is even more possible given the likely type of incoming member and its associated type of connection. You must consider the type of connection between the incoming members and the column since these can have a significant influence on the ability of the member to provide restraint to one, none or both column flanges. For example, consider a long fin plate connection for beams framing into the column web where the beam stops outside the column flange tips to ease detailing. The fin plate is very slender and the beam end is remote from the column flanges such that it may not be able to provide any restraint to torsional or torsional flexural buckling. The fact that a slab is usually present may mitigate this. You are expected to override the ineffective restraint.

Tekla Structural Designer always assumes full restraint at the base and at the roof level when carrying out buckling design checks – you are warned on validation if your restraint settings do not reflect this. Restraints are considered effective on a particular plane providing they are within ±45° to the local coordinate axis system.

Effective lengths

In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length in any way. You are expected to adjust the strut buckling effective length factor (up or down) as necessary. Different values can apply in the major and minor axis.

NOTE It is assumed that you will make a rational and “correct” choice for the effective lengths between restraints. The default value for the effective length factor of 1.0L may be neither correct nor safe.

The torsional and torsional flexural buckling effective length factor (1.0L) can not be changed.

Any strut buckling effective length can take the type “Continuous” to indicate that it is continuously restrained over that length. There is no facility for specifying torsional, or torsional flexural buckling effective lengths as “Continuous”.

There is no guidance in EC3 on the values to be used for effective length factors for beam-columns.

For general columns - The minimum theoretical value of effective length factor is 0.5 and the maximum is infinity for columns in rigid moment resisting (RMR) frames. Practical values for simple columns are in the range 0.7 to 2.0 (see For simple columns below). In theory, values less than 1.0 can be chosen for non-sway frames or for sway frames in which the effects of sway are taken into account using either the amplified forces method or P-Delta analysis. However, EC3 states that when second-order effects are included in this way then the design “may be based on a buckling length equal to the system length” i.e. an effective length factor of 1.0. The program default of 1.0 matches this requirement but allows you flexibility for special situations.

One such situation might be in RMR frames where the principal moments due to frame action preventing sway are in one plane of the frame. There will often be little or no moment out-of-plane and so, if using the amplified forces method, the amplification of these moments has little effect on the overall design. Nevertheless the stability out-of-plane can still be compromised by the lack of restraint due to sway sensitivity in that direction. In such cases a value of greater then 1.0 (or substantially greater) may be required. Similarly, in simple construction where only eccentricity moments exist, it is only the brace forces that 'attract' any amplification. Thus for the column themselves the reduced restraining effect of a sway sensitive structure may require effective length factors greater than 1.0.

For Simple columns - There is no concept of simple columns in EC3 and hence no information on effective lengths either. However, reference can be made to the “NCCI” on the subject of simple construction but none of this includes the clear guidance on effective lengths of simple columns that was included as Table 22 in BS 5950-1: 2000. Again the program defaults the effective length factor to 1.0

Compression resistance

The relevant buckling resistances are all calculated from Equation 6.47.

These consist of the flexural buckling resistance about both the major and minor axis i.e. $N_{b,y,Rd}$ and $N_{b,z,Rd}$ over the buckling lengths L_{yy} and L_{zz} and where required the buckling resistance in the torsional or flexural-torsional modes, $N_{b,x,Rd}$.

All section types are checked for flexural buckling. It is only hollow sections that do not need to be checked for torsional and torsional-flexural buckling.

Lateral torsional buckling (Columns: EC3 Eurocode)

Effective lengths

The value of effective length factor is entirely your choice. The default value is 1.0 and is editable for flanges A & C. Any individual segment (for either flange) can be 'continuously restrained' in which case no lateral torsional buckling (LTB) check is carried out for that flange over that segment.

For a level to be treated as torsional restraint it must have both A and C restraint **and** also be restrained for compression buckling in both the major and minor axis.

There is no specific factor for destabilizing loads - you can however adjust the 'normal' effective length factor to allow for such effects.

Lateral torsional buckling resistance

The LTB resistance is calculated from Equation 6.55.

LTB does not need to be checked for the following sections:

- circular and square hollow sections,
- equal and unequal flanged I/H sections loaded in the minor axis only.

Combined buckling (Columns: EC3 Eurocode)

The column must be restrained laterally in two directions, and torsionally at the top and bottom of the 'design length'. This equates to LTB restraint to faces A and C and restraint to major and minor axis compression buckling all being coincident. A design length is allowed to have intermediate restraint and if the restraint requirements are not met at a particular floor then the design length does not have to be between adjacent floors. Thus a stack can 'jump' floors or sheeting rails can be attached. It is assumed that the restraints for compression buckling are fully capable of forcing the buckled shape. Hence, the compression buckling resistance is based on the restrained lengths whilst the LTB resistance ignores the intermediate restraint and hence is based on the full design length.

NOTE It is conservative to ignore the intermediate restraints in this latter case.

Loading within the design length is allowed.

Effective lengths

Effective lengths for flexural (i.e. strut major and strut minor) and lateral torsional buckling are as described in the appropriate section above.

Combined buckling resistance

The combined buckling resistance is checked in accordance with Equations 6.61 and 6.62. Both equations are evaluated at the ends of the design length and, except for simple columns, at the position of maximum moment, if that lies elsewhere.

Eccentricity moments due to beam end reactions are added to the “real” moments due to frame action:

- in the first case the uniform moment factors are calculated from the real moments and applied to the real moments. Eccentricity moments are only added if they are more critical.
- in the second case all moments are “combined” and all uniform moment factors are based on the combined moments and applied to them.

WARNING Equations 6.61 and 6.62 are limited to doubly symmetric sections and do not consider torsional or torsional flexural buckling. Should either of these buckling modes govern the compression buckling check, you should consider very carefully whether the calculations provided by Tekla Structural Designer for combined buckling can be considered valid.

Serviceability limit state (Columns: EC3 Eurocode)

The column is assessed for sway and the following values are reported for each stack:

- Sway X and α_{critx}
- Sway Y and α_{crity}
- Sway X-Y

Depending on the reported α_{crit} the column is classified as Sway or Non sway accordingly.

NOTE A sway assessment is only performed for the column if the Alpha Crit Check box is checked on the Column Properties dialog. If very short columns exist in the building model these can distort the overall sway classification for the building. For this reason you may apply engineering judgement to uncheck the Alpha Crit Check box for those columns for which a sway assessment would be inappropriate.

Steel brace design to EC3 (Eurocode)

Design method

Unless explicitly stated all brace calculations are performed in accordance with the relevant sections of BS EN 1993-1-1:2005 (Ref. 1) (herein abbreviated to EC3) and the associated National Annex.

A basic knowledge of the design methods for braces in accordance with the design code is assumed.

Classification

No classification is required for braces in tension.

Braces in compression are classified according to Table 5.2 as either: Class 1, Class 2, Class 3 or Class 4.

Class 4 sections are not allowed.

Axial tension

An axial tension capacity check is performed according to clause 6.2.3.(1)

The following points should be noted:

- Clause 6.2.3 (3) - is not considered

- Clause 6.2.3 (4) - is not considered
- Clause 6.2.3 (5) - is not considered

Axial compression

An axial compression capacity check is performed according clause 6.2.4.(1)

Compression buckling

If axial compression exists, the member is also assessed according to clause 6.3.1.1(1) for flexural buckling resistance about both the major and minor axis i.e. $N_{b,y,Rd}$ and $N_{b,z,Rd}$ over the buckling lengths L_{yy} and L_{zz} and where required the torsional, or flexural-torsional buckling resistance, $N_{b,x,Rd}$.

For single and double angles (both equal and unequal) there is also a compression buckling check about the v-v axis, over the buckling length L_{vv}. For single angles, L_{vv} is the system length L, while for double angles L_{vv} is L/3.

All section types are checked for flexural buckling. It is only hollow sections that do not need to be checked for torsional and torsional-flexural buckling.

Different effective length factors can be applied for flexural buckling in the major and minor axis. For single and double angles an effective length factor can also be applied in the v-v axis. The default effective length is 1.0L in all 3 cases. You are expected to adjust the effective length factor (up or down) as necessary.

The torsional and torsional flexural buckling effective length factor (1.0L) can not be changed.

Steel single, double angle and tee section design to EC3 (Eurocode)

Design method (Angles and tees: EC3 Eurocode)

The EC3 (Ref. 1) design method adopted is dictated by the member characteristic type:

- “Beam”, “Truss member top” or “Truss member bottom” characteristic:
    - Member is designed for axial tension, compression, shear, bending and combined forces - consistent with the method detailed in Steel Beam Design to EC3 (page 1889)

- “Brace”, “Truss internal” or “Truss member side” characteristic:
  - Member is designed for axial tension, compression and compression buckling only - consistent with the method detailed in Steel Brace Design to EC3 (page 1926)

NOTE Additional Angle and tee limitations (page 1928)have to be considered when designing these sections to the above design methods.

Angle and tee limitations (EC3 Eurocode)

In the current version when designing tees, single, and double angles to EC3, the following checks remain beyond scope:

	Tee	Angle	Double angle
Classification	ok	ok	ok
Axial tension	ok	ok	ok
Axial compression	ok	ok	ok
Shear	ok	ok	ok
Buckling	ok	ok	ok
Combined strength	ok	ok	ok
LTB	Beyond scope	ok	Beyond scope
Combined buckling	Beyond scope	Beyond scope	Beyond scope
Deflection	ok	ok	ok

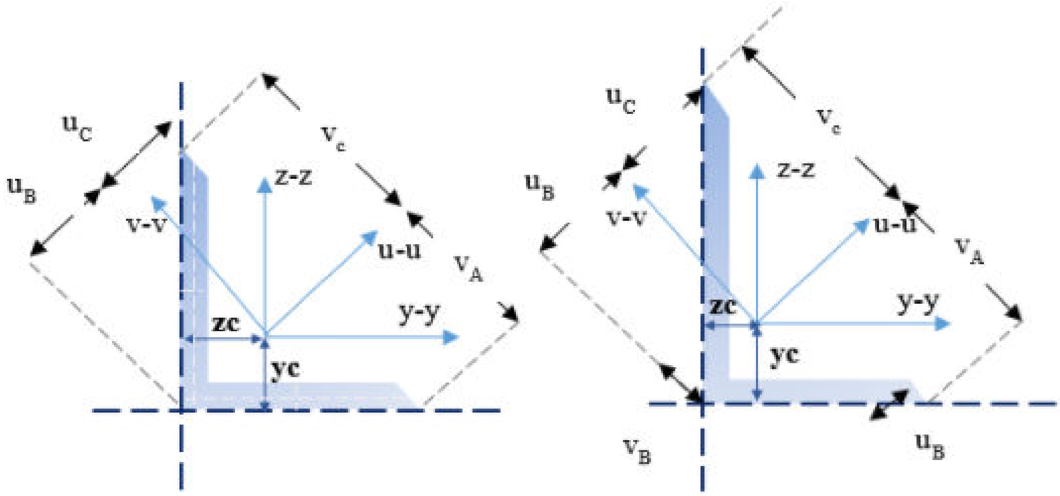
In addition, the following limitations apply:

- All sections and in particular single angles are assumed to be effectively loaded through the shear center such that no additional torsion moments are developed. In addition no direct allowance is made for 'destabilizing loads'.
- Design excludes bending of the outstand leg of single and double angles loaded eccentrically e.g. supporting masonry.
- Conditions of restraint can be defined as top and bottom flange for lateral torsional buckling LTB. It is upon these that the buckling checks will be based. For the current release intermediate LTB restraints are omitted (i.e. only fully restrained for LTB, or unrestrained).
- Single, double angles and tee sections subject to moment with high shear are beyond scope.

Section axes (Angles and tees: EC3 Eurocode)

For all sections:

- y-y is the axis parallel to the flanges (major axis)
- z-z is the axis perpendicular to the flanges (minor axis)
- for Single angles and Double angles
    - z-z parallel to long side (leg) - single angles
    - z-z parallel to long side (leg) - double angles with long leg back to back
    - z-z parallel to short side (leg) - double angles with short leg back to back
- u-u is the major principal axis for single angles
- v-v is the minor principal axis for single angles



Single angles - Section axes

Design procedures (Angles and tees: EC3 Eurocode)

This section includes key notes and assumptions made for the EC3 (Ref. 1) design of tees and angle sections.

Classification checks

For axial compression and bending both the web and flange (Leg 1 and Leg 2) are classified as Class 1, Class 2, Class 3 or Class 4 and the worst of the two is the resultant classification for that cross section.

The rules from Table 5.2 (sheet 2 of 3) of EC3 are used for tee sections. In particular the rules of “Part subject to compression” are used to classify the tee section since these are more conservative compared to the limits of “Part subject to bending and compression”.

For double angles and single angles the rules from Table 5.2 (sheet 3 of 3) of EC3 are used.

NOTE Class 4 section classification is only allowed for tees, double angles and single angles.

Axial tension check

Section 6.2.3 of EC3 is used for this design check.

Axial compression check

Section 6.2.4 of EC3 is used for this design check.

Effective length:

The value of effective length factor is entirely at the user's choice. The default value is generally 1.0 although for truss members, there are special settings for the effective length depending upon the type of section and its position in the truss.

Different values can apply in the major and minor axis. Coincident strut restraint points in these two directions define the length for torsional and torsional flexural buckling and this can also have an effective length factor (this is assumed to be 1.0 and cannot be changed).

There is no guidance in EC3 on the values to be used for effective length factors for beam-columns although Annex BB does contain some information on the effective lengths to be used in trusses but not for single, double angles and tees.

It is the responsibility of the user to adjust the value from 1.0 (for the effective length factor) and to justify such a change on the compression page.

For tees:

Check:

1. the buckling length in the major axis – Use L_{yy} = L * major factor
2. the buckling length in the minor axis – Use L_{zz} = L * minor factor
3. the buckling length for the torsional mode – Use L_{xx} = 1.00 * minor factor

For single and double angles:

Check:

1. the buckling length in the major axis – Use L_{yy} = L * major factor
2. the buckling length in the minor axis – Use L_{zz} = L * minor factor
3. the buckling length for the torsional mode – Use L_{xx} =1.00 * L
    a. Double angles – Check as single angle
        1. Use $L_y = L_{yy}/ 3$
        2. Use $L_z = L_{zz}/ 3$

3. Use $L_x = L_{xx}/ 3$

b. Double angles – Check as double angle

4. the buckling length for the principal axis, v-v – Use $L_{vv} = 1.00 * L$

a. Double angles – Check as single angle

1. Use $L_v = L_{vv}/3$

For double angles for (4) & (3a) minor principal axis buckling & torsional buckling respectively – half of the axial force and half of the double angle area is used.

Shear check

Section 6.2.6 of EC3 is used for this design check.

Moment check

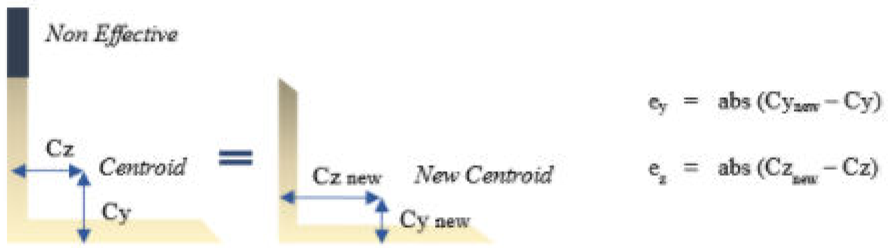
Section 6.2.5 of EC3 is used for this design check.

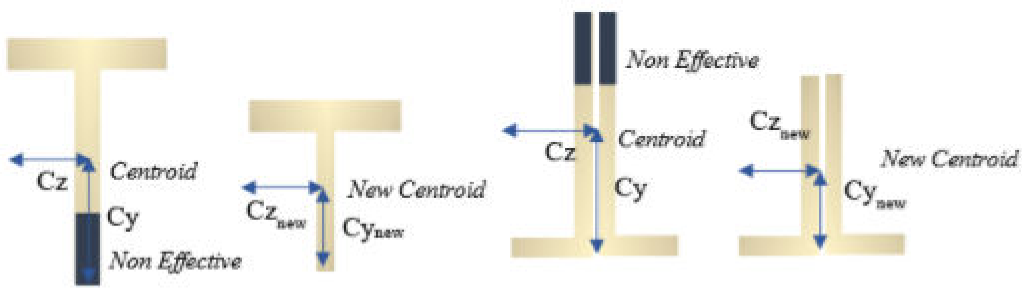
NOTE	Tees, double angles and single angles are designed as Class 4. Equation 6.15 is used for class 4 slender sections.

NOTE	Tees, double angles and single angles subject to moment with high shear are beyond scope.

Moment capacity for Class 4 slender sections:

Class 4 sections are designed as Class 3 effective sections.





Hence, additional moments are induced in the member due to the shift of the centroid of the effective cross-section compared to that of the gross section when under axial compression only.

Thus:

$\Delta M_{Ed,y} = e_y \times N_{Ed,max}$

$\Delta M_{Ed,z} = e_z \times N_{Ed,max}$

Where:

$N_{Ed,max}$ is the max compressive force in the span.

For tees and double angles $e_y = 0$. Hence, total minor design moment = minor design moment.

Where:

e_y and e_z = the shift of the centroid of the effective area A_{eff} relative to the centre of gravity of the gross cross section

$e_y = abs(cy_{new} - c_y)$

$e_z = abs(cz_{new} - c_z)$

So finally, a total moment is obtained for which the moment design check is performed:

$M_{total\ y} = Abs(M_{Ed,y}) + Abs(\Delta M_{Ed,y})$

$M_{total\ z} = Abs(M_{Ed,z}) + Abs(\Delta M_{Ed,z})$

Single angles - asymmetric sections:

Single angles with continuous lateral – torsional restraint along the length are permitted to be designed on the basis of geometric axis (y, z) bending.

Single angles without continuous lateral – torsional restraint along the length are designed using the provision for principal axis (u, v) bending since we know that the principal axes do not coincide with the geometric ones.

$\Delta M_u = \Delta M_y \times \cos\vartheta + \Delta M_z \times \sin\vartheta$

$\Delta M_v = -\Delta M_y \times \sin\vartheta + \Delta M_z \times \cos\vartheta$

Note that when principal axis design is required for single angles and the classification is Class 4, all moments are resolved into the principal axes (total moment in the principal axes u-u and v-v).

Combined bending and axial check

Section 6.2.9 of EC3 is used for this design check.

For Class 3 - Equation 6.42 is applied:

$Abs\ (N_{Ed} / A) + abs\ (M_{y,Ed} / W_{el,min,y}) + abs\ (M_{z,Ed} / W_{el,min,z}) \leq f_y / \gamma_{M0}$

For Class 4 - Equation 6.43 is applied:

Abs (N_{Ed}/A_{eff}) + (abs ($M_{y,Ed}$) + abs ($\Delta M_{y,Ed}$)) / $W_{eff,min,y}$ + abs ($M_{z,Ed}$) + abs ($\Delta M_{z,Ed}$)) / $W_{eff,min,z} \leq f_y / \gamma_{M0}$

Note that total moments are used when the section classification is Class 4.

For Class 4 cross section capacity - Equation 6.44 is applied.

Lateral torsional buckling check

NOTE LTB check for tees and double angles is currently beyond scope.

EC3 is completely silent on LTB check for asymmetric sections such as single angles and mono-symmetric sections such as double angles and tees.

Hence we follow the approach of the Blue Book (Ref. 10 (page 1934)):

Firstly we calculate the equivalent slenderness coefficient (φ) (From Blue book) and the equivalent slenderness λ_{LT} (BS approach).

Then we find the non-dimensional slenderness $\overline{\lambda_{LT}}$ in order to follow the EC design approach.

Conservatively we have taken: $\overline{\lambda_{LT}} = \lambda_{LT}/\lambda_1$

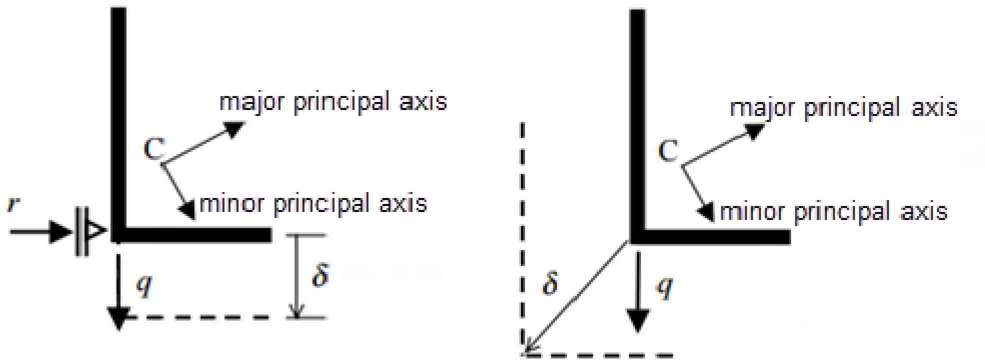
NOTE All intermediate LTB restraints for single angles, double angles and tees are ignored.

Combined buckling check

In the current version this check is beyond scope for single angles, double angles and tees.

Deflection of single angles (Eurocode)

If a single angle is continuously restrained the major geometric moment and major geometric section properties are used in the general equation governing the beam deflection.



Single angle deflections (continuously restrained, unrestrained)

However, because single angle geometric axes are not coincident with the principal axes; a different procedure is required if the angle is not continuously restrained, the procedure being as follows:

1. External loads are transposed from the geometric axes to the principal axes.
2. The deflection equations are used to calculate deflections in the principal axes.
3. These principal axis deflections are then transposed to geometric axes again.

References to EC3 and EC4 (Eurocode)

1. **British Standards Institution**. BS EN 1993-1-1:2005. Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings. **BSI 2005**.
2. **British Standards Institution**. NA to BS EN 1993-1-1:2005. UK National Annex to Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings. **BSI 2005**.
3. **National Standards Authority of Ireland**. I.S EN 1993-1-1 National Annex. Irish National Annex (informative) to Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings. **NSAI 2007**.
4. **British Standards Institution**. BS EN 1994-1-1:2004. Eurocode 4: Design of composite steel and concrete structures - Part 1-1: General rules and rules for buildings. **BSI 2005**.
5. **British Standards Institution**. NA to BS EN 1994-1-1:2004. UK National Annex to Eurocode 4: Design of composite steel and concrete structures - Part 1-1: General rules and rules for buildings. **BSI 2005**.
6. **British Standards Institution**. BS EN 1992-1-1:2004. Eurocode 2: Design of concrete structures. General rules and rules for buildings. **BSI 2004**.

7. **The Steel Construction Institute**. Publication 076. Design Guide on the Vibration of Floors. **SCI 1989**.
8. **The Steel Construction Institute**. Publication P355. Design of Composite Beams with Large Web Openings. **SCI 2011**.
9. **The Steel Construction Institute**. Publication 068. Design for openings in the webs of composite beams. **SCI 1987**.
10. **The Steel Construction Institute and The British Constructional Steelwork Association Ltd**. Publication P363. Steel Building Design: Design Data. **SCI and BCSA 2009**.

Concrete design to EC2 (Eurocode)

Tekla Structural Designer designs reinforced concrete members to a range of international codes. This reference guide specifically describes the design methods applied in the software when BS EN 1992-1-1:2004 (Ref. 1) (page 1989) is selected.

Unless explicitly noted otherwise, all clauses, figures and tables referred to are from BS EN 1992-1-1:2004

Within the remainder of this guide BS EN 1992-1-1:2004 is referred to as EC2.

General parameters (EC2)

Shrinkage and Creep

The following design parameters can be specified individually as part of each member's properties set.

Permanent Load Ratio

This is the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Reinforcement Anchorage Length Parameters

Max. Bond Quality Coefficient

Acceptable input range 0.5 to 1.0

In the bond stress calculation (Cl 8.4.2), the bond quality coefficient $\eta 1$ can be either 1.0 or 0.7 depending on section depth. Where 0.7 is used the bond strength is reduced and laps are extended.

Specifying a maximum of 1.0 for the Bond Quality Coefficient allows the coefficient to vary between 0.7 and 1.0 as required, hence lap lengths will vary accordingly.

Some users may prefer to specify a maximum of 0.7 (which actually fixes the coefficient at 0.7), the effect is to standardise on the use of extended lap lengths throughout. Further conservatism can be introduced in all lap lengths by using a value as low as 0.5.

Plain Bars Bond Quality Modifier

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for plain bars.

In Tekla Structural Designer a factor "T" has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for plain bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.5 would be reasonable.)

Type-1 Bars Bond Quality Modifier

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for Type 1 bars.

In Tekla Structural Designer a factor "T" has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for Type 1 bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.8 would be reasonable.)

Concrete beam design to EC2 (Eurocode)

The topics in this section describe how the software applies BS EN 1992-1-1:2004 (Ref. 1) (page 1989) to the design of reinforced concrete beams.

Limitations (concrete beam: EC2)

The following general exclusions apply:

- Seismic design,
- Consideration of fire resistance. [You are however given full control of the minimum cover dimension to the reinforcement and are therefore able to take due account of fire resistance requirements.]
- Openings in the beam web.
- Bundled bars.
- Design for minor axis bending and shear.
- Design for axial forces.

In addition, for beams classified as "deep beams":

- all beams with a ratio of 1.5 < span/overall depth ≤ 3.0 are designed but with an appropriate Warning
- beams with a ratio of span/overall depth ≤ 1.5 are *Beyond Scope*

Slender beams (concrete beam: EC2)

Second order effects associated with lateral instability may be ignored if beams are within the geometric limits given by the following;

$L_{0t} \leq 50*b_{comp}/(h/b_{comp})^{1/3}$

and

$h/b_{comp} \leq 2 .5$

where

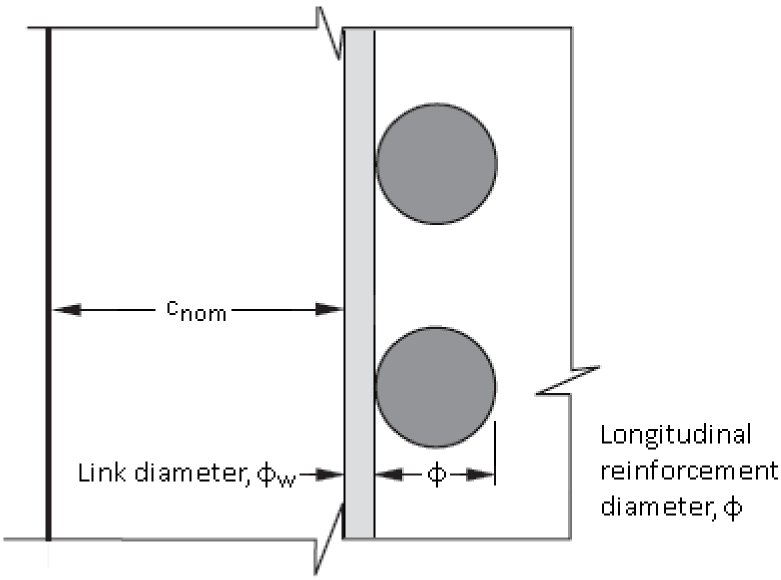
L_{0t}	=	the distance between torsional restraints, which in Tekla Structural Designer is taken as the distance between the faces of the supports

h	=	the total overall depth of the beam at the centre of L_{0t}
b_{comp}	=	the width of the compression flange of the beam (= b_w for rectangular sections and b_{eff} for flanged beams)

If either of the above checks fail then a Warning is displayed.

Cover to reinforcement (concrete beam: EC2)

The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.



You are required to set a minimum value for the nominal cover, $c_{nom, u}$, for the top, bottom, sides and ends of each beam in the beam properties.

These values are then checked against the nominal limiting cover, $c_{nom,lim}$ which depends on the diameter of the reinforcement plus an allowance for deviation, Δc_{dev} (specified in Design Options > Beam > General Parameters).

Generally, the allowance for deviation, Δc_{dev} is a NDP.[1] The recommended value is 10mm, but under strict controls it can be reduced to 5mm.

If $c_{nom,u} < c_{nom,lim}$ then a warning is displayed in the calculations.

[1] BS EN 1992-1-1:2004 cl 4.4.1.3 (1)P

Design parameters for longitudinal bars (EC2)

For each of these parameters, any user defined limits (as specified on the appropriate Reinforcement Settings page within Design Options) are considered in addition to the EC2 or NA recommendations.

Minimum diameter of reinforcement for pad and strip bases

For design in accordance with EC2 Recommendations;

φ_{min}	=	8 mm

For design in accordance with UK NA, Irish NA and Malaysian NA;

φ_{min}	=	8 mm

For design in accordance with Singapore NA;

φ_{min}	=	10 mm

Maximum diameter of reinforcement

At Section 8.8 of BS EN 1992-1-8:2004, additional rules are specified when "large diameter bars" are used in the design. A large diameter bar is defined as being a bar with a diameter larger than φ_{large} where φ_{large} is an NDP value.

For design in accordance with EC2 Recommendations;

φ_{large}	=	32 mm

For design in accordance with UK NA, Irish NA, Malaysian NA and Singapore NA;

φ_{large}	=	40 mm

In the current release the provisions of Section 8.8 are not implemented. If the design results in a bar size with $\varphi > \varphi_{large}$ then a warning is displayed.

NOTE Clause 7.3.3 (2) indicates that cracking can be controlled either by restricting the bar diameter or the max spacing. Tekla Structural Designer adopts the latter approach using Table 7.3N - therefore the maximum bar diameters specified in Table 7.2N are not checked.

Minimum distance between bars

The minimum clear horizontal distance between individual parallel bars, $s_{cl,min}$, is given by;[1]

[1] BS EN 1992-1-1:2004 Section 8.2(2)

$s_{cl,min}$		≥	MAX [k_1*φ, d_g+k_2, $s_{cl,u,min}$, 20 mm]

where

k_1		=	the appropriate NDP
k_2		=	the appropriate NDP
d_g		=	the maximum size of aggregate
φ		=	the maximum diameter of adjacent bars, $φ_i$ and $φ_j$
$s_{cl,u,min}$		=	user specified minimum clear distance between bars

NOTE To allow you to make decisions regarding access for concrete compaction or size of aggregate, a value for the minimum clear distance between bars can be specified on the appropriate Reinforcement Settings page within Design Options - separate values being set for bars in the top of a beam and for those in the bottom of a beam.

The minimum clear vertical distance between horizontal layers of parallel bars, $s_{cl,min}$, is given by;

$s_{cl,min}$		≥	MAX[k_1*φ, d_g+k_2, 20 mm]

For design in accordance with UK NA,EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;

k_1		=	1.0
k_2		=	5.0 mm

Maximum spacing of tension bars

The maximum centre to centre bar spacing for crack control, $s_{cr,max}$, is dependent on the maximum allowable crack width, w_{max}, specified in the beam properties from a menu of values which are: 0.20mm, 0.30mm or 0.40mm with a default value of 0.30mm.

The service stress in the reinforcement, $σ_s$, is given by;

$σ_s$		=	($A_{s,reqd}$/$A_{s,prov}$)*(f_{yk}/$γ_s$)*R_{PL}

where

$A_{s,reqd}$	=	area of reinforcement required for the maximum design Ultimate Limit State moment, M_{Ed}
$A_{s,prov}$	=	area of reinforcement provided
R_{PL}	=	permanent load ratio

In the beam properties you are required to supply a value for the permanent load ratio, R_{PL}. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.

The maximum allowable centre to centre bar spacing, $s_{cr,max}$, is then obtained from table 7.3N (shown below) by looking up the calculated value of the service stress in the reinforcement, σ_s, using interpolation between values of σ_s

Steel Service Stress, σ_s (N/mm^2)	Max Allowable bar Spacing, $s_{cr,max}$		
	w_{max} = 0.40 mm	w_{max} = 0.30 mm	w_{max} = 0.20 mm
≤ 160	300	300	200
200	300	250	150
240	250	200	100
280	200	150	50
320	150	100	Warning
360	100	50	Warning
>360	Warning	Warning	Warning

Maximum spacing of tension bars (slabs not exceeding 200mm)

In accordance with clause 7.3.3(1) of EC2 for slabs not exceeding 200mm in overall depth and not subjected to significant axial tension the maximum limit on centre to centre bar spacing is governed by clause 9.3 only and there is no need to perform specific checks on the bar spacings to control cracking. These limits are applied to all slabs and then the additional limit in the next section are applied to slabs greater than 200mm thick.

From clause 9.3 the maximum limit on bar spacings can be somewhat subjective so these limits will be user definable with conservative defaults as follows :-

Principal bars (NDP) (cl. 9.3.1.1(3))

s_{max} = 2h but ≤ 250mm

Secondary bars (NDP) (cl. 9.3.1.1(3))

s_{max} = 3h but ≤ 400mm

Bars are classed as secondary if both the following are true:

1. The design moment for bars in this direction is lower than the design moment for bars in the other direction.
2. The calculated reinforcement requirement based on the design moment is less than the minimum reinforcement requirement.

Minimum area of reinforcement

The minimum area of longitudinal tension reinforcement, $A_{s,min}$, is given by;[2]

$A_{s,min}$	≥	MAX[k_{min1}*b_w*d*(f_{ctm}/f_{yk}), k_{min2}*b_w*d]

where

k_{min1}	=	the appropriate NDP value
k_{min2}	=	the appropriate NDP value
f_{ctm}	=	mean value of the axial tensile strength of the concrete
f_{yk}	=	characteristic yield strength of the reinforcement

For design in accordance with UK NA,EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;

k_{min1}	=	0.26
k_{min2}	=	0.0013

NOTE Note that there is no requirement to have a minimum area of compression reinforcement.

The minimum area of longitudinal tension reinforcement for crack control, $A_{s,min,cr}$ is given by;[3]

$A_{s,min,cr}$	≥	0.4*k*f_{ctm}*A_{ct}/σ_s

[2] BS EN 1992-1-1:2004 Section 9.2.1.1(1)

[3] BS EN 1992-1-1:2004 Section 7.3.2(2)

where

k	=	1.0 when h ≤ 300
		0.65 when h ≥ 800
f_{ctm}	=	mean value of axial tensile strength of concrete
		$0.30*f_{ck}^{(2/3)}$ for concrete grades ≤ C50/60
		$2.12*\ln(1+((f_{ck}+8)/10))$ for concrete grades > C50/60
σ_s	=	the interpolated reinforcement service stress from appropriate for the bar spacing of the reinforcement provided
A_{ct}	=	area of concrete in tension just before formation of first crack
	=	b*y where y = the distance of the Elastic NA from bottom of beam

The minimum area of longitudinal tension reinforcement required, $A_{s,min,reqd}$, is then given by;

$A_{s,min,reqd}$	≥	MAX ($A_{s,min}$, $A_{s,min,cr}$)

Maximum area of reinforcement

The maximum area of longitudinal tension reinforcement, $A_{st,max}$, is given by;[4]

$A_{st,max}$	≤	$k_{max}*A_c$

The maximum area of longitudinal compression reinforcement, $A_{sc,max}$, is given by;

$A_{sc,max}$	≤	$k_{max}*A_c$

where

[4] BS EN 1992-1-1:2004 Section 9.2.1.1(3)

k_{max}	=	the appropriate NDP value
A_c	=	the cross sectional area of the beam
	=	$h*b_w$

For design in accordance with UK NA,EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;

k_{max}	=	0.04

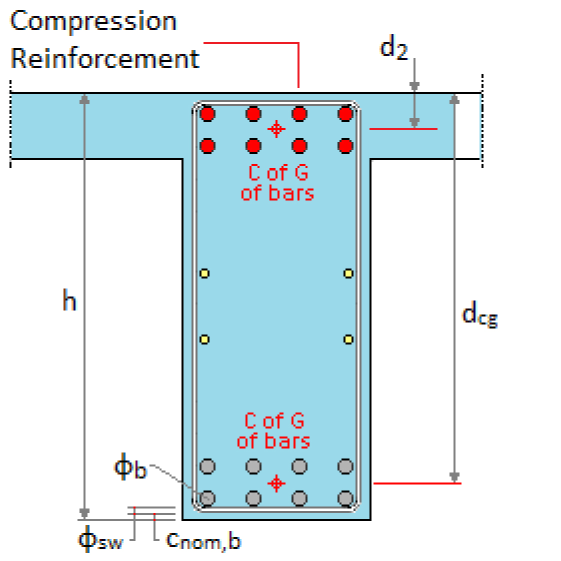
Side reinforcement (concrete beam: EC2)

To control cracking in beams with a total depth ≥ 1000 mm, side bars are provided in the side faces of the beam as per BS EN 1992-1-1:2004 Section 7.3.3(3).

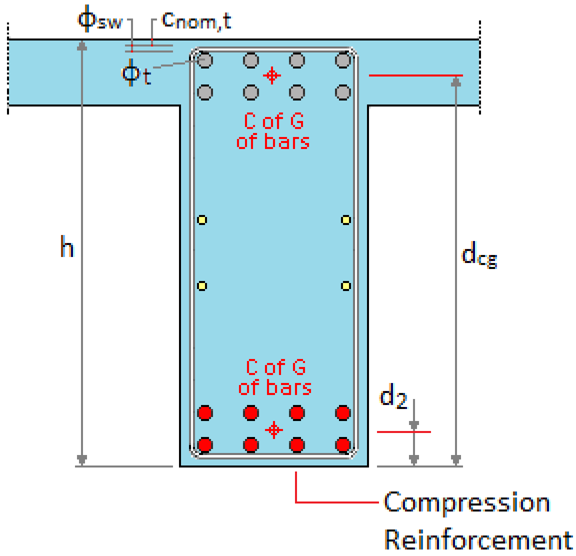
Effective depth of section (concrete beam: EC2)

For the design of the longitudinal tension reinforcement, the effective depth of a section, d is defined as the distance from the extreme concrete fibre in compression to the centre of gravity of the longitudinal tension reinforcement.

For the design of the longitudinal compression reinforcement, the effective depth in compression, d_2 is defined as the distance from the extreme fibre in compression to the centre of gravity of the longitudinal compression reinforcement.



Tension Reinforcement in Bottom of Beam



Tension Reinforcement in Top of Beam

Design for bending for rectangular sections (beams and slabs: EC2)

Calculate the value of *K* from;

$K = M_{Ed}/(f_{ck}*b_w*d^2)$

Then calculate the limiting value of K, known as *K'* from;

$K' = (2*\eta*\alpha_{cc} /\gamma_C)*(1 - \lambda*(\delta - k_1)/(2*k_2))*(\lambda*(\delta - k_1)/(2*k_2))$ for $f_{ck} \leq 50$ N/mm^2

$K' = (2*\eta*\alpha_{cc} /\gamma_C)*(1 - \lambda*(\delta - k_3)/(2*k_4))*(\lambda*(\delta - k_3)/(2*k_4))$ for $f_{ck} > 50$ N/mm^2

where

k_i	=	moment redistribution factors
δ	=	moment redistribution ratio (= 1.0 in the current release)
γ_C	=	the NDP partial safety factor for concrete
α_{cc}	=	coefficient to take account of long term effects on compressive strength of concrete
λ	=	1. 8 for $f_{ck} \leq 50$ N/mm^2
	=	1. 8-(f_{ck}-50)/400 for 50 < $f_{ck} \leq 90$ N/mm^2
η	=	1. 0 for $f_{ck} \leq 50$ N/mm
	=	1. 0-(f_{ck}-50)/200 for 50 < $f_{ck} \leq 90$ N/mm^2

For design in accordance with **UK NA**, **Irish NA**, **Malaysian NA** and **Singapore NA**;

$\gamma_C = 1.5$

$\alpha_{cc} = 0.85$

For design in accordance with **EC2 Recommendations**;

$\gamma_C = 1.5$

$\alpha_{cc} = 1.0$

IF K ≤ K' THEN compression reinforcement is not required.

Calculate the lever arm, *z* from;

$z = MIN(0.5*d*[1 + (1 - 2*K/(\eta*\alpha_{cc}/\gamma_C))^{0.5}], 0.95*d)$

The area of tension reinforcement required is then given by;

$A_{st,reqd} = M_{Ed}/(f_{yd}*z)$

where

$f_{yd} = f_{yk}/\gamma_S$

γ_S = the NDP partial safety factor for reinforcement

The depth to the neutral axis, x_u is given by;

$x_u = 2*(d-z)/\lambda$

For design in accordance with **UK NA, EC2 Recommendations**, **Irish NA**, **Malaysian NA** and **Singapore NA**;

$\gamma_S = 1.15$

IF K > K' THEN compression reinforcement is required.

Calculate the depth to the neutral axis from;

$x_u = d*(\delta-k_1)/k_2$ for $f_{ck} \leq 50$ N/mm^2

$x_u = d*(\delta-k_3)/k_4$ for $f_{ck} > 50$ N/mm^2

Calculate the stress in the reinforcement from;

$f_{sc} = MIN(E_s*\varepsilon_{cu3}*(1-(d_2/x_u), f_{yd})$

where

d_2= the distance from the extreme fibre in compression to the c of g of the compression reinforcement

Calculate the limiting bending strength, *M'* from;

$M' = K'*f_{ck}*b_w*d^2$

Calculate the lever arm from;

$z = 0.5*d*[1 + (1 - 2*K'/(\eta*\alpha_{cc}/\gamma_C))^{0.5}]$

The area of compression reinforcement required, $A_{s2,reqd}$ is given by;

$A_{s2,reqd} = (M_{Ed}-M')/(f_{sc}*(d-d_2))$

The area of tension reinforcement required, $A_{st,reqd}$ is given by;

$A_{st,reqd} = M'/(f_{yd}*z) + A_{s2,reqd}*f_{sc}/f_{yd}$

Design for bending for flanged sections (concrete beam: EC2)

IF $h_f < 0.1*d$ THEN treat the beam as rectangular.

$h_f = MIN(h_{f,side1}, h_{f,side2})$

where

$h_{f,sidei}$ = the depth of the slab on side " *i*" of the beam

Calculate the value of *K* from;

$K = M_{Ed}/(f_{ck}*b_{eff}*d^2)$

Calculate the lever arm, *z* from;

$z = MIN(0.5*d*[1 + (1 - 2*K/(\eta*\alpha_{cc}/\gamma_C))^{0.5}], 0.95*d)$

Calculate the depth of the rectangular stress block, *λ*x* from;

$\lambda*x = 2*(d-z)$

IF $\lambda*x \leq h_f$ THEN the rectangular compression block is wholly in the depth of the flange and the section can be designed as a rectangular section by setting $b_w = b_{eff}$.

IF $\lambda*x > h_f$ THEN the rectangular compression block extends into the rib of the flanged section and the following design method is to be used.

The design bending strength of the flange, M_f is given by;

$M_f = f_{cd}*h_f*(b_{eff}-b_w)*(d-0.5*h_f)$

The area of reinforcement required to provide this bending strength, $A_{sf,reqd}$ is given by;

$A_{sf,reqd} = M_f/(f_{yd}*(d-0.5*h_f))$

The remaining design moment, $(M_{Ed}-M_f)$ is then taken by the rectangular beam section.

Calculate the value of *K* from;

$K = (M_{Ed}-M_f)/(f_{ck}*b_w*d^2)$

Then calculate the limiting value of K, known as *K'* from;

$K' = (2*\eta*\alpha_{cc}/\gamma_C)*(1 - \lambda*(\delta - k_1)/(2*k_2))*(\lambda*(\delta - k_1)/(2*k_2))$ for $f_{ck} \leq 50$ N/mm^2

$K' = (2*\eta*\alpha_{cc}/\gamma_C)*(1 - \lambda*(\delta - k_3)/(2*k_4))*(\lambda*(\delta - k_3)/(2*k_4))$ for $f_{ck} > 50$ N/mm^2

IF K ≤ K' THEN compression reinforcement is not required.

Calculate the lever arm, *z* from;

$z = MIN(0.5*d*[1 + (1 - 2*K/(\eta*\alpha_{cc}/\gamma_C))^{0.5}], 0.95*d)$

The area of tension reinforcement required is then given by;

$A_{sr,reqd} = (M_{Ed}-M_f)/(f_{yd}*z)$

The total area of tension reinforcement required, $A_{st,reqd}$ is then given by;

$A_{st,reqd} = A_{sf,reqd}+A_{sr,reqd}$

The depth to the neutral axis, x_u is given by;

$x_u = 2*(d-z)/\lambda$

IF K > K' THEN compression reinforcement is required.

Calculate the depth to the neutral axis from;

$x_u = d*(\delta-k_1)/k_2$ for $f_{ck} \leq 50$ N/mm^2

$x_u = d*(\delta-k_3)/k_4$ for $f_{ck} > 50$ N/mm^2

Calculate the stress in the reinforcement from;

$f_{sc} = MAX(E_s*\varepsilon_{cu3}*(1-(d_2/x_u), f_{yd})$

where

d_2 = the distance from the extreme fibre in compression to the c of g of the compression reinforcement

Calculate the limiting bending strength, *M'* from;

$M' = K'*f_{ck}*b_w*d^2$

Calculate the lever arm from;

$z = 0.5*d*[1 + (1 - 2*K'/(\eta*\alpha_{cc}/\gamma_C))^{0.5}]$

The area of compression reinforcement required, $A_{s2,reqd}$ is given by;

$A_{s2,reqd} = (M_{Ed}-M_f-M')/(f_{sc}*(d-d_2))$

The area of tension reinforcement required, $A_{sr,reqd}$ is given by;

$A_{sr,reqd} = M'/(f_{yd}*z) + A_{s2,reqd}*f_{sc}/f_{yd}$

The total area of tension reinforcement required, $A_{st,reqd}$ is then given by;

$A_{st,reqd} = A_{sf,reqd}+A_{sr,reqd}$

Design shear resistance (concrete beam: EC2)

The design value of the shear resistance of a concrete section with vertical shear reinforcement, $V_{Rd,max}$ is given by;

$V_{Rd,max} = 0.9*\alpha_{cw}*b_w*d*\nu_1*f_{cwd}/(\cot\theta + \tan\theta)$

where

$\theta = MIN \{\theta_{max}, MAX[0.5*\sin^{-1}[2*V_{Ed,max}/(\alpha_{cw}*b_w*0.9*d*\nu_1*f_{cwd})], \theta_{min}]\}$

f_{cwd}[1]$= \alpha_{ccw}*f_{ck}/\gamma_C$

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;**

α_{cw} = 1.0 (assuming no axial load in the beam)

$\alpha_{ccw} = 1.0$

$\gamma_C = 1.5$

$\nu_1 = 0.6*(1 - (f_{ck}/250))$ f_{ck} in N/mm²

The limits of θ are given by $1 \leq \cot\theta \leq 2.5$ which gives;

$\theta_{max} = \tan^{-1}1$

$\theta_{min} = \tan^{-1}(0.4)$

IF $V_{Ed,max} > V_{Rd,max}$

where

$V_{Ed,max}$ = the maximum design shear force acting anywhere on the beam

THEN the shear design process FAILS since the section size is inadequate for shear (the compression strut has failed at the maximum allowable angle).

The design shear capacity of the minimum area of shear links actually provided, V_{nom} is given by[2];

$V_{nom} = (A_{sw,min,prov} /s_l) * 0.9 * d * f_{ywd} * \cot\theta$

where

$A_{sw,min, prov}$ is the area of shear reinforcement provided to meet the minimum requirements.

$f_{ywd} = f_{ywk}/\gamma_S$

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA** the limiting values of θ are given by;

$1 \leq \cot\theta \leq 2.5$

---

[1] Eqn (3.15) BS EN 1992-1-1:2004 Section 3.1.6(1)P

[2] BS EN 1992-1-1:2004 Section 6.2.3(3) Eqn (6.8)

and: γ_S = 1.15

The maximum possible value for the shear resistance provided by this area of shear reinforcement will be when the angle of the compression strut is the minimum value i.e. cotθ = 2.5 and therefore V_{nom} can be simplified to;

$V_{nom} = (A_{sw,min,prov}/s_l) * 2.25 * d * f_{ywd}$

In any region, *i*;

IF

$V_{Ed,i} > V_{nom}$

where

$V_{Ed,i}$ = the maximum shear in region *i*

THEN shear links are required in the region.

For designed shear links in shear region *Si*, first calculate the angle of the compression strut from;

$\theta_{Si} = MIN\{\theta_{max}, MAX[0.5*\sin^{-1}[2*V_{Ed,Si} /(\alpha_{cw}*b_w*0.9*d*\nu_1*f_{cd})], \theta_{min}]\}$

The area of links required in shear region *Si* is then given by;

$(A_{sw,reqd}/s)_{Si} = V_{Ed,Si}/(0.9*d*f_{ywd}*\cot\theta_{Si})$

where

$V_{Ed,Si}$ = the maximum shear force in shear region *Si*

Minimum area of shear reinforcement (concrete beam: EC2)

The minimum area of shear reinforcement required, $A_{sw,min}$ is given by[1];

$A_{sw,min} = MAX[s_l*\rho_{w,min}*b_w, A_{sw,min,u}]$

where

s_l = the spacing of the shear reinforcement along the longitudinal axis of the beam

$A_{sw,min,u}$ = the total minimum area of the shear reinforcement calculated from data supplied i.e. maximum spacing across the beam, minimum link diameter and number of legs

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA**;

$\rho_{w,min} = (0.08*\sqrt{(f_{ck})})/f_{yk}$

[1] BS EN 1992-1-1:2004 Section 9.2.2(5) Eqn (9.4)

Spacing of shear reinforcement (concrete beam: EC2)

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA** the longitudinal spacing, s_l between the legs of shear reinforcement is given by;

$s_{l,min,u} \leq s_l \leq MIN[0.75*d, s_{l,max,u}]$

where

$s_{l,max,u}$ = the maximum longitudinal spacing specified

$s_{l,min,u}$ = the minimum longitudinal spacing specified

If compression reinforcement is required for bending, for design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA** the longitudinal spacing, s_l between the legs of shear reinforcement is given by;

$s_{l,min,u} \leq s_l \leq MIN\{MIN[0.75*d, 15*\Phi_{comp}] , s_{l,max,u}\}$

where

Φ_{comp} = the minimum diameter of the compression reinforcement

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA** the transverse spacing, s_t between the legs of shear reinforcement is given by;

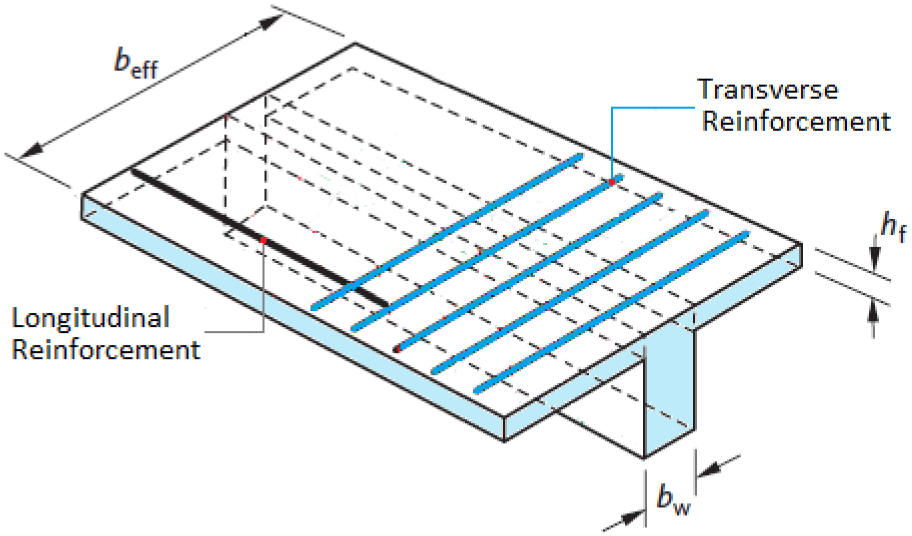
$s_t \leq MIN[0.75*d, 600, s_{t,max,u}]$

where

$s_{t,max,u}$ = the maximum link leg spacing across the beam

Shear between flanges and web of flanged beams (concrete beam: EC2)

The shear strength of the interface between the flanges and the web of a flanged beam is checked and, if necessary, transverse reinforcement is provided as shown in the diagram below.[1]



Additional tension reinforcement (concrete beam: EC2)

In BS EN 1992-1-1:2004, the method of designing for vertical shear is based on a truss analogy with a diagonal strut acting at an angle θ. This strut action means that there must be a tension force developed in the longitudinal reinforcement which is additional to that arising from bending action.

To resist this tension force, an area of reinforcement additional to that required to resist bending is required.

The total area of longitudinal tension reinforcement in each of the regions then becomes;

$A_{stt,reqd,i} = A_{st,reqd,i} + A_{swa,reqd,i}$

where

$A_{st,reqd,i}$ = the area of longitudinal reinforcement required to resist bending as appropriate in region " *i*"

$A_{swa,reqd,i}$ = the area of longitudinal reinforcement required to resist the additional tension force from vertical shear in region " *i*"

---

[1] BS EN 1992-1-1:2001 Section 6.2.4

Design values of shear resistance and torsional resistance moment (Concrete beam: EC2)

The design value of the shear resistance of a concrete section with vertical shear reinforcement, $V_{Rd,max}$ is given by;

$V_{Rd,max} = 0.9*\alpha_{cw}*b_w*d*\nu_1*f_{cwd}/(\cot\theta + \tan\theta)$

where

$\theta = MIN\{\theta_{max}, MAX[0.5*\sin^{-1}[2*V_{Ed,max}/(\alpha_{cw}*b_w*0.9*d*\nu_1*f_{cwd})], \theta_{min}]\}$

$f_{cwd} = \alpha_{ccw}*f_{ck}/\gamma_C$

The maximum design value of the torsional resistance moment, $T_{Rd,max}$ is given by;

$T_{Rd,max} = 2*\nu_1*\alpha_{ccw}*f_{cwd}*A_k*t_{ef}*\sin\theta*\cos\theta$

where

$A_k = (h-t_{ef})*(b_w-t_{ef})$

and

$t_{ef} = MAX(A/u, 2*(h-d_o))$[1]

where

$A = h*b_w$

u = the outer circumference of the cross-section

$= 2*(h+b_w)$

d_o = the effective depth of the outer layer of longitudinal reinforcement

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;**

$\alpha_{cw} = 1.0$

$\alpha_{ccw} = 1.0$

$\gamma_C = 1.5$

$\nu_1 = 0.6*(1 - f_{ck}/250)$ f_{ck} in N/mm^2

The limits of θ are given by $1 \le \cot\theta \le 2.5$ which gives;

$\theta_{max} = \tan^{-1}1$

$\theta_{min} = \tan^{-1}(0.4)$

The design value of the torsional resistance moment of a concrete section with no shear reinforcement, $T_{Rd,c}$ is given by[2];

Eqn (3.15) BS EN 1992-1-1:2004 Section 3.1.6(1)P

[1] BS EN 1992-1-1:2004 Section 6.3.2(1)

$T_{Rd,c} = 2*A_k*t_{ef}*f_{ctd}$

where

f_{ctd} = the design tensile strength of the concrete

$= \alpha_{ct}*f_{ctk,0.05}/\gamma_C$

If the maximum torsional moment acting on the beam, $T_{Ed,max}$ is less than the ignorable torque limit then no further calculations are necessary.

Otherwise:

If $(T_{Ed,max,i}/T_{Rd,max}) + (V_{Ed,max,i}/V_{Rd,max}) \leq 1.0$

then the torsion design process can proceed.

ELSE the torsion design FAILS since the section size is inadequate for torsion.

Additional reinforcement for torsion (Concrete beam: EC2)

The design value of the shear resistance of a concrete section with no shear reinforcement, $V_{Rd,c}$ is given by;[1]

$V_{Rd,c} = \nu_{min} * b_w * d$

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;**

$C_{Rd,c} = 0.18/\gamma_C$

$\gamma_C = 1.5$

$\nu_{min} = 0.035 * k^{1.5} * f_{ck}{}^{0.5}$

where

k = MIN(1 + √(200/d), 2.0) *d* in mm

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA;**

$\alpha_{ct} = 1.0$

$\gamma_C = 1.5$

If $(T_{Ed,max}/T_{Rd,c}) + (V_{Ed,max}/V_{Rd,c}) \leq 1.0$

THEN no additional longitudinal reinforcement for torsion is required.

IF $(T_{Ed,max}/T_{Rd,c}) + (V_{Ed,max}/V_{Rd,c}) > 1.0$

THEN additional longitudinal reinforcement for torsion, $A_{slT,reqd}$ is required in some or all regions.

---

[2] BS EN 1992-1-1:2004 Eqn (6.26) with $\tau_{t,i} = f_{ctd}$

[1] The design value of the shear resistance is calculated ignoring the longitudinal reinforcement as it is not known if this reinforcement is adequately anchored beyond the point under consideration. This is a conservative approach.

The additional longitudinal reinforcement is given by;

$A_{slT,reqd} = (T_{Ed} * u_k * \cot\theta)/(2 * A_k * f_{yd})$

where

$u_k = 2*((h-t_{ef})+(b_w-t_{ef}))$

This reinforcement is **in addition** to that required for bending and tension arising from vertical shear and it is distributed in each of the four faces of the beam in proportion to the length of the face of the cross-section.

The area of the additional link reinforcement that is required to resist torsion is given by;

$A_{swt}/s = (T_{Ed})/(2 * A_k * 0.9 * f_{ywd} * \cot\theta)$ per leg

Deflection check (beam and slab: EC2)

The deflection of reinforced concrete beams is not directly calculated and the serviceability of the beam is measured by comparing the calculated limiting span/effective depth ratio *L/d* to the maximum allowable values as given by;[1]

IF $\rho \le \rho_0$

$(L/d)_{max} = K_{ss}*f_1*f_2*(11+1.5*(f_{ck})^{1/2}*(\rho_0/\rho)+3.2*(f_{ck})^{1/2}*((\rho_0/\rho)-1)^{3/2})*(500*A_{st,prov}/(f_{yk}*A_{st,reqd}))$

IF $\rho > \rho_0$

$(L/d)_{max} = K_{ss}*f_1*f_2*(11+1.5*(f_{ck})^{1/2}*(\rho_0/(\rho-\rho'))+(1/12)*(f_{ck})^{1/2}*(\rho'/\rho_0)^{1/2})*(500*A_{st,prov}/(f_{yk}*A_{st,reqd}))$

where

ρ	=	the designed tension reinforcement ratio at mid-span (or at support for cantilevers) required to resist bending
	=	$A_{st.reqd}/(b_w*d)$ for rectangular beams
	=	$A_{st,reqd}/(b_{eff}*d)$ for flanged beams
ρ'	=	the designed compression reinforcement ratio at mid-span (or at support for cantilevers) required to resist bending
	=	$A_{s2,reqd}/(b_w*d)$ for rectangular beams

[1]: This definition of effective length will return conservative results when the width of the support is greater than the depth of the beam - see BS EN 1992-1-1:2004 Section 5.3.2.2(1)

[1] BS EN 1992-1-1:2004 Section 7.4.2

	=	$A_{s2,reqd}$ /(b_{eff} *d) for flanged beams
$A_{st,reqd}$	=	the designed area of tension reinforcement at mid-span (or at support for cantilevers) required to resist bending
$A_{st,prov}$	=	MIN(the area of tension reinforcement provided at mid-span (or at support for cantilevers), f_3 *$A_{st,reqd}$)
$A_{s2,reqd}$	=	the designed area of compression reinforcement at mid-span (or at support for cantilevers) required to resist bending
f_1	=	1. 0 for rectangular beams
	=	1. 0 for flanged beams with $b_{eff}/b_w \le 3.0$
	=	1. 8 for flanged beams with $b_{eff}/b_w > 3.0$
f_2	=	1. 0 IF $L_{eff} \le 7$ m
	=	7/L_{eff} IF L_{eff} > 7 m with L_{eff} in metre units
L_{eff}	=	the length of the beam between the centre of its supports[1]
f_3	=	an NDP factor as given below
K_{ss}	=	the structural system factor which is an NDP and is given below

[1]: This definition of effective length will return conservative results when the width of the support is greater than the depth of the beam - see BS EN 1992-1-1:2004 Section 5.3.2.2(1)

For design in accordance with **EC2 Recommendations** the NDP value of f_3 is given by[3];

f_3 = 1.5

For design in accordance with **UK NA, Irish NA, Malaysian NA and Singapore NA** the NDP value of f_3 is given by[4];

f_3 = 1.5

For design in accordance with **UK NA, EC2 Recommendations, Irish NA, Malaysian NA and Singapore NA** the NDP value of K_{ss} is given by the following table:

Span Detail	LH End Fixity	RH End Fixity	K_{ss}
LH End Span	Fixed	Fixed	1.3

[3] BS EN 1992-1-1:2004 is silent on the recommended value to use therefore adopt 1.5 since if f_3 is greater than 1.5 no benefit arises.

[4] For Irish NA refer to Table NA.3 and for other others refer to Table NA.5

Span Detail	LH End Fixity	RH End Fixity	K_{ss}
	Fixed	Pinned	1.0
	Pinned	Fixed	1.3
	Pinned	Pinned	1.0
Internal Span	Fixed	Fixed	1.5
	Fixed	Pinned	1.3
	Pinned	Fixed	1.3
	Pinned	Pinned	1.0
RH End Span	Fixed	Fixed	1.3
	Fixed	Pinned	1.3
	Pinned	Fixed	1.0
	Pinned	Pinned	1.0
Cantilever			0.4

Concrete column design to EC2 (Eurocode)

The topics in this section describe how the software applies BS EN 1992-1-1:2004 (Ref. 1) (page 1989) to the design of reinforced concrete columns.

Limitations (Concrete column: EC2)

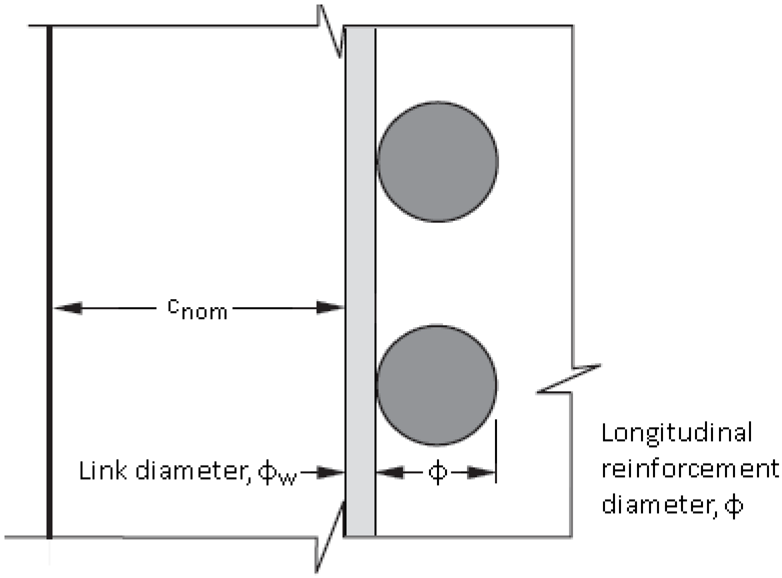
The longitudinal and transverse reinforcement requirements of clause 9.5 are applied to all columns, including columns where the larger dimension is greater than 4 times the smaller dimension - this is conservative.

The following general exclusions also apply:

- Seismic design,
- Consideration of fire resistance. [You are however given full control of the minimum cover dimension to the reinforcement and are therefore able to take due account of fire resistance requirements.],
- Chamfers,
- Multi-stack reinforcement lifts.

Cover to Reinforcement (Concrete column: EC2)

The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.



You are required to set a minimum value for the nominal cover, $c_{nom,\,u}$, for each column in the column properties.

These values are then checked against the nominal limiting cover, $c_{nom,lim}$ which depends on the diameter of the reinforcement plus an allowance for deviation, Δc_{dev} (specified in Design Options > Column > General Parameters).

Generally, the allowance for deviation, Δc_{dev} is a NDP. The recommended value is 10mm, but under strict controls it can be reduced to 5mm.

If $c_{nom,u} < c_{nom,lim}$ then a warning is displayed in the calculations.

Design parameters for longitudinal bars (concrete column: EC2)

For some of the longitudinal reinforcement design parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Column > Reinforcement Layout.

Minimum Longitudinal Bar Diameter

For design in accordance with **EC2 Recommendations**;

$\varphi_{long,min}$ = 8mm

BS EN 1992-1-1:2004 cl 4.4.1.3 (1)P

For design in accordance with **Malaysian NA**;

$\varphi_{long,min}$ = 10m

For design in accordance with **UK NA, Irish NA** and **Singapore NA**;

$\varphi_{long,min}$ = 12mm

Minimum Longitudinal Bar Spacing

For design to[1] **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$s_{cl,min}$ ≥ MAX[maximum longitudinal bar diameter, 20mm, d_g + 5mm]

Where d_g is the maximum aggregate size.

Maximum Longitudinal Bar Spacing

You are given control over this value by specifying an upper limit in Design Options > Column > Reinforcement Layout.

Minimum Longitudinal Total Steel Area

For design to[2] **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

If N_{Ed} >= 0 (i.e. compression)

$A_{sl,min}$ = MAX[(0.1 * N_{Ed}) / f_{yd}, 0.2% * column area]

Else

$A_{sl,min}$ = 0.45% * column area

NOTE It has been decided that in the tension case, in the absence of clear guidance by EC2, it is responsible and conservative to adopt the 0.45% used by BS8110.

Maximum Longitudinal Total Steel Area

For design to[3] **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$A_{sl,max}$ = 4% * column area (8% in lap regions)

Long Term Compressive Strength Factor

For design in accordance with[4] **UK NA, Irish NA, Malaysian NA** and **Singapore NA**;

[1] BS EN 1992-1-1:2004 Section 8.2

[2] BS EN 1992-1-1:2004 Section 9.5.2(2)

[3] BS EN 1992-1-1:2004 Section 9.5.2(3)

[4] BS EN 1992-1-1:2004 Section 3.1.6(1)

$\alpha_{cc} = 0.85$

For design in accordance with **EC2 Recommendations**;

$\alpha_{cc} = 1.0$

Design Concrete Compressive Strength for Shear

For design to[4] **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$f_{cd} = \alpha_{cc} * f_{ck} / \gamma_C$

Ultimate axial load limit (column and wall: EC2)

This limit is when the section is under pure compression (i.e. no moment is applied). It is observed that for non-symmetric arrangements, applying a small moment in one direction may increase the maximum axial load that can be applied to a section because the peak of the N-M interaction diagram is shifted away from the N-axis (i.e. the zero moment line). Checking that the axial load does not exceed the ultimate axial load limit of the section ensures that there is always a positive moment limit and a negative moment limit for the applied axial load for the section.

The ultimate axial load limit of the section, assuming a rectangular stress distribution, is calculated from:

$N_{max} = (RF * A_c * f_{cd} * \eta) + \sum(A_{s,i} * f_{s,i})$

Given that,

$A_c = A - \sum A_{s,i}$

$f_{s,i} = \varepsilon_c * E_{s,i}$

Where

RF is the concrete design reduction factor, (this is a fixed value of 0.9 which cannot be changed)

A is the overall area of the section,

A_c is the area of concrete in the section,

$A_{s.i}$ is the area of bar *i*,

f_{cd} is the design compressive strength of the concrete,

η is a reduction factor for the design compressive strength for high strength concrete for the rectangular stress distribution,

ε_c is the strain in the concrete at reaching the maximum strength,

$f_{s,i}$ is the stress in bar *i* when the concrete reaches the maximum strength,

$E_{s,i}$ is the modulus of elasticity of the steel used in bar *i*.

---

[4] BS EN 1992-1-1:2004 Section 3.1.6(1)

NOTE The concrete design reduction factor RF originates from EC2 section 3.1.7(3): "Note: If the width of the compression zone decreases in the direction of the extreme compression fibre, the value η_{fcd} should be reduced by 10%"

In Tekla Structural Designer the RF factor is applied in both the axial-moment interaction check and the ultimate axial resistance check (even though there is no extreme compression fibre in this latter calculation) so that the ultimate axial resistance matches the peak position of the interaction diagram - its inclusion creates a conservative result.

Effective length calculations (column and wall:EC2)

Clear Height

The clear height is the clear dimension between the restraining beams at the bottom of the stack and the restraining beams at the top of the stack. The clear height may be different in each direction.

If, at an end of the stack, no effective beams or flat slab to include are found, then the clear height includes the stack beyond this restraint, and the same rules apply for finding the end of the clear height at the end of the next stack (and so on).

Effective Length

The effective length, l_0 is calculated automatically - you also have the ability to override the calculated value.

From EC2, cl. 5.8.3.2, the equations for calculating the effective length are as follows.

For stacks designated as "braced", the effective length is given by[1]:

$l_0 = 0.5 * l * \sqrt{(1 + (k_1 / (0.45 + k_1)))} * \sqrt{(1 + (k_2 / (0.45 + k_2)))}$

In addition Tekla Structural Designer imposes the following limits for stacks that are designated as braced:

$5 \le l_0 / l \le 1$

For stacks designated as "bracing", the effective length is the larger of[1]:

$l_0 = l * \sqrt{(1 + (10 * k_1 * k_2 / (k_1 + k_2)))}$

Or

$l_0 = l * (1 + (k_1 / (1 + k_1))) * (1 + (k_2 / (1 + k_2)))$

Where

[1] BS EN 1992-1-1:2004 Section 5.8.3.2(3)

k_1 and k_2 are the relative flexibilities of rotational restraints at ends 1 and 2 respectively, in the direction under consideration. Which way the ends are numbered is irrelevant to the result. The program uses the bottom end of the stack as end 1 and the top end as end 2.

The value of *k*, which may refer to either k_1 or k_2 depending on which end of the stack is being examined, is defined by[1]:

$k = (\theta / M) * (E * I / l)$

Where

M is the moment applied to the restraining members by the buckling member or members,

θ is the rotation of the joint at the end of the stack considered for the bending moment *M*,

*(E * I / l)* is the bending stiffness of the compression member or members considered to be buckling.

It is recommended to take "θ / M" for the beams as "l / (2 * E * I)" .

The standard approximation [2] for "θ / M" is between "l / (4 * E * I)" and "l / (3 * E * I)", so to allow for cracking the value is increased. Also, "E * I / l" is the sum of the stiffness of column stacks joining at the connection.

The above equation then becomes:

$k = \sum(E * I / l)_{cols} / \sum(2 * E * I / l)_{beams}$

If there are any adjacent stacks beyond the joint at the end of the restrained length under consideration, then it must be considered whether these adjacent stacks are likely to contribute to the deflection or restrain it. If the stiffness are similar then the stiffness of the adjacent stacks can be ignored, and the guidance in PD6687 suggests that this range of similarity of stiffness can be taken as 15% above or below the stiffness of the stack being designed. Therefore:

If

$1.85 \leq \sum((E * I / l)_{\text{stacks beyond this joint}}) / (E * I / l)_{\text{stack under consideration}} \leq 1.15$

Then

$\sum(E * I / l)_{cols} = (E * I / l)_{\text{stack under consideration}}$

Else

$\sum(E * I / l)_{cols} = (E * I / l)_{\text{stack under consideration}} + \sum(E * I / l)_{\text{stacks beyond this joint}}$

These stacks can be part of the same column length or another column length.

Note that as the restrained length may be multiple stacks, "E * I" for this stack are the values for the stack being designed, and l is the restrained length. For

---

[1] BS EN 1992-1-1:2004 Section 5.8.3.2(3)

[2] PD 6687-1:2010 Section 2.11.2

the stacks beyond the restraint, "E * I" are the values for the stack attached to the restraint, and l is the restrained length that the stack exists within.

Any beams framing into the end of the stack within 45 degrees of the axis being considered are said to be restraining beams for the stack in that direction.

There is a lower limit [3] for the value of k:

$k \geq 0.1$

Additionally, Tekla Structural Designer imposes an upper limit:

$k \leq 20$

For bracing stacks, a warning is displayed when the calculated value of k exceeds this limit.

Fixed Column Base

k = 0.1 for fixed bases in Tekla Structural Designer. There is no clear guidance in EC2, but the Concrete Centre guidance suggests that this is suitable.

NOTE If you have set the bottom of the column to be "fixed" but the support as "pinned". The program will always assume that the support is fixed and therefore only ever consider the fixity applied to the column.

Pinned Column End

In any situation where the end of a column anywhere in the structure is pinned, k = 20.

No Effective Beams Found

If no effective beams are found to restrain the end of the stack in the direction in question, then the program will consider whether there is a flat slab restraining the stack at this end. If a flat slab is found it will either be considered as a restraint, or not, in each direction at each end of the stack - this is controlled by checking the option Use slab for stiffness calculation... located as a Stiffness setting in the column properties. If there are no effective beams and there is no flat slab (or any flat slab is not to be considered), then the program looks for the far end of the stack on the other side of the joint, and look at the restraints there, and so on until a restraint with an effective beam or flat slab to be considered is found.

If the stack is restrained by a flat slab, then the slab will be considered to act as a beam in this direction - note that it is one beam in the direction and NOT a beam on each side of the column.

If the stack is an end stack and there are no supports, beams or flat slabs considered to restrain the stack at this end in the direction, the end is therefore free in this direction and k = 20.

---

[3] BS EN 1992-1-1:2004 Section 5.8.3.2(3)

Slenderness ratio

For columns: the slenderness ratio, λ of the restrained length about each axis is calculated as follows:

$\lambda = l_0 / i = l_0 / \sqrt{(I / A)}$

Where

l_0 is the effective height of the stack,

i is the radius of gyration of the stack section about the axis under consideration,

I is the second moment of area of the stack section about the axis,

A is the cross-sectional area of the stack section.

The slenderness ratio λ is then checked against the limiting slenderness ratio λ_{lim} in each direction. If the slenderness is less than this limit, then the member is short and slenderness effects are ignored, otherwise it is slender.

For walls: since the wall panel has a rectangular plan shape, the above calculation can be simplified:

In-plane,

Slenderness, $l_y = l_{0,y} / i_y$

Where

Radius of gyration, $i_y = l_w/(12)^{0.5}$

$l_{0,y}$ is the effective length,

l_w is the length of wall panel

Out-of-plane,

Slenderness, $l_z = l_{0,z} / i_z$

Where

Radius of gyration, $i_z = h_w/(12)^{0.5}$

$l_{0,z}$ is the effective length

h_w is the thickness of wall panel

Limiting slenderness ratio

$\lambda_{lim} = 20 * A * B * C / \sqrt{n}$

Where

$A = 1 / (1 + (0.2 * \varphi_{ef})) \geq 0.7$

$B = \sqrt{(1 + (2 * \omega))} \geq 1.1$

$C = 1.7 - r_m$

Where

φ_{ef} is the effective creep ratio,

$\omega = A_s * f_{yd} / (A_c * f_{cd})$,

f_{yd} is the design yield strength of the reinforcement,

f_{cd} is the design compressive strength of the concrete,

A_s is the total area of longitudinal reinforcement,

$n = N_{Ed} / (A_c * f_{cd})$,

N_{Ed} is the design axial force between restrained floor levels in this direction,

$r_m = M_{1.1} / M_{2.1}$

$M_{1.1}$ and $M_{2.1}$ are the first order moments at the ends of the stack about the axis being considered, with $|M_{2.1}| \geq |M_{1.1}|$.

If $M_{1.1}$ and $M_{2.1}$ cause tension in the same side of the stack then r_m is positive and $C \leq 1.7$. If the converse is true then the stack is in double curvature, and it follows that r_m is negative and $C > 1.7$.

For braced stacks in which the first order moments arise only from transverse loads (lateral loading is significant) or imperfections ($M_{imp.1} > |M_{2.1}|$), C must be taken as 0.7,

For

bracing stacks, C must be taken as 0.7,

For restrained lengths encompassing more than one stack, C is taken as 0.7.

The effective creep ratio, φ_{ef}, is derived as follows:

$f_{cm} = f_{ck} + 8$ (N/mm^2)

$h_0 = 2 * A_g / u$

Where

u is the section perimeter in contact with the atmosphere (assumed to be the full section perimeter),

A_g is the gross section area.

$\alpha_1 = (35 / f_{cm})^{0.7}$

$\alpha_2 = (35 / f_{cm})^{0.2}$

$\alpha_3 = (35 / f_{cm})^{0.5}$

If $f_{cm} \leq 35$ N/mm^2,

$\beta_H = (1.5 * (1 + (1.2 * RH))^{18} * h_0) + 250 \leq 1500$

Else,

$\beta_H = (1.5 * (1 + (1.2 * RH))^{18} * h_0) + (250 * \alpha_3) \leq 1500 * \alpha_3$

Where

RH is the relative humidity, which is set under Design parameters in the column properties.

$\beta_c(t , t_0) = ((t - t_0) / (\beta_H + t - t_0))^{0.3}$

$\beta_{t0} = 1 / (0.1 + t_0^{\ 0.2})$

$\beta_{fcm} = 16.8 / \sqrt{f_{cm}}$

Where

t_0 is the age of column loading and defaults to 14 days, if required it can be changed under Design parameters in the column properties.

If $f_{cm} \leq 35$ N/mm^2,

$\varphi_{RH} = 1 + ((1 - (RH / 100)) / (0.1 * h_0^{\ 1/3}))$

Else,

$\varphi_{RH} = (1 + (((1 - (RH / 100)) / (0.1 * h_0^{\ 1/3})) * \alpha_1)) * \alpha_2$

Then,

$\varphi_0 = \varphi_{RH} * \beta_{fcm} * \beta_{t0}$

$\varphi(\infty , t_0) = \varphi_0 * \beta_c(\infty , t_0)$

If $\varphi(\infty , t_0) \leq 2$ and $\lambda < 75$ and $M_{max.1} / N_{Ed} \geq h$ and $\omega \geq 0.25$,

$\varphi_{ef} = 0$

Else

$\varphi_{ef} = \varphi(\infty, t_0) * R_{PL}$

Where

$M_{max.1}$ is the largest first order moment in the restrained length in this direction,

N_{Ed} is the design axial force in the restrained length in this direction,

R_{PL} is the permanent load ratio.

You are required to supply a value for the permanent load ratio which is located under Design parameters in the column properties. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.

Tekla Structural Designer assumes that t ∞ (t-infinity) is equal to 70 years (25550 days).

Pre-selection of Bracing Contribution

The significant parameter within the slenderness criteria is a choice of how a wall, or column, is contributing to the stability of the structure.

In-plane direction, a wall is usually considered to be a bracing member. Out-of-plane direction, a wall is usually considered to be braced by other stabilizing members. These are the default settings but can be edited.

Overview of second order effects (concrete column: EC2)

For 'isolated' columns and walls, EN1992-1-1 (EC2) allows for second order effects and member imperfections in a number of ways,

- It specifies a minimum level of member imperfection along with a conservative value - see Clause 5.2 (7).
- It provides for the additional moment due to slenderness (member buckling) using one of two methods. One method (the (Nominal) Stiffness Method) increases the first-order moments in the column using an amplifier based on the elastic critical buckling load of the member - see Clause 5.8.7.3. The second method (the (Nominal) Curvature Method) calculates the 'second-order' moment directly based on an adjustment to the maximum predicted curvature that the column section can achieve at failure in bending - see Clause 5.8.8.
- The impact of the slenderness is increased or decreased depending upon the effective length factor for the member. For braced members this will be ≤ 1.0 and for unbraced (bracing) members it will be ≥ 1.0 see Clause 5.8.3.2.

Finally, EC2 also requires consideration of a minimum moment based on the likelihood that the axial load cannot be fully concentric see Clause 6.1 (4).

Minimum moment (Clause 6.1 (4))

The minimum moment about each axis, M_{min} is calculated. When using the Curvature Method, M_2 is added to the minimum moment. When using the Stiffness Method M_2 is calculated from $M_{min} \times \pi^2/(8(\alpha_{cr} - 1))$ and added to M_{min}.

If for any design combination and design position the minimum moment including second-order moment is greater than the overall design moment then the former is used when comparing the values on the locus of moment of resistance. Note that the minimum might be governing about neither axis, one axis or both axes.

Member imperfections (Clause 5.2 (7))

The imperfection moment is calculated using the eccentricity, $e_i = l_0/400$, and it is conservatively assumed that it increases the first-order moments irrespective of sign. In the case of the Stiffness Method the imperfection moment is added before the moment magnifier is applied. It is applied to both braced and bracing columns/walls.

Curvature Method (Clause 5.8.8)

This method is only applied to symmetrical, rectangular and circular sections and is equally applicable to columns and walls. The second-order moment, M_2 (= N_{Ed} e_2), is calculated but the resulting design moment is only used if it is less than that calculated from the Stiffness Method. It is applied in the same manner as that for the Stiffness Method to both braced and bracing columns.

Stiffness Method (Clause 5.8.7)

This method is applied to all columns and walls.

For braced columns the second-order moment M_2 is calculated from:

M_2	=	$M_{e.1}$ x π^2 /(8 x (N_B /N_{Ed} - 1))
Where,		
$M_{e.1}$	=	the maximum first-order moment in the mid-fifth
N_B	=	the buckling load of the column based on nominal stiffness and the effective length
	=	π^2 EI/l_0^2
N_{Ed}	=	the maximum axial force in the design length

When a point of zero shear occurs inside the mid-fifth or does not exist in the member length, the value of M_2 is added algebraically to the first-order moments at the ends but only if this increases the first-order moment. At the mid-fifth position M_2 is always "added" in such a way as to increase the first-order mid-fifth moment.

When a point of zero shear occurs within the member length and is outside the mid-fifth, the second-order moments is taken as the greater of that calculated as above and that calculated as per Clause 5.8.7.3 (4) by multiplying all first-order moments by the amplifier,

1/(1 - N_{Ed}/N_B)

For bracing columns the second-order moments are calculated in the same way as braced columns except that in the determination of the amplifier, the buckling load is based on bracing effective lengths. These are greater than 1.0L and hence produce more severe amplifiers.

Second-order analysis

When second-order analysis is selected then both braced and bracing columns are treated the same as if first-order analysis were selected. If the second-order analysis is either the amplified forces method or the rigorous method then this approach will double count some of the global P-Δ effects in columns that are determined as having significant lateral loads. Also, when it is a rigorous second-order analysis there is some double counting of member P-δ effects in both braced and bracing columns.

Design moment calculations (column and wall:EC2)

For each combination and for each analysis model the end moments in the two local member directions, "1" and "2" are established. From these and the local load profile, the moment at any position and the maximum axial force in the member can be established.

Step 1 - the amplifier

Calculate the "amplifier" due to buckling in each of Direction 1 and Direction 2 from Equ. 5.28 and Equ. 5.30 of EC2 as[1],

$k_{5.28}$	=	$1 + \pi^2 /(8*(N_B /N_{Ed} - 1))$
$k_{5.30}$	=	$1 + 1/(N_B /N_{Ed} - 1)$

Where

N_B	=	the (Euler) buckling load in the appropriate direction
	=	$\pi^2 EI/l_o^2$
l_o	=	the effective length in the appropriate direction which for braced columns will be ≤ 1.0L and for unbraced columns ≥ 1.0L
N_{Ed}	=	the maximum axial compressive force in the column length under consideration (stack)

NOTE When N_{Ed} ≤ zero i.e. tension, $k_{5.28}$ and $k_{5.30}$ are 1.0.

Step 2 - minimum moment

Calculate the minimum moment due to non-concentric axial force in each of the two directions from,

$M_{min.1}$		$= \| N_{Ed} \| * MAX(h/30, 20)$

Where

h	=	the major dimension of the column in the appropriate direction
N_{Ed}	=	the maximum axial force (compression or tension) in the column length under consideration (stack)

Step 3 - imperfection moment

Calculate the "first-order" and "second-order" imperfection moment in Direction 1 and Direction 2 as,

[1] Direction 1 and Direction 2 are referring here to the member local axes

$M_{imp.1}$	=	$N_{Ed} * e_i$
$M_{imp.2}$	=	$M_{imp.1} * k_{5.28}$

Where

$M_{imp.1}$	=	the "first-order" imperfection moment in a given direction
$M_{imp.2}$	=	the "second-order" imperfection moment in a given direction
e_i	=	the effective length in the appropriate direction divided by 400
	=	l_o /400
N_{Ed}	=	the maximum axial compressive force in the column length under consideration (stack)

NOTE When N_{Ed} ≤ zero i.e. tension, $M_{imp.1}$ and $M_{imp.2}$ are zero.

Step 4 - second-order moment, curvature method

For rectangular and circular sections the second-order moment, $M_{2.curv}$, using the Curvature Method is calculated for each direction.

$M_{2.curv}$	=	$N_{Ed} * e_2$

Where

e_2	=	the deflection due to the maximum curvature achievable with the given axial force
	=	$(1/r)\ l_o^2 /c$
N_{Ed}	=	the maximum axial compressive force in the column length under consideration (stack)

NOTE When N_{Ed} ≤ zero i.e. tension, $M_{2.curv}$ is zero.

Step 5 - second-order moment, stiffness method

For all section shapes, the second-order moment, $M_{2.stiff}$, using the Stiffness Method is calculated in each direction based on the maximum first-order moment in the mid-fifth of the column, $M_{e.1}$, in the appropriate direction.

$M_{2.stiff}$	=	$M_{e.1} * (\pi^2 /(8*(N_B /N_{Ed} - 1))$

Where

$M_{e.1}$	=	the maximum absolute moment in the mid-fifth of the column length under consideration (stack) in the appropriate direction
N_{Ed}	=	the maximum axial compressive force in the column length under consideration (stack)

NOTE When $N_{Ed} \leq$ zero i.e. tension, $M_{2.stiff}$ is zero.

Step 6 - lateral loading classification

For the current design combination, for each direction using the member analysis routines, check for point(s) of zero shear within the column length. If none exist or are within the mid-fifth of the column length then this design case is designated as having lateral loads that are "not significant". Else the lateral loads are considered as "significant".

Step 7 - design moment at top

Calculate the design moment at the top of the column in each direction (for both braced and unbraced columns) taking into account if lateral loads are "significant", or "not significant".

Step 8 - design moment at bottom

Calculate the design moment at the bottom of the column in each direction (for both braced and unbraced columns) taking into account if lateral loads that are "significant", or "not significant".

Step 9 - design moment in mid-fifth

Calculate the design moment in the mid-fifth of the column in each direction (for both braced and unbraced columns) taking into account if lateral loads that are "significant", or "not significant".

Design for combined axial and bending (column and wall:EC2)

Tekla Structural Designer designs the column for an applied axial force and applied bending about one or both axes of the section. In the case of bi-axial bending, a resultant moment is created for the combination of the applied moments.

$$\sqrt{(M^2_{major} + M^2_{minor})} \,/\, \sqrt{(M^2_{major,res} + M^2_{minor,res})} \leq 1.0$$

Where

M_{major}	=	Moment about the major axis
M_{minor}	=	Moment about the minor axis
$M_{major,res}$	=	Moment of resistance about the major axis
$M_{minor,res}$	=	Moment of resistance about the minor axis

Tekla Structural Designer adopts the above approach in preference to the simplified method specified in equation 5.39 of EC2 as it has a wider range of application.

Design parameters for shear (column and wall:EC2)

For some of the shear design parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Column > Reinforcement Layout.

Minimum Shear Link Diameter

$\varphi_{w,min}$ = MAX[6mm, 0.25 * largest longitudinal bar diameter]

Maximum Span Region Shear Link Spacing

For design to **UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [1]

$\varphi_{w,max}$ = MIN[20 * smallest longitudinal bar diameter, lesser column dimension, 400mm]

NOTE For UK NA when concrete class > C50/60 there are separate calculations in clause 9.5.3(3). These are not implemented but a warning is displayed in this situation.

For design to **EC2 Recommendations:**

$\varphi_{w,max}$ = MIN[20 * smallest longitudinal bar diameter, lesser column dimension, 400mm]

Support Region Shear Link Spacing

For support regions, the maximum link spacing is reduced by 40%. [2]

---

[1] BS EN 1992-1-1:2004 Section 9.5.3(3)

[2] Maximum BS EN 1992-1-1:2004 Section 9.5.3(4)

Long Term Compressive Strength Factor for Shear, α_{ccw}

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [3]

$\alpha_{ccw} = 1.0$

Design Concrete Compressive Strength for Shear, f_{cwd}

For design to **UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**:

$f_{cwd} = \alpha_{ccw} * \text{MIN}(f_{ck}, 50) / \gamma_C$

For design to **EC2 Recommendations:**

$f_{cwd} = \alpha_{ccw} * f_{ck} / \gamma_C$

Factor $C_{Rd,c}$ [4]

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**:

$C_{Rd,c} = 0.18 / \gamma_C$

Factor k_1 [5]

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**:

$k_1 = 0.15$

Cracked Concrete Reduction Factor, v

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [6]

$v = 0.6 * (1 - (f_{ck} / 250))$

Cracked Concrete Reduction Factor, v_1

For design to **UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [7]

$v_1 = v * (1 - (0.5 * \cos(\alpha))$

α is the inclination of links.

Note that links in columns are always assumed to be at 90° to column direction.

Therefore $v_1 = v$

---

[3] BS EN 1992-1-1:2004 Section 3.1.6(1)

[4] BS EN 1992-1-1:2004 Section 6.2.2(1)

[5] BS EN 1992-1-1:2004 Section 6.2.2(1

[6] BS EN 1992-1-1:2004 Section 6.2.2(6)

[7] BS EN 1992-1-1:2004 Section 6.2.3(3)

For design to **EC2 Recommendations**:

$v_1 = v$

Minimum Shear Reinforcement Ratio, $\rho_{w,min}$

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [8]

$0.08 * \sqrt{(f_{ck})} / f_{yk}$

Maximum Angle of Compression Strut, θ_{max}

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [9]

$\cot(\theta) = 1$

$\theta = 45°$

Minimum Angle of Compression Strut, θ_{min}

For design to[10] **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**:

$\cot(\theta) = 2.5$

$\theta = 21.8°$

Angle of Compression Strut, θ

For design to **UK NA, Irish NA, Malaysian NA** and **Singapore NA**: [10]

If $N_{Ed} \geq 0$ i.e. compression

$\theta = 0.5 * \arcsin(2 * v_{Ed} / (0.9 * \alpha_{cw} * v_1 * f_{cwd}))$

else

$\cot(\theta) = 1.25$

$\theta = 38.7°$

For design to**EC2 Recommendations**:

$\theta = 0.5 * \arcsin(2 * v_{Ed} / (0.9 * \alpha_{cw} * v_1 * f_{cwd}))$

Stress State Factor, α_{cw}

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**:

If $\sigma_{cp} \leq 0$

$\alpha_{cw} = 1.0$

---

[8] BS EN 1992-1-1:2004 Section 9.2.2(5)

[9] BS EN 1992-1-1:2004 Section 6.2.3(2)

[10] clause 6.2.3(2)

else if $0 < \sigma_{cp} \leq 0.25 * f_{cd}$

$\alpha_{cw} = 1.0 + (\sigma_{cp} / f_{cd})$

else if $0.25 * f_{cd} < \sigma_{cp} \leq 0.5 * f_{cd}$

$\alpha_{cw} = 1.25$

else if $0.5 * f_{cd} < \sigma_{cp} \leq f_{cd}$

$\alpha_{cw} = 2.5 * (1.0 - (\sigma_{cp} / f_{cd}))$

Minimum Shear Strength , v_{min}

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**: [11]

$035 * k^{1.5} * f_{ck}^{\ 0.5}$

Concrete wall design to EC2 (Eurocode)

The topics in this section describe how the software applies BS EN 1992-1-1:2004 (Ref. 1) (page 1989) to the design of reinforced concrete walls.

Limitations (Concrete wall: EC2)

The requirements of clause 9.6 are applied to all walls, irrespective of their length to thickness ratio. (Isolated compression members with a length to thickness ratio less than 4 should be defined as columns rather than walls.)

The following general exclusions also apply:

- Seismic design,
- Consideration of fire resistance. [You are however given full control of the minimum cover dimension to the reinforcement and are therefore able to take due account of fire resistance requirements.],
- Multi-stack reinforcement lifts.

Cover to Reinforcement (Concrete wall: EC2)

For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.

For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/confinement transverse reinforcement that may be present.

---

[11] clause 6.2.2(1)

You are required to set a minimum value for the nominal cover, $c_{nom,u}$, for each wall in the wall properties.

These values are then checked against the nominal limiting cover, $c_{nom,lim}$ which depends on the diameter of the reinforcement plus an allowance for deviation, Δc_{dev} (specified in Design Options > Wall > General Parameters).

Generally, the allowance for deviation, Δc_{dev} is a NDP . The recommended value is 10mm, but under strict controls it can be reduced to 5mm.

If $c_{nom,u} < c_{nom,lim}$ then a warning is displayed in the calculations.

Design Parameters for Vertical Bars (Concrete wall: EC2)

F or some of the vertical bar parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified inDesign Options > Wall > Reinforcement Layout.

NOTE In the following, the concrete area is the gross area of the general wall, or the gross area of the mid zone if one exists. For the end zone the design criteria for a reinforced concrete column element applies.

Minimum Vertical Bar Diameter

For design in accordance with **EC2 Recommendations**;

$\varphi_{v,min}$ = 8mm

For design in accordance with **UK NA, Irish NA, Malaysian NA** and **Singapore NA**;

$\varphi_{v,min}$ = 12mm

Minimum Vertical Bar Spacing

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$s_{cl,min} \geq$ MAX[maximum longitudinal bar diameter, 20mm, d_g + 5mm]

Where d_g is the maximum aggregate size.

Maximum Vertical Bar Spacing

You are given control over this value by specifying an upper limit in Design Options > Wall > Reinforcement Layout.

Minimum Reinforcement Area

Total minimum area of vertical reinforcement, $A_{s,\ min} = \rho_{v,min} * A_{cg}$

BS EN 1992-1-1:2004 cl 4.4.1.3 (1)P

BS EN 1992-1-1:2004 Section 8.2

Where

A_{cg} = gross area of the concrete wall

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$\rho_{v, min}$ = 0.002

Where 2 layers are specified distributed equally to each face, this is a minimum of 0.001*A_{cg} placed at each face.

You are given control over the minimum reinforcement ratio value via a user limit in Design Options > Wall > Reinforcement Layout (default 0.004).

For walls subjected to "predominantly out-of-plane bending", the minimum area rules for "slabs" apply if they are more critical than the above, [cl 9.3 and reference to cl 9.2.1.1 (1) (2) and (3)], so an additional check for any value of minor axis bending is applied.

$A_{s,min}$ = max [(2*0.26*f_{ctm}*l_{wp}*d/f_{yk}) , (2*0.0013*l_{wp}*d) , ($\rho_{v,min}$*A_{cg})]

This applies for the general wall length, or the mid zone if it exists.

For a general wall panel length, l_{wp} = l_w

Gross area of the wall, A_{cg} = l_w*h_w

For a mid zone panel length, l_{wp} = l_{mz}

Gross area of the mid zone, $A_{cg,mz}$ = l_{mz}*h_w

Effective depth of the cross section, d, is the dimension of the extreme concrete compression fibre to the centroid of reinforcement layer on the tension side, which for a wall is the line of the vertical reinforcement.

It does not apply for the end zones, since these are subject to the minimum reinforcement requirements as a column section.

Gross area of each end zone, $A_{cg,ez}$ = l_{ez}*h_w

Length of each end zone, l_{ez}

Design Parameters for Horizontal Bars (Concrete wall: EC2)

For some of the horizontal bar parameters, additional user defined limits can be applied - where this is the case minimum and maximum values are specified in Design Options > Wall > Reinforcement Layout.

Minimum Horizontal Bar Diameter

The suggested minimum for design in accordance with **EC2 Recommendations, UK NA, Irish NA, Malaysian NA** and **Singapore NA**;

$\varphi_{h,min}$ = 8mm

BS EN 1992-1-1:2004 Section 9.5.2(2)

Minimum Horizontal Bar Spacing

For design to **EC2 Recommendations, UK NA, Irish NA**, **Malaysian NA** and **Singapore NA**;

$s_{cl,min} \geq$ MAX[maximum longitudinal bar diameter, 20mm, d_g + 5mm]

Where d_g is the maximum aggregate size.

Maximum Horizontal Bar Spacing

To satisfy the slab condition if "predominantly out-of-plane bending";

Limiting maximum horizontal spacing, $s_{cr,max}$= min (3*h_w , 400 mm)

You are given control over this value by specifying a user limit in Design Options > Wall > Reinforcement Layout.

Concrete slab design to EC2 (Eurocode)

The topics in this section describe how the software applies BS EN 1992-1-1:2004 (Ref. 1) (page 1989) to the design of reinforced concrete slabs.

Pad and strip base design to EC2 (Eurocode)

The topics in this section describe how the software applies BS EN 1992-1-1:2004 (Ref. 1) (page 1989) to the design of pad and strip bases.

Checks performed (pad and strip base:EC2)

The checks performed for both directions are:

- EC7 - Max soil bearing pressure must not exceed allowable bearing pressure.
- EC2 - Provided steel must be greater than As,min for both vertical directions.
- EC2 - Provided bar spacing must be inside the limiting spacing
- EC2 - Provided bar size must be inside the limiting sizes
- EC2 - Check for bending moment capacity
- EC2 - Check for shear capacity
- EC2 - Punching check at column face
- EC2 - Punching check at critical perimeter
- EC7 - Check for overturning forces - not in the current release
- EC7 - Check for sliding

BS EN 1992-1-1:2004 Section 8.2

- EC7 - Check for uplift

NOTE EC7 - Check for overturning forces are beyond scope of the current release.

Foundation Bearing Capacity (pad and strip base:EC2)

Annex A of EC7 allows bearing capacity to be checked using two sets of partial factors: A1 and A2.

In Tekla Structural Designer the bearing capacity check is performed on STR load combinations using set A1 and on GEO load combinations using set A2.

Alternatively, an option is also provided to check a **Presumed Bearing Resistance** in accordance with EN1997-1cl.6.5.2.4).

Check for Pad Base Bearing Capacity

Total vertical force:

F_{dz}	=	$\gamma_G *(F_{swt} + F_{soil} + F_{sur,G}) + \gamma_Q * F_{sur,Q} - F_{z,sup}$

Moment about X axis:

$M_{x,c}$	=	$M_{x,sup} + F_{z,sup} * y_1 + h*F_{y,sup}$

Moment about Y axis:

$M_{y,c}$	=	$M_{y,sup} + F_{z,sup} * x_1 + h*F_{x,sup}$

Where:

L_x		Length of foundation in X-direction =	
L_y		Length of foundation in Y-direction =	
A		$L_x * L_y$ = Foundation area =	

h		Depth of foundation =	
h_{soil}		Depth of soil above the foundation =	
l_x		Length of column/wall in X-direction =	
l_y		Length of column/wall in Y-direction =	
x_1		Offset in X-axis. (Distance between centre of the pad to the centre of the support in X-direction) =	
y_1		Offset in Y-axis. (Distance between centre of the pad to the centre of the support in Y-direction) =	
γ_G		1.35 = Permanent partial factor - unfavourable action =	when Set A1 used
		1.0 = Permanent partial factor - unfavourable action =	when Set A2 used
γ_Q		1.5 = Variable partial factor - unfavourable action =	when Set A1 used
		1.3 = Variable partial factor - unfavourable action =	when Set A2 used
F_{swt}		$A * h * \gamma_{conc}$ = foundation self-weight	

		=	
F_{soil}		$(A - A_c)*h_{soil}*\gamma_{soil}$ = Unfactored load from soil =	
γ_{soil}		Density of soil - user input =	
$F_{sur,G}$		$(A - A_c)*sc_G$ = Unfactored load from surcharge for permanent load case =	
$F_{sur,Q}$		$(A - A_c)*sc_Q$ = Unfactored load from surcharge for variable load case =	
sc_G		Surcharge in permanent load case - user input =	
sc_Q		Surcharge in variable load case - user input =	
A_c		cross section of the column/wall =	
$F_{z,sup}$		Vertical load acting on support in STR/GEO limit states- (from analysis) =	
$M_{x,sup}$		Moment acting on support around X-axis in STR/GEO limit states- from analysis =	
$M_{y,sup}$		Moment acting on support around Y-axis in STR/GEO limit states - from analysis =	

$F_{x,sup}$		Horizontal force acting on support X-direction in STR/GEO limit_states - from analysis	
$F_{y,sup}$		Horizontal force acting on support Y-direction in STR/GEO limit_states - from analysis	

Eccentricity in X-direction:

e_x	=	$M_{y,c}$ / F_{dz}

Eccentricity in Y-direction:

e_y	=	$M_{x,c}$ / F_{dz}

Uniform rectangular stress distribution method

Effective length in X-direction:

L'_x		$L_x - 2e_x$ =	when $e_x > 0$
L'_x		$L_x + 2e_x$ =	when $e_x < 0$

Effective length in Y-direction:

L'_y		$L_y - 2e_y$ =	when $e_y > 0$
L'_y		$L_y + 2e_y$ =	when $e_y < 0$

Design bearing pressure:

f_{dz}		F_{dz} / (L'_x * L'_y) =

Presumed bearing capacity method

If

abs(ex) / Lx + abs(ey) / Ly		1.167 ≤

Then Base reaction acts within middle third - no loss of contact and:

Pad base pressures:

q_1		$F_{dz}/A - 6*M_{y,c}/(L_x*A) + 6*M_{x,c}/(L_y*A)$ =
q_2		$F_{dz}/A - 6*M_{y,c}/(L_x*A) - 6*M_{x,c}/(L_y*A)$ =
q_3		$F_{dz}/A + 6*M_{y,c}/(L_x*A) + 6*M_{x,c}/(L_y*A)$ =
q_4		$F_{dz}/A + 6*M_{y,c}/(L_x*A) - 6*M_{x,c}/(L_y*A)$ =

Max base pressure:

q_{max}		$\max(q_1, q_2, q_3, q_4)$ =

Else base reaction acts outside middle third - loss of contact.

In this case the pressure calculations are more complex - in Tekla Structural Designer these are done using sets of equations presented in an article by Kenneth E. Wilson published in the Journal of Bridge Engineering in 1997.

NOTE **Seismic combinations:** The presumed bearing capacity method uses SLS combinations in the bearing checks - however as there is no clear Eurocode guidance on service factors for seismic combinations, in Tekla Structural Designer they are not currently assigned. If using the presumed bearing capacity method, to avoid the check being performed for zero loading you are advised to consider which service factors might be appropriate and update the seismic combinations manually.

Check for Strip Base Bearing Capacity

The principles used in the strip base bearing capacity calculations are similar to those for pad foundations. Only the direction X is checked (around Y-axis) using segment widths.

Design bearing pressure:

f_{dz}		$F_{dz} / (L'_x * L_y)$ =

Design for bending (pad and strip base:EC2)

Bending design calculations are performed for the STR load combinations.

For tension on the bottom face of the foundation, the design bending moment may be taken as that at the face of the column or wall and may therefore be less than the peak bending moment.

The bending capacity check follows the same basic principle as used for beams, see: Design for bending for rectangular sections (beams and slabs: EC2) (page 1945).

Design for shear (pad and strip base:EC2)

Pad base shear design check

Calculate tension reinforcement ratio (cl 6.2.2(1)):

ρ_l	=	$\min(A_{sl} / (L*d), 0.02)$

where

A_{sl}	=	area of tension reinforcement
L	=	unit width of foundation in which A_{sl} is provided
d	=	effective depth of reinforcement in direction considered

$F_{y,sup}$	=	Horizontal force acting on support Y-direction in STR/GEO limit states - from analysis

Calculate k (cl. 6.2.2(1)):

k	=	$\min(1 + (200mm/d)^{1/2}, 2.0)$

Calc min shear strength (NDP) (cl. 6.2.2(1)):

v_{min}	=	$1.035k^{3/2}f_{ck}{}^{1/2}$	EC2 recommendations
v_{min}	=	$1.035k^{3/2}f_{ck}{}^{1/2}$	for UK, Irish, Malaysian and Singapore NA

Calculate resistance without shear reinforcement for X and Y directions (cl. 6.2.2(1)):

$v_{Rd.c,x}$	=	$\max(C_{Rd.c} * k * (100\ N^{2}/mm^{4} * \rho_{l} * f_{ck})^{1/3}, v_{min})$
$v_{Rd.c,y}$	=	$\max(C_{Rd.c} * k * (100\ N^{2}/mm^{4} * \rho_{l} * f_{ck})^{1/3}, v_{min})$

Maximum allowable shear resistance (cl. 6.2.2(6));

$v_{Rd,max}$	=	$1.5 * v * f_{cd}$

where

f_{cd}	=	concrete design compressive strength
v	=	cracked concrete reduction factor

If applied design shear force is less than or equal to the shear resistance i.e $v_{Ed} \leq v_{Rd} = \min(v_{Rd,c}, v_{Rd,max})$ the foundation thickness is adequate for beam shear.

Strip base shear design check

The principle of the strip base shear design check is similar to that for the pad base. Only the direction X is checked (around Y-axis) using segment widths.

Check for sliding (pad and strip base:EC2)

The check for sliding is carried out for pad foundations only.

If there is no horizontal force acting on foundation check for sliding is not required.

Sliding resistance (EC7 Section 6.5.3) - forces are defined from STR combinations.

Horizontal Forces on foundation for each direction:

F_{dx}	=	$F_{x,sup}$
F_{dy}	=	$F_{y,sup}$

where

$F_{x,sup}$	=	factored horizontal force acting on support in X-dir. (from analysis)
$F_{y,sup}$	=	factored horizontal force acting on support in Y-dir. (from analysis)

Horizontal force on foundation:

H_d	=	$[abs(F_{dx})^2 + abs(F_{dy})^2]^{0.5}$

Sliding resistance verification (Section 6.5.3)

Sliding resistance (exp.6.3b and table A.5):

$R_{H.d}$	=	$[F_{zG,d} + \gamma_{Gf} * F_{swt}] * \tan(\delta k) / \gamma_{R.h}$

where

δ_k	=	factored horizontal force acting on support in X-dir.	
$\gamma_{R.h}$	=	1.1 (set R2)	EC7 recomm. and for Irish and Malaysian NA
$\gamma_{R.h}$	=	1.0 (set R1)	for UK, Singapore NA
γ_{Gf}	=	1.0 (permanent favourable action)	
$F_{zG,d}$	=	Vertical load acting on support in STR/GEO limit states where favourable actions considered.	

Check for uplift (pad and strip base:EC2)

For combinations producing tension at the support the tension value is compared to the stabilizing loads. Auto-design can automatically increment the base size to achieve a passing status.

Pile cap design to EC2 (Eurocode)

The forces acting on a pile cap are applied to the foundation at the foundation level. The foundation can take axial load and bi-axial shear and moment.

Pile cap design is divided between pile design (pile capacity check) and structural design of the pile cap which includes bending, shear and punching shear design checks.

Pile cap calculations are performed in accordance with BS EN 1992-1-1:2004 (Ref. 1) (page 1989).

Bottom reinforcement is designed to the Eurocode - (EC base, UK NA, Irish NA, Singapore NA or Malaysia NA).

Pile capacity (pile cap:EC2)

Annex A of EC7 allows bearing capacity to be checked using two sets of partial factors: A1 and A2.

In Tekla Structural Designer the bearing capacity check is performed on STR load combinations using set A1 and on GEO load combinations using set A2.

Pile capacity passes if:

$R_{c,d}$	≥	$P_n \geq - R_{t,d}$
Where:		
$R_{c,d}$	=	Pile design compression resistance
$R_{t,d}$	=	Pile design tension resistance
P_n	=	Pile load

Design for bending (pile cap:EC2)

The pile cap is treated as a beam in bending, where the critical bending moments for the design for the bottom reinforcement are taken at the face of the column.

Bending design calculations are performed for the STR load combinations.

The bending capacity check follows the same basic principle as used for beams, see: Design for bending for rectangular sections (beams and slabs: EC2) (page 1945).

Design for shear (pile cap:EC2)

Shear design calculations are performed for the STR load combinations.

Determination of Design Shear Stress

Shear stress acting on side 1 in direction X	$v_{Ed,x1}$	$= \Sigma P_{n,1} / (d_x * L_y)$

Shear stress acting on side 2 in direction X	$v_{Ed,x2}$	$\Sigma P_{n,2} / (d_x * L_y)$
Shear stress acting on side 1 in direction Y	$v_{Ed,y1}$	$\Sigma P_{n,1} / (d_y * L_x)$
Shear stress acting on side 2 in direction Y	$v_{Ed,y2}$	$\Sigma P_{n,2} / (d_y * L_x)$

Maximum allowable shear resistance

$v_{Rd,max} = 0.5 * \nu * f_{cd}$ [1]

where:

$f_{cd} = \alpha_{cc} * f_{ck} / \gamma_c$

$\alpha_{cc} = 1.0$ (EC2); $\alpha_{cc} = 0.85$ (supported NAs)

$\gamma_c = 1.5$

$v = 0.6 * [1 - (f_{ck}/250)]$

Check for Shear

The shear capacity check procedure is identical to that for pad bases, see:
Design for shear (pad and strip base:EC2) (page 1984)

Checks for limiting parameters (pile cap:EC2)

Check for distance of pile cap overhang

Check pile edge distance "e" for pile "i" in a pile group for both directions:

The check passes if:

If min $e_i > e_{min,user}$

Check for minimum pile spacing

Check centre to centre spacing "s" between piles "i" and "j" in a pile group:

The check passes if:

If $s_{ij} > s_{min,user}$

where

$s_{min,user}$ = user input

Check for maximum pile spacing

Check centre to centre maximum spacing "s" between piles "i" and "j" in a pile group:

The check passes if:

If $s_{ij} < s_{max,user}$

$s_{max,user}$ = user input

[1] BS EN 1992-1-1:2004 Section 6.2.2(6)

Other checks

The remaining checks are identical to those for pad bases, see: Design parameters for longitudinal bars (EC2) (page 1938)

References EC2

1. British Standards Institution. BS EN 1992-1-1:2004. Eurocode 2: Design of concrete structures. General rules and rules for buildings. BSI 2004.
2. British Standards Institution. NA to BS EN 1992-1-1:2004. Eurocode 2: Design of concrete structures. General rules and rules for buildings. BSI 2005.

Vibration of floors to SCI P354

These topics describe the SCI P354 floor vibration calculations that can be performed in Tekla Structural Designer.

The following topics are covered:

Introduction to floor vibration (P354)

This handbook describes the SCI P354 floor vibration calculations that can be performed in Tekla Structural Designer.

With the advent of long span floors, multiple openings in webs, minimum floor depth zones etc. the vibration response of floors in multi-storey buildings under normal occupancy has increasingly become of concern to clients and their Engineers and Architects.

Detailed guidance on the subject is available through the SCI Publication P354 Design of Floors for Vibration: A New Approach (page 2006)

This handbook describes the method for the assessment of floor vibration in accordance with P354 that has been adopted in Tekla Structural Designer. The method seeks to establish, with reasonable accuracy, the response of the floor to dynamic excitation expected in offices of normal occupancy. This excitation is almost solely based on occupants walking. With appropriate design criteria, the approach is likely to be equally applicable to sectors other than offices.

The existing solution to checking this type of criterion - a simple calculation of the natural frequency of an individual beam - is felt in many cases to be insufficiently accurate. More importantly, such calculations do not consider two important factors,

- the natural frequency is only the 'response side' of the equation. The 'action' side of the equation is also important i.e. the dynamic excitation - this is the activity that might cause an adverse response from the floor. Walking, dancing and machine vibration are all on the 'action' side of the equation and are all very different in their potential effect.
- the natural frequency of an isolated beam is exactly that and takes no account of the influence (good or bad) of the surrounding floor components. In particular, with composite floors, the slabs will force other beams to restrict or sympathize with the beam under consideration.

The culmination of the calculations carried out by Tekla Structural Designer is a "Response Factor". It is important to note that this response factor,

- is not a truly real value of the response of the actual floor since the complex nature of real building layouts are idealized into standard 'cases'.
- is compared with certain limits that have been recommended by industry experts for a limited classification of building type. They are not arbitrary but are not absolute either (cf. calculated deflection and deflection limits)
- is relatively insensitive. That is, a twofold change in the response factor will only just be perceptible to the occupants (cf. logarithmic scale of sound power levels, dbA).
- could be over-conservative particularly for those buildings where tight requirements are imposed.

Notwithstanding the above, this approach is another tool at your disposal that could enable you to spot a problem before the floor is built and prevent the first steps of the client into his new building proving a disaster!

You should find that the check is simple to operate, but it will require you to make choices that may be unfamiliar to you. The purpose of this handbook is to assist you in becoming familiar with the requirements of the check and to assist you in making reasonable judgments regarding the input required.

Scope of floor vibration (P354)

The reference upon which Tekla Structural Designer's floor vibration check is based is the main limiting factor with regard to scope. This is SCI Publication P354 (page 2006) There are no doubt many other texts that deal with vibration problems in buildings, and indeed there is a British Standard dealing with the evaluation of human exposure to vibration in buildings, BS 6472: 1992 (page 2006) . However this SCI publication has distilled this wider knowledge into readily usable design guidance that is specifically aimed at floors in multi-storey buildings of normal occupancy.

You are able to define an area on a particular floor level that is to be subject to the vibration response analysis and design. The layout of beams in real multi-storey buildings can be of almost any configuration. **The methodology adopted in P354 is only applicable to regular structures which by and large have to be created from rectilinear grids.** It is your responsibility to make an appropriate selection of the beams etc. that are to be the basic components of the idealized case.

As you proceed through the input making your selections, Tekla Structural Designer will, where it is possible to do so, interrogate the underlying model and retrieve the appropriate data. Once all the data has been assembled, you are then able to perform the check, after which a detailed set of results will be available for review. If you are unhappy with the outcome of your choices you can close the results window and make alternative selections by editing the Floor Vibration Check item properties.

Limitations and Assumptions of floor vibration to P354

The scope is primarily defined by the reference design document (page 2006) but the following additional limitations and assumptions should be noted.

- The design guidance is based on composite floors acting compositely with the steel beams. It is unclear whether the design approach is directly applicable to non-composite construction.
- For simplicity and to avoid the necessity of Tekla Structural Designer having to identify all the beams in the area selected for vibration assessment, the component of the unit mass from the self-weight of the beams is ignored. This will lead to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous). Note, however, that beam self-weight is included in the calculation of beam deflection but only when the self-weight loadcase is included in the load combination.
- Cantilever beams are excluded from the analysis.
- Cold formed sections are excluded.
- Precast slabs are excluded.

Design philosophy of P354 floor vibration

General

The Engineer ensures the safety of building occupants by satisfying all design criteria at the Ultimate Limit State. Similarly, the health of building occupants is partly taken care of when deflection limits at the Serviceability Limit State are satisfied (although this Limit State does have other purposes than simply the health of occupants).

However, for floors that are subject to cyclic or sudden loading, it is the human perception of motion that could cause the performance of a floor to be found unsatisfactory. Such perception is usually related to acceleration levels. In most practical building structures, the reaction of the occupants to floor acceleration varies between irritation and a feeling of insecurity. This is based on the instinctive human perception that motion in a 'solid' building indicates inadequacy or imminent failure.

The working environment also affects the perception of motion. For busy environments, where the occupant is surrounded by the activity that is producing the vibrations, the perception of motion is reduced. In contrast, for quieter environments (such as laboratories and residential dwellings), where the source of vibration is unseen, the perception of motion is significantly heightened.

The design philosophy to ensure that the potential for such human response is minimized, has a number of facets,

- the **dynamic excitation** causing the vibration i.e. the disturbing force profile, which is force and time dependent. For the sorts of building and occupancy considered here, this is the act of walking.
- the **required performance**. This depends upon the type of environment. As discussed above this, in turn, depends upon the involvement of the occupant in the generation of the vibration and also on the nature of the occupancy. The latter is important for laboratories carrying out delicate work, or operating theatres, for example.
- the **provided performance**. This is the "Response Factor" and is dependent on the system natural frequency and, more importantly, the participating mass. The latter is driven mainly by the selection of an area of floor that is reasonable and appropriate.

Dynamic excitation

In a classical spring-mass system that includes a (viscous) damper, when a simple force is applied to the mass to extend (or contract) the spring, the mass moves up and down (oscillates). This movement is significant at first but eventually reduces to zero due to the resistance offered by the damper. In a floor system in a building,

- the mass is the self-weight of the floor and any other loading that is present for the majority of the time that the occupants could be exposed to vibration effects,
- the spring is the stiffness of the floor system, which will have a number of different component beams (secondary and primary) and the floor slab,
- the damper is provided by a number of elements that are able to absorb energy from the free vibration of the system. There will be energy absorbed,

- within connections, since they behave 'better' than the ideal that is assumed
- from losses due to the unsymmetrical nature of real buildings e.g. grid layout, and dispersion of loads from furnishings and contents
- from components such as partitions that are out-of-plane of the vibration and interfere with the 'mode'.

The determination of the contribution of each of these components as they affect real floor systems is given in detail in later sections. These describe the 'response' side of the floor system. In order to establish the required performance of the system the 'input' must also be defined i.e. that event, events or continuum that is the 'dynamic excitation'.

In the simple example described at the start of this section the 'input' was simply a force that caused a displacement to the system and was then released. This might be equivalent to a person jumping off a chair onto the floor. However, in the context of the concerns over the vibration of floors, it is not this sort of input that is of interest. The main concern is the excitation of the floor brought about by walking.

Unlike the simple example, walking produces loading that is cyclic. This loading can be idealized into a series of sine curves of load against time. Each curve is an exact multiple of the walking frequency called harmonics. When one of these harmonics of the cyclic loading coincides with the natural frequency of the floor system then resonance is set up. The consequence of resonance that is detected, and may disturb occupants, is the associated peak acceleration. For the first harmonic, the peak acceleration is dependent upon the applied force (the weight of one standard person multiplied by a factor, α_n), the mass of the system (the self-weight of the floor plate plus other loading that could be considered as permanent), and the amount of damping in the system (the damping ratio, ζ). The factor, α_n, is known as a Fourier coefficient and links the magnitude of the applied force in any harmonic of the walking function to the weight of one standard person. It has been established experimentally for different activities and different activity frequencies.

Hence, the dynamic excitation of a floor is dependent upon the forcing function due to walking and its relationship to the natural frequency of the floor system. It is the level of the peak acceleration that this generates that is particularly important in determining the performance of the floor.

Required performance

The required performance of a floor system is very dependent upon the potential response of humans. Human response is a very complex subject since there is no such thing as a 'standard human'. The perception of vibration will differ from person to person, their body mass varies significantly and the body's reaction will depend upon age, gender etc. The human response has been studied and the accepted wisdom is embodied in BS 6472: 1992, Guide to evaluation of human exposure to vibration in buildings (1 Hz to 80 Hz) (page 2006)

It may be remembered that it is the acceleration of the floor system that the human perceives. BS 6472: 1992 provides a series of curves one of which is the 'base' limit of (vertical) acceleration against frequency (of the floor). Within the practical range of frequencies dealt with, a single value of the 'base' limit on acceleration is given as 0.005 m/s^2. This single value holds

- down to 3 Hz but no floor should be allowed to have a system natural frequency below this value anyway
- up to 10 Hz. Such a large value would be unusual but beyond that point there is a simple linear relationship between the base limit of acceleration and the natural frequency within an extended but just practical range.

The accelerations acceptable for different use of buildings are described using the 'base' limits. Multiplying factors are used to increase the base acceleration limit according to the intended use of the building. The multiplying factors are referred to as 'response factors' in the SCI guidance. Thus the target acceleration of the floor under consideration is the root mean square acceleration multiplied by the response factor. This design condition is turned on its head to give a 'provided response factor' that is then compared with the 'required response factor'. The required response factor is the measure of the "Required performance" and is given in the SCI guidance as,

- R = 8 for a workshop
- R = 8 for a general office
- R = 2 for a residential building during day time use

You should choose a required response factor based on both engineering judgement and the advice given in P354. In particular it may be noted that, "changing R by a factor of 2 is equivalent only to the most marginal change to human perception".

Provided performance

It is in establishing the provided performance that most of the design calculations are required. The object of these calculations is to determine the 'required response factor'.

The start point is the calculation of the natural frequency of the floor system. This is established from the individual component frequencies for each of two possible shape modes, namely the Secondary Beam Mode and the Primary Beam Mode. The natural frequencies of the individual components can be adjusted to allow for boundary conditions e.g. two spans continuous. The fundamental frequency, f_0, is the lower value for the two modes considered. A minimum natural frequency is given in SCI P354 of 3.0 Hz.

Next the 'modal mass' is required. This is dependent upon the physical size of the floor plate selected and an effective width and/or length that is itself dependent on the natural frequency of the floor. The modal mass has by far the largest influence on the response factor provided.

The 'Resonance Build-up Factor' makes allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. This has an upper limit of 1.0 and can be taken conservatively as 1.0. The calculation requires the 'damping ratio' - this is a user input.

The resonance build-up factor, the damping ratio, the modal mass, and the weight of a 'standard person' along with an appropriate Fourier coefficient are used to calculate the peak acceleration.

The final determination of the response factor provided requires the 'root mean square' acceleration. The rms acceleration has two formulations depending upon the fundamental, system frequency. The response factor is a very simple calculation.

The design condition is simply,

$R_{prov} \leq R_{reqd}$

Provided performance P354 floor vibration

System frequency

Deflections

For the primary beam, the base maximum simply supported deflection, δ_{PBSS}, is derived from the analysis model with no allowance for boundary conditions.

For the secondary beam, the base maximum simply supported deflection, δ_{SBSS}, is derived from the analysis model and the maximum deflection for a fixed end condition, δ_{SBFE}, is calculated from,

$$\delta_{SBFE} = m*b*L_{SB}^4 /(384*E_S *I_{SB}) + m*b*L_{SB}^{\ 2} /(24*G*A_y)$$

Where		
m	=	unit mass in kN/mm^2
b	=	secondary beam spacing in mm
L_{SB}	=	span of the secondary beam in mm
I_{SB}	=	the inertia of the secondary beam in mm 4
E_S	=	the steel modulus in kN/mm^2
G	=	the steel shear modulus in kN/mm^2
A_y	=	the major axis shear area in mm^2

For the slab, the base maximum deflection for a fixed end condition, δ_{SlabFE}, is calculated from,

δ_{SlabFE} = $m*L_{Slab}$ 4/(384*E_C *I_{Slab})

Where

m = unit mass in kN/mm ^{2}

L_{Slab} = span of the slab in mm

I_{Slab} = the inertia of the slab in mm 4/mm

EC = the dynamic concrete slab modulus in kN/mm^2

= E_s *1.1/α_{short}

These base, maximum simply supported deflections for both primary and secondary beams, δ **SS, derived from the analysis model, can be adjusted to cater for boundary conditions for 'two-span continuous' or 'three-span continuous' cases to give δ_{barSS}.

For 'two span continuous' the adjusted deflection is taken from P354 as,

δ_{barSS} = MIN[(0.4 + k_M /k_S * (1 + 0.6 * L_S 2/L_M^2))/(1 + k_M /k_S), 1.0] * δ_{**SS}

Where

k_M = the 'stiffness' of the critical span selected by the user (primary or secondary beam as appropriate)

= I_M /L_M

k_S = the stiffness of the adjoining span selected by the user (primary or secondary beam as appropriate)

= IS/L_S

L_M = the span of the critical span selected by the user (primary or secondary beam as appropriate)

L_S = the span pf the adjoining span selected by the user (primary or secondary beam as appropriate)

I_M = the inertia of the critical span selected by the user (primary or secondary beam as appropriate)

I_S = the inertia of the adjoining span selected by the user (primary or secondary beam as appropriate)

For 'three span continuous' the adjusted deflection is taken from P354 as,

δ_{barSS} = MIN[(0.6 + 2 * k_M /k_S * (1 + 1.2 * L_S^2 /L_M^2))/(3 + 2 * k_M /k_S), 1.0] * δ_{**SS}

Where

k_M = the 'stiffness' of the critical (middle) span selected by the user (primary or secondary beam as appropriate)

= I_M / L_M

k_S = the stiffness of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)

= I_S / L_S

L_M = the span of the critical (middle) span selected by the user (primary or secondary beam as appropriate)

L_S = the span of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)

I_M = the inertia of the critical (middle) span selected by the user (primary or secondary beam as appropriate)

I_S = the inertia of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)

Secondary Beam Mode

In this mode the primary beams form nodal lines (zero deflection) about which the secondary beams vibrate. The slab is assumed to be continuous over the secondary beams so a fixed end condition is used.

$\delta_{SBmode} = \delta_{barSBSS} + \delta_{SlabFE}$

and

$f_{SBmode} = 18/\sqrt{\delta_{SBmode}}$

Primary Beam Mode

In this mode the primary beams vibrate about the columns as simply supported beams whilst the secondary beams and slabs are taken to be fixed ended

$\delta_{PBmode} = \delta_{barPBSS} + \delta_{SBFE} + \delta_{SlabFE}$

and

$f_{PBmode} = 18/\sqrt{\delta_{PBmode}}$

System Frequency

The natural frequency of the system, f_0, is calculated from,

f_0	=	MIN{ f_{SBmode} , f_{PBmode} }

Limitations

The absolute minimum natural frequency of the floor system is limited to 3.0 Hz. Where the floor system frequency is below these limits the design fails.

Similarly, no single element within the floor structure should have a fundamental frequency less than 3.0 Hz. Three additional checks are therefore carried out and their results only published if there is a Fail. These checks are,

f_{PBSS} =18/ √ δ_{PBSS} must be ≥ 3 else the design Fails

f_{SBSS} =18/ √ δ_{SBSS} must be ≥ 3 else the design Fails

f_{SlabFE} =18/ √ δ_{SlabFE} must be ≥ 3 else the design Fails

Modal mass

The 'modal mass' is the effective mass participating in the vibration of the floor. In accordance with SCI P354, it is taken as the 'unit mass' multiplied by the effective plan area of the floor participating in the motion as given by,

M	=	m * L_{eff} * S

Where		
m	=	the unit mass in kg/m^2
L_{eff}	=	the effective floor length
S	=	the effective floor width

Where		
L_{eff}	=	09*(1.10)$^{ny-1}$ *(E*I_{SB}/(m*b*f_0 ^{2}))$^{0.25}$ but ≤ n_y*L_y

Where		
n_y	=	number of bays (≤ 4) in the direction of the secondary beam span
EI_{SB}	=	dynamic flexural rigidity of the composite secondary beam (in Nm^2 when m is in kg/m^2)
b	=	floor beam spacing (in m)

f_0 = system, natural frequency from above

Ly = span of the secondary beam (in m)

and

S = $\eta*(1.15)^{nx-1} *(E*I_{Slab} /(m* f_0^2))0.25$ but $\leq n_x *L_x$

Where			
n_x	=	number of bays (≤ 4) in the direction of the primary beam span	
EI_{Slab}	=	dynamic flexural rigidity of the slab (in Nm^2 when m is in kg/m^2) system,	
f_0	=	natural frequency from above	
L_x	=	span of the primary beam (in m)	

Where			
η	=	frequency factor	
	=	0.5	for $f_0 < 5$ Hz
	=	$21*f_0 - 0.55$	for 5 Hz $\leq f_0 \leq$ 6 Hz
	=	0.71	for $f_0 > 6$ Hz

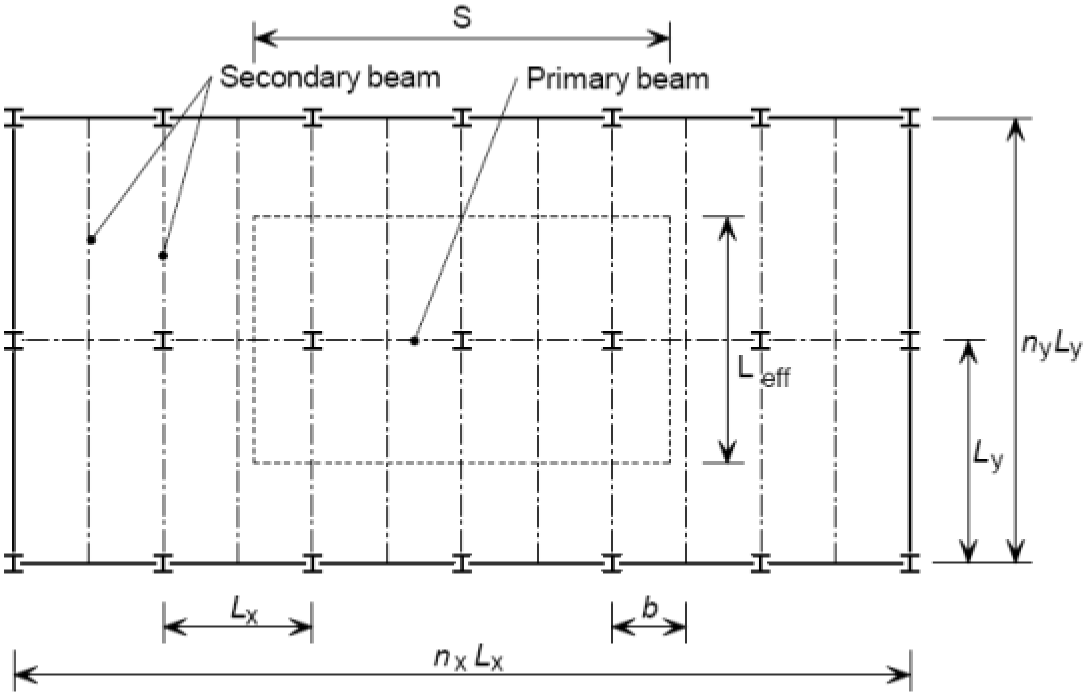


Figure 1: Definition of variables used to establish effective modal mass

Mode Shape Factor

As previously described, there are two main mode shapes which relate to the lowest frequencies - a secondary beam mode and a primary beam mode. The lowest frequency of the two modes is used and the mode shape factors is determined using the same mode.

There are two mode shape factors, μ_e at the point of excitation and μ_r at the point of response.

If the response and excitation points are unknown, or if a general response for the whole floor is required, μ_e and μ_r can conservatively be taken as 1.

Tekla Structural Designer will not calculate the values of these mode shape factors, and will default to 1.0 but also gives you the option of providing values to be used.

Resonance Build-up Factor

The 'resonance build-up factor' makes an allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. Hence, a 'walking time' is required and is calculated from the 'walking distance' (see:Maximum corridor length) divided by the 'walking velocity'.

First it is necessary to calculate the walking velocity as given by Equation 16 of SCI P354,

V	=	$67*f_p^2 - 4.83*f_p + 4.5$	for f_p in the range 1.7 to 2.4 Hz

Where

f_p	=	the pace (walking) frequency supplied by the user

The resonance build-up factor is taken from Equation 37 of SCI P354,

ρ	=	$1 - e^{(-2*\pi*\zeta\ Lp*fp\ /V)}$

Where

ζ	=	the damping ratio
L_p	=	the walking distance
V	=	the walking velocity given above

Note that the resonance build-up factor has an upper bound of 1.0 and may, conservatively be set to 1.0.

Resonance Acceleration

Low Frequency Floors

For system frequencies between 3 Hz and 10 Hz, the root mean square (rms) acceleration is calculated from,

$a_{w,rms}$	=	$\mu_e * \mu_r * 0.1*Q*W*\rho/(2 * \sqrt{2}*M * \zeta)$

Where

μ_e & μ_r	=mode shape factors	
Q	=the person's weight taken as 745.6 N (76 kg)	
M	=the modal mass (kg)	
	=the damping ratio	
ρ	=the resonance build-up factor	
W	=the appropriate code-defined weighting factor for the human perception of vibrations, based on the fundamental frequency, f_0	
	$=f_0 /5$	for $2 \le f_0 < 5$

$=1.0$	for $5 \le f_0 \le 16$
$=16/f_0$	for $f_0 > 16$

High Frequency Floors

For system frequencies greater than 10 Hz, the root mean square (rms) acceleration is calculated from the following expression, which assumes that the floor exhibits a transient response,

$$a_{w,rms} = 2*\pi* \mu e* \mu r * 185*Q*W /(M*f_0{}^{0.3}*700* \sqrt{2})$$

Response Factor

The 'base curves' in BS 6472: 1992 are given in terms of root mean square (rms) acceleration

The provided response factor is then calculated from,

$$R_{prov} = a_{w,rms} /0.005$$

The 'required response factor', R_{reqd}, is a user input and leads to the final design condition,

$$R_{prov} \le R_{reqd}$$

In SCI P354 the recommended Response Factors derive from BS 6472: 1992, where they are called 'Multiplying Factors' and are reproduced in SCI P354 as Tables 5.2 and 5.3.

Vibration Dose Values

When the floor has a higher than acceptable response factor, the acceptability of the floor may be assessed by considering the intermittent nature of the dynamic forces. This is accomplished by carrying out a Vibration Dose Value [VDV] analysis.

This method calculates the number of times an activity (for example walking along a corridor) will take place during an exposure period, n_a, from,

$$n_a = (1/T_a)*(VDV/(0.68*a_{w,rms}))^4$$

where

T_a	=	the duration of the activity
	=	L p/V if L p is known OR
	=	value supplied by user if L p is not known
VDV	=	VDV value supplied by user, (default 0.4).

Typical VDV values are shown below:

Vibration dose limits ($m/s^{1.75}$) for z-axis vibration specified by BS 6472			
Place	**Low probability of adverse comment**	**Adverse comment possible**	**Adverse comment probable**
buildings 16 h day	0.2 to 0.4	0.4 to 0.8	0.8 to 1.6
buildings 8 h night	0.13	0.26	0.51

Input requirements for P354 floor vibration

General

The simplified method for the analysis of the vibration of floors given in the SCI Publication P354, on which the Tekla Structural Designer check is based, is only applicable to regular structures which, by and large, are created from rectilinear grids.

Of course the floor layouts of 'real' multi-storey buildings are rarely uniform and Tekla Structural Designer therefore provides you with the opportunity to select the more irregular floor areas to be assessed with grids that are other than rectilinear.

In so far as the selection of the beams to be used in the analysis is concerned, only beams with Non-Composite or Composite attributes are valid for selection and, within these confines, you are able to:

• select a single beam

• select a beam span as critical plus an adjoining span (in a two or three span configuration)

In all cases, and subject to the above restrictions, which beams from the selected area of floor are chosen is entirely at your discretion and under your judgement, but it is expected that the beams chosen will be those that are typical, common or the worst case. Irrespective, Tekla Structural Designer will take these beams as those that form the idealized floor layout. There is no validation on what the you select (although there is some validation on which beams are selectable i.e. beams which have no slab for part of their length, beams from angle sections, beams with no adjoining span when a 2-span configuration is chosen, and beams with no adjoining span at both ends when a 3-span configuration is chosen will not be selectable).

Data Derived from Tekla Structural Designer (P354)

Note that, where appropriate, the derived data is for each design combination under SLS loads only.

Unit mass

The unit mass in kg/m^2 is used to establish the 'participating mass' of the floor - that is the mass of floor and its permanent loading that has to be set in motion during vibration of the floor. It is taken as the slab self-weight (and to be accurate, the beam self-weight), other permanent 'Dead' loads and the proportion of the 'Imposed' loads that can be considered as permanent. The latter is usually taken as 10% and, whilst this is the default, the value is editable since imposed storage loads, for example, would warrant a higher value.

The unit mass is obtained by summing all the loads (or the appropriate percentage in the case of imposed loads) that act over or in the selected area. This includes any blanket, area, line and spot loads that are present within the selected area. The component of any of these load types that lie outside of the selected area are ignored. Nodal loads directly on columns are also ignored. The total load is then divided by the area selected.

The slab self-weight will usually be in the Slab Dry loadcase - note that in the case of composite slabs this includes the weight of decking. The beam self-weight is in a separate protected loadcase. For simplicity this component of the unit mass is ignored. This leads to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous).

Note that the use of imposed load reductions has no effect on the floor vibration check.

Slab data

If there are more than one set of slab attributes in the selected area then you have to choose which of these it is appropriate to use. From the designated slab attributes the following information/data is obtained,

• the un-transformed inertia in c m 4 per metre width. For profiled decking this takes account of the concrete in the troughs and is independent of the direction of span of the decking.

• the short-term modular ratio for normal or lightweight concrete as appropriate.

If the designated slab attributes are for a 'generic' slab, then you are asked for the inertia and the dynamic modular ratio.

Secondary beam data

When these are non-composite beams, the inertia is obtained from the sections database. When these beams are of composite construction the inertia is the gross, uncracked composite inertia based on the dynamic modular ratio that is required. Steel joist inertias from the database are assumed to be 'gross' inertias of the chords and are editable. Following guidance contained in AISC Steel Design Guide 11 (page 2006), section 3.6, the gross steel joist inertia is factored by quantity C_r and displayed as the 'effective' inertia in the results viewer.

The span of the critical/base beam and the adjoining beams is required.

The deflection of the critical beam under the permanent loads is required. To calculate this value, the deflection under the Dead loads and the appropriate percentage of the Imposed load deflection is summed.

Primary beam data

The same data is required as that for the secondary beams.

Floor plate data

The dimensions of the floor plate in the idealized cases are defined in one direction by the number of secondary beam bays and in the orthogonal direction by the number of primary beam bays. In practice, given that the idealized case may not attain, the floorplate dimensions are derived from the slab items you select as participating in the mass.

User Input Data (P354)

Secondary Beam Spacing

You must confirm the spacing of the secondary beams - an average value when the spacing is non-uniform.

Proportion of Imposed Loads

You are required to specify the proportion of the imposed loads that is to be used in the vibration analysis.

Number of bays used to establish Modal mass

You are required to specify the number of bays in the direction of the secondary beam span, n y, and the number of bays in the direction of the primary beam span, n x, that are to be used to establish the modal mass. The number of bays ranges from 1 to 4 for both directions.

Mode Shape Factors

You are required to specify the mode shape factors, μ_e and μ_r, which are to be used in the evaluation of the root mean square response acceleration. The default value is 1.0 for both variables.

Damping ratio

Floors do not vibrate as a free mass but have some damping i.e. dissipation of the energy in the system. Four values of damping ratio are recommended in P354 as a percentage,

- 0.5%, for fully welded steel structures, e.g. staircases,
- 1.1%, for completely bare floors or floors where only a small amount of furnishings are present,
- 3.0%, for normal, open-plan, well-furnished floors (the default),
- 4.5%, for a floor where the designer is confident that partitions will be appropriately located to interrupt the relevant mode(s) of vibration i.e. the partition lines are perpendicular to the main vibrating elements of the critical mode shape.

Since an even higher damping ratio might be justified for storage floors for example, a range of up to 10% is offered.

Maximum corridor length

This is used in the calculation of the "Resonance Build-up Factor" that makes an allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. Hence, a "walking time" is required and is calculated from the "walking distance" (maximum corridor length) divided by the "walking velocity".

The designer will often not know, reliably, the maximum corridor length. The default is therefore taken as the longer of the floor plate dimensions.

If the designer does not wish to estimate the maximum corridor length or accept the default, then the Resonance Build-up Factor can be set to 1.0 by selecting Not known for the maximum corridor length. This sets the Resonance Build-up Factor to 1.0.

Walking Pace

The walking frequency (pace) must be selected in the range 1.7 to 2.4 Hz. This range is equivalent to a walking velocity of 2.5 to 5.7 mph (4.0 to 9.1 kph). Walking velocities less than and greater than this are achievable - slow walking 1.0 to 1.5 mph (1.6 to 2.4 kph) or running 6.0 to 12.0 mph (9.6 to 19.2 kph). However, the range of validity of the formula for calculating the walking velocity is given as that quoted. Thus any consequent value outside of the range 1.7 to 2.4 Hz is given a Warning that this is outside of the range given in Equation 16 of SCI P354. The default value is 1.8 Hz.

Resonance build-up factor

This is calculated data and has an upper bound of 1.0. However, you are able to specify that the calculations should use 1.0 perhaps because there is insufficient information at the time to make a more accurate and reliable estimate (see: Maximum corridor length). Setting the value to 1.0 is conservative.

Required Response Factor

You must enter the response factor that you expect the floor to achieve. This will be based on your engineering judgement and the advice given in P354. Tables 5.2 and 5.3 of that publication give a range of values with the common values being 2, 4, and 8.

Vibration Dose Value (VDV)

You have to specify the VDV value to be used if this analysis is performed (see: Vibration Dose Values (page 1995)).

Vibration of floors to SCI P354 references

1. **British Standards Institution.** BS 6472: 1992 Guide to the evaluation of human response to vibration in buildings (1Hz to 80 Hz). **BSI 1992.**

2. **The Steel Construction Institute.** Design of Floors for Vibration: A New Approach. **SCI P354. 2007**

3. **AISC Steel Design Guide Series.**11: Floor Vibrations Due to Human Activity. **AISC 2003 re-print.**

14.3 British Standards

Loading (British Standards)

This handbook provides a general overview of how loadcases and combinations are created in Tekla Structural Designer when a British Standards (BS) head code is applied. The Combination Generator for BS loading is also described.

Load cases (British Standards)

Loadcase types (British Standards)

The following load case types can be created:

Loadcase Type	Calculated Automatically	Include in the Combination Generator	Imposed Load Reductions	Pattern Load
self weight (beams, columns and walls)	yes/no	yes/nol	N/A	N/A
slab wet	yes/no	N/A	N/A	N/A
slab dry	yes/no	yes/no	N/A	N/A
dead	N/A	yes/no	N/A	N/A
imposed	N/A	yes/no	yes/no	yes/no
roof imposed	N/A	yes/no	N/A	N/A
wind	N/A	yes/no	N/A	N/A

Loadcase Type	Calculated Automatically	Include in the Combination Generator	Imposed Load Reductions	Pattern Load
snow	N/A	yes/no	N/A	N/A
snow drift	N/A	yes/no	N/A	N/A
temperature	N/A	N/A	N/A	N/A
settlement	N/A	N/A	N/A	N/A
seismic	N/A	yes	N/A	N/A

As shown above, self weight loads can all be determined automatically. However, other gravity load cases have to be applied manually as you build the structure.

Self weight (British Standards)

Self weight - excluding slabs loadcase

Tekla Structural Designer automatically calculates the self weight of the structural beams/columns for you. The **Self weight - excluding slabs** loadcase is pre-defined for this purpose. Its loadcase type is fixed as “Selfweight”. It cannot be edited and by default it is added to each new load combination.

Self weight of concrete slabs

Tekla Structural Designer expects the wet and dry weight of concrete slab to be defined in separate loadcases. This is required to ensure that members are designed for the correct loads at construction stage and post construction stage.

The **Slab self weight** loadcase is pre-defined for the dry weight of concrete post construction stage, its loadcase type is fixed as “Slab Dry”.

There is no pre-defined loadcase for the wet weight of concrete slab at construction stage, but if you require it for the design of any composite beams in the model the loadcase type should be set to “Slab Wet”.

Tekla Structural Designercan automatically calculate the above weights for you taking into account the slab thickness, the shape of the deck profile and wet/dry concrete densities. It does not explicitly take account of the weight of any reinforcement but will include the weight of decking. Simply click the **Calc Automatically** check box when you create each loadcase. When calculated in this way you can’t add extra loads of your own into the loadcase.

If you normally make an allowance for ponding in your slab weight calculations, Tekla Structural Designer can also do this for you. After selecting the composite slabs, you are able to review the slab item properties - you will find two ways to add an allowance for ponding (under the slab parameters heading). These are:

- as a value, by specifying the average increased thickness of slab

- or, as a percentage of total volume.

Using either of these methods the additional load is added as a uniform load over the whole area of slab.

Imposed and roof imposed loads (British Standards)

Imposed load reductions

Reductions can be applied to imposed loads to take account of the unlikelihood of the whole building being loaded with its full design imposed load. Reductions can not however be applied to roof imposed loads.

Imposed loads are only automatically reduced on:

- Columns of any material
- Concrete walls, mid-pier or meshed

Tekla Structural Designer does not automatically apply imposed load reductions to floors. For steel beams, concrete beams, slabs and mats it is however possible to define the level of imposed load reduction manually via the beam/slab item properties.

This is particularly relevant for the design of transfer beams/slabs:

- The imposed load reduction for beams, slabs and mats is intended to work with loads applied from columns acting on the beam or slab when the slab is acting in transfer or for a mat foundation supporting a column. (The theory being that if you want to design the columns for the reduced axial load, you should also design the supporting member for the reduced axial load applied by the column.)
- The engineer would need to work out the reduction of the axial load in the column and apply this as a the reduction percentage, i.e. if the raw axial load in the column is 100kN and the reduced load is 60kN, the reduction is 40%. You would than apply the 40% reduction to the transfer beam/slab or mat as well.
- The reduction is not applied to loads for analysis - it is a post-analysis process which does not affect the analysis results. It does not get applied solely to the imposed load applied directly to the beam or slab panel, but instead is applied to the design moment used in the beam/slab or mat design process.

Wind loads (British Standards)

The BS 6399-2 Wind wizard...

NOTE The **Wind Wizard** used for automatic wind loadcase generation is fully described in the Wind Modelling Engineer’s Handbook.

The **Wind Wizard…** is run to create a series of static forces that are combined with other actions due to dead and imposed loads in accordance with BS6399-2:1997.

The following assumptions/limitations exist:

- The shape of the building meets the limitations allowed for in the code.
- It must be a rigid structure.
- The structure must be either enclosed or partially enclosed.
- Parapets and roof overhangs are not explicitly dealt with.

For further information on the wind loading capabilities of Tekla Structural Designer refer to the Wind Modelling Engineer's Handbook.

Simple wind loading

If use of the **Wind Wizard** is not appropriate for your structure then wind loads can be applied via element or structure loads instead.

Combinations (British Standards)

Once your load cases have been generated as required, you then combine them into load combinations; these can either be created manually, by clicking **Add…** - or with the assistance of the Combinations Generator, by clicking **Generate…**

NOTE For the British Standard codes we are assuming that the wind load applied in manually defined combinations, or via the combination generator, satisfies the minimum horizontal load requirement (BS5950 Cl 2.4.2.3 (1% of factored dead load) and BS8110 Cl 3.1.4.2 (1.5% characteristic dead weight)). If this is not the case, i.e. the wind load is less than the minimum proportion of dead load specified in the code, then you need to consider manually creating a minimum horizontal load combination.

Manually defined combinations (British Standards)

As you build up combinations manually, the combination factors are automatically adjusted as load cases are added and removed from the combination.

Notional horizontal forces (NHFs) (British Standards)

NHF's are automatically derived from the loadcases within the current combination, their magnitude being calculated in accordance with BS5950 cl

2.4.2.3 as 0.5% of the factored vertical load that passes through any beam/column intersection in the structure.

NOTE BS8110 cl 3.1.4.2 has a requirement for notional horizontal load “NHL” This does NOT equate to the NHF requirement described above. The calculation of “NHL” as defined in BS8110 is beyond scope in the current version of Tekla Structural Designer.

They are applied to the structure in the building directions 1 and 2 as follows:

- NHF Dir1+
- NHF Dir1-
- NHF Dir2+
- NHF Dir2-

The net result is that any combination is able to have up to 2 Notional Loads applied within it - one from Dir1 (+ or -) and one from Dir2 (+ or -). Note however, that Dir1+ can not be added with Dir1- (and similarly Dir2+ can not be added with Dir2-).

Combination generator (British Standards)

Accessed via the **Generate...** command, this automatically sets up combinations for both strength and serviceability.

Combination generator - Combinations

The first page of the generator lists the combinations applicable (with appropriate strength factors).

The following basic load combinations are created:

- 1.4 (Dead) + 1.6 (Imposed or Snow)
- 1.2 (Dead) + 1.2 (Imposed or Snow) + 1.2 (Wind)
- 1.0 (Dead) + 1.4 (Wind)

NOTE Temperature and settlement load case types are not included in the **Generate...** command - these need to be added manually.

The combination names are generated automatically.

Combination generator - Service

This page indicates which combinations are to be checked for serviceability and the factors applied.

The following basic load combinations are created:

- 1.0 (Dead) + 1.0 (Live or Snow)
- 1.0 (Dead) + 0.8 (Live or Snow) + 0.8 (Wind)
- 1.0 (Dead) + 1.0 (Wind)

Combination generator - NHF

The last page is used to set up the notional horizontal forces. You can specify NHF's and factors in each of four directions. For each direction selected, a separate NHF combination will be generated.

Any combination with wind in is automatically greyed.

Click **Finish** to see the list of generated combinations.

Combination classes (British Standards)

Having created your combinations you classify them as: Construction Stage, Gravity, Lateral, or Modal Mass.

NOTE If generated via the Combinations generator they are classified for you automatically.

Then (where applicable) you indicate whether they are to be checked for strength or service conditions, or both. You also have the option to make any of the combinations inactive.

Construction stage combination (British Standards)

A Construction Stage load combination is only required for the purpose of designing any composite beams within the model. It is distinguished from other combinations by setting its “Class” to Construction Stage. Typically this combination would include a loadcase of type “Slab Wet”, (not “Slab Dry”), other loadcases being included in the combination as required.

NOTE The Slab Wet loadcase type should not be included in any other combination.

Gravity combination (British Standards)

These combinations are considered in both the Gravity Sizing and Full Design processes.

They are used in the Gravity Sizing processes as follows:

- Design Concrete (Gravity) - concrete members in the structure are automatically sized (or checked) for the gravity combinations
- Design Steel (Gravity) - steel members in the structure are automatically sized (or checked) for the gravity combinations.
- Design All (Gravity) - all members in the structure are automatically sized (or checked) for the gravity combinations.

They are also used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the gravity combinations.

- Design Steel (All) - steel members in the structure are automatically sized (or checked) for the gravity combinations.
- Design All (All) - all members in the structure are automatically sized (or checked) for the gravity combinations.

Lateral combinations (British Standards)

These combinations are **not** used in the Gravity Sizing processes.

They are used during the Full Design processes as follows:

- Design Concrete (All) - concrete members in the structure are automatically sized (or checked) for the lateral combinations.
- Design Steel (All) - steel members in the structure which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.
- Design All (All) - all concrete members and all steel members which have not been set as Gravity Only are automatically sized (or checked) for the lateral combinations.

Modal mass combinations (British Standards)

For modal analysis, you are required to set up specific “modal mass” combinations. Provided these combinations are active they are always run through the modal analysis.

NOTE It is always assumed that all loads in the load cases in the combination are converted to mass for modal analysis. You are permitted to add lumped mass directly to the model.

Steel design to BS 5950

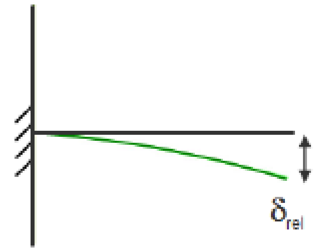
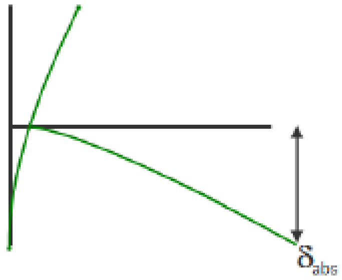
Tekla Structural Designer designs steel members and composite members to a range of international codes. This reference guide specifically describes the design methods applied when the steel design and composite design resistance codes are set as BS 5950-1 and BS 5950-3.1 respectively.

Unless explicitly noted otherwise, all clauses, figures and tables referred to are from BS 5950-1:2000 (Ref. 2); apart from the Composite Beam section, within which references are to BS 5950-3.1:2010 (Ref. 1) unless stated.

Basic principles (BS 5950)

Deflection checks

Tekla Structural Designer calculates both ***relative*** and ***absolute*** deflections. Relative deflections measure the internal displacement occurring within the length of the member and take no account of the support settlements or rotations, whereas absolute deflections are concerned with deflection of the structure as a whole. The absolute deflections are the ones displayed in the structure deflection graphics. The difference between ***relative*** and ***absolute*** deflections is illustrated in the cantilever beam example below.

	
Relative Deflection	Absolute Deflection

Relative deflections are given in the member analysis results graphics and are the ones used in the member design.

Steel beam design to BS 5950

Design method (Beams: BS 5950)

Unless explicitly stated all calculations are in accordance with the relevant sections of BS 5950-1:2000. You may find the handbook and commentary to the Code of Practice published by the Steel Construction Institute useful.

Steel beam limitations and assumptions (Beams: BS 5950)

The following limitations apply:

- continuous beams (more than one span) must be co-linear in the plane of the web within a small tolerance (sloping in elevation is allowed),
- rolled doubly symmetric prismatic sections (that is I- and H-sections), doubly symmetric hollow sections (i.e. SHS, RHS and CHS), and channel sections are fully designed,
- single angle, double angles and tees are designed, but certain checks are beyond scope, (see Angle and tee limitations)
- plated beams are fully designed provided the section type is either “Plated Beam” or “Plated Column”. All other plated section types (“Rolled I Sections

with Plates", "Double Rolled I Sections" etc.) are only analysed but not designed.

- Fabsec beams (with or without openings) are excluded.

The following assumptions apply:

- All supports are considered to provide torsional restraint, that is lateral restraint to both flanges. This cannot be changed. It is assumed that a beam that is continuous through the web of a supporting beam or column together with its substantial moment resisting end plate connections is able to provide such restraint.
- If, at the support, the beam oversails the supporting beam or column then the detail is assumed to be such that the bottom flange of the beam is well connected to the supporting member and, as a minimum, has torsional stiffeners provided at the support.
- In the Tekla Structural Designer model, when not at supports, coincident restraints to both flanges are assumed when one or more members frame into the web of the beam at a particular position and the cardinal point of the centre-line model of the beam lies in the web. Otherwise, only a top flange or bottom flange restraint is assumed. Should you judge the actual restraint provided by the in-coming members to be different from to what has been assumed, you have the flexibility to edit the restraints as required.
- Intermediate lateral restraints to the top or bottom flange are assumed to be capable of transferring the restraining forces back to an appropriate system of bracing or suitably rigid part of the structure.
- It is assumed that you will make a rational and "correct" choice for the effective lengths between restraints for both LTB and compression buckling. **The default value for the effective length factor of 1.0 may be neither correct nor safe.**

Ultimate limit state (strength) (Beams: BS 5950)

The checks relate to doubly symmetric prismatic sections (that is rolled I- and H-sections), to singly symmetric sections i.e. channel sections and to doubly symmetric hollow sections i.e. SHS, RHS and CHS. Other section types are not currently covered.

The strength checks relate to a particular point on the member and are carried at regular intervals along the member and at "points of interest".

Classification (Beams: BS 5950)

General

The classification of the cross section is in accordance with BS 5950-1: 2000.

Steel beams can be classified as:

- Plastic Class = 1

- Compact Class = 2
- Semi-compact Class = 3
- Slender Class = 4

Class 4 sections are only acceptable for angle and tee sections.

Sections with a Class 3 web can be taken as Class 2 sections (Effective Class 2) providing the cross section is equilibrated to that described in Clause 3.5.6 where the section is given an "effective" plastic section modulus, S_{eff}. For rolled I and H sections in the UK, this gives no advantage in pure bending since the web d/t is too small. Hence for beams there is likely to be little advantage in using this approach since the axial loads are generally small, this classification is therefore not implemented.

All unacceptable classifications are either failed in check mode or rejected in design mode.

Hollow sections

The classification rules for SHS and RHS relate to "hot-finished hollow sections" only ("cold-formed hollow sections" are not included in this release).

WARNING	Important Note. The classification used to determine M_b is based on the maximum axial compressive load in the relevant segment length. Furthermore, the Code clearly states that this classification should (only) be used to determine the moment capacity and lateral torsional buckling resistance to Clause 4.2 and 4.3 for use in the interaction equations. Thus, when carrying out the strength checks, the program determines the classification at the point at which strength is being checked.

Shear capacity (Beams: BS 5950)

The shear check is performed according to BS 5950-1: 2000 Clause 4.2.3. for the absolute value of shear force normal to the x-x axis (F_{vx}) and normal to the y-y axis (F_{vy}), at the point under consideration.

Shear buckling

When the web slenderness exceeds 70ε shear buckling can occur in rolled sections. There are very few standard rolled sections that breach this limit. Tekla Structural Designer will warn you if this limit is exceeded, but will not carry out any shear buckling checks.

Moment capacity (Beams: BS 5950)

The moment capacity check is performed according to BS 5950-1: 2000 Clause 4.2.5 for the moment about the x-x axis (M_x) and about the y-y axis (M_y), at the point under consideration. The moment capacity can be influenced by the magnitude of the shear force ("low shear" and "high shear" conditions). The maximum absolute shear to either side of a point load is examined to determine the correct condition for the moment capacity in that direction.

NOTE Not all cases of high shear in two directions combined with moments in two directions along with axial load are considered thoroughly by BS 5950-1: 2000. The following approach is adopted by Tekla Structural Designer:

- if high shear is present in one axis or both axes and axial load is also present, the cross-section capacity check is given a Beyond Scope status. The message associated with this status is “High shear and axial load are present, additional hand calculations are required for cross-section capacity to Annex H.3”. Tekla Structural Designer does not perform any calculations for this condition.
- if high shear and moment is present in both axes and there is no axial load (“biaxial bending”) the cross-section capacity check is given a Beyond Scope status and the associated message is, “High shear present normal to the y-y axis, no calculations are performed for this condition.”
- if high shear is present normal to the y-y axis and there is no axial load, the y-y moment check and the cross-section capacity check are each given Beyond Scope statuses. The message associated with this condition is, “High shear present normal to the y-y axis, no calculations are performed for this condition.”

Axial capacity (Beams: BS 5950)

The axial capacity check is performed according to BS 5950-1: 2000 Clause 4.6.1 using the gross area and irrespective of whether the axial force is tensile or compressive. This check is for axial compression capacity and axial tension capacity. Compression resistance is a buckling check and as such is considered under Compression Resistance.

Cross-section capacity (Beams: BS 5950)

The cross-section capacity check covers the interaction of axial load and bending to clause 4.8.2 and 4.8.3.2 appropriate to the type (for example – doubly symmetric) and classification of the section. Since the axial tension capacity is not adjusted for the area of the net section then the formula in clause 4.8.2.2 and 4.8.3.2 are the same and can be applied irrespective of whether the axial load is compressive or tensile.

The Note in Moment capacity also applies here.

Ultimate limit state (buckling) (Beams: BS 5950)

Lateral torsional buckling resistance, clause 4.3 (Beams: BS 5950)

For beams that are unrestrained over part or all of a span, a Lateral torsional buckling (LTB) check is required either:

- in its own right, clause 4.3 check,
- as part of an Annex G check,
- as part of a combined buckling check to 4.8.3.3.1, 4.8.3.3.2 or 4.8.3.3.3, (see Member buckling resistance, clause 4.8.3.3.1, Member buckling resistance, clause 4.8.3.3.2 (page 2020), and Member buckling resistance, clause 4.8.3.3.3 (page 2021), respectively)

This check is not carried out under the following circumstances:

- when bending exists about the minor axis only,
- when the section is a CHS or SHS,
- when the section is an RHS that satisfies the limits given in Table 15 of BS 5950-1: 2000.

For sections in which LTB cannot occur (the latter two cases above) the value of M_b for use in the combined buckling check is taken as the full moment capacity, M_{cx}, not reduced for high shear in accordance with clause 4.8.3.3.3 (c), equation 2 (See Member buckling resistance, clause 4.8.3.3.3).

Effective lengths

The value of effective length factor is entirely at your choice. The default value is 1.0 for “normal” loads and 1.2 for “destabilizing loads”. Different values can apply in the major and minor axis.

Lateral torsional buckling resistance, Annex G (Beams: BS 5950)

This check is applicable to I- and H-sections with equal or unequal[1] flanges.

The definition of this check is the out-of-plane buckling resistance of a member or segment that has a laterally unrestrained compression flange and the other flange has intermediate lateral restraints at intervals. It is used normally to check the members in portal frames in which only major axis moment and axial load exist. Although not stated explicitly in BS 5950-1: 2000, it is taken that the lateral torsional buckling moment of resistance, M_b, from the Annex G check can be used in the interaction equations of clause 4.8.3.3 (combined buckling).

Since this is not explicit within BS 5950-1: 2000 a slight conservatism is introduced. In a straightforward Annex G check the axial load is combined with major axis moment. In this case both the slenderness for lateral torsional buckling and the slenderness for compression buckling are modified to allow for the improvement provided by the tension flange restraints (λ_{LT} replaced by λ_{TB} and λ replaced by λ_{TC}). When performing a combined buckling check in accordance with 4.8.3.3 the improvement is taken into account in determining the buckling resistance moment but not in determining the compression resistance. If the incoming members truly only restrain the tension flange, then you should switch off the minor axis strut restraint at these positions.

The original source research work for the codified approach in Annex G used test specimens in which the tension flange was continuously restrained. When a segment is not continuously restrained but is restrained at reasonably frequent intervals it can be clearly argued that the approach holds true. With only one or two restraints present then this is less clear. BS 5950-1: 2000 is clear that there should be “at least one intermediate lateral restraint” (See Annex G.1.1). Nevertheless, you are ultimately responsible for accepting the adequacy of this approach.

For this check Tekla Structural Designer sets m_t to 1.0 and calculates n_t. The calculated value of n_t is based on M_{max} being taken as the maximum of M_1 to M_5, and not the true maximum moment value where this occurs elsewhere in the length. The effect of this approach is likely to be small. If at any of points 1 - 5, R >1[2], then the status of the check is set to Beyond Scope.

Reference restraint axis distance, a

The **reference restraint axis distance** is measured between some reference axis on the restrained member - usually the centroid - to the axis of restraint - usually the centroid of the restraining member. The measurement is shown diagramatically in Figure G.1 of BS 5950-1: 2000.

Tekla Structural Designer does not attempt to determine this value automatically. Instead, by default, it uses half the depth of the restrained section, and you can specify a value to be added to, or subtracted from, this at each restraint point. **You are responsible for specifying the appropriate values for each restraint position. The default value of 0mm may be neither correct nor safe**.

[1] Unequal flanged sections are not currently included.

[2] Which could happen since R is based on Z and not S.

Compression resistance (Beams: BS 5950)

For most structures, all the members resisting axial compression need checking to ensure adequate resistance to buckling about both the major- and minor-axis. Since the axial force can vary throughout the member and the buckling lengths in the two planes do not necessarily coincide, both are checked. Because of the general nature of a beam, it may not always be safe to assume that the combined buckling check will always govern. Hence the compression resistance check is performed independently from the other strength and buckling checks.

Effective lengths

The value of effective length factor is entirely at your choice. The default value is 1.0 for “normal” loads and 1.2 for “destabilizing loads”. Different values can apply in the major and minor axis.

Beams are less affected by sway than columns but the effectiveness of the incoming members to restrain the beam in both position and direction is generally less than for columns. Hence, it is less likely that effective length factors greater than 1.0 will be required but equally factors less than 1.0 may not easily be justified. Nevertheless, it is your responsibility to adjust the value from 1.0 and to justify such a change.

Please note that the requirements for slenderness limits in (for example l/r ≤ 180) are no longer included in BS 5950-1: 2000. Consequently Tekla Structural Designer does not carry out such checks. Accordingly, for lightly loaded members you should ensure that the slenderness ratio is within reasonable bounds to permit handling and erection and to provide a reasonable level of robustness.

Member buckling resistance, clause 4.8.3.3.1 (Beams: BS 5950)

This check is used for channel sections. Such sections can be Class 1, 2 or 3 Plastic, Compact or Semi-compact (Class 4 Slender sections and Effective Class 2 sections are not allowed in this release).

Note that, whilst this check could be used for any section type dealt with in the subsequent sections, the results can never be any better than the alternatives but can be worse.

Two formula are provided in clause 4.8.3.3.1, both are checked; the second is calculated twice – once for the top flange and once for the bottom flange.

See also the *Important Note* at the end of Member buckling resistance, clause 4.8.3.3.2.

Only one value of F is used, the worst anywhere in the length being checked. If the axial load is tensile, then F is taken as zero.

If this check is invoked as part of an Annex G check, and thus M_b is governed by Annex G, then m_{LT} is taken as 1.0.

Member buckling resistance, clause 4.8.3.3.2 (Beams: BS 5950)

This check is used for Class 1, 2 and 3 Plastic, Compact and Semi-compact rolled I- and H-sections with equal flanges (Class 4 Slender sections and Effective Class 2 sections are not included in this release).

Three formula are provided in clause 4.8.3.3.2 (c) to cover the combined effects of major and minor axis moment and axial force. These are used irrespective of whether all three forces / moments exist. Clause 4.9 deals with biaxial moment in the absence of axial force, clause 4.8.3.3.2 (c) can also be used in such cases by setting the axial force to zero.

All three formula in clause 4.8.3.3.2 (c) are checked; the second is calculated twice – once for each flange.

Only one value of F is used, the worst anywhere in the length being checked. If the axial load is tensile, then F is taken as zero.

WARNING	Important Note. Clause 4.8.3.3.4 defines the various equivalent uniform moment factors. The last three paragraphs deal with modifications to these depending upon the method used to allow for the effects of sway. This requires that for sway sensitive frames the uniform moment factors, m_x, m_y and m_{xy}, should be applied to the non-sway moments only. In this release there is no mechanism to separate the sway and non-sway moments, Tekla Structural Designer adopts a conservative approach and sets these 'm' factors equal to 1.0 if the frame is sway sensitive (in either direction). This is doubly conservative for sway-sensitive unbraced frames since it is likely that all the loads in a design combination and not just the lateral loads will be amplified. In such a case, both the sway and non-sway moments are increased by k_{amp} and neither are reduced by the above “m” factors. The calculation of m_{LT} is unaffected by this approach, and thus if the second equation of clause 4.8.3.3.2 (c) governs, then the results are not affected.

Member buckling resistance, clause 4.8.3.3.3 (Beams: BS 5950)

This check is used for Class 1, 2 and 3 Plastic, Compact and Semi-compact hollow sections (Class 4 Slender sections and Effective Class 2 sections are not included in this release).

Four formula are provided in clause 4.8.3.3.3 (c) to cover the combined effects of major and minor axis moment and axial force. These are used irrespective of whether all three forces / moments exist. Clause 4.9 deals with biaxial moment in the absence of axial force, clause 4.8.3.3.3 (c) can also be used in such cases by setting the axial force to zero.

The second and third formula are mutually exclusive – that is the second is used for CHS, SHS and for RHS when the limits contained in Table 15 are **not** exceeded. On the other hand the third formula is used for those RHS that exceed the limits given in Table 15. Thus only three formula are checked; the first, second and fourth or the first, third and fourth. Either the second or third (as appropriate) is calculated twice – once for each “flange”.

Only one value of F is used, the worst anywhere in the length being checked. If the axial load is tensile, then F is taken as zero.

See also the *Important Note* at the end of Member buckling resistance, clause 4.8.3.3.2.

Natural frequency checks (SLS) (Beams: BS 5950)

Tekla Structural Designer calculates the approximate natural frequency of the beam based on the simplified formula published in the Design Guide on the vibration of floors (Ref. 6) which states that Natural frequency = 18 / √ δ

In line with the calculation of natural frequency of 18 / √ δ for a pin ended beam with applied UDL, we calculate δ as the maximum static instantaneous deflection based upon the composite inertia (using the short term modular ratio) but not modified for the effects of partial interaction as:

$\delta = \%max\ \delta_{self+slab} + \%max\ \delta_{other\ dead} + \%max\ \delta_{live}$

The engineer can specify:

- Percentage self wt + slab deflection (default 100%)
- Percentage other dead deflection (default 100%)
- Percentage live load deflection (default 10%)
- Factor of increased dynamic stiffness of concrete flange (default 1.1)

Web openings (Beams: BS 5950)

Circular openings as an equivalent rectangle

Each circular opening is replaced by equivalent rectangular opening, the dimensions of this equivalent rectangle for use in all subsequent calculations are:

- d_o' = 0.9 * opening diameter
- l_o = 0.45 * opening diameter

Properties of tee sections

When web openings have been added, the properties of the tee sections above and below each opening are calculated in accordance with Section 3.3.1 of SCI P355 (Ref. 10 (page 2047)) and Appendix B of the joint CIRIA/SCI Publication P068 (Ref. 5 (page 2047)). The bending moment resistance is calculated separately for each of the four corners of each opening.

Design

The following calculations are performed where required for web openings:

- Axial resistance of tee sections
- Classification of section at opening
- Vertical shear resistance
- Vierendeel bending resistance
- Web post horizontal shear resistance
- Web post bending resistance
- Web post buckling resistance
- Lateral torsional buckling

• Deflections

Deflections

The deflection of a beam with web openings will be greater than that of the same beam without openings. This is due to two effects,

- the reduction in the beam inertia at the positions of openings due to primary bending of the beam,
- the local deformations at the openings due to Vierendeel effects. This has two components - that due to shear deformation and that due to local bending of the upper and lower tee sections at the opening.

The primary bending deflection is established by 'discretising' the member and using a numerical integration technique based on 'Engineer's Bending Theory' - M/I = E/R = σ/y. In this way the discrete elements that incorporate all or part of an opening will contribute more to the total deflection.

The component of deflection due to the local deformations around the opening is established using a similar process to that used for cellular beams which is in turn based on the method for castellated beams given in the SCI publication, “Design of castellated beams. For use with BS 5950 and BS 449".

The method works by applying a 'unit point load' at the position where the deflection is required and using a 'virtual work technique to estimate the deflection at that position.

For each opening, the deflection due to shear deformation, δ_s, and that due to local bending, δ_{bt}, is calculated for the upper and lower tee sections at the opening. These are summed for all openings and added to the result at the desired position from the numerical integration of primary bending deflection.

Note that in the original source document on castellated sections, there are two additional components to the deflection. These are due to bending and shear deformation of the web post. For castellated beams and cellular beams where the openings are very close together these effects are important and can be significant. For normal beams the openings are likely to be placed a reasonable distance apart. Thus in many cases these two effects will not be significant. They are not calculated for such beams but in the event that the openings are placed close together a warning is given.

Composite beam design to BS 5950

Design method (Composite beams: BS 5950)

Unless explicitly stated all calculations are in accordance with the relevant sections of BS 5950-3.1:1990+A1:2010 (Ref. 1). You may find the handbook and commentary to the Code of Practice published by the Steel Construction Institute (Ref. 3 and 4) useful.

Construction stage design checks

When you use Tekla Structural Designer to design or check a beam for the construction stage (the beam is acting alone before composite action is achieved) the following conditions are examined in accordance with BS 5950-1:2000:

- section classification (Clause 3.5.2),
- shear capacity (Clause 4.2.3),
- moment capacity:
  - Clause 4.2.5.2 for the low shear condition,
  - Clause 4.2.5.3 for the high shear condition,
  - lateral torsional buckling resistance (Clause 4.3.6),

    NOTE This condition is only checked in those cases where the profile decking or precast concrete slab (at your request) does not provide adequate restraint to the beam.

- web openings,
- Westok checks,
  - Shear horizontal,
  - Web post buckling,
  - Vierendeel bending,
- construction stage total load deflection check.

Composite stage design checks

When you use Tekla Structural Designer to design or check a beam for the composite stage (the beam and concrete act together, with shear interaction being achieved by appropriate shear connectors) the following Ultimate Limit State and Serviceability Limit State conditions are examined in accordance with BS 5950 : Part 3 : Section 3.1 : 1990 (unless specifically noted otherwise).

Ultimate limit state checks

- section classification (Clause 4.5.2), depending on whether adequate connection is achieved between the compression flange and the slab. The section classification allows for the improvement of the classification of the section if the appropriate conditions are met,
- vertical shear capacity (BS 5950-1:2000 - Clause 4.2.3),
- longitudinal shear capacity (Clause 5.6) allowing for the profiled metal decking, transverse reinforcement and other reinforcement which has been defined,

- number of shear connectors required (Clause 5.4.7) between the point of maximum moment and the end of the beam, or from and between the positions of significant point loads,
- moment capacity:
    - Clause 4.4.2 for the low shear condition,
    - Clause 5.3.4 for the high shear condition,
- web openings.

Serviceability limit state checks

- service stresses (Clause 6.2),
- - concrete
    - steel top flange and bottom flange
- deflections (Clause 6.1.2)
    - self-weight
    - SLAB loadcase,
    - dead load,
    - imposed load[1],
    - total deflections,

  natural frequency check (Clause 6.4).

[1] This is the only limit given in BS 5950 : Part 3 : Section 3.1 : 1990.

Construction stage design (Composite beams: BS 5950)

Tekla Structural Designer performs all checks for this condition in accordance with BS 5950-1:2000 (Ref. 2)

Section classification (Composite beams: BS 5950)

Cross-section classification is determined using Table 11 and clause 3.5.

The classification of the section must be Plastic (Class 1), Compact (Class 2) or Semi-compact (Class 3).

Sections which are classified as Slender (Class 4) are beyond the scope of Tekla Structural Designer.

Member strength checks (Composite beams: BS 5950)

Member strength checks are performed at the point of maximum moment, the point of maximum shear, the position of application of each point load, and at each side of a web opening as well as all other points of interest along the beam.

Shear capacity

Shear capacity is determined in accordance with clause 4.2.3. Where the applied shear force exceeds 60% of the capacity of the section, the high shear condition applies to the bending moment capacity checks (see below).

Bending moment capacity

This is calculated to clause 4.2.5.2 (low shear at point) or clause 4.2.5.3 (high shear at point) for plastic, compact and semi-compact sections.

Lateral torsional buckling checks (Composite beams: BS 5950)

BS 5950 : Part 3 : Section 3.1 : 1990 states that lateral torsional buckling checks are not required when the angle between the direction of span of the beam and that of the profile decking is greater than or equal to 45°.

When the angle is less than this, then lateral torsional buckling checks will normally be required. Tekla Structural Designerallows you to switch off these checks by specifying that the entire length between the supports is continuously restrained against lateral torsional buckling.

If you use this option you must be able to provide justification that the beam is adequately restrained against lateral torsional buckling during construction.

When the checks are required you can position restraints at any point within the length of the main beam and can set the effective length of each sub-beam (the portion of the beam between one restraint and the next) either by giving factors to apply to the physical length of the beam, or by entering the effective length that you want to use. Each sub-beam which is not defined as being continuously restrained is checked in accordance with clause 4.3.8 and Annex B of BS 5950-1:2000.

Deflection checks (Composite beams: BS 5950)

Tekla Structural Designer calculates relative deflections. (See: Deflection checks)

The following deflections are calculated for the loads specified in the construction stage load combination:

- the dead load deflections i.e. those due to the beam self weight, the Slab Wet loads and any other included dead loads,
- the imposed load deflections i.e. those due to construction live loads,
- the total load deflection i.e. the sum of the previous items.

The loads are taken as acting on the steel beam alone.

The “Service Factor” (default 1.0), specified against each load case in the construction combination is applied when calculating the above deflections.

If requested by the user, the total load deflection is compared with either a span-over limit or an absolute value The initial default limit is span/200.

NOTE Adjustment to deflections. If web openings have been defined, the calculated deflections are adjusted accordingly. See: Web openings (page 2022)

Composite stage design (Composite beams: BS 5950)

Tekla Structural Designer performs all checks for the composite stage condition in accordance with BS 5950-3.1:1990+A1:2010 unless specifically noted otherwise.

Equivalent steel section - Ultimate limit state (ULS) (Composite beams: BS 5950)

An equivalent steel section is determined for use in the composite stage calculations by removing the root radii whilst maintaining the full area of the section. This approach reduces the number of change points in the calculations while maintaining optimum section properties.

Section classification (ULS) (Composite beams: BS 5950)

For section classification purposes the true section is used. Tekla Structural Designer classifies the section in accordance with the requirements of BS 5950-1:2000 except where specifically modified by those of BS 5950-3.1:1990+A1:2010.

There are a small number of sections which fail to meet a classification of compact at the composite stage. Although BS 5950-3.1:1990+A1:2010 covers the design of such members they are not allowed in this release of Tekla Structural Designer.

Member strength checks (ULS) (Composite beams: BS 5950)

Member strength checks are performed at the point of maximum moment, the point of maximum shear, the position of application of each point load, and at each side of a web opening as well as all other points of interest along the beam.

Shear capacity (Vertical)

is determined in accordance with clause 4.2.3 of BS 5950-1:2000. Where the applied shear force exceeds 50% of the capacity of the section, the high shear condition applies to the bending moment capacity checks (see below).

Shear capacity (Longitudinal)

the longitudinal shear resistance of a unit length of the beam is calculated in accordance with clause 5.6. You can set the position and attachment of the decking and details of the reinforcement that you want to provide. Tekla Structural Designer takes these into account during the calculations.

The following assumptions are made:

- the applied longitudinal shear force is calculated at the centre-line of the beam, and at the position of the lap (if known). If the position of the lap is not known, then the default value of 0mm should be used (that is the lap is at the centre-line of the beam) as this is the worst case scenario.
- the minimum concrete depth is assumed for calculating the area of concrete when the profile decking and beam spans are parallel,
- the total concrete area is used when the profile decking and beam spans are perpendicular,
- the overall depth of the slab is used for precast concrete slabs. that is the topping is assumed to be structural and any voids or cores are ignored.

In the calculations of the longitudinal shear resistance on the beam centre-line and at the lap, the areas used for the reinforcement are shown in the following table.

Decking angle	Reinforcement type	Area used
perpendicular	transverse	that of the single bars defined or for mesh the area of the main wires[1]
	other	that of the single bars defined or for mesh the area of the main wires[1]
parallel	transverse	that of the single bars defined or for mesh the area of the main wires[1]
	other	single bars have no contribution, for mesh the area of the minor wires[2]

[1]These are the bars that are referred to as longitudinal wires in BS 4483: 1998 Table 1

[2]These are the bars that are referred to as transverse wires in BS 4483: 1998 Table 1

If the decking spans at some intermediate angle (α) between these two extremes then the program calculates:

- the longitudinal shear resistance as if the sheeting were perpendicular, v_1,
- the longitudinal shear resistance as if the sheeting were parallel, v_2,
- then the modified longitudinal shear resistance is calculated from these using the relationship, $v_1 \sin^2(\alpha) + v^2 \cos^2(\alpha)$.

Moment capacity

for the low shear condition the plastic moment capacity is determined in accordance with clause 4.4.2. For the high shear condition the approach given in clause 5.3.4 is adopted.

The overall depth of the slab is used for precast concrete slabs. That is the topping is assumed to be structural and any voids / cores are ignored.

In this calculation the steel section is ***idealized*** to one without a root radius so that the position of the plastic neutral axis of the composite section can be determined correctly as it moves from the flange into the web.

Shear connectors (ULS) (Composite beams: BS 5950)

Tekla Structural Designer checks shear connectors to clause 5.4.7. It calculates the stud reduction factor based on the number of studs in a group.

Tekla Structural Designer always uses 2 * e (and not b_r) in the calculation of k for perpendicular profiles, and always uses b_r for parallel cases.

For angled cases two values of k are calculated and summed in accordance with clause 5.4.7.4. In this instance Tekla Structural Designer uses 2 * e for the calculation of k1 and b_r for the calculation of k2.

WARNING Caution: The value of e (when used) can have a very significant effect on the value of k. This can have a dramatic effect on the number of studs required for a given beam size. Alternatively for a fixed layout of studs this can have a significant effect on the required beam size.

Optimize shear connection

Stud optimization is a useful facility since there is often some over conservatism in a design due to the discrete changes in the size of the section.

If you choose the option to optimize the shear studs, then Tekla Structural Designer will progressively reduce the number of studs either until the minimum number of studs to resist the applied moment is found, until the minimum allowable interaction ratio (for example 40% for beams with a span less than 10 m) is reached or until the minimum spacing requirements are reached. This results in partial shear connection.

The degree of shear connection is checked at the point of maximum bending moment or the position of a point load if at that position the maximum utilization ratio occurs.

NOTE During the selection process, in auto design mode point load positions are taken to be “significant” (i.e. considered as positions at which the maximum utilization could occur) if they provide more than 10% of the total shear on the beam. For the final configuration and for check mode all point load positions are checked.

To determine if the degree of shear connection is acceptable Tekla Structural Designer applies the following rules:

- If the degree of shear connection at the point of maximum moment is less than the minimum permissible shear connection, then this generates a FAIL status,
- If the point of maximum utilization ratio occurs at a point that is not the maximum moment position and the degree of shear connection is less than the minimum permissible shear connection, then this generates a WARNING status,
- If the degree of shear connection at any other point load is less than the minimum permissible shear connection, then this does not affect the status in any way.

NOTE The percentage degree of shear connection is always calculated by the program as a proportion of the maximum concrete force and not simply N_a/N_p as in the code.

Section properties - serviceability limit state (SLS) (Composite beams: BS 5950)

BS 5950-3.1:1990+A1:2010 indicates that the Serviceability Limit State modular ratio for all SLS calculations should be based upon an effective modular ratio derived from the proportions of long term loading in the design combination being considered.

Tekla Structural Designer therefore calculates the deflection for the beam based on the properties as tabulated below.

Loadcase type	Properties used
self-weight	bare beam
Slab	bare beam
Dead	composite properties calculated using the modular ratio for long term loads
Live	composite properties calculated using the effective modular ratio appropriate to the long term load percentage for each load. The deflections for all loads in the loadcase are calculated using the principle of superposition.
Wind	composite properties calculated using the modular ratio for short term loads
Total loads	these are calculated from the individual loadcase loads as detailed

Loadcase type	Properties used
	above again using the principle of superposition

Stress checks (SLS) (Composite beams: BS 5950)

Tekla Structural Designer calculates the worst stresses in the extreme fibres of the steel and the concrete at serviceability limit state for each load taking into account the proportion which is long term and that which is short term. These stresses are then summed algebraically. Factors of 1.00 are used on each loadcase in the design combination (you cannot amend these). The stress checks assume that full interaction exists between the steel and the concrete at serviceability state.

Deflection checks (SLS) (Composite beams: BS 5950)

Tekla Structural Designer calculates relative deflections. (See: Deflection checks)

The composite stage deflections are calculated in one of two ways depending upon the previous and expected future load history:

- the deflections due to all loads in the Slab Dry loadcase and the self-weight of the beam are calculated based on the inertia of the steel beam alone (these deflections will not be modified for the effects of partial interaction).

  NOTE It is the Slab Dry deflection alone which is compared with the limit, if any, specified for the Slab loadcase deflection. See: Web openings (page 2022)

- the deflections for all loads in the other loadcases of the Design Combination will be based on the inertia of the composite section allowing for the proportions of the particular load that are long or short term (see above). When necessary these will be modified to include the effects of partial interaction in accordance with clause 6.1.4.

  NOTE It is the deflection due to imposed loads alone (allowing for long and short term effects) which is limited within the code. Tekla Structural Designer also gives you the deflection for the Slab loadcase which is useful for pre-cambering the beam. The beam Self-weight, Dead and Total deflections are also given to allow you to be sure that no component of the deflection is excessive.

  NOTE Adjustment to deflections - If web openings have been defined, the calculated deflections are adjusted accordingly.

Circular openings as an equivalent rectangle

Each circular opening is replaced by equivalent rectangular opening, the dimensions of this equivalent rectangle for use in all subsequent calculations are:

d_o' = 0.9 * opening diameter

l_o = 0.45 * opening diameter

Properties of tee sections

When web openings have been added, the properties of the tee sections above and below each opening are calculated in accordance with Section 3.3.1 of SCI P355 (Ref. 10) and Appendix B of the joint CIRIA/SCI Publication P068 (Ref. 5). The bending moment resistance is calculated separately for each of the four corners of each opening.

Design at construction stage

The following calculations are performed where required for web openings:

- Axial resistance of tee sections
- Classification of section at opening
- Vertical shear resistance
- Vierendeel bending resistance
- Web post horizontal shear resistance
- Web post bending resistance
- Web post buckling resistance
- Lateral torsional buckling
- Deflections

Design at composite stage

The following calculations are performed where required for web openings:

- Axial resistance of concrete flange
- Vertical shear resistance of the concrete flange
- Global bending action - axial load resistance
- Classification of section at opening
- Vertical shear resistance
- Moment transferred by local composite action
- Vierendeel bending resistance
- Web post horizontal shear resistance

- Web post bending resistance
- Web post buckling resistance
- Deflections

Deflections

The deflection of a beam with web openings will be greater than that of the same beam without openings. This is due to two effects,

- the reduction in the beam inertia at the positions of openings due to primary bending of the beam,
- the local deformations at the openings due to Vierendeel effects. This has two components - that due to shear deformation and that due to local bending of the upper and lower tee sections at the opening.

The primary bending deflection is established by 'discretising' the member and using a numerical integration technique based on 'Engineer's Bending Theory' - M/I = E/R = σ/y. In this way the discrete elements that incorporate all or part of an opening will contribute more to the total deflection.

The component of deflection due to the local deformations around the opening is established using a similar process to that used for cellular beams which is in turn based on the method for castellated beams given in the SCI publication, “Design of castellated beams. For use with BS 5950 and BS 449".

The method works by applying a 'unit point load' at the position where the deflection is required and using a 'virtual work technique to estimate the deflection at that position.

For each opening, the deflection due to shear deformation, δ_s, and that due to local bending, δ_{bt}, is calculated for the upper and lower tee sections at the opening. These are summed for all openings and added to the result at the desired position from the numerical integration of primary bending deflection.

Note that in the original source document on castellated sections, there are two additional components to the deflection. These are due to bending and shear deformation of the web post. For castellated beams and cellular beams where the openings are very close together these effects are important and can be significant. For normal beams the openings are likely to be placed a reasonable distance apart. Thus in many cases these two effects will not be significant. They are not calculated for such beams but in the event that the openings are placed close together a warning is given.

Steel column design to BS 5950

Design method (Columns: BS 5950)

Unless explicitly stated all calculations are in accordance with the relevant sections of BS 5950-1: 2000. You may find the handbook and commentary to the Code of Practice published by the Steel Construction Institute useful.

Ultimate limit state (strength) (Columns: BS 5950)

The checks relate to doubly symmetric prismatic sections i.e. rolled I- and H-sections and to doubly symmetric hot-finished hollow sections i.e. SHS, RHS and CHS. Other section types are not currently covered.

The strength checks relate to a particular point on the member and are carried out at 5th points and "points of interest", (i.e. positions such as maximum moment, maximum axial etc.)

Classification (Columns: BS 5950)

General

The classification of the cross section is in accordance with BS 5950-1: 2000.

Steel columns can be classified as:

- Plastic Class = 1
- Compact Class = 2
- Semi-compact Class = 3
- Slender Class = 4

Class 4 sections are not allowed.

Sections with a Class 3 web can be taken as Class 2 sections (Effective Class 2) providing the cross section is equilibrated to that described in Clause 3.5.6 where the section is given an "effective" plastic section modulus, S_{eff}. This approach is not adopted in the current version of Tekla Structural Designer.

All unacceptable classifications are either failed in check mode or rejected in design mode.

Hollow sections

The classification rules for SHS and RHS relate to "hot-finished hollow sections" only ("cold-formed hollow sections" are not included in this release).

WARNING important. The classification used to determine M_b is based on the maximum axial compressive load in the relevant segment length. Furthermore, the code clearly states that this classification should (only) be used to determine the moment capacity and lateral torsional buckling resistance to clause 4.2 and 4.3 for use in the interaction equations. Thus, when carrying out the strength checks, the program determines the classification at the point at which strength is being checked.

Shear capacity (Columns: BS 5950)

The shear check is performed according to BS 5950-1: 2000 clause 4.2.3. for the absolute value of shear force normal to the x-x axis and normal to the y-y axis, F_{vx} and F_{vy}, at the point under consideration.

Shear buckling

When the web slenderness exceeds 70e shear buckling can occur in rolled sections. There are very few standard rolled sections that breach this limit. Tekla Structural Designerwill warn you if this limit is exceeded, but will not carry out any shear buckling checks.

Moment capacity (Columns: BS 5950)

The moment capacity check is performed according to BS 5950-1: 2000 clause 4.2.5 for the moment about the x-x axis and about the y-y axis, M_x and M_y, at the point under consideration. The moment capacity can be influenced by the magnitude of the shear force (“low shear” and “high shear” conditions). The maximum absolute shear to either side of a point of interest is used to determine the moment capacity for that direction.

High shear condition about x-x axis

The treatment of high shear is axis dependent. In this release for CHS, if high shear is present, the moment capacity about the x-x axis is not calculated, the check is given a Beyond Scope status and an associated explanatory message.

High shear condition about y-y axis

For rolled sections in the current release, if high shear is present normal to the y-y axis then the moment capacity about the y-y axis is not calculated, the check is given a Beyond Scope status and an associated explanatory message.

For hollow sections, there is greater potential for the section to be used to resist the principal moments in its minor axis. Of course for CHS and SHS there is no major or minor axis and so preventing high shear arbitrarily on one of the two principal axes does not make sense. Nevertheless, if high shear is present normal to the y-y axis then in this release the moment capacity about the y-y axis is not calculated, the check is given a Beyond Scope status and an associated explanatory message.

Note

Not all cases of high shear in two directions combined with moments in two directions along with axial load are considered thoroughly by BS 5950-1: 2000.

The following approach is adopted by Tekla Structural Designer:

- if high shear is present in one axis or both axes and axial load is also present, the cross-section capacity check is given a Beyond Scope status. The message associated with this status is “High shear and axial load are present, additional hand calculations are required for cross-section capacity to Annex H.3”. Tekla Structural Designer does not perform any calculations for this condition.
- if high shear and moment is present in both axes and there is no axial load (“biaxial bending”) the cross-section capacity check is given a Beyond Scope status and the associated message is, “High shear present normal to the y-y axis, no calculations are performed for this condition.”
- if high shear is present normal to the y-y axis and there is no axial load, the y-y moment check and the cross-section capacity check are each given Beyond Scope statuses. The message associated with this condition is, “High shear present normal to the y-y axis, no calculations are performed for this condition.”

Axial capacity (Columns: BS 5950)

The axial capacity check is performed according to BS 5950-1: 2000 clause 4.6.1 using the gross area and irrespective of whether the axial force is tensile or compressive. This check is for axial compression capacity and axial tension capacity. Compression resistance is a buckling check and as such is considered under Compression resistance.

Cross-section capacity (Columns: BS 5950)

The cross-section capacity check covers the interaction of axial load and bending to clause 4.8.2 and 4.8.3.2 appropriate to the type (for example – doubly symmetric) and classification of the section. Since the axial tension capacity is not adjusted for the area of the net section then the formula in clause 4.8.2.2 and 4.8.3.2 are the same and can be applied irrespective of whether the axial load is compressive or tensile.

The Note in Moment capacity (page 2035)also applies here.

Ultimate limit state (buckling) (Columns: BS 5950)

Lateral torsional buckling resistance, Clause 4.3 (Columns: BS 5950)

For columns that are unrestrained over part or all of a span, a Lateral torsional buckling (LTB) check is required either:

- in its own right, clause 4.3 check,
- as part of an Annex G check,

- as part of a combined buckling check to clause 4.8.3.3.2 or 4.8.3.3.3, (See: Member buckling resistance, Clause 4.8.3.3.2 (page 2039), and Member buckling resistance, clause 4.8.3.3.3 (page 2039)).

This check is not carried out under the following circumstances:

- when bending exists about the minor axis only,
- when the section is a CHS or SHS,
- when the section is an RHS that satisfies the limits given in Table 15 of BS 5950-1: 2000.

For sections in which LTB cannot occur (the latter two cases above) the value of M_b for use in the combined buckling check is taken as the full moment capacity, M_{cx}, not reduced for high shear in accordance with clause 4.8.3.3.3 (c), equation 2 (See: Member buckling resistance, Clause 4.8.3.3.2 (page 2039)).

Destabilising loads are excluded from Tekla Structural Designer, this is justified by the rarity of the necessity to apply such loads to a column. If such loads do occur, then you can adjust the “normal” effective length to take this into account although you can not achieve the code requirement to set m_{LT} to 1.0.

Effective lengths

The value of effective length factor is entirely at your choice. The default value is 1.0.

Lateral torsional buckling resistance, Annex G (Columns: BS 5950)

This check is applicable to I- and H-sections with equal or unequal[1] flanges.

The definition of this check is the out-of-plane buckling resistance of a member or segment that has a laterally unrestrained compression flange and the other flange has intermediate lateral restraints at intervals. It is used normally to check the members in portal frames in which only major axis moment and axial load exist. Although not stated explicitly in BS 5950-1: 2000, it is taken that the lateral torsional buckling moment of resistance, M_b, from the Annex G check can be used in the interaction equations of clause 4.8.3.3 (combined buckling).

Since this is not explicit within BS 5950-1: 2000 a slight conservatism is introduced. In a straightforward Annex G check the axial load is combined with major axis moment. In this case both the slenderness for lateral torsional buckling and the slenderness for compression buckling are modified to allow for the improvement provided by the tension flange restraints (λ_{LT} replaced by λ_{TB} and λ replaced by λ_{TC}). When performing a combined buckling check in accordance with 4.8.3.3 the improvement is taken into account in determining the buckling resistance moment but not in determining the compression resistance. If the incoming members truly only restrain the tension flange, then you should switch off the minor axis strut restraint at these positions.

The original source research work for the codified approach in Annex G used test specimens in which the tension flange was continuously restrained. When a segment is not continuously restrained but is restrained at reasonably frequent intervals it can be clearly argued that the approach holds true. With only one or two restraints present then this is less clear.BS 5950-1: 2000 is clear that there should be “at least one intermediate lateral restraint” (See Annex G.1.1). **Nevertheless, you are ultimately responsible for accepting the adequacy of this approach.**

For this check Tekla Structural Designer sets mt to 1.0 and calculates n_t. The calculated value of n_t is based on M_{max} being taken as the maximum of M_1 to M_5, and not the true maximum moment value where this occurs elsewhere in the length. The effect of this approach is likely to be small. If at any of points 1 - 5, R >1 [2], then Tekla Structural Designer sets the status of the check to Beyond Scope.

Reference restraint axis distance, a

The *reference restraint axis distance* is measured between some reference axis on the restrained member - usually the centroid - to the axis of restraint - usually the centroid of the restraining member. The measurement is shown diagramatically in Figure G.1 of BS 5950-1: 2000.

Tekla Structural Designer does not attempt to determine this value automatically, since such an approach is fraught with difficulty and requires information from you which is only used for this check. Instead, by default, Tekla Structural Designer uses half the depth of the restrained section, and you can specify a value to be added to, or subtracted from, this at each restraint point. **You are responsible for specifying the appropriate values for each restraint position. The default value of 0mm may be neither correct nor safe.**

[1]Unequal flanged sections are not currently included.

[2] Which could happen since R is based on Z and not S.

Compression resistance (Columns: BS 5950)

For most structures, all the members resisting axial compression need checking to ensure adequate resistance to buckling about both the major- and minor-axis. Since the axial force can vary throughout the member and the buckling lengths in the two planes do not necessarily coincide, both are checked. Because of the general nature of a column, it may not always be safe to assume that the combined buckling check will always govern. Hence the compression resistance check is performed independently from all other strength and buckling checks.

Effective lengths

The value of effective length factor is entirely at your choice. The default value is 1.0. Different values can apply in the major and minor axis.

The minimum theoretical value is 0.5 and the maximum infinity for columns in rigid moment resisting (RMR) frames. Practical values for simple columns are in the range 0.7 to 2.0. Values less than 1.0 can be chosen for non-sway frames or for sway frames in which the effects of sway are taken into account using the amplified moments method. However, there is a caveat on the value of effective length factor even when allowance is made for sway.

In particular for RMR frames, the principal moments due to frame action preventing sway are in one plane of the frame. There will often be little or no moment out-of-plane and so amplification of these moments has little effect. Nevertheless the stability out-of-plane can still be compromised by the lack of restraint due to sway sensitivity in that direction. In such cases a value of greater then 1.0 (or substantially greater) may be required. Similarly, in simple construction where only eccentricity moments exist, it is only the brace forces that “attract” any amplification. Thus for the column themselves the reduced restraining effect of a sway sensitive structure may require effective length factors greater than 1.0 as given in Table 22 of BS 5950-1: 2000.

Member buckling resistance, Clause 4.8.3.3.2 (Columns: BS 5950)

This check is used for Class 1, 2 and 3 Plastic, Compact and Semi-compact rolled I- and H-sections with equal flanges (Class 4 Slender sections and Effective Class 2 sections are not included in this release).

Three formulae are provided in clause 4.8.3.3.2 (c) to cover the combined effects of major and minor axis moment and axial force.These are used irrespective of whether all three forces / moments exist. Clause 4.9 deals with biaxial moment in the absence of axial force, clause 4.8.3.3.2 (c) can also be used in such cases by setting the axial force to zero.

All three formulae in clause 4.8.3.3.2 (c) are checked; the second is calculated twice – once for Face A and once for Face C.

Only one value of F is used, the worst anywhere in the length being checked. If the axial load is tensile, then F is taken as zero.

Important Note

Clause 4.8.3.3.4 defines the various equivalent uniform moment factors. The last three paragraphs deal with modifications to these depending upon the method used to allow for the effects of sway. This requires that for sway sensitive frames the uniform moment factors, m_x, m_y and m_{xy}, should be applied to the non-sway moments only. In this release there is no mechanism to separate the sway and non-sway moments, Tekla Structural Designer adopts the only conservative approach and sets these “m” factors equal to 1.0 if the frame is sway sensitive (in either direction). This is doubly conservative for sway-sensitive unbraced frames since it is likely that all the loads in a design combination and not just the lateral loads will be amplified. In such a case, both the sway and non-sway moments are increased by kamp and neither are reduced by the above “m” factors. The calculation of m_{LT} is unaffected by this approach, and thus if the second equation of clause 4.8.3.3.2 (c) governs, then the results are not affected.

Member buckling resistance, Clause 4.8.3.3.3 (Columns: BS 5950)

This check is used for Class 1, 2 and 3 Plastic, Compact and Semi-compact hollow sections (Class 4 Slender sections and Effective Class 2 sections are not included in this release).

Four formulae are provided in clause 4.8.3.3.3 (c) to cover the combined effects of major and minor axis moment and axial force. These are used irrespective of whether all three forces / moments exist. Clause 4.9 deals with biaxial moment in the absence of axial force, clause 4.8.3.3.3 (c) can also be used in such cases by setting the axial force to zero.

The second and third formulae are mutually exclusive – that is the second is used for CHS, SHS and for RHS when the limits contained in Table 15 are **not** exceeded. On the other hand the third formula is used for those RHS that exceed the limits given in Table 15. Thus only three formulae are checked; the first, second and fourth or the first, third and fourth. Either the second or third (as appropriate) is calculated twice – once for Face C and once for Face A.

Only one value of F is used, the worst anywhere in the length being checked. If the axial load is tensile, then F is taken as zero.

See also the **Important Note** at the end of Member buckling resistance, Clause 4.8.3.3.2 (page 2039).

Serviceability limit state (Columns: BS 5950)

The column is assessed for sway and the following values are reported for each stack:

- Sway X (mm) and λ_{critx}
- Sway Y (mm) and λ_{crity}
- Sway X-Y (mm)

Depending on the reported λ_{crit} the column is classified as Sway or Non sway accordingly.

NOTE A sway assessment is only performed for the column if the Lambda Crit Check box is checked on the Column Properties dialog.

If very short columns exist in the building model these can distort the overall sway classification for the building. For this reason you may apply engineering judgement to uncheck the Lambda Crit Check box for those columns for which a sway assessment would be inappropriate

Steel brace design to BS 5950

Design method

Unless explicitly stated all brace calculations are in accordance with the relevant sections of BS 5950-1:2000 (Ref. 2).

A basic knowledge of the design methods for braces in accordance with the design code is assumed.

Classification

No classification is required for braces in tension.

Braces in compression are classified according to Clause 3.5 as either: Class 1, Class 2, Class 3 or Class 4.

Class 4 sections are not allowed.

Hollow sections

The classification rules for SHS and RHS relate to “hot-finished hollow sections” only (“cold-formed hollow sections” are not included in this release).

Axial Tension

An axial tension capacity check is performed according to clause 4.6.

Axial compression

An axial compression capacity check is performed according clause 4.7.

Compression buckling

If axial compression exists, the member is also assessed according to clause 4.7 with all relevant sub-clauses.

The default effective length in each axis is 1.0L.

Steel single, double angle and tee section design to BS 5950

Design method (Angles and tees: BS 5950)

The design method adopted is dictated by the member characteristic type:

- “Beam”, “Truss member top” or “Truss member bottom” characteristic:
    - Member is designed for axial tension, compression, shear, bending and combined forces - consistent with the method detailed in Steel beam design to BS 5950 (page 2014)
- “Brace”, “Truss internal” or “Truss member side” characteristic:

- Member is designed for axial tension, compression and compression buckling only - consistent with the method detailed in Steel brace design to BS 5950

NOTE Additional Angle and tee limitations (page 2042) have to be considered when designing these sections to the above design methods.

Angle and tee limitations (BS 5950)

In the current version when designing tees, single, and double angles to BS 5950, the following checks remain beyond scope:

	Tee	Angle	Double Angle
Classification	ok	ok	ok
Axial tension	ok	ok	ok
Axial compression	ok	ok	ok
Shear	ok	ok	ok
Bending	ok	ok	ok
Combined strength	ok	ok	ok
LTB	ok	ok	Beyond scope
Combined buckling	ok	ok	Beyond scope
Deflection	ok	ok	ok

In addition, the following limitations apply:

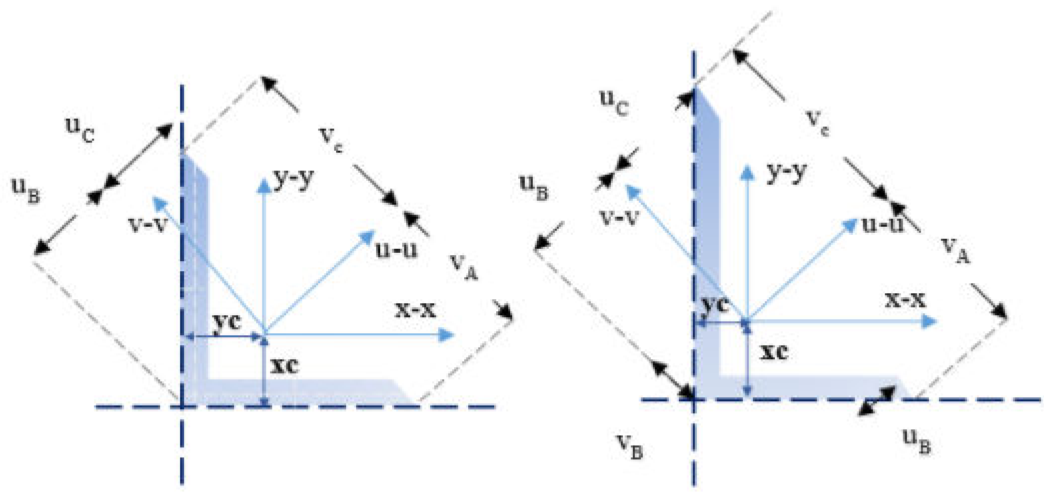
- All sections are assumed to be effectively loaded through the shear centre such that no additional torsion moments are developed. In addition no direct allowance is made for 'destabilizing loads'.
- Design excludes bending of the outstand leg of double angles loaded eccentrically e.g. supporting masonry.
- Conditions of restraint can be defined as top and bottom flange for lateral torsional buckling. It is upon these that the buckling checks are based. For the current release intermediate LTB restraints are omitted (i.e. only fully restrained for LTB, or unrestrained).
- Double angles and tee sections subject to moment with high shear are beyond scope.

Section axes (Angles and tees: BS 5950)

For all sections:

- x-x is the axis parallel to the flanges (major axis)

- y-y is the axis perpendicular to the flanges (minor axis)
- for Single angles and Double angles
    - y-y parallel to long side (leg) - single angles
    - y-y parallel to long side (leg) - double angles with long leg back to back
    - y-y parallel to short side (leg) - double angles with short leg back to back
- u-u is the major principal axis for single angles
- v-v is the minor principal axis for single angles



Single angles - Section axes

Design procedures (Angles and tees: BS 5950)

This section includes key notes and assumptions made for the British Standard design of tees and angle sections.

Classification checks

For axial compression and bending both the web and flange (Leg 1 and Leg 2) are classified as Class 1, Class 2, Class 3 or Class 4 and the worst of the two is the resultant classification for that cross section.

The rules from Table 11 and 12 of BS 5950-1:2000 apply for the classification of these sections.

NOTE Class 4 section classification is only allowed for tees, double angles and single angles.

Axial tension check

Section 4.6 of BS 5950 is used for this design check.

Axial compression check

Section 4.6 of BS 5950 is used for this design check.

Shear check

Section 4.2.3 of BS 5950 is used for this design check.

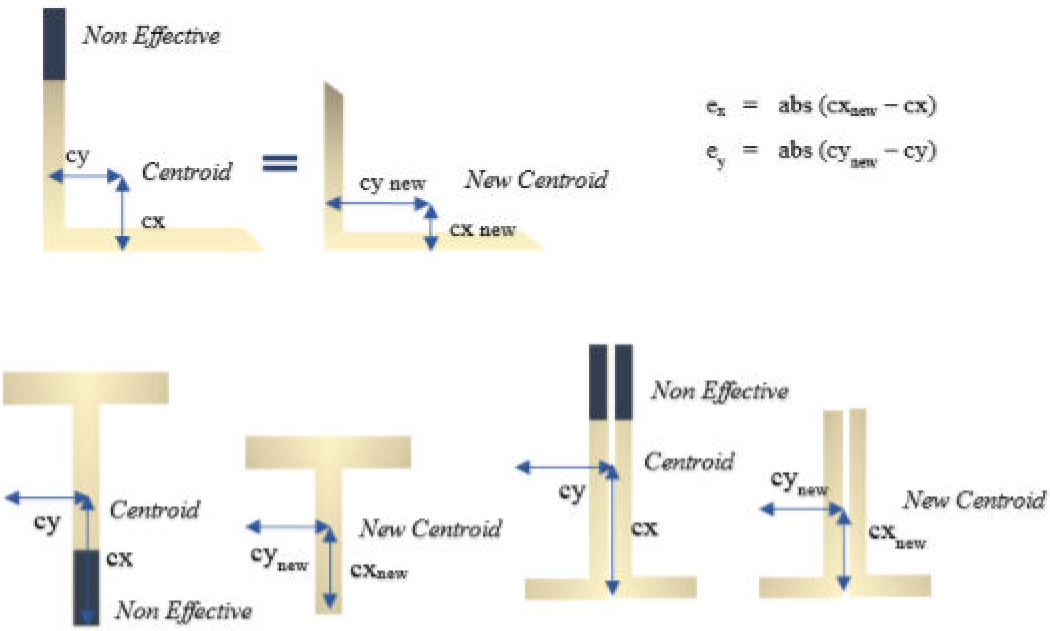
Moment check

Section 4.2.5 of BS 5950 is used for this design check.

NOTE Tees, single angles and double angles are designed as Class 4.

Moment capacity for Class 4 slender sections:

Class 4 sections are designed as Class 3 effective sections.



Hence, additional moments are induced in the member due to the shift of the centroid of the effective cross-section compared to that of the gross section when under axial compression only.

Thus:

$\Delta M_{Ed,x} = e_x \times F_c$

$\Delta M_{Ed,y} = e_y \times F_c$

Where:

F_c is the max compressive force in the span.

For tees and double angles $e_x = 0$. Hence, total minor design moment = minor design moment.

Where:

e_x and e_y = the shift of the centroid of the effective area A_{eff} relative to the centre of gravity of the gross cross section

$e_x = abs(cx_{new} – c_x)$

$e_y = abs(cy_{new} – c_y)$

So finally, a total moment is obtained for which the moment design check is performed:

$M_{total\ x} = Abs(M_{Ed,x}) + Abs(\Delta M_{Ed,x})$

$M_{total\ y} = Abs(M_{Ed,y}) + Abs(\Delta M_{Ed,y})$

Single angles - asymmetric sections:

Single angles **with continuous lateral – torsional restraint** along the length are permitted to be designed on the basis of **geometric axis (x, y) bending**.

Single angles **without continuous lateral – torsional restraint** along the length are designed using the provision for **principal axis (u, v) bending** since we know that the principal axes do not coincide with the geometric ones.

$\Delta M_u = \Delta M_x \times \cos\vartheta + \Delta M_y \times \sin\vartheta$

$\Delta M_v = -\Delta M_x \times \sin\vartheta + \Delta M_y \times \cos\vartheta$

Note that when principal axis design is required for single angles and the classification is Class 4, all moments are resolved into the principal axes (total moment in the principal axes u-u and v-v).

NOTE	Tees, single angles and double angles subject to moment with high shear are beyond scope.

Combined bending and axial check

Section 4.8.3 of BS 5950 is used for this design check.

For Class 3:

$Abs\ (F_c\ /\ A_gP_y) + abs\ (M_{x,Ed}\ /M_{cx}) + abs\ (M_{y,Ed}/W_{el,min,y}) \leq 1.0$

For Class 4:

$Abs\ (F_c/A_{eff}p_y) + (abs\ (M_{x,Ed}) + abs\ (\Delta M_{x,Ed}))\ /\ M_{cx} + abs\ (M_{y,Ed}) + abs\ (\Delta M_{y,Ed}))\ /\ M_{cy} \leq 1.0$

Note that total moments are used when the section classification is Class 4.

Lateral torsional buckling check

Section 4.3 of BS 5950 is used for this design check.

NOTE	This check is beyond scope for double angles.

In the case of a beam with continuous lateral torsional restraint along its length this check is not performed. The lateral torsional resistance is considered adequate.

For beams that are unrestrained, a Lateral torsional buckling (LTB) check is required, either:

- In its own right check for LTB, clause 4.3, and B2.8 for tee sections and B.2.9 for angle sections in BS 5050-1: 2000.
- As part of combined buckling, clause 4.8 “Members with combined moment and axial force”, 4.8.3.3, for single Angles I3 and I4 sections

This check is not performed when bending exists about the minor axis only

---

NOTE Conditions of restraint can be defined as top and bottom flange for lateral torsional buckling. It is upon these that the buckling checks are based. All intermediate LTB restraints for tees and single angles are ignored.

Combined buckling check

NOTE This check is beyond scope for double angles.

Single angles:

Clause I.4 - For beam with continuous lateral torsional restraint or for equal single angle sections with b/t ≤ 15ε a combined buckling check is performed according to clause I.4.3 - the simplified method.

For any other case clause 4.8.3.3.1 is used with the moments being resolved into the principal axes u-u and v-v. Two formula are provided in clause 4.8.3.3.1, both are checked

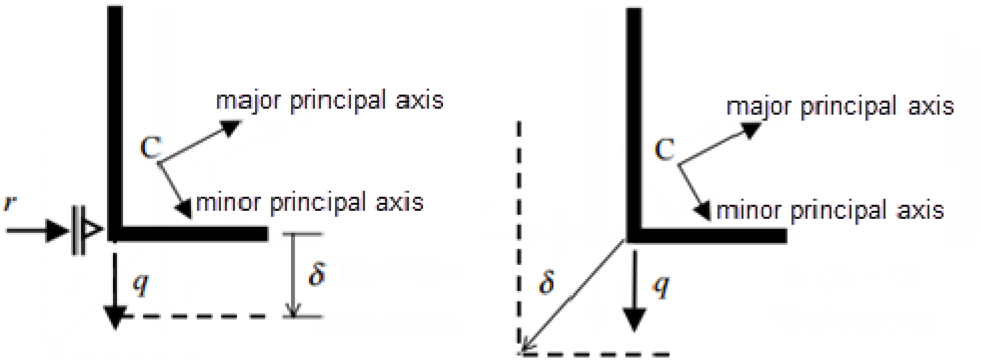
Tees:

Clause 4.8.3.3.1 is used. Two formula are provided in clause 4.8.3.3.1, both are checked.

- If the axial load is tensile, then F is taken as zero
- Only one value of F is used, the worst anywhere in the length being checked
- Class 4 slender sections are allowed

Deflection of single angles

If a single angle is continuously restrained the major geometric moment and major geometric section properties are used in the general equation governing the beam deflection.



Single angle deflections (continuously restrained, unrestrained)

However, because single angle geometric axes are not coincident with the principal axes; a different procedure is required if the angle is not continuously restrained, the procedure being as follows:

1. External loads are transposed from the geometric axes to the principal axes.
2. The deflection equations are used to calculate deflections in the principal axes.
3. These principal axis deflections are then transposed to geometric axes again.

References (BS 5950)

1. **British Standards Institution**. BS 5950-3.1:1990+A1:2010: Structural use of steelwork in building . Design in composite construction. Code of practice for design of simple and continuous composite beams. **BSI 2010**.
2. **British Standards Institution**. BS 5950 : Structural use of steelwork in building; Part 1. Code of practice for design in simple and continuous construction: hot rolled sections. **BSI 2000**.
3. **The Steel Construction Institute**. Publication 078. Commentary on BS 5950 : Part 3 : Section 3.1 : 1990. **SCI 1989**.
4. **The Steel Construction Institute**. Publication 055. Design of Composite Slabs and Beams with Steel Decking. **SCI 1989**.
5. **The Steel Construction Institute**. Publication 068. Design for openings in the webs of composite beams. **SCI 1987**.
6. **The Steel Construction Institute**. Publication 076. Design Guide on the Vibration of Floors. **SCI 1989**.
7. **The Steel Construction Institute**. Publication 100. Design of Composite and Non-Composite Cellular Beams. **SCI 1990**.

8. **The Steel Construction Institute**. Joints in Steel Construction. Simple Connections. **SCI/BCSA 2002. Publication P212**.
9. **The Steel Construction Institute**. Joints in Steel Construction. Moment Connections. **SCI/BCSA 1995. Publication P207**.
10. **The Steel Construction Institute**. Publication P355. Design of Composite Beams with Large Web Openings. **SCI 2011**.

14.4 Australian Standards

Loading to AS/NZS 1170.0 and AS 1170.1 (Australian Standards)

This handbook provides a general overview of how loadcases and combinations are created in Tekla Structural Designer when an Australian Standards (AS) head code is applied. The **Combination Generator** for AS loading is also described.

Load cases (Australian Standards)

Loadcases types (Australian Standards)

The following load case types can be created:

Loadcase Type	Calculated Automatically	Include in the Combination Generator	Imposed Load Reductions	Pattern Load
self weight (beams, columns and walls)	yes/no	yes/no	N/A	N/A
slab wet	yes/no	N/A	N/A	N/A
slab dry	yes/no	yes/no	N/A	N/A
dead	N/A	yes/no	N/A	N/A
imposed	N/A	yes/no	yes/no	yes/no

Loadcase Type	Calculated Automatically	Include in the Combination Generator	Imposed Load Reductions	Pattern Load
roof imposed	N/A	yes/no	N/A	N/A
wind	N/A	yes/no	N/A	N/A
snow	N/A	yes/no	N/A	N/A
snow drift	N/A	yes/no	N/A	N/A
temperature	N/A	N/A	N/A	N/A
settlement	N/A	N/A	N/A	N/A

As shown above, self weight loads can all be determined automatically. However, other gravity load cases have to be applied manually as you build the structure.

Self weight (Australian Standards)

Self weight - excluding slabs loadcase

Tekla Structural Designer automatically calculates the self weight of the structural beams/columns for you. The **Self weight - excluding slabs** loadcase is pre-defined for this purpose. Its loadcase type is fixed as “Selfweight”. It cannot be edited and by default it is added to each new load combination.

Self weight of concrete slabs

Tekla Structural Designer expects the wet and dry weight of concrete slab to be defined in separate loadcases. This is required to ensure that members are designed for the correct loads at construction stage and post construction stage.

The **Slab self weight** loadcase is pre-defined for the dry weight of concrete post construction stage, its loadcase type is fixed as “Slab Dry”.

There is no pre-defined loadcase for the wet weight of concrete slab at construction stage, but if you require it for the design of any composite beams in the model the loadcase type should be set to “Slab Wet”.

Tekla Structural Designercan automatically calculate the above weights for you taking into account the slab thickness, the shape of the deck profile and wet/dry concrete densities. It does not explicitly take account of the weight of any reinforcement but will include the weight of decking. Simply click the **Calc Automatically** check box when you create each loadcase. When calculated in this way you can’t add extra loads of your own into the loadcase.

If you normally make an allowance for ponding in your slab weight calculations, Tekla Structural Designer can also do this for you. After selecting the composite slabs, you are able to review the slab item properties - you will find two ways to add an allowance for ponding (under the slab parameters heading). These are:

- as a value, by specifying the average increased thickness of slab
- or, as a percentage of total volume.

Using either of these methods the additional load is added as a uniform load over the whole area of slab.

Imposed and roof imposed loads (Australian Standards)

Imposed load reductions

Reductions can be applied to imposed loads to take account of the unlikelihood of the whole building being loaded with its full design imposed load. Reductions can not however, be applied to roof imposed loads.

Imposed loads are only automatically reduced on:

- Columns of any material
- Concrete walls, mid-pier or meshed

Tekla Structural Designer does not automatically apply imposed load reductions to floors. For steel beams, concrete beams, slabs and mats it is however possible to define the level of imposed load reduction manually via the beam/slab item properties.

Wind loads (Australian Standards)

The AS 1170.2 Wind wizard...

NOTE	The **Wind Wizard** is not included in this release.

Simple wind loading

Simple wind loads can be applied via element or structure loads.

Combinations (Australian Standards)

Once your load cases have been generated as required, you then combine them into load combinations; these can either be created manually, by clicking **Add...** - or with the assistance of the Combinations Generator (page 2051), by clicking **Generate...**

Manually defined combinations (Austalian Standards)

As you build up combinations manually, the combination factors are automatically adjusted as load cases are added and removed from the combination.

Notional horizontal forces (NHFs) (Australian Standards)

NHF's are automatically derived from the loadcases within the current combination, their magnitude being calculated as 0.2% of the factored vertical load that passes through any beam/column intersection in the structure.

NOTE The values of the NHFs may vary for each load combination.

They are applied to the structure in the building directions 1 and 2 as follows:

- NHF Dir1+
- NHF Dir1-
- NHF Dir2+
- NHF Dir2-

The net result is that any combination is able to have up to 2 Notional Loads applied within it - one from Dir1 (+ or -) and one from Dir2 (+ or -). Note however, that Dir1+ can not be added with Dir1- (and similarly Dir2+ can not be added with Dir2-).

Combination generator (Australian Standards)

Accessed via the **Generate...** command, this automatically sets up combinations for both strength and serviceability.

Combination generator - Combinations

The first page of the generator lists the combinations applicable (with appropriate strength factors).

The following basic load combinations are created:

- 1.35 (Permanent)
- 1.2 (Permanent) + 1.5 (Imposed)
- 1.2 (Permanent) + 1.5 (Ψ_l * Long-term Imposed)
- 1.2 (Permanent) + 1.0 (Wind) + 1.0 (Ψ_C * Imposed)
- 0.9 (Permanent) + 1.0 (Wind)

NOTE Temperature and settlement load case types are not included in the **Generate...** command - these need to be added manually.

The combination names are generated automatically.

Combination generator - Service

This page indicates which combinations are to be checked for serviceability and the factors applied.

The following basic load combinations are created:

- 1.0 (Permanent)
- 1.0 (Ψ_S * Imposed)

- 1.0 (Ψ_l * Imposed)
- 1.0 (Wind)

Combination generator - NHF

The last page is used to set up the notional horizontal forces. You can specify NHF's and factors in each of four directions. For each direction selected, a separate NHF combination will be generated.

Any combination with wind in is automatically greyed.

Click **Finish** to see the list of generated combinations.

Combination classes (Australian Standards)

Having created your combinations you classify them as either Gravity combinations or Lateral combinations, and also (where applicable) indicate whether they are to be checked for strength or service conditions, or both.

NOTE If generated via the Combinations Generator they are classified for you automatically.

You also have the option to make any of the combinations inactive.

Steel design to AS 4100

Tekla Structural Designer designs steel members and composite members to a range of international codes. This reference guide specifically describes the design methods applied when the steel design and composite design resistance codes are set as AS 4100 and AS 2327.1 respectively.

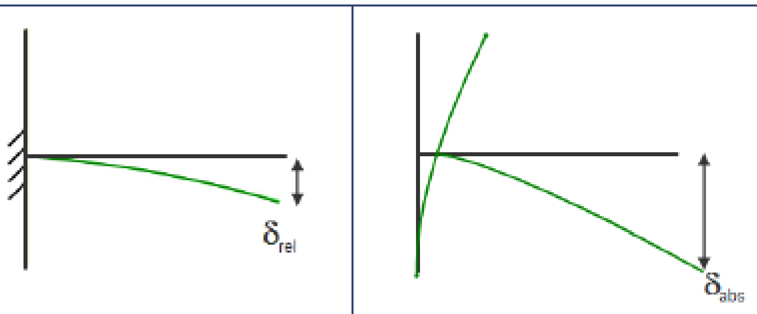
Unless explicitly noted otherwise, all clauses, figures and tables referred to are from AS 4100-1998/Amdt 1-2012 (Ref. 1); apart from the Composite Beam section, within which references are to AS 2327.1-2003 (Ref. 2) unless otherwise stated.

Basic principles (AS 4100)

Deflection checks (AS 4100)

Relative and Absolute Deflections

Tekla Structural Designer calculates both ***relative*** and ***absolute*** deflections. Relative deflections measure the internal displacement occurring within the length of the member and take no account of the support settlements or rotations, whereas absolute deflections are concerned with deflection of the structure as a whole. The absolute deflections are the ones displayed in the structure deflection graphics. The difference between ***relative*** and ***absolute*** deflections is illustrated in the cantilever beam example below.



Relative Deflection	Absolute Deflection

Relative deflections are given in the member analysis results graphics and are the ones used in the member design.

Steel beam design to AS 4100

Design method (Beams: AS 4100)

Unless explicitly stated all calculations are in accordance with the relevant sections of AS 4100 (Ref. 1). You may find the Commentary (Ref. 3) to the Standard published by Standards Australia International useful.

Steel beam limitations and assumptions (Beams: AS 4100)

The following limitations apply:

- continuous beams (more than one span) must be co-linear in the plane of the web within a small tolerance (sloping in elevation is allowed),
- only doubly symmetric prismatic sections (that is rolled or plated I- and H-sections) and channel sections are fully designed,

The following assumptions apply:

- All supports are considered to provide torsional restraint, that is lateral restraint to both flanges. This cannot be changed. It is assumed that a beam that is continuous through the web of a supporting beam or column together with its substantial moment resisting end plate connections is able to provide such restraint.
- If, at the support, the beam oversails the supporting beam or column then the detail is assumed to be such that the bottom flange of the beam is well

connected to the supporting member and, as a minimum, has torsional stiffeners provided at the support.

- In the Tekla Structural Designer model, when not at supports, coincident restraints to both flanges are assumed when one or more members frame into the web of the beam at a particular position and the cardinal point of the center-line model of the beam lies in the web. Otherwise, only a top flange or bottom flange restraint is assumed. Should you judge the actual restraint provided by the in-coming members to be different from to what has been assumed, you have the flexibility to edit the restraints as required.
- Intermediate lateral restraints to the top or bottom flange are assumed to be capable of transferring the restraining forces back to an appropriate system of bracing or suitably rigid part of the structure.
- It is assumed that you will make a rational and “correct” choice for the effective lengths between restraints for both LTB and compression buckling. **The default value for the effective length factor of 1.0 may be neither correct nor safe.**

Ultimate limit state (strength) (Beams: AS 4100)

The checks relate to doubly symmetric prismatic sections (that is rolled and welded I- and H-sections), to singly symmetric sections i.e. Channel sections, and to doubly symmetric hollow sections i.e. CHS, RHS and SHS. Other section types are not currently covered.

The strength checks relate to a particular point on the member and are carried out at regular intervals along the member and at 'points of interest'.

Classification (Beams: AS 4100)

General

The classification of the cross section is in accordance with AS 4100. Beams can be classified for flexure about either principal axis as:

- Compact
- Non-compact
- Slender

Slender sections about either axis will not be designed in Tekla Structural Designer.

All unacceptable classifications are either failed in check mode or rejected in design mode.

Shear capacity (Beams: AS 4100)

The shear check is performed according to AS 4100 Clause 5.11.

For rolled and welded I- and H-sections, and for Channel sections, an approximately uniform shear stress distribution is assumed when calculating major axis shear capacity (to Clause 5.11.2). For these same sections a non-uniform shear stress distribution is assumed when calculating minor axis shear capacity (to Clause 5.11.3), with a shear stress ratio $f^*_{vm} / f^*_{va} = 1.5$

For hollow sections a non-uniform shear stress distribution is assumed when calculating both major and minor axis shear capacity (to Clause 5.11.3). For a CHS section the nominal shear yield capacity is taken per Clause 5.11.4, while RHS and SHS sections assume a shear stress ratio $f^*_{vm} / f^*_{va} = 3 * (2 * b + d) / [2 * (3 * b + d)]$ for major axis shear capacity and $f^*_{vm} / f^*_{va} = 3 * (2 * d + b) / [2 * (3 * d + b)]$ for minor axis shear capacity.

Shear buckling

For rolled and welded I- and H-sections, and for Channel sections, about the major axis, and also for RHS and SHS sections about both axes, when the shear panel depth to thickness ratio exceeds $82/\sqrt{(f_y/250)}$ then the shear buckling capacity will be calculated per Clause 5.11.5.1 assuming an unstiffened shear panel.

Note that for rolled and welded I- and H-sections, and for channel sections, f_y will be taken as the yield strength of the web based on t_w.

Moment capacity (Section) (Beams: AS 4100)

The (section) moment capacity check is performed according to AS 4100 clause 5.1 for the moment about the x-x axis (Mx) and about the y-y axis (My), at the point under consideration.

For (member) moment capacity refer to the section Lateral torsional buckling resistance (Member moment capacity).

Note that for all section types, the effective section modulus about the major axis (Z_{ex}) will be based on the minimum slenderness ratio considering both flange and web. Internally Tekla Structural Designer will calculate the following:

- flange slenderness ratio, $z_f = (\lambda_{ey} - \lambda_{ef}) / (\lambda_{eyf} - \lambda_{epf})$
- web slenderness ratio, $z_w = (\lambda_{eyw} - \lambda_{ew}) / (\lambda_{eyw} - \lambda_{epw})$

For sections which have flexure major class either Compact or Non-compact, the effective section modulus about the major axis (Zex) will then be calculated by:

- $Z_{ex} = Z_x + [MIN(z_f, z_w, 1.0) * (Z_c - Z_x)]$ where $Z_c = MIN(S_x, 1.5 * Z_x)$

Note that for Channel sections under minor axis bending:

- if there is single curvature with the flange tips in compression then Z_{ey} will be based on Z_{eyR}

- if there is single curvature with the web in compression then Z_{ey} will be based on Z_{eyL}
- if there is double curvature then Z_{ey} will be based on the minimum of Z_{eyR} and Z_{eyL}

Combined bending & shear capacity (Section) (Beams: AS 4100)

The combined bending & shear capacity check is performed according to AS 4100 clause 5.12.3, assuming bending is resisted by the whole of the cross-section, for the coincident major shear and moment about the x-x axis (M_x) and minor shear and moment about the y-y axis (M_y), at the point under consideration.

Note that if the (section) moment capacity is found to be less than the design moment then the combined bending & shear check will automatically be set as Fail.

Axial capacity (Section) (Beams: AS 4100)

The (section) axial capacity check is performed according to AS 4100 clause 6.1 for axial compression, or clause 7.1 for axial tension, using the gross cross-section area for A_n in both cases.

Note that member (axial compression) capacity is a buckling check and as such is considered under the heading Compression buckling

Ultimate limit state (buckling) (Beams: AS 4100)

Lateral torsional buckling resistance (Member moment capacity) (Beams: AS 4100)

For beams with major axis bending, a Lateral torsional buckling (LTB) check is required, except in the following circumstances:

- when the segment critical flange is continuously restrained for LTB, or
- when bending exists about the minor axis only, or
- when the section is a CHS, or
- when the segment length satisfies the relevant limit given in clause 5.3.2.4 of AS 4100

In the latter case, when calculating the limiting LTB length, the ratio β_m will be taken as -0.8 if the segment has major axis bending induced by transverse load within its length, and the ratio of end moments otherwise.

The LTB resistance (member moment capacity) check is performed according to AS 4100 clause 5.6

Note that the moment modification factor α_m will be calculated from the equation given in AS 4100 clause 5.6.1.1 (a) (iii) except for cantilevers, where

α_m will be 0.25 if the free end moment is greater than the ignore forces major moment, and 1.0 otherwise.

The twist restraint factor k_t will be determined by consideration of the LTB cross-section restraints at either end of the segment, per Table 5.6.3(1) of AS 4100

The load height factor k_l will default to 1.4 for a non-cantilever and 2.0 for a cantilever.

The lateral rotation factor k_r will default to 1.0.

Compression buckling resistance (Member capacity under axial compression) (Beams: AS 4100)

For most structures, all the members resisting axial compression need checking to ensure adequate resistance to buckling about both the major and minor axis. Since the axial force can vary throughout the member and the strut buckling lengths in the two planes do not necessarily coincide, both axes are checked. Because of the general nature of a beam-column, it may not always be safe to assume that the combined actions check will always govern. Hence the compression resistance check is performed independently from the other strength and buckling checks.

The compression buckling resistance (member capacity under axial compression) check is performed according to AS 4100 clause 6.3

The default value of effective length factor is 1.0. Different values can apply in the major and minor axis. Beams are less affected by sway than columns but the effectiveness of the incoming members to restrain the beam in both position and direction is generally less than for columns. Hence, it is less likely that effective length factors greater than 1.0 will be required but equally factors less than 1.0 may not easily be justified. Nevertheless, it is your responsibility to adjust the value from 1.0 and to justify such a change.

Combined actions resistance (Beams: AS 4100)

NOTE Important Note. Clause 8.2 of AS 4100 defines the design bending moments to be used in the combined actions checks as either amplified moments from a first order linear elastic analysis or the moments resulting directly from a second order elastic analysis. Tekla Structural Designer will not provide amplified moments from a first order linear elastic analysis and you are expected to switch to second order analysis to complete the design for combined actions.

Combined actions resistance - Section capacity (Beams: AS 4100)

The combined actions section capacity check is performed according to AS 4100 clause 8.3

The higher tier equations will be used automatically if the conditions for their use are met.

Note that if the design axial force exceeds the design axial section capacity then the check will automatically be set as Fail.

In the section capacity check, the design forces are those which are coincident at any one point along the member.

Combined actions resistance - Member capacity (Beams: AS 4100)

The combined actions member capacity check is performed according to AS 4100 clause 8.4

The higher tier equations will be used automatically if the conditions for their use are met.

In the higher tier equation for M_i, the ratio β_m will be based on the relevant strut length; if the strut length has bending induced by transverse load within its length then β_m will be taken as -1.0, and the ratio of end moments otherwise.

In the higher tier equation for M_{ox}, the ratio β_m will be based on the LTB segment length, and taken as the ratio of end moments.

Note that if the design axial force exceeds the design axial member capacity then the check will automatically be set as Fail.

In the member capacity check, the design forces are the maxima in the design length being considered, where the design lengths are based on the major and minor strut lengths within a loop of LTB lengths.

Therefore, since any one design length will comprise both major and minor strut lengths, the design axial force for each design length will be taken as the maximum axial compression or axial tension force from the major and minor strut lengths considered together.

Since both axial compression and axial tension are to be considered, but make use of different equations, then in cases where both axial forces exist within a design length the compression equations and tension equations will both be evaluated and the worst case of the two will be reported.

Note that in bi-axial bending cases, zero axial force will be treated as compression.

Web openings (Beams: AS 4100)

The checks for beams with web openings are not included in this release.

Serviceability limit state (Beams: AS 4100)
Beams are assessed for deflection. Only the total load deflection is active by default, with a span/over value assigned of 250 per Table B1 of AS 4100.

Composite beam design to AS 2327.1
The design of composite beams is not included in this release.

Steel column design to AS 4100

Design method (Columns: AS 4100)

Unless explicitly stated all calculations are in accordance with the relevant sections of AS 4100 (Ref. 1). You may find the Commentary (Ref. 3) to the Standard published by Standards Australia International useful.

Ultimate limit state (strength) (Columns: AS 4100)

The checks relate to doubly symmetric prismatic sections (that is rolled and welded I- and H-sections), to singly symmetric sections i.e. Channel sections, and to doubly symmetric hollow sections i.e. CHS, RHS and SHS. Other section types are not currently covered. The strength checks relate to a particular point on the member and are carried out at regular intervals along the member and at 'points of interest'.

Hollow sections

The checks for CHS, RHS and SHS relate to “hot-finished hollow sections” only - “cold-formed hollow sections” are not included in this release.

Classification (Columns: AS 4100)

The flexural classification of the cross section is in accordance with AS 4100 Columns can be classified for flexure about either principal axis as:

- Compact
- Non-compact
- Slender

Slender sections about either axis will not be designed in Tekla Structural Designer.

All unacceptable classifications are either failed in check mode or rejected in design mode.

Shear capacity (Columns: AS 4100)

The shear check is performed according to AS 4100 clause 5.11.

For rolled and welded I- and H-sections, and for channel sections, an approximately uniform shear stress distribution is assumed when calculating major axis shear capacity (to clause 5.11.2). For these same sections a non-uniform shear stress distribution is assumed when calculating minor axis shear capacity (to clause 5.11.3), with a shear stress ratio $f^*_{vm} / f^*_{va} = 1.5$

For hollow sections a non-uniform shear stress distribution is assumed when calculating both major and minor axis shear capacity (to clause 5.11.3). For a CHS section the nominal shear yield capacity is taken per clause 5.11.4, while RHS and SHS sections assume a shear stress ratio $f^*_{vm} / f^*_{va} = 3 * (2 * b + d) / [2 * (3 * b + d)]$ for major axis shear capacity and $f^*_{vm} / f^*_{va} = 3 * (2 * d + b) / [2 * (3 * d + b)]$ for minor axis shear capacity.

Shear buckling

For rolled and welded I- and H-sections, and for channel sections, about the major axis, and also for RHS and SHS sections about both axes, when the shear panel depth to thickness ratio exceeds $82/\sqrt{(f_y/250)}$ then the shear buckling capacity will be calculated per clause 5.11.5.1 assuming an unstiffened shear panel.

Note that for rolled and welded I- and H-sections, and for channel sections, f_y will be taken as the yield strength of the web based on t_w.

Moment capacity (section) (Columns: AS 4100)

The (section) moment capacity check is performed according to AS 4100 clause 5.1 for the moment about the x-x axis (M_x) and about the y-y axis (M_y), at the point under consideration.

For (member) moment capacity refer to the section Lateral torsional buckling resistance (Member moment capacity).

Note that for all section types, the effective section modulus about the major axis (Z_{ex}) will be based on the minimum slenderness ratio considering both flange and web. Internally Tekla Structural Designer will calculate the following:

- flange slenderness ratio, $z_f = (\lambda_{ey} - \lambda_{ef}) / (\lambda_{eyf} - \lambda_{epf})$
- web slenderness ratio, $z_w = (\lambda_{eyw} - \lambda_{ew}) / (\lambda_{eyw} - \lambda_{epw})$

For sections which have flexure major class either Compact or Non-compact, the effective section modulus about the major axis (Z_{ex}) will then be calculated by:

- $Z_{ex} = Z_x + [MIN(z_f, z_w, 1.0) * (Z_c - Z_x)]$ where $Z_c = MIN(S_x, 1.5 * Z_x)$

Note that for Channel sections under minor axis bending:

- if there is single curvature with the flange tips in compression then Z_{ey} will be based on Z_{eyR}
- if there is single curvature with the web in compression then Z_{ey} will be based on Z_{eyL}
- if there is double curvature then Z_{ey} will be based on the minimum of Z_{eyR} and Z_{eyL}

Eccentricity Moments

Eccentricity moment will be added algebraically to the coincident real moment (at top or bottom of column stack) only if the resulting 'combined' moment has a larger absolute magnitude than the absolute real moment alone.

The resulting 'combined' design moment (major and/or minor) will be that used in moment capacity, combined bending & shear, LTB, and combined actions checks.

Combined bending and shear capacity (section) (Columns: AS 4100)

The combined bending and shear capacity check is performed according to AS 4100 clause 5.12.3, assuming bending is resisted by the whole of the cross-section, for the coincident major shear and moment about the x-x axis (M_x) and minor shear and moment about the y-y axis (M_y), at the point under consideration. The design moments may include eccentricity moments - see Moment capacity (section): Eccentricity Moments.

Note that if the (section) moment capacity is found to be less than the design moment then the combined bending and shear check will automatically be set as Fail.

Axial capacity (section) (Columns: AS 4100)

The (section) axial capacity check is performed according to AS 4100 clause 6.1 for axial compression, or Clause 7.1 for axial tension, using the gross cross-section area for A_n in both cases.

Note that member (axial compression) capacity is a buckling check and as such is considered under the heading Compression buckling resistance (Member capacity under axial compression).

Ultimate limit state (buckling) (Columns: AS 4100)

Lateral torsional buckling resistance (Member moment capacity) (Columns: AS 4100)

For beams with major axis bending, a Lateral Torsional Buckling (LTB) check is required, except in the following circumstances:

- when the segment critical flange is continuously restrained for LTB, or
- when bending exists about the minor axis only, or
- when the section is a CHS, or
- when the segment length satisfies the relevant limit given in clause 5.3.2.4 of AS 4100

In the latter case, when calculating the limiting LTB length, the ratio β_m will be taken as -0.8 if the segment has major axis bending induced by transverse load within its length, and the ratio of end moments otherwise.

The LTB resistance (member moment capacity) check is performed according to AS 4100 clause 5.6

Note that the moment modification factor α_m will be calculated from the equation given in AS 4100 clause 5.6.1.1 (a) (iii) except for cantilevers, where α_m will be 0.25 if the free end moment is greater than the ignore forces major moment, and 1.0 otherwise.

The design moment may include eccentricity moment - see Moment Capacity (Section): Eccentricity Moments - but note in particular that the ratio β_m and the moment modification factor α_m will be based on real moments only.

The twist restraint factor k_t will be determined by consideration of the LTB cross-section restraints at either end of the segment, per Table 5.6.3(1) of AS 4100

The load height factor k_l will default to 1.4 for a non-cantilever and 2.0 for a cantilever.

The lateral rotation factor k_r will default to 1.0.

Compression buckling resistance (Member capacity under axial compression) (Columns: AS 4100)

For most structures, all the members resisting axial compression need checking to ensure adequate resistance to buckling about both the major and minor axis. Since the axial force can vary throughout the member and the strut buckling lengths in the two planes do not necessarily coincide, both axes are checked. Because of the general nature of a beam-column, it may not always be safe to assume that the combined actions check will always govern. Hence the compression resistance check is performed independently from the other strength and buckling checks.

The compression buckling resistance (member capacity under axial compression) check is performed according to AS 4100 clause 6.3

The default value of effective length factor is 1.0 Different values can apply in the major and minor axis. Beams are less affected by sway than columns but the effectiveness of the incoming members to restrain the beam in both position and direction is generally less than for columns. Hence, it is less likely that effective length factors greater than 1.0 will be required but equally factors less than 1.0 may not easily be justified. Nevertheless, it is your responsibility to adjust the value from 1.0 and to justify such a change.

Combined actions resistance (Columns: AS 4100)

NOTE important Note. Clause 8.2 of AS 4100 defines the design bending moments to be used in the combined actions checks as either amplified moments from a first order linear elastic analysis or the moments resulting directly from a second order elastic analysis. Tekla Structural Designer will not provide amplified moments from a first order linear elastic analysis and you are expected to switch to second order analysis to complete the design for combined actions.

Combined actions resistance - Section capacity (Columns: AS 4100)

The combined actions section capacity check is performed according to AS 4100 clause 8.3

The higher tier equations will be used automatically if the conditions for their use are met.

Note that if the design axial force exceeds the design axial section capacity then the check will automatically be set as Fail.

In the section capacity check, the design forces are those which are coincident at any one point along the member.

Combined actions resistance - Member capacity (Columns: AS 4100)

The combined actions member capacity check is performed according to AS 4100 clause 8.4

The higher tier equations will be used automatically if the conditions for their use are met.

In the higher tier equation for M_i, the ratio β_m will be based on the relevant strut length; if the strut length has bending induced by transverse load within its length then β_m will be taken as -1.0, and the ratio of end moments otherwise.

In the higher tier equation for M_{ox}, the ratio β_m will be based on the LTB segment length, and taken as the ratio of end moments, using real moments only.

In the member capacity check, the design forces are the maxima in the design length being considered, where the design lengths are based on the major and minor strut lengths within a loop of LTB lengths.

Therefore, since any one design length will comprise both major and minor strut lengths, the design axial force for each design length will be taken as the maximum axial compression or axial tension force from the major and minor strut lengths considered together.

Note that if the design axial force exceeds the design axial member capacity then the check will automatically be set as Fail.

Since both axial compression and axial tension are to be considered, but make use of different equations, then in cases where both axial forces exist within a design length the compression equations and tension equations will both be evaluated and the worst case of the two will be reported.

Note that in bi-axial bending cases, zero axial force will be treated as compression.

Serviceability limit state (Columns: AS 4100)

The column is assessed for sway and the following values are reported for each stack:

- Sway X (mm) and λ_{critx}
- Sway Y (mm) and λ_{crity}
- Twist i.e. Sway X-Y (non-dimensional ratio)

Depending on the reported λ_{crit} the column is classified as Sway or Non sway accordingly.

NOTE A sway assessment is only performed for the column if the Lambda Crit Check box is checked on the Column Properties dialog.

If very short columns exist in the building model these can distort the overall sway classification for the building. For this reason you may apply engineering judgement to uncheck the Lambda Crit Check box for those columns for which a sway assessment would be inappropriate

---

Steel brace design to AS 4100

Design method (Braces: AS4100)

Unless explicitly stated all brace calculations are in accordance with the relevant sections of AS 4100 (Ref. 1).

A basic knowledge of the design methods for braces in accordance with the design code is assumed.

Hollow sections

The checks for CHS, RHS and SHS relate to “hot-finished hollow sections” only - “cold-formed hollow sections” are not included in this release.

Classification (Braces: AS 4100)

No classification is required for braces.

Axial capacity (section) (Braces: AS 4100)

The (section) axial capacity check is performed according to AS 4100 clause 6.1 for axial compression, or clause 7.1 for axial tension, using the gross cross-section area for A_n in both cases.

Note that member (axial compression) capacity is a buckling check and as such is considered under the heading Compression Buckling.

Compression buckling resistance (Member capacity under axial compression) (Braces: AS 4100)

The compression buckling resistance (member capacity under axial compression) check is performed according to AS 4100 clause 6.3

The default effective length factor in each axis is 1.0

References (AS 4100)

1. **Standards Australia International**. AS 4100-1998/Amdt 1-2012: Steel structures. **SAI 2012**.
2. **Standards Australia International**. AS 2327.1-2003: Composite structures. Part 1: Simply supported beams. **SAI 2003**.
3. **Standards Australia International**. AS 4100 Supp1-1999: Steel structures – Commentary. (Supplement to AS 4100-1998). **SAI 1999**.
4. **Standards Australia International/Standards New Zealand**. AS/NZS 1170.0:2002 (Including Amendments Nos.1,2,4 and 5). Structural design actions. Part 0: General principles. **SAI/NZS 2011**.
5. **Standards Australia International/Standards New Zealand**. AS/NZS 1170.1:2002 (Including Amendments Nos.1 and 2). Structural design actions. Part 1: Permanent, imposed and other actions. **SAI/NZS 2009**.
6. **Standards Australia International/Standards New Zealand**. AS/NZS 1170.2:2011 (Including Amendments Nos.1 and 2). Structural design actions. Part 2: Wind actions. **SAI/NZS 2012**.

15 Tekla Structural Designer reference

This section contains reference information mainly on settings, options, and sign conventions, but with with various other miscellaneous reference topics included also.

15.1 Properties

This section describes properties of the main object types.

Structure geometric/analytical properties:

Member properties:

Other object properties:

Structure Properties

Use the **Structure** properties to view or modify the basic properties of the structure.

Property	Description
Building Direction Rotation	Defines the building direction relative to the Global Axis System The default (0 degrees) aligns the building direction 1 arrow with the global X axis and the direction 2 arrow with the global Y axis. Entering a positive value rotates the Building Direction arrows clockwise about positive Global Z, a negative value rotates anti-clockwise. The limiting values are +45 degrees and -45 degrees. (If you enter larger values they will be capped at these limits). The building direction arrows are always at 90 degs to each other.
Show Building Arrows	When shown, building direction arrows are displayed in all 2D and 3D Views.
Building Direction Labels	The labels to be used for the building direction arrows. The options are: • Dir 1/2 • Dir H/V • Dir X/Y
Consequence class	Defines k_{FI} used in strength combinations.

Property	Description
	• CC3 - k_{FI} = 1.1 • CC2 - k_{FI} = 1.0 • CC1 - k_{FI} = 0.9 **NOTE** Applies to **Finland (Eurocode)** head code only
Reliability class	Defines γ_d used in strength combinations. • RC3 - γ_{FI} = 1.0 • RC2 - γ_{FI} = 0.91 • RC1 - γ_{FI} = 0.83 **NOTE** Applies to **Sweden (Eurocode)** head code only
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs. **NOTE** To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.
Shell Uniformity Factor	Defines the shell mesh uniformity for two way spanning slabs. (100% = maximum uniformity).
Semi-Rigid Mesh Size	Defines the mesh size for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Uniformity Factor	Defines the semi-rigid mesh uniformity for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type for roof panels, and slabs when modeled as semi-rigid diaphragms The options are: • QuadDominant • QuadOnly • Triangular
Wall Mesh Horizontal Size	Defines the horizontal mesh size for all meshed walls in the model - but can be overridden in individual wall properties.
Wall Mesh Vertical Size	Defines the vertical mesh size for all meshed walls in the model - but can be overridden in individual wall properties.

Property	Description
Wall Mesh Type	Defines the mesh type for all meshed walls in the model, (but can be overridden in individual wall properties): The options are: • Quad only • Tri only • Quad dominant

Level Properties

Use the **Level** properties to view or modify the basic properties of a level.

General	
Level	The height of the construction level above the base level
Floor	By setting a construction level to be a Floor you are indicating that it is a major level in the building. Floor levels are used to determine items such as your inter story height and positions from which column splices are laid out. If a level is not set to be a floor then no live load reductions will be accounted for in the beams at that level, or in the columns supporting that level. There can certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs, • K Bracing - you require a construction level for the intermediate bracing connection points, • steps in the building floor levels. Where you define a level which is clearly not a floor, then you should not check the floor box.
Type	The level type can be: • T.O.S = Top of Steel • S.S.L = Structural Slab Level • T.O.F = Top of Foundation **NOTE** Slabs are modeled above the level when it is set to T.O.S or T.O.F but below the level when it is set to S.S.L
Short Name	Each construction level should be given a unique reference. Typically this might be a storey number, 1, 2 , 3 etc.

<table>
<tr><th colspan="2">General</th></tr>
<tr><td>Long Name</td><td>Each construction level can also be given a name to further assist identification. 'First Floor', or 'Mezzanine' etc.</td></tr>
<tr><td>Name</td><td>Automatically generated from the short and long name. By default this will be used as the name in the Structure tree</td></tr>
<tr><td>Mesh 2-way Slabs in 3D Analysis</td><td>• On = 2-way slabs are meshed in the 3D building analysis and grillage chasedown analysis in addition to the FE chasedown analysis.
• Off = 2-way slabs are only meshed in the FE chasedown analysis.

NOTE The Sub Model in which the Level is contained determines the mesh parameters that are applied.</td></tr>
<tr><td>Include in Export</td><td>Determine whether Level should be included in CXL or IFC export.
• Floor Only - include only if defined as a Floor.
• Yes - always include.
• No - never include.

NOTE The lowest Level will always be exported.</td></tr>
<tr><td>Show grids in plane view</td><td>Controls whether the Grid & Construction Lines checkbox in Scene Content has any effect in plane views:
• On = The Scene Content Grid & Construction Lines checkbox switches grid & construction lines on and off
• Off = Grid & construction lines are always off, irrespective of the Scene Content Grid & Construction Lines checkbox setting</td></tr>
<tr><td>Show grids in 3D view</td><td>Controls whether the Grid & Construction Lines checkbox in Scene Content has any effect in 3D views:
• On = The Scene Content Grid & Construction Lines checkbox switches grid & construction lines on and off for the specified level
• Off = Grid & construction lines are always off for the specified level, irrespective of the Scene Content Grid & Construction Lines checkbox setting

NOTE This setting does not control the display of Architectural Grids which have a separate checkbox in Scene Content.</td></tr>
</table>

General	
Decomposition> Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between analyses: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Decomposition	
Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between analyses: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Frame Properties

Use the **Frame** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified frame

Property	Description
	• Off = Grid & construction lines are always off for the specified frame irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the frame view can be opened or not.

Slope Properties

Use the **Slope** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified slope • Off = Grid & construction lines are always off for the specified slope irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the slope view can be opened or not.

Sub Model Properties

Use the **Sub Models** properties to view or modify the selected sub model.

Property	Description	
Override model's	Select this check box in order to override the Structure meshing properties in the current sub model.	
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs in the sub model.	
Shell Uniformity Factor	Defines the shell mesh uniformity factor for two way spanning slabs in the sub model.	
Slab Mesh Type	Defines the shell mesh type used in the sub model. These options are: • Quad only • Tri only • Quad dominant	
Semi-Rigid Mesh Size	Defines the semi-rigid mesh size when slabs are modeled as semi-rigid diaphragms in the sub model.	
Semi-Rigid Uniformity Factor	Defines the semi-rigid uniformity factor when slabs are modeled as semi-rigid diaphragms in the sub model.	
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type when slabs are modeled as semi-rigid diaphragms in the sub model. The options are: • Quad only • Tri only • Quad dominant	

Beam properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: Concrete member design and detailing groups (page 1281) , Steel member design groups (page 1206)

General	
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active	Clearing this option makes single span beams inactive in the analysis. See: Inactive members (page 482) **NOTE** Only displayed for single span members
Material type	• Steel • Concrete • Timber • General • Cold formed • Cold rolled
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 2110)
Autodesign	See: Concrete member autodesign (page 1280), Steel member autodesign (page 1205)
Design section order (steel only)	The design order file from which a section size will be selected. **NOTE** Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 1006)
Select bars starting from (concrete only)	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: Concrete member autodesign (page 1280), Steel member autodesign (page 1205)
Gravity only	Controls whether the member is defined for gravity combinations only, or gravity plus lateral: • On = designed for gravity combinations only

General	
	• Off = designed for gravity and lateral combinations See: Designing individual members for gravity only (page 1204)
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 422)
Major snap level, Minor snap level (not concrete)	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset (not concrete)	Used to offset the member from the snap point in the major and minor axis.
Allow automatic join end 1, end 2 (concrete only)	When this check box is selected - the end in question will be automatically joined to a suitable connecting concrete beam end during design process or when the 'Beam Lines' command is run, (providing the Beam Lines limiting criteria specified in Model Settings are met.)

All spans	
Section	The section size
Concrete type, Grade/Class	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line

All spans	
	elements that replace the curved member in the solver model. See: Analysis Model settings (page 2271)
Top flange cont. rest. (steel and composite beams only)	Define if the top flange is continuously restrained.
Alignment (concrete only)	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Free end 1, end 2	When this check box is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam releases (page 2084)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when ‘Fixity’ is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed

All spans	
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Overview of load reductions (page 1172)
Reduce imposed loads by (All other Head Codes)	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of load reductions (page 1172)
Deflection limits (steel only)	
Apply span \factor	With this option checked, the limit can be defined as a Relative span/factor.
Apply abs. limit	With this option checked, the limit can be defined as absolut value.

All spans	
Limit for immediate live load deflection, Limit for total deflection affecting sensitive finishes Calculate total deflection at design time Calculate deflection after installation of finishes (ACI only)	These options control how the deflection is calculated.
Torsion (steel only)	
Check for torsion, Apply rotational limit	Used to specify if the member should be checked for torsion, and also to apply a rotational limit if required. See: Steel beam torsion (page 1223)
Camber (steel only)	
Apply camber to this beam	Used to specify a camber to the beam if required. See: Camber (page 1217)
Natural frequency (non-composite and composite beams only)	
Calculate natural frequency	For composite beams this is fixed to on (for EC and BS headcodes only) and is utilized during autodesign. The calculated natural frequency is displayed in the results viewer.
Check natural frequency against minimum	For composite beams this is fixed to on (for EC and BS headcodes only). For non-composite beams it can be used in conjunction with the 'Calulate natural frequency' property to impose an optional 2-step control, ‘calculate’ and ‘check’, which allows for the natural frequency value to be calculated (and displayed in Results Viewer) with or without the check. While the US headcode has no requirement for this check, it can still be requested if required. When performed, a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse

All spans	
	particularly for long span beams and does not consider the response side of the behaviour i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer has the option to perform a 1st Order Modal Analysis. See: • Natural frequency checks (SLS) (Beams: EC4 Eurocode) (page 1900) • Natural frequency checks (SLS) (Beams: BS 5950) (page 2021)
Minimum natural frequency	The minimum value against which the natural frequency is checked (default 4Hz).
Include self weight (beam & slab) Include other dead loads Include live loads (US) Include imposed loads (other headcodes)	The engineer can specify the percentages of each of these loads to be included in the calculation of the maximum static instantaneous deflection, δ
Factor for increased dynamic stiffness of the concrete flange	For composite beams this factor is applied to the beam's short term modular ratio
Effective width Calculate effective width	For composite beams the effective width can either be entered directly or calculated from the geometry. See: • Effective width used in the design - Head Code: ACI/AISC (page 1230) • Effective width used in the design - Head Code: Eurocode (page 1232) • Effective width used in the design - Head Code: BS (page 1234)
Size constraints (steel only)	

All spans	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam. See: Size constraints (page 1205)
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See: Size constraints (page 1205)
Instability factor (steel only)	
Prevent out of plane instability	Define if out of plane stability is prevented. See: Instability factor (page 1217)
Design control (concrete only)	
Structure supporting sensitive finishes (ACI and Eurocode only)	Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting this property.
	ACI - The deflection method applied to the beam depends on this setting as follows: • beams not required to support sensitive finishes adopt the simplified method. • beams required to support sensitive finishes adopt the rigorous method.
	Eurocode - The f2 parameter used in the deflection check depends on this parameter as follows: • When selected: f2 is calculated as MIN[1,7/L_{eff}] • When unselected: f2 will be taken as 1.0.
Increase reinforcement if deflection check fails (Eurocode BS and IS only)	Select in order to increase the reinforcement during the auto-design process if the deflection check fails.

All spans	
Permissible increase in reinforcement	Specify the max percentage increase in reinforcement that is allowed in order to satisfy the deflection check.
Consider flanges	Select in order to consider flanges in the concrete beam design calculations - once checked additional fields are displayed for specifying an allowance for openings. Flange dimensions can only be calculated by editing the beam once it has been positioned and slabs have been defined. (In this case a 'Calculate flanges' button is also displayed, this can be clicked in order to automatically calculate the flange dimensions based on the adjoining slabs.) See: Flanged concrete beams (page 1297)
Include flanges in analysis	**NOTE** This property is only displayed when the 'Consider flanges' option has been selected. Select this check box to use flanged beam properties when the analysis is performed. See: Flanged concrete beams (page 1297)
Isolated beam (ACI only)	**NOTE** This property is only displayed when the 'Consider flanges' option has been selected. Select this check box in order to apply ACI 318 clause 8.12.4. When the check is performed, if the flange geometry does not meet the requirements the flanges are ignored. See: Flanged concrete beams (page 1297)
User defined flange (left/ right)	**NOTE** This property is only displayed when 'Include flanges in analysis' has been checked. If you clear this check box the flange depth and the effective flange width are determined automatically. If you select the check box, two new properties are displayed for defining 'Flange width' and 'Flange depth'. See: Flanged concrete beams (page 1297)
Ignore lateral instability (Eurocode only)	This option allows you to ignore lateral instability for slender spans to EC2 clause 5.9(1). • When selected: the slender span check is excluded from design

All spans	
	• When unselected (default): the slender span check is included.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.
Design parameters (concrete only)	
Nominal cover beam top edge, bottom edge, section side, beam ends	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.Different values of nominal cover can be specified to the beam edges, sides and ends.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Fire check (steel only, Eurocode only)	
Check for fire resistance	**NOTE** This property is only displayed for non-composite, simply supported rolled steel beams to Eurocodes. On • When the check is requested, the additional properties shown below are required. See: Fire check (Eurocode only) (page 1223) Off • no check is performed
Load reduction factor	0.65, or user input value.
Required time of fire exposure	R15, R30, R60, R90, or R120
Exposure	Exposed on three, or four sides

Protected	On
	• When the protected option is selected, the following fire protection material details are required:
	  • Shape - Contour, or Hollow encasement
	  • Thickness
	  • Thermal conductivity
	  • Specific heat
	  • Density
	• The check is performed using the 'protected' time interval for critical temperature iteration, as specified in Design Settings
	Off
	• The check is performed using the 'unprotected' time interval for critical temperature iteration, as specified in Design Settings

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Reinforcement (concrete beams only)	
Rib type - vertical, Class - longitudinal	Specifies the longitudinal reinforcement properties
Rib type - link, Class - link	Specifies the link properties.
Top longitudinal bar pattern, Bottom longitudinal bar pattern	Choose from standard patterns (which can be setup in Design Options) to control the top and bottom bar arrangement when the beam is auto-designed. See: Concrete beam design aspects (page 1289)

Span 1, 2, 3 etc.	
	In a multi-span beam properties can be entered for a specific span, over-riding those defined at the **All spans** level.

Beam releases

Releases at the two ends of a beam span can be set as follows:

- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fully fixed** - with the following major (My), and minor (Mz) **stiffness** sub-options to allow definition of partial fixity in each direction if required:
  - **Fixed** - Encastré, all degrees of freedom fixed.
  - **Spring linear** - allows you to specify a linear spring stiffness value.
  - **Partially fixed** - allows you to specify partial fixity as a percentage of a fully fixed connection (%4EI/L)
- **Moment (pin Mz)** - Major axis moment connection, and pinned about the minor axis.
- **Moment (pin My)** - Minor axis moment connection, and pinned about the major axis.
- **Fully fixed (free end)** - Denotes a cantilever end. It is achieved by selecting Free end.

  (In a single span beam this box can only be checked if the opposite end is fully fixed.)
- **Continuous** - This setting is automatically applied when a continuous beam is created and effectively creates a non-editable fully fixed connection between the spans of the continuous member. The connection can only be edited by splitting the beam.

In addition to the above release options you are also able to apply a torsional release at either end by checking the appropriate box. Similarly an axial release can be applied to beams of all materials apart from concrete.

NOTE **Moment (pin My)** this (unusual) release type is not available in the Properties Window and can only be specified as follows:

1. Right-click the beam to display the context menu.
2. Choose Edit
3. From the Beam Property Dialog open the Releases page.
4. Check the Mz and uncheck the My degree of freedom at the desired end as required.

Brace properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: Steel member design groups (page 1206)
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes the brace inactive in the analysis. See: Inactive members (page 482)
Material type	• Steel • Timber • General • Cold formed
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 2110)
Autodesign	See: Concrete member autodesign (page 1280), Steel member autodesign (page 1205)
Design section order (steel only)	The design order file from which a section size will be selected.

General	
	NOTE Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 1006)
Rotation	Rotation of the member about its local x axis.
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 422)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

All spans	
Section	The section size
Grade	The material grade
Connection	• Bolted • Welded
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor	Only displayed if the brace is tension only (default 0.0). Any value entered is not used unless you also check the ‘Use relaxation factors for tension only elements’ box in Analysis Options.

All spans	
	NOTE It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis Settings (page 2278)
Releases	
Fixity end 1, end 2	• Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Include force in eccentricity moment end 1, end 2	Eccentricity moments in steel and precast columns as a result of beam connection eccentricities do not also consider the brace connection eccentricities unless this property is checked for the appropriate end of the brace. • When checked ON the brace axial force is resolved into a vertical force, multiplied by the connection eccentricity, to obtain a connection eccentricity moment. • When checked OFF the brace axial force does not contribute to the total connection eccentricity moment. See: • Steel column connection eccentricity moments - overview (page 1256) • Precast column connection eccentricity moments - overview (page 1558)

<table>
<tr><th colspan="2">All spans</th></tr>
<tr><td colspan="2">Load reductions</td></tr>
<tr><td>KLL

(Head Code ACI/AISC)</td><td>Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10.

See: Overview of load reductions (page 1172)</td></tr>
<tr><td>Reduce imposed loads by

(All other Head Codes)</td><td>Although the percentage of imposed load reduction is not determined automatically, this property allows you to specify the percentage manually.

• reducible loadcases are reduced

• combinations incorporating reducible loadcases are reduced

See: Overview of load reductions (page 1172)</td></tr>
<tr><td colspan="2">Compression</td></tr>
<tr><td>Effectve length factor y-y,

z-z</td><td></td></tr>
<tr><td colspan="2">Tension</td></tr>
<tr><td>Net area</td><td>Specified as an effective net area or a percentage value.</td></tr>
<tr><td colspan="2">Size constraints (steel only)</td></tr>
<tr><td>Max depth,

Min depth,

Max width,

Min width</td><td>Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.</td></tr>
<tr><td colspan="2">Seismic</td></tr>
<tr><td>In a seismic force resisting system</td><td>If this is the case, select the check box, and then specify the SFRS direction and type.

NOTE Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.</td></tr>
</table>

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Column properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: Concrete member design and detailing groups (page 1281) , Steel member design groups (page 1206)
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.

General	
Plane	Indicates the grid along which the wall is placed.
Characteristic	Column
Material type	• Steel • Concrete • Timber • General • Cold formed • Cold rolled
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 2110)
Autodesign	See: Concrete member autodesign (page 1280), Steel member autodesign (page 1205)
Design section order (steel only)	The design order file from which a section size will be selected. **NOTE** Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 1006)
Select bars starting from (concrete only)	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: Concrete member autodesign (page 1280), Steel member autodesign (page 1205)
Gravity only	Controls whether the member is defined for gravity combinations only, or gravity plus lateral: • On = designed for gravity combinations only • Off = designed for gravity and lateral combinations See: Designing individual members for gravity only (page 1204)
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column:

General	
	• 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Assume extra floors supported	Enter the number of extra floors supported.
Simple column (steel only)	See: Simple columns (page 1251)

All stacks	
Section	The section size
Concrete type, Grade/Class (Concrete columns only)	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Alignment (concrete only)	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Face A cont. rest., Face C cont. rest. (steel only)	Indicates continuous restraint to Face A and/or Face C. See: Steel column restraints (page 1255)
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 2097)
Axial load release top	An axial (Fx) release is only allowed at the top of the column.
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release.
Size constraints (steel only)	
Max depth,	Size Constraints are only applicable when Autodesign is checked. They

All stacks	
Min depth, Max width, Min width	allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.
Instability factor (steel only)	
Prevent out of plane instability	Define if out of plane stability is prevented. See: Instability factor (page 1217)
Design control (concrete only)	
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/ drift sensitivity calculations are also influenced by this assumption.
Apply rigid zones	Unless cleared, rigid zones are automatically created at the connection between the column and any connecting beams.
Design parameters (Eurocode only)	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See: Design parameters (Eurocode only) (page 2096)
Confinement reinforcement (concrete only)	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness (concrete only)	

All stacks	
Direction 1, Direction 2	• Braced • Bracing
Direction 1 effective length factor, Direction 2 effective length factor	• Calculated • User input value
Stiffness (concrete only)	
Use slab for calculation upper, lower	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See: Stiffness (page 1300)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing

All stacks	
	the check box located under that stack only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don’t want this check to be performed you can either clear the check box located under ‘All Stacks’ to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/shear capacity ratio in Dir 1 Shear demand/shear capacity ratio in Dir 2 (ASCE7 code only)	When working to the ASCE7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Reinforcement	
Rib type - vertical, Class - vertical	Specifies the vertical reinforcement properties
Rib type - link/ confinement, Class - link/ confinement	Specifies the link/confinement properties.

Stack 1, 2, 3 etc.	
	In a multi-stack column properties can be entered for a specific stack, over-riding those defined at the **All stacks** level.

Stack 1, 2, 3 etc.	
Splice, Splice offset	Used to indicate a splice at the bottom of a stack, and the offset above the floor level. See: Splice and splice offset (page 1261)

Level 1, 2, 3 etc.	
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Overview of load reductions (page 1172)
Count the floor as being supported (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.
Eccentricity (steel only)	
Apply face A eccentricity, etc.	See: Steel column connection eccentricity moments (page 1256)

Design parameters (Eurocode only)

Located under the Design parameters heading in the column properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Column releases

Releases in columns can be set as follows:

Moment Releases

- **Pinned** - Pinned to the adjacent stack and connecting members. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fixed** - fully fixed
- **Moment (pin My)** - Pinned about the y axis only
- **Moment (pin Mz)** - Pinned about the z axis only.

NOTE While **Moment (pin My)** and **Moment (pin Mz)** releases are less usual, for continuous columns composed of two or more stacks, they have been provided in response to customer feedback, to allow any/all column stacks to be pinned in one direction whilst the other direction remains fixed.

When such a release is applied, a Warning status is issued in the Column Property dialog. This is intentional and does not prevent analysis or design -

as the Warning tooltip states, it is to prompt the engineer to consider the connection details at this location in the column.

Translational releases

- Fy and Fz releases are prevented in all stacks i.e. no ‘roller’ connections.
- Fx (axial) release allowed at the top of the topmost stack only.
- Mx (torsional release allowed in all stacks.

Concrete meshed and mid-pier wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	• Meshed Shear Wall • Mid-pier Shear Wall **NOTE** Mid-pier shear walls must be rectangular in a vertical plane. Meshed shear walls can be vertical or sloping.
Fabrication	Choice of: • Cast-in-place • Precast **NOTE** Design of precast members is beyond scope in the current release
Autodesign	• Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process. See: Concrete member autodesign (page 1280)
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is ‘on’. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size.

General	
	• Current - the auto design commences from the current bar arrangement. See: Concrete member autodesign (page 1280)
Assume extra floors supported	Enter the number of extra floors supported.
Rotation (mid-pier only)	• 0° - wall spans horizontally • 90° - wall spans vertically
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Overview of load reductions (page 1172)
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Grade	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back

All panels	
	• Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.
End 1 extension	The amount the wall is extended or trimmed back from end 1. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 429)
End 2 extension	The amount the wall is extended or trimmed back from end 2. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 429)
Reinforcement Layers	Number of layers of reinforcement to be used in the wall: • 1 • 2
Releases	
Minor Top	• Fixed • Pinned

All panels	
	• Continuous (incoming members pinned) **NOTE** The ‘Continuous’ option is only available for FE meshed walls.
Minor Bottom	• Fixed • Pinned • Continuous (incoming members pinned) **NOTE** The ‘Continuous’ option is only available for FE meshed walls.
Design parameters	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See: Design parameters (Eurocode only) (page 2096)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this

All panels	
	calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the check box located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the check box located under that panel only.
Confinement reinforcement	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness	
Major (Minor)	• Braced • Bracing
Effective length factor direction Major (Minor)	• Calculated • User input value
Stiffness	

All panels	
Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See: Stiffness (page 1308)
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/shear capacity ratio in Dir 1 Shear demand/shear capacity ratio in Dir 2 (ASCE7 code only)	When working to the ASCE7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit.

Utilization ratio	
	Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Reinforcement	
Include end zones	• Cleared - the wall is designed without end zones of reinforcement • Checked - the wall is designed with end zones of reinforcement
Wall zone	
Form, type class	Specifies the wall zone reinforcement properties

Reinforcement	
End zone	
Form, type class	Specifies the end zone reinforcement properties (if end zones have been requested).

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing (meshed walls only)	
Override Model's	Select this check box to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only • Quad dominant See: How meshed walls are represented in solver models (page 750)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Concrete core properties

General	
Name	The automatically generated name for the core.
User Name	Can be used to override the automatically generated name if required.
Rotation	Defines the axis system for reporting the core results: • Dir 1/2 - Main Building Directions • Principal 1/2 - Major and minor local axis • Angle - in Global Coordinate System

Level 1, Level 2 etc.	
Centroid above, Centroid below	Every core will have at least 2 core levels, with a core section for each level. The centroids of the core sections are reported above and below each level.

General wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	• Meshed Shear Wall
Material type	• General

General	
Assume extra floors supported	Enter the number of extra floors supported.
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades that are available for General materials are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/

All panels	
	drift sensitivity calculations are also influenced by this assumption.
End 1 extension	The amount the wall is extended or trimmed back from end 1. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 429)
End 2 extension	The amount the wall is extended or trimmed back from end 2. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 429)
Releases	
Minor Top	• Fixed • Pinned • Continuous (incoming members pinned)
Minor Bottom	• Fixed • Pinned • Continuous (incoming members pinned)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation

All panels	
	to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the check box located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the check box located under that panel only.

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing	
Override Model's	Select this check box to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only • Quad dominant See: How meshed walls are represented in solver models (page 750)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Member characteristic, construction and fabrication properties

The following table lists the available permutations of characteristic, construction and fabrication for each material type.

Material	Characteristic	Construction	Fabrication	Notes
Concrete	Beam	Concrete beam	Cast-in-place	
			Post tensioned	Not designed
			Precast	Not designed

Material	Characteristic	Construction	Fabrication	Notes
		Coupling beam	Cast-in-place	Coupling beams are the only beams considered by the Assisted core function. As they may require additional design checks beyond those currently made in the program they are not designed.
	Column	Concrete column	Cast-in-place	
			Precast	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Steel	Beam	Non-composite beam	Rolled	
			Plated	
			Westok cellular	
			Westok plated	
			Fabsec	Not designed
		Composite beam	Rolled	
			Plated	
			Westok cellular	Not designed
			Westok plated	Not designed
			Fabsec	Not designed
	Column	Non-composite column	Rolled	
			Plated	
			Concrete filled	Not designed
			Concrete encased	Not designed
		Composite column	Rolled	Not designed
			Plated	Not designed
	Brace	Steel brace	Rolled	
	Steel joist	Steel joist	Steel joist	Standard, Special or Girder types
	Gable post	Steel gable post	Rolled	
	Parapet post	Steel Parapet post	Rolled	
	Truss member top	Steel Truss member top	Rolled	

Material	Characteristic	Construction	Fabrication	Notes
	Truss member bottom	Steel Truss member bottom	Rolled	
	Truss member side	Steel Truss member side	Rolled	
	Truss member internal	Steel Truss member internal	Rolled	
	Analysis Element	Not applicable	Not applicable	Not designed
Cold formed	Beam	Cold formed beam member	Cold formed	
	Column	Cold formed column member	Cold formed	
	Brace	Cold formed brace member	Cold formed	
	Gable post	Steel gable post	Cold formed	
	Parapet post	Steel Parapet post	Cold formed	
	Truss member top	Steel Truss member top	Cold formed	
	Truss member bottom	Steel Truss member bottom	Cold formed	
	Truss member side	Steel Truss member side	Cold formed	
	Truss member internal	Steel Truss member internal	Cold formed	
	Analysis Element	Not applicable	Not applicable	Not designed
Cold rolled	Beam	Cold rolled beam member	Cold rolled	Not designed

Material	Characteristic	Construction	Fabrication	Notes
	Column	Cold rolled column member	Cold rolled	Not designed
	Eaves beam	Eaves beam	Cold rolled	Not designed
	Purlin	Purlin	Cold rolled	Not designed
	Rail	Rail	Cold rolled	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Timber	Beam	Timber beam	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Column	Timber column	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Brace	Timber brace	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Gable post	Timber gable post	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member top	Timber Truss member top	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member bottom	Timber Truss member bottom	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed

Material	Characteristic	Construction	Fabrication	Notes
	Truss member side	Timber Truss member side	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member internal	Timber Truss member internal	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
General	Beam	General material beam	Not applicable	Not designed
	Column	General material column	Not applicable	Not designed
	Brace	General material brace	Not applicable	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed

Slab item properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	This property is used for the following where appropriate: • Span direction for 1-way load decomposition • To determine the 2D solver element local axes in the solver model

General	
	• Bar direction for Slab on Beam and Flat Slabs. Different angles can be specified for different panels within the slab. See: Rotation angle for panels (page 1380)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For panels in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures.
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the mat item.
User Name	Can be used to override the automatically generated name if required.
Slab type	• Precast • Steel deck • Timber deck • Composite slab • Foundation mat
Deck type	The deck type depends on the slab type as follows: • Precast concrete planks: Precast, • Steel plate: Steel deck, • Timber: Timber deck, • Profiled metal decking: Composite slab,

Slab general	
	• Reinforced concrete: Slab on beams, Flat slab, Foundation mat • Post tension: Slab on beams, Flat slab, Foundation mat
Decomposition	Decomposition choices depend on the slab type as follows: • One-way only: Precast, Timber deck, Composite slab • One-way or two way: Steel deck, Slab on beams • Two-way only: Flat slab, Foundation mat

Slab parameters	
Slab properties(general)	
Overall depth	Specifies the slab thickness.
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Slab properties (concrete slabs)	
Concrete type	• Normal • Lightweight
Grade/Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Long term elastic modulus = Ecm divided by... composite slab and precast only)	Factor by which the short term elastic modulus is divided to obtain the long term modulus .
Ponding allowance	If required, choose either value or percentage and then specify actual ponding allowance accordingly.

Slab parameters	
option, Ponding allowance value (composite slab only)	
Apply ponding to construction stage (composite slab only)	• On = ponding allowance applied at construction stage
Apply ponding to composite stage (composite slab only)	• On = ponding allowance applied at composite stage
Decking properties (where applicable)	
Country, Manufacturer, Reference, Gauge	Defines properties of decking (where applicable).
Reinforcement in slab for crack control or fire requirements (composite slab, and precast with topping specified only)	
Type, Rib type, Mesh type Class, Bar size	Defines reinforcement for crack control or fire requirements
Topping details(precast only)	
Topping	• None • Non-structural • Structural
Depth	Depth of topping when topping other than 'None' is selected.
Design parameters (Head Code Eurocode)	
Permanent load ratio	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Deflection parameters	

Slab parameters	
Restraint constant (ACI only)	See: Restraint constant (ACI) (page 1389)
Restraint type (Eurocode only)	See: Restraint type (EC2) (page 1388)
Cement class (Eurocode only)	See: Concrete Properties (Eurocode) (page 1390)
Material properties	
Modulus of elasticity, Poisson's ratio, Shear moduls, Coefficient of thermal expansion	Properties specific to timber and steel decks.

Design parameters	
Adjustment ratio direction X ,Adjustment ratio direction Y (slab on beam only)	These factors are applied to the enclosing lengths in X and Y in order to manually adjust the X and Y direction span in the span-effective depth check. See Slab on beam idealized panels (page 1378)
Enclosing length X, Enclosing length Y (slab on beam only)	The automatically calculated span length in the X direction, and Y direction. See Slab on beam idealized panels (page 1378)
Adjusted length X, Adjusted length X (slab on beam only)	The adjusted span lengths in the X and Y directions. See Slab on beam idealized panels (page 1378)
Edge category start X, Edge category start Y (slab on beam only)	The assumed support condition at the start of the span in the X direction, and Y direction. See Slab on beam idealized panels (page 1378)

Design parameters	
Edge category end X, Edge category end Y (slab on beam only)	The assumed support condition at the end of the span in the X direction, and Y direction. See Slab on beam idealized panels (page 1378)
Average stiffness ratio (Slab on beam ACI/AISC only)	In the current release of Tekla Structural Designerr the average stiffness ratio is a user defined value. (Default value = 1.0). The way in which the minimum thickness is calculated directly depends on this value.

Live/Imposed load reduction	
Reduce live/ imposed loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Utilization ratio	
Apply (to autodesign)	On

Utilization ratio	
	• When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

See also

Foundation mat properties (page 2120)

Foundation mat properties

General	
Name	The automatically generated name for the mat item.

General	
User Name	Can be used to override the automatically generated name if required.
Foundation type	Mat
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the mat. See: Rotation angle for panels (page 1380)
Include in diaphragm	If this option is cleared, the mat item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For panels in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Plane	Indicates the level at which the mat is placed.

Slab general	
Name	The parent foundation mat name.
User Name	Can be used to override the automatically generated name if required.
Slab type	***Foundation mat***
Deck type	• Reinforced concrete • Post tension **NOTE** Design of post tensioned slabs is beyond scope in the current release.
Decomposition	Two-way

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Grade/Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Design parameters (Head Code Eurocode)	
Permanent load ratio	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Deflection parameters	
Restraint constant	See: Restraint constant (ACI) (page 1389)
Restraint type (Eurocode only)	See: Restraint type (EC2) (page 1388)
Cement class (Eurocode only)	See: Concrete Properties (Eurocode) (page 1390)
Soil parameters	
Use Ground Bearing Springs	Select to specify that the mat is ground bearing.
Allowable Bearing Pressure	The allowable bearing pressure is only required when the 'Use Ground Bearing Springs' option is selected.

Slab parameters	
Ground Stiffness The following properties are only displayed when the 'Use Ground Bearing Springs' option is selected.	
Type	• Spring Linear • Spring Non-Linear
Stiffness	Only displayed when the Type is set to 'Spring Linear'.
Stiffness -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Tension limit -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Stiffness +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Compression limit +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Horizontal Support	• Fixed • Free • Spring
% of vertical spring stiffness	This field is displayed when the Horizontal Support is set to 'Spring'.

Live/Imposed load reduction	
Reduce live/ imposed loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Slab/Mat overhang properties

General	
LengthofBeam	• When checked, the overhang is created by a single click on a supporting beam along the the slab edge. • When unchecked, the overhang is created by clicking two points along the slab edge.
EdgeParallel	• When checked, the overhang is created with a straight edge. • When unchecked, the overhang is created with a curved edge.
Curvature	Specifies the overhang curvature, (when EdgeParallel is unchecked).
Tapered	• When checked, the overhang varies in width. (Width1 to Width2) • When unchecked, the overhang is a constant width (Width1).
Width1	The width at end 1 of the overhang.
Width2	The width at end 2 of the overhang, (when tapered).

Pad base strip base and pile cap properties

Property	Description
General	
Foundation Type	Isolated Pad Base (not editable). Isolated Pile Cap (not editable).
Auto-design depth	• When Autodesign is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing depth is retained and Tekla Structural Designer determines if it is sufficient.

Property	Description
	NOTE In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the depth will be checked or designed according to the auto-design depth setting.
Auto-design size	• When Autodesign is selected an iterative procedure is used to determine the base size. If the bearing check fails the size is increased until either the check passes, or the maximum side length specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing size is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the size will be checked

Property	Description
	or designed according to the Auto-design size setting.
Auto-design piles (Pile cap only)	Autodesign piles automatically determines the quantity and size of piles. • Cleared - the specified number of piles at their specified positions will be checked during the design process. • Checked - the number of piles and their positions under the pile cap will automatically determined during the design process. When this check box is selected, the actual procedure used will depend on the Pile auto-design method that has been specified in Design Options > Concrete Foundations > Isolated Foundations Piles The **Minimise number of piles** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded the pile with the next smallest load capacity in the catalogue is selected and the process starts again. This is repeated until the pile loading check passes. The **Minimise pile capacity** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded one pile is added to the pile group and the process starts again. This is repeated until the pile loading check passes.
Select size/depth starting from	This option only appears when either Autodesign depth or Autodesign size

Property	Description
	is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in Design Options. For both options the auto-design depth increment is that specified in Design Options> Foundations Isolated Foundations> Foundation Size
Autodesign reinforcement	This setting applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. If enabled, auto-design of reinforcement occurs after the base size and depth have been established. • When Autodesign reinforcement is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. **NOTE** If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar

Property	Description
	size gets increased until the largest bar size reached. • When Autodesign reinforcement is disabled (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base the reinforcement is always designed (irrespective of the Autodesign reinforcement setting).
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Autodesign reinforcement is selected. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Foundation Parameters	

Property	Description
Eccentricity in Y direction	Specifies the eccentricity of the base in the Y direction. (For strip bases this is along the wall)
Eccentricity in X direction	Specifies the eccentricity of the base in the X direction. (For strip bases this is across the wall)
Rotation angle	Specifies the rotation of the base: • Pad base: relative to that of the support (which is controlled by the rotation angle specified for column). • Strip base: the angle of the base about global Z.
Shape Pad base and pile cap only)	Specifies the base shape in plan: • Square • Rectangular
Length in Y direction (Pad base and pile cap only)	Specifies the size of the pad in the Y direction.
Length in X direction (Pad base and pile cap only)	Specifies the size of the pad in the X direction.
Width (Strip base only)	Specifies the width of the base
Depth	Specifies the depth of the base.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Use loaded area (Pad base only)	Selecting this option allows you to specify a user override of the punching shear area - defined using the Loaded Area Breadth and Loaded Area Depth properties. This feature can be used in both pad bases and pile caps to, for example, consider unmodeled base plates or pedestals.
Sliding in Y prevented / Sliding in X prevented (Pad base only)	If sliding is prevented by some other means (e.g. if attached to a floor slab in one or other direction) then this

Property	Description
	can be indicated by selecting the appropriate direction. **TIP** If the direction is not clear make the base non-rectangular so that length in X and hence X-direction is clearer. Selecting this option sets the shear force for the particular direction to zero. This affects the sliding check, but actually affects all checks because there is no additional overturning moment developed due to shear at top of base.
Overturning about X prevented / Overturning about Y prevented (Pad base only)	This option only appears when the Sliding in Y prevented / Sliding in X preventedoption is selected. Selecting this option indicates that overturning is also prevented by some other means for the selected direction, which sets the support moment for that direction to zero. This affects all checks because there is no overturning moment considered.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Piles (Pile cap only)	
Pile type	Pile types that have previously been specified in the Pile Catalogue are available for selection. You can choose <New...> to add additional types.
User defined arrangement	Specifies the eccentricity of the pile cap in the X direction.

Property	Description
Pile arrangement	Opens the Pile arrangement dialog for specifying the number of piles, and spacing, pile type and principal direction.
Number of piles	Specifies the number of piles.
Principal direction	Specifies the principal direction.
Pile spacing	The dimension between pile centers.
Shape	For three piles only, you can choose to specify either a triangular. or rectangular pile cap.
Reinforcement	
Type	• Mesh • Loose Bars • None
Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here.
Top, Bottom, Side cover	Nominal cover to reinforcement.
Soil Parameters	
Soil unit weight	Soil unit weight
Characteristic friction angle	Characteristic friction angle
Presumed bearing resistance (EC Head Code)	The presumed bearing resistance is only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is selected in Design Options.
Surcharge	
Soil Surcharge depth Other permanent Permanent surcharge load	Drop-list controls allow each surcharge load to be assigned to a specific load case. The partial safety factors defined in the design combinations for these

Property	Description
Other variable Variable surcharge load	cases are then used for these loads in all design calculations. The assigned load cases are initially set to “None” and must be user-selected. Where no appropriate Dead or Live load case exist in the model and so no load case can be assigned, zero partial factor values are used in the calculations.
Design shear strength of soil angle (ACI & BS Head Codes)	Design shear strength of soil angle
Allowable bearing capacity (ACI & BS Head Codes)	Allowable bearing capacity
Bearing capacity A1 - STR (EC Head Code)	Allowable bearing capacity A1. Only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is unselected in Design Options.
Bearing capacity A2 - GEO (EC Head Code)	Allowable bearing capacity A2. Only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is unselected in Design Options.
UDA	A customizable list of the attributes that can be applied to individual members and panels.

Bearing wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Bearing Wall **NOTE** Bearing walls must be rectangular in a vertical plane.

General	
Material Type	Choice of: • Concrete • Timber • General
Assume extra floors supported	Enter the number of extra floors supported.
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall.

All panels	
	NOTE The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Shear only wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Shear Only Wall **NOTE** Shear only walls must be rectangular in a vertical plane.
Material Type	Choice of: • Concrete • Timber

General	
	• General
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall. **NOTE** The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User

All panels	
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Spring stiffness	The stiffness to be used in the analysis. See: How shear only walls are represented in solver models (page 757)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Wall Panel Properties

Use the**Wall Panel**properties to view or modify the properties of a wall panel.

The **Wall Panel** command is used to create a wall panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

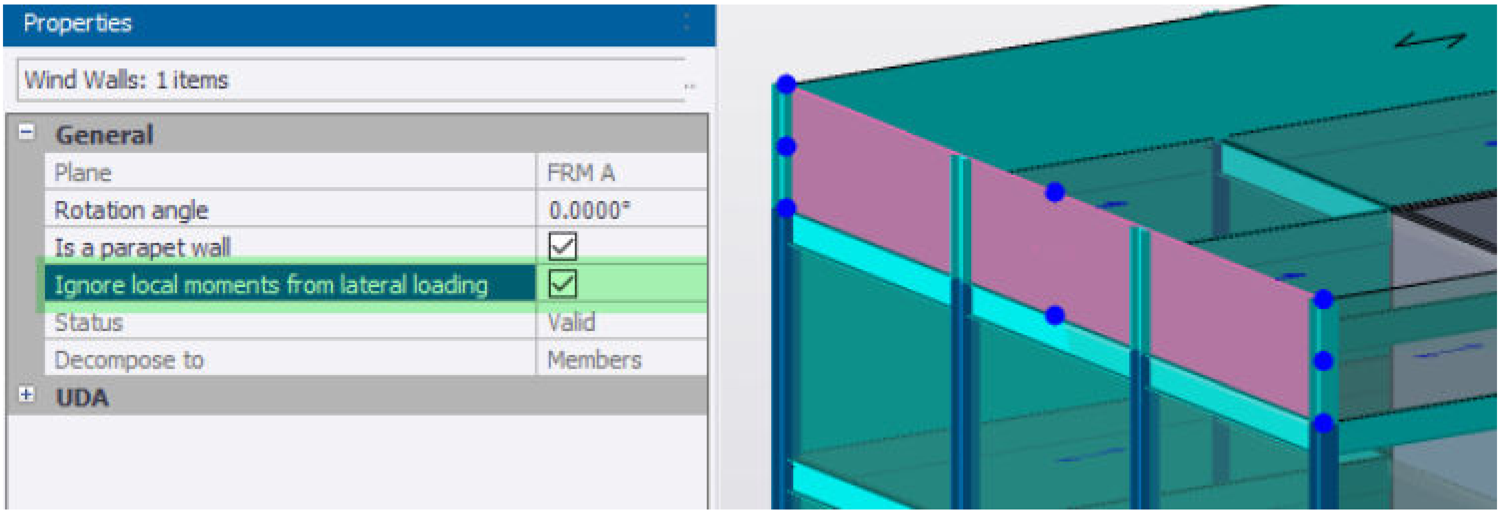
Property	Description
General	
Plane	Describes the plane in which the panel was placed.
Rotation angle	Describes the panel span direction as an angle, 0° is horizontal and 90° is vertical.
Is a parapet wall	Check to indicate the panel is to be treated as a parapet in the wind analysis. See: Parapet wall panel load decomposition (page 2139)
Ignore local moments from lareral loading	This option is omly available when the “Is a parapet wall” setting is selected. When checked, only a lateral point load is applied to the top of a supporting column. See: Parapet wall panel load decomposition (page 2139)

Property	Description	
Gap	Where the funneling gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funneling. A zero gap value explicitly means ignore funneling, for example where this building and the adjacent one are sheltered by upwind buildings.	
Solidity	If you indicate that the wall panel is a parapet, then you also need to indicate the Solidity of the parapet. (Walls that are not parapets automatically adopt a solidarity of 1.0).	
Status	Indicates whether the panel is valid or not.	
Decompose to	Loads can be set to be decomposed to: • Members • Nodes (Default) • Rigid Diaphragms **NOTE** The decompose to nodes option only applies to those loads generated by the wind wizard. It does not apply to those loads applied directly to panels.	
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels.	

Parapet wall panel load decomposition

For the special case of parapet wall panels spanning horizontally to parapet posts supported by columns, the panel load is decomposed as a lateral point load and point moment - which would be produced by a UDL on the post - to the top of the supporting column.

In some circumstances the engineer may wish only the lateral load but not a moment to be applied. This is catered for using the option “Ignore local moments from lateral loading” which is available in the wind wall properties only when the “Is a parapet wall” setting is checked on. When the “Ignore local moments from lateral loading” setting is checked on, only a lateral point load is applied to the top of a supporting column. No moment is applied.



NOTE

- The special decomposition of point loads - with or without a point moment - to supporting columns only occurs when all the following are true; the wind panel spans horizontally to parapet posts, the "Is a parapet wall" setting is checked on for the wall panel, the parapet post is supported by a column.
- In both the following cases a UDL will be decomposed to the members onto which the parapet wind panel directly spans;
  - The member supporting the parapet post is NOT a column.
  - The member onto which the parapet wind panel spans is not a parapet post (e.g. is an extended column or vertical beam).

Roof Panel Properties

Use the **Roof Panel** properties to view or modify the properties of a wall panel.

The **Roof Panel** command is used to create a roof panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

Property	Description
General	
Plane	Describes the level at which the panel was placed.
Rotation angle	This value determines the following where appropriate: • Span direction for 1-way load decomposition • Orientation of semi-rigid 2D elements in the Solver Model

Property	Description
Include in diaphragm	The options are: • Checked The roof panel is meshed to form a diaphragm of semi-rigid 2D elements. • Unchecked (Default) No diaphragm is defined on the roof panel.
Thickness	Roof panel thickness **NOTE** Only displayed if "Include in diaphragm" is checked
Youngs Modulus	Youngs Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Shear Modulus	Shear Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Temperature coefficient	Temperature coefficient. **NOTE** Only displayed if "Include in diaphragm" is checked
Divide Stiffness by	Used to adjust the roof panel stiffness. **NOTE** Only displayed if "Include in diaphragm" is checked
RoofType	The options are: • Default (Default) • Flat • Monopitch • Duopitch • Hip Gable • Hip Main • Mansard **NOTE** The default option assigns the roof panel as flat (if the slope < 5 deg) or Monopitch (if the slope is > 5 deg). You can more accurately specify the roof panel by choosing the appropriate option from the list.

Property	Description
	NOTE If the RoofType is changed after the wind model has already been established, you will need run  **Update Zones** to reinstate the zoning.
[+] UDA	Customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1026)

Support properties

Use the Support properties to view or modify the properties of a support.

When a support is first created, it's properties are taken as those displayed in the **Properties** window at that time.

You can either set the properties prior to placing the support, or select a support in the model afterwards to review or modify its properties.

Property	Description
General	
Name	Automatically created name based on the grid point location. **NOTE** This property is only displayed when editing an existing support
User Name	You can enter a user name to replace the automatically created name if required. **NOTE** This property is only displayed when editing an existing support
Plane	Describes the level at which the support was placed. **NOTE** This property is only displayed when editing an existing support
3 Grid Points	The options are: • Checked A user defined coordinate system is applied to the support. (After clicking where you want to create

Property	Description
	the support, the second click defines the x direction and the third click defines the y direction.) • Unchecked (Default) Support properties are defined in accordance with the global coordinate system. **NOTE** This property is only displayed when creating a new support
Fx, Fy, Fz	The translational degrees of freedom can be set as either Free, or Fixed in each direction.
Mx, My, Mz	The rotational degrees of freedom can be set as either Free, or Fixed in each direction.
Angles	
Inclination, Azimuth and Rotation	When creating new supports, the angles are calculated automatically depending on the placement method (3 Grid Points Checked/Unchecked). When editing existing supports, the angles can be edited in order to redefine the direction in which the support acts.
Translational stiffness x, y, and z	
Type	In order to define a translational spring in a particular direction, the translational degree of freedom in the same direction must first be set to Free. The available types are: • Release - (i.e. zero translational stiffness) • Spring Linear • Spring Non-linear
Stiffness	The options are: • Spring Linear A single stiffness value is entered, which acts in both the positive and negative directions. • Spring Non-linear Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must

<table>
<tr><th>Property</th><th>Description</th></tr>
<tr><td></td><td>always be entered as a positive value, for both +ve and -ve directions).</td></tr>
<tr><td>Rotational stiffness x, y, z</td><td></td></tr>
<tr><td>Type</td><td>In order to define a rotational spring in a particular direction, the rotational degree of freedom in the same direction must first be set to Free. The available options are:
Release - (i.e. zero rotational stiffness)
Spring Linear
Spring Non-linear
Nominally Pinned
Nominally Free</td></tr>
<tr><td>Stiffness</td><td>The options are:
Spring Linear
A single stiffness value is entered, which acts in both the positive and negative directions.
Spring Non-linear
Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.</td></tr>
<tr><td>Stiffness Percentage</td><td>The options are:
Nominally Pinned - 10%
(i.e. 10% * 4 EI/L)
Nominally Fixed - 100%
(i.e. 100% * 4 EI/L)
NOTE In the above equations, L is the length from the base of the column to the level of the next column stack that is denoted as a floor, or it is the distance to the top of the column if shorter.</td></tr>
<tr><td>Fmax -ve and Fmax +ve</td><td>For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).</td></tr>
</table>

See also

Create supports (page 490)

Patch properties

General	
Name	The automatically generated name for the patch.
User Name	Can be used to override the automatically generated name if required.
Type	• Column • Beam • Wall
Lx	Specifies the size of the patch in the X direction.
Ly	Specifies the size of the patch in the Y direction.
Associated Slab Panel	Where a patch sits over more than one panel then an automatic choice is made, but you are able to override this if required by selecting one of the alternative panels from a drop-down list.
Align to Panel Reinforcement	When this check box is selected, the calculated strip reinforcement is aligned with the background reinforcement in the Associated Slab Panel.
Local X Angle	The angle of the X axis reinforcement is only editable ifAlign to Panel Reinforcement is cleared.
Surface	Specifies the reinforcement to be associated with and designed by the patch: • Top or Bottom (cannot be both).
Autodesign	This setting applies to all strips in both directions, but reinforcement in either direction can still be set to none - see below. • Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process.
Select bars starting from	This option controls the starting point for auto-design procedures. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement.

General	
Consider Strips	This setting controls which strips are to be designed by the patch. • X • Y • X and Y
Consider patch surface moments only	Off (default) • patch reinforcement is designed in the surface of the slab at which the patch has been specified AND the slab reinforcement is also checked in the opposite surface. On • patch reinforcement is designed in the surface of the slab at which the patch has been specified, but the slab reinforcement in the opposite surface is NOT checked. **NOTE** The only situation when you might need to activate this option is if the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face.(When the other patch is designed additional reinforcement will be provided at that surface, so this check is not required for the original patch).

Strips in X, Strips in Y	
Center, Left and Right Strip	
Width	The left and right strip widths can be specified independently: the center strip width is recalculated accordingly and cannot be edited. (By default the center strip covers half the panel, so that the left and right strips each cover a quarter of the panel.)
Design Force	• Average (of all the FE nodal values within the strip). • Maximum (of all the FE nodal values within the strip).
Reinforcement	
Combine with Panel Reinforcement	When this check box is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this check box is selected, the cover is set to be the same as that in the Associated Slab Panel.

Strips in X, Strips in Y	
Outer Bar Direction as Panel	When this check box is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.

Reinforcement	
Combine with Panel Reinforcement	When this check box is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this check box is selected, the cover is set to be the same as that in the Associated Slab Panel.
Outer Bar Direction as Panel	When this check box is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.
Reinforcement	This setting is used to specify whether bars or mesh are to be used in each direction. • Mesh • Bars XY • Bars X • Bars Y • None (If Mesh is selected an extra setting then allows you to specify if main bars are in X or Y.)

Reinforcement in X, Reinforcement in Y, or Mesh	
Bar Size, spacing, Mesh type etc.	The actual reinforcement provided in each of the strips is indicated here.

Punching check properties

General	
Tension Reinforcement	This setting identifies the slab reinforcement to be used in the punching check calculation. • Top • Bottom
Center	The check location (not editable).

General	
Column Drop	Indicates if the check considers a Column Drop (not editable).
Beta - User limit (Head Code Eurocode)	When this check box is selected, a minimum value of Beta = 1.15 is applied to all internal columns.
User factor for Vt (Head Code BS)	When this check box is selected, the user factor for Vt is applied.
u0 - user reduction	Can be used to manually specify a reduction in the length of the u0 perimeter to account for undefined openings.
u1 - user reduction	Can be used to manually specify a reduction in the length of the u1 perimeter to account for undefined openings.
Check Status and Ratio	Indicates the status of the checks for each calculated perimeter and the overall check ratio.

Loaded Perimeter	
Length	Indicates the length of the u0 Loaded Perimeter (not editable).
Reduced length	The reduced length of u0 after accounting for openings.
BEquiv, DEquiv, BBound, BBound, Bounding Perimeter	Refer to the Concrete Design Reference Guide for the current Head Code for the appropriate definition of these terms.
d Effective Depth	Indicates the average effective depth to the tension reinforcement (not editable). d = (dy + dz) / 2 where dy and dz are the effective depths in the two orthogonal directions. There is a value of d for top steel and a different value for bottom steel. Note this definition changes in the presence of a drop panel. This information is only available if the reinforcement is known in each direction.
Slab Override	When an override is applied the slabs in each direction can be de-activated in the check. In this way the Loaded Perimeter Position can be edited. **NOTE** In the typical case of punching checks around a column, the slab 'y' & 'z', 'positive' & 'negative' are defined by the local axis system of the column. This

Loaded Perimeter	
	can be displayed by displaying the Local Axes for 1D Elements in Scene Content.

Control Perimeter	
Length	Indicates the length of the u1 Control Perimeter (not editable).
Reduced length	The reduced length of u1 after accounting for openings.
[-] Design Input (Head Code Eurocode and ACI)	
Use Reinforcement	Select this check box in order to apply a default punching reinforcement arrangement that can then either be checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the check box when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.

Control Perimeter	
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the 'Arrangement Type' is Circular)
Number of studs per column face - Y direction	The number of stud rails adjacent to the column face in the local Y direction.
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 786)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Design Input (Eurocode and ACI only)	
Use Reinforcement	Select this check box in order to apply a default punching reinforcement arrangement that can then either be checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the check box when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the 'Arrangement Type' is Circular)

Design Input (Eurocode and ACI only)	
Number of studs per column face - Y direction	The number of stud rails adjacent to the column face in the local Y direction.
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

Result strip properties

General	
Name	The automatically generated name for the strip.
User Name	Can be used to override the automatically generated name if required.
Start Width	The total strip width at the first point picked when creating the strip.
End Width	The total strip width at the second point picked when creating the strip.
Result Type	Determines how the strip result is calculated: • Average • Centreline • Maximum
Number of Stations	The number of stations per metre along the strip.
Number of Points	The number of points per metre across the strip at each station.

15.2 Ribbon commands

- Home ribbon (page 2217)
- BIM Integration ribbon (page 2207)
- Model ribbon (page 2222)

* Edit ribbon (page 2211)
* Load ribbon (page 2218)
* Analyse ribbon (page 2203)
* Design ribbon (page 2208)
* Slab Deflection ribbon (page 2232)
* Foundations ribbon (page 2213)
* Report ribbon (page 2228)
* Draw ribbon (page 2211)
* Windows ribbon (page 2232)

Commands A - Z

A-Z listing of commands.

0-9

* **1st Order Linear** - Runs a linear static analysis
* **1st Order Non-Linear** - Runs a nonlinear analysis with loading applied in a single step
* **1st Order RSA Seismic** - Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis
* **1st Order Modal** - Runs an unstressed modal analysis to determine the structure's natural frequencies
* **2nd Order Linear** - Runs a 2-stage P-Delta analysis
* **2nd Order Non-Linear** - Runs a nonlinear analysis with loading applied in a single step
* **2nd Order Buckling** - Runs a linear buckling analysis to determine a structure's susceptibility to buckling
* **2nd Order RSA Seismic** - Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis
* 3D DXF Import (page 2180) - Import a model from a 3D DXF File

A

- ADAPT Export (page 2180) - Export a model to ADAPT
- Add Group (page 268)
- **Analyse All** - Performs a slab deflection analysis of all submodels regardless of current view/selection
- **Analyse All (Static)** - Performs a full series of analyses that would be carried out as part of 'Design All (Static)' but with no design. Includes all relevant analyses. Analysis type (e.g. second-order non-linear) is the same as set in the design settings
- **Analyse Current** - Performs a slab deflection analysis for the current sub model
- **Analyse Selected** - Opens dialog allowing analysis of selected submodels.
- Area (page 2180) - Create a polygon area mat by selecting perimeter points and including a defined overhang dimension
- Area Load (page 2180) - Applies an area load to any panel. Area loads entirely cover an individual panel
- Autodesk Revit Export (page 2180) - Export the structural BIM model to Autodesk Revit

B

- Bays (page 2180) - Create a mat by selecting bays bounded by grid lines
- Beam (page 2181) - Create a beam
- **Beam End Forces** - Create and save beam end forces drawing for an active 2D view for the currently selected loading
- Beam Lines (page 2181) - Joins existing concrete beams to make a continuous concrete beam
- **Beam Schedule** - Configure and then create a concrete beam schedule for members in an active 2D view
- Bearing Wall (page 2181) - Creates a bearing wall
- Brace (page 2181) - Create a brace

C

- **Calalogue** - Add and delete piles from your pile catalogue or edit their properties (e.g. compression and tension capacity)
- Cellbeam Export (page 2181) - Export a beam to Westok Cellbeam
- Cellbeam Import (page 2181) - Import a beam from Westok Cellbeam

- Check Floor Vibration (page 2181) - Perform all floor vibration checks in the model
- Check Group (page 270)
- Check member (page 2238)
- Check model (page 2239)
- Check model patches (page 2239)
- Check panel (page 2240)
- Check plane (page 2240)
- Check plane patches (page 2241)
- Check plane slabs (page 2241)
- Check punching shear (page 2242)
- Check Punching Shear (panel) (page 804)
- Check member (page 2238)
- Check members (page 2239)
- Check model slabs (page 2240)
- **Check sub structure**
- Check sub structure slabs (page 797)
- Check wall (page 2246)
- Check walls (page 2247)
- **Close** - Close the model
- Cloud Export (page 2182) - Export the model to the cloud
- Column (page 2183) - Create a column
- Column Drop (page 2183) - Create a column drop
- **Column Schedule** - Configure and then create a concrete column schedule
- **Column Splice Loads** - Create and save steel column splice loads drawings for an active 2D frame view for the currently selected loading
- **Combination** - Opens the **Loading** dialog at the Combinations page
- Construction Levels (page 2248) - Opens the Construction Levels dialog (page 2402) allowing you to define the levels required in order to construct your model
- Construction Line (page 2183) - Create a straight construction line between two points
- Copy (page 2183) - Select items then copy
- Copy Loads (page 2182) - Copy between objects in the current case, or copy from one case to another

- Cores (page 2183) - Assign existing concrete members as part of a concrete core
- Create Infills (page 2183) - Creates a pattern of infill members in the selected bay
- Create new core (page 435)
- Create Sub Structure (page 1031)
- Create sub structure group (page 1034)
- Cutting Planes (page 2183) - Move/Activate/Deactivate planes to isolate a smaller visible working area within the model

D

- **Decomposition** - Decompose loads from 1-way and 2-way slabs onto supporting members.
- **Decomposition** (Wind Load) - Decompose the wind loads on to the structure
- **Deflection Checks** - Opens the Slab Deflection Check Catalogue for defining the default check requirements associated with Check Lines (also associated with events)
- Delete (page 1033) - Delete Selected Items
- **Delete Seismic** - Delete all of the seismic data entered in the seismic wizard along with the horizontal spectrum, seismic loadcases and seismic load combinations
- **Delete Snow** - Deletes the snow model data previously defined using the Snow Wizard
- **Delete Wind** - Deletes the wind model data previously defined using the Wind Wizard
- Design All (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all members in the model based on their individual 'auto-design' setting
- Design All (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting
- Design All (RSA) (page 2185) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting
- Design Concrete (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all concrete members in the model based on their individual 'auto-design' setting
- Design Concrete (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all

concrete members in the model based on their individual 'auto-design' setting

- Design Concrete (RSA) (page 2184) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting
- Design Connection (page 275)
- Design Group (page 270)
- Design Mats (page 2186) - Design ground bearing and piled mat foundations for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting
- Design member (page 2248)
- Design members (page 2249)
- Design model (page 2250)
- Design model patches (page 2250)
- Design model slabs (page 2251)
- Design Pad Bases (page 2186) - Design pad bases to all valid combinations based on their individual 'auto-design' settings
- Design panel (page 2251)
- Design Patches (page 2185) - Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting
- Design Patches (page 2185) - Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting
- Design Pile Caps (page 2186) - Design pile caps to all valid combinations based on their individual 'auto-design' settings
- Design plane (page 2252)
- Design plane slabs (page 2253)
- Design plane patches (page 2252)
- Design Punching Shear (page 2185) - Check of all punching shear checks in the model.
- Design Punching Shear (page 2185) - Perform a check or design (according to individual autodesign settings) of all punching checks in the model
- Design punching shear (page 2253)
- Design Punching Shear (panel) (page 805)
- Design Slabs (page 2185) - Designs two-way concrete slabs for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting

- Design Steel (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all steel members in the model based on their individual 'auto-design' setting
- Design Steel (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting
- Design Steel (RSA) (page 2184) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting
- Design sub structure (page 786)
- Design sub structure slabs (page 799)
- Design wall (page 2261)
- Design walls (page 2261)
- Dimension (page 2187) - Create dimensions between appropriate points which can be included on drawings
- Dissociate (page 438)
- **Drawing Management** - Opens a dialog for the generation and laying out of multiple drawings on to a single drawing sheet. The dialog can also be used to manage drawing revisions

E

- Edit (page 275)
- **Edit Footer** - Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers
- **Edit Header** - Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers
- Element (page 2187) - Create analysis elements
- **Envelope** - Opens the **Loading** dialog at the Envelopes page
- **Event Sequences** - Submodel Event Sequences give access to override aspects of the model event sequence on a submodel basis
- Export Connection to Tekla Connection Designer (page 275)
- Export Connection to IDEA StatiCa (page 345)

F

- **Filters** - Set up filters to be used when generating a new report
- Find (page 353) - Find and select objects by typing a part of their name

- **Foundation Layout** - Create and save foundation layout drawing for an active 2D view.
- **Foundation Reactions** - Create and save support reactions drawings for an active 2D view for the currently selected loading
- Frame (page 2187) - Create a frame when in a 3D view (when a 2D view is displayed, this is inactive). The drop list underneath can be used to delete existing frames
- Free Points (page 2189) - Add, move, or delete points at specific co-ordinates to facilitate creation of other objects.
- Full UDL (page 2189) - Applies a full length UDL load to members

G

- **General Arrangement** - Create and save general arrangement drawing for an active 2D view
- Generate Detail Drawing... (page 976)
- Grid Line (page 2189) - Create a straight grid line between two points
- **Groups Window** - Open/close the groups tree window. Manage groups for general design and reinforced concrete detailing. Review objects assigned user defined attributes.

H, I, J

- **Horizontal Spectrum** - Display the horizontal spectrum generated by the seismic wizard
- IFC Export (page 2189) - Export a physical representation of the structural BIM model in IFC format
- Join (page 2189) - Joins members to make a longer continuous member
- Joist (page 2190) - Create a joist

L

- Level Load (page 2190) - Applies a level load to all slabs at the current level
- Line Load (page 2190) - Applies line load to any panel
- **Loadcases** - Opens the **Loading** dialog at the Loadcases page
- **Load Groups** - Opens the **Loading** dialog at the Groups page
- **Loading Window** - Open/close the loading tree window. This lists loadcases and combinations in a hirearchical way, and allows to review and delete applied loads and provides information and total load and reaction summations for each load combination.

- **Loading Plan** - Create and save loading information drawings for an active 2D view for the currently selected loading
- **Local Drift Snow** - Manually apply Local Drift Snow loads

M

- Mat Opening (page 2190) - Create an opening in a mat foundation
- Mat Overhang (page 2196) - Create an overhang at the edge of a mat
- Mat Split (page 2197) - Split mats into smaller panels for the purposes of pattern loading and mat design
- Mat Join (page 2197) - Join mats to create larger panels for the purposes of pattern loading and mat design
- Transfer property sets between models (page 1025) - Import, Export and Delete property sets
- **Manage View Configurations** - Opens or deletes saved View Configurations
- Measure (page 2190) - Measures the distance between any existing construction points or intersection points
- Measure Angle (page 2190) - Measures angles between existing points in 2D Views
- **Member Report** - Create or edit the chapters and options (content) of member reports
- Merge Planes (page 2190) - Merge two closely spaced planes
- Meshed Wall (page 2191) - Create a reinforced concrete shear wall characterized analytically by a mesh of 2D solver elements located along the wall insertion line and extending to all four corners
- **Mesh Slabs** - Slabs are meshed automatically for FE Load Decomposition, FE Chasedown, and also if specified in the 3D Analysis. This command is only used if you need to mesh slabs manually
- Mid-pier Wall (page 2192) - Create a reinforced concrete shear wall characterized analytically by a single vertical pier object of identical properties as the wall section, located at the centre of the wall and extending from the top to the bottom level
- Minimum Area (page 2192) - Create a minimum polygon area mat under selected objects (window to select all objects) including a defined overhang dimension
- Mirror (page 2192) - Select items then mirror
- **Model Report** - Create or edit the chapters and options (content) of model reports

- **Model Settings** - Opens the Model Settings dialog (page 2422) to specify settings for the current project
- Moment Load (page 2191) - Applies a Moment load to members
- Move (page 2191) - Select items then move
- Move Model (page 2191) - Moves the entire model to a new origin
- Move DXF Shadow (page 2191) - Move the dxf shadow to a new origin

N, O

- New (page 2192) - Create a new blank model
- Nodal Load (page 2192) - Applies a load to a node
- **Open** - Open a Tekla Structural Designer model
- Open view (page 2262)
- Open solver view (page 2262)

P

- Pad Base Column (page 2192) - Create a pad foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist
- **Page Setup** - Edit the paper size, orientation and margins
- Patch Column (page 2193) - Creates a localized slab reinforcement patch at a slab-column connection
- Patch Load (page 2193) - Applies a patch load to any panel
- Perimeter Load (page 2193) - Applies a load around the external perimeter of all slabs at a given level
- Pile (page 2193) - Create a pile under a mat foundation.
- Pile Array (page 2193) - Create a pile array under a mat foundation
- Pile Cap Column (page 2194) - Create a pile cap foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist
- Point Load (page 2194) - Applies a point load to any panel
- Polygon Load (page 2194) - Applies a polygonal load to any panel
- Point Load (page 2194)- Applies a Point load to members
- Portal Frame (page 2195) - Create a portal frame
- **Process Window** - Open/close the process window. This displays in a sequence each step of the analysis and design of the model and their statuses, as well as other useful information about program content.

- Project - Wiki (page 2194) - Opens the Project Wiki dialog for recording properties associated with the project
- **Properties Window** - Open/close the properties window. Input, review and edit properties related to ribbon commands, tree view items and individual or multiple objects selected in active views.
- Punching Check (page 2194) - Creates a punching check associated with the punching object

R

- Rationalize (page 2195) - Open a dialog to remove unused grids, planes, frames etc.
- Rectangular (page 2195) - Create a minimum rectangular area mat under selected objects including a defined overhang dimension (window to select all objects). Set the angle of the rectangle, or use absolute minimal area option
- Regroup Members (page 268)
- Remove Group (page 270)
- Rename (page 1033)
- Rename Group (page 269)
- Renumber (page 999)
- **Report Index** - Open/close the report index window. This displays the bookmarks for the active report view which can be used to locate and display a specific section within it, and allows bookmarks to be easily moved around the report.
- Reverse (page 2195) - Reverse Wall Panel Faces (inside becomes outside), or Reverse Beam Directions
- Robot Export (page 2195) - Export the analysis model to Autodesk Robot
- Roof Panel (page 2196) - Create roof panels for distributing loads and/or introducing diaphragm action

S

- **Save** - Save the model
- **Save As** - Saves the currently open model to a new name, or to a template
- **Schedule Management** - Opens a dialog for the generation of concrete beam, column and wall schedules on drawing sheets. The dialog can also be used to manage schedule revisions
- **Scene Content** - Open/close the scene content window. Manage by entity the displayed content in the 2D and 3D scene views.

- **Seismic Wizard** - Opens the Seismic wizard dialog for defining the parameters required for determining the seismic loading and load cases on the structure
- Select in visible views (page 2262)
- Set As Default Group (page 268)
- **Settings** (Design) - Opens the Design Settings dialog
- **Settings** (Drawing)- Opens the Drawing Settings dialog
- **Settings** (Report) - Change the report appearance
- **Settings** (Slab Deflection) - Open the slab deflection settings dialog
- Settlement Load (page 2196) - Applies a Settlement load (a translation or a rotation) to a support.
- Show references... (page 2262)
- **Show Report** - Display the selected report style on the screen
- **Simple Wind** - Create a Simple Wind Load that will be applied to rigid diaphragms
- **Slab/Mat Detailing** - Create and save slab or mat detailing drawings for an active 2D view
- Slab Load (page 2196) - Applies a slab load to all slab panels that constitute an individual slab.
- Slab on Beams (page 2196) - Create a slab
- Slab Opening (page 2196) - Create a rectangular or circular slab opening
- Slab Overhang (page 2196) - Create an overhang at the edge of a slab
- Slab Split (page 2197) - Split slab items into smaller panels for the purposes of pattern loading and slab design
- Slab Join (page 2197) - Join slabs to create larger panels for the purposes of pattern loading and slab design
- Sloped Plane (page 2197) - Create a sloped plane when in a 3D view (when a 2D view is displayed, this is inactive). The droplist underneath can be used to delete existing sloped planes
- **Snow Wizard** - Opens the Snow wizard dialog which is used to automate snow loadcase generation
- Split (page 2198) - Splits existing continuous members
- Split Group (page 268)
- Strip (page 2198) - Create a strip of mat between selected points with a defined width and end extension
- Strip Base Wall (page 2198) - Create a strip foundation under a concrete shear wall. These can only be created at the supported level of the wall and are only valid while the associated wall exists

- Structural BIM Import (page 2198) - Import a model from a Neutral File
- STAAD Export (page 2198) - Export the analysis model to STAAD
- **Status Window** - Open/close the status tree window. This displays information and statuses of modelling, stages of analyses, design and BIM.
- **Structure Window** - Open/close the structure tree window. This lists structural and non-structural components present in the model grouped by commonality in a hierarchical way, and allows for selective control over objects in the structure.
- Support (page 2199) - Create additional supports underneath existing members

T

- **Tabular Data** - Displays the analysis model data in spreadsheets for review/editing
- TCD Export (page 2199) - Export connections to Tekla Connection Designer
- Tekla Structures Export (page 2199) - Export the structural BIM model to Tekla Structures
- TEL File Import (page 2199) - Import a file that has been saved in the 'TEL' file format.
- Temperature Load (page 2200) - Applies a temperature load (a global rise in temperature) to individual elements/panels, selected elements/panels, or to all elements/panels
- Torsion Full UDL (page 2201) - Applies a Torsion Full UDL to members
- Torsion UDL (page 2201) - Applies a Torsion UDL to members
- Torsion VDL (page 2201) - Applies a Torsion VDL to members.
- TPFD Export (page 2199) - Export portal frames to Tekla Portal Frame Designer
- Trapezoidal Load (page 2201) - Applies a Trapezoidal load to members
- Truss (page 2200) - Runs the **Truss Wizard** to define a truss

U

- UDL (page 2201) - Applies a UDL load to members
- **Uniform Snow** - Manually apply Uniform Snow loads
- Update Connections (page 274)
- **Update Patterns** - If load patterns have been applied and the building geometry or loading is subsequently modified, this command should be run to ensure the load patterns reflect these changes

- **Update Snow Loads** - Update Snow Loads
- **Update Zones** - Recalculates the zoning details

V, W

- Variable Patch Load (page 2202) - Applies variable patch load to any panel
- Variable Area Load (page 2202) - Applies a variable area load to any panel. Area loads entirely cover an individual panel
- Vibration Check (page 2202)- Create a floor vibration check
- Walk (page 2203) - Walk through 3D views
- Wall Opening (page 2203) - Create a rectangular opening in an existing meshed shear wall when in a Frame
- View
- Wall Panel (page 2202) - Create wall panels for distributing loads
- **Wall Schedule** - Configure and then create a concrete wall schedule
- **Wind Loadcases** - Opens the Wind Loadcases dialog for defining the details of each wind loadcase
- **Wind Window** - Open/close the wind model tree window. This lists wind directions and loadcases in a hierachical way, and offers information about wind loadcases and pressure zone calculation status as well as control over wind model directions.
- **Wind Wizard** - The Wind Wizard dialog is used for defining the information that is required in order to calculate the wind loading on the structure

Ribbon commands

Descriptions of commands on the Ribbon.

Home tab

- New (page 2192) - Create a new blank model
- **Open** - Open a Tekla Structural Designer model
- **Close** - Close the model
- **Save** - Save the model
- **Save As** - Saves the currently open model to a new name, or to a template
- Project Wiki (page 2194) - Opens the Project Wiki dialog for recording properties associated with the project
- **Model Settings** - Opens the Model Settings dialog (page 2422) to specify settings for the current project

- Transfer property sets between models (page 1025) - Import, Export and Delete property sets
- **Manage View Configurations** - Opens or deletes saved View Configurations
- Find (page 353) - Find and select objects by typing a part of their name
- Walk (page 2203) - Walk through 3D views

BIM Integration tab

- Structural BIM Import (page 2198) - Import a model from a Neutral File
- TEL File Import (page 2199) - Import a file that has been saved in the 'TEL' file format
- 3D DXF Import (page 2180) - Import a model from a 3D DXF File
- Trimble Connect (page 2199) - Access Trimble's cloud-based project collaboration tool
- Tekla Structures Export (page 2199) - Export the structural BIM model to Tekla Structures
- TCD Export (page 2199) - Export connections to Tekla Connection Designer
- TPFD Export (page 2199) - Export portal frames to Tekla Portal Frame Designer
- Autodesk Revit Export (page 2180) - Export the structural BIM model to Autodesk Revit
- IFC Export (page 2189) - Export a physical representation of the structural BIM model in IFC format
- Cellbeam Export (page 2181) - Export a beam to Westok Cellbeam
- Cellbeam Import (page 2181) - Import a beam from Westok Cellbeam
- FBEAM Export (page 330) - Export a beam to FBEAM
- FBEAM Import (page 330) - Import a beam from FBEAM
- ADAPT Export (page 2180) - Export a model to ADAPT
- STAAD Export (page 2198) - Export the analysis model to STAAD
- Robot Export (page 2195) - Export the analysis model to Autodesk Robot
- Cloud Export (page 2182) - Export the model to the cloud

Model tab

- Construction Levels (page 2248) - Opens the Construction Levels dialog (page 2402) allowing you to define the levels required in order to construct your model

- Frame (page 2187) - Create a frame when in a 3D view (when a 2D view is displayed, this is inactive). The drop list underneath can be used to delete existing frames
- Sloped Plane (page 2197) - Create a sloped plane when in a 3D view (when a 2D view is displayed, this is inactive). The droplist underneath can be used to delete existing sloped planes
- Grid Line (page 2189) - Create a straight grid line between two points
- Construction Line (page 2183) - Create a straight construction line between two points
- Column (page 2183) - Create a column
- Beam (page 2181) - Create a beam
- Brace (page 2181) - Create a brace
- Joist (page 2190) - Create a joist
- Truss (page 2200) - Runs the **Truss Wizard** to define a truss
- Portal Frame (page 2195) - Create a portal frame
- Meshed Wall (page 2191) - Create a reinforced concrete shear wall characterized analytically by a mesh of 2D solver elements located along the wall insertion line and extending to all four corners
- Mid-pier Wall (page 2192) - Create a reinforced concrete shear wall characterized analytically by a single vertical pier object of identical properties as the wall section, located at the centre of the wall and extending from the top to the bottom level
- Wall Opening (page 2203) - Create a rectangular opening in an existing meshed shear wall when in a Frame View
- Cores (page 2183) - Assign existing concrete members as part of a concrete core
- Slab on Beams (page 2196) - Create a slab
- Slab Opening (page 2196) - Create a rectangular or circular slab opening
- Slab Overhang (page 2196) - Create an overhang at the edge of a slab
- Column Drop (page 2183) - Create a column drop
- Slab Split (page 2197) - Split slab items into smaller panels for the purposes of pattern loading and slab design
- Slab Join (page 2197) - Join slabs to create larger panels for the purposes of pattern loading and slab design
- Roof Panel (page 2196) - Create roof panels for distributing loads and/or introducing diaphragm action
- Wall Panel (page 2202) - Create wall panels for distributing loads
- Support (page 2199) - Create additional supports underneath existing members

- Element (page 2187) - Create analysis elements
- Measure (page 2190) - Measures the distance between any existing construction points or intersection points
- Measure Angle (page 2190) - Measures angles between existing points in 2D Views
- Dimension (page 2187) - Create dimensions between appropriate points which can be included on drawings
- Bearing Wall (page 2181) - Creates a bearing wall
- Validate (page 2201) - Validate the structure for physical model issues.

Edit tab

- Copy (page 2183) - Select items then copy
- Copy Loads (page 2182) - Copy between objects in the current case, or copy from one case to another
- Move (page 2191) - Select items then move
- Mirror (page 2192) - Select items then mirror
- **Delete** - Delete Selected Items
- Join (page 2189) - Joins members to make a longer continuous member
- Split (page 2198) - Splits existing continuous members
- Reverse (page 2195) - Reverse Wall Panel Faces (inside becomes outside), or Reverse Beam Directions
- Beam Lines (page 2181) - Joins existing concrete beams to make a continuous concrete beam
- Cutting Planes (page 2183) - Move/Activate/Deactivate planes to isolate a smaller visible working area within the model
- Move Model (page 2191) - Moves the entire model to a new origin
- Rationalize (page 2195) - Open a dialog to remove unused grids, planes, frames etc.
- Move DXF Shadow (page 2191) - Move the dxf shadow to a new origin
- Create Infills (page 2183) - Creates a pattern of infill members in the selected bay
- Merge Planes (page 2190) - Merge two closely spaced planes
- Free Points (page 2189) - Add, move, or delete points at specific co-ordinates to facilitate creation of other objects.

Load tab

Structure

- **Loadcases** - Opens the **Loading** dialog at the Loadcases page
- **Load Groups** - Opens the **Loading** dialog at the Groups page
- **Combination** - Opens the **Loading** dialog at the Combinations page
- **Envelope** - Opens the **Loading** dialog at the Envelopes page
- **Update Patterns** - If load patterns have been applied and the building geometry or loading is subsequently modified, this command should be run to ensure the load patterns reflect these changes

Wind, Snow & Seismic

- **Wind Wizard** - The Wind Wizard dialog is used for defining the information that is required in order to calculate the wind loading on the structure
- **Update Zones** - Recalculates the zoning details
- **Wind Loadcases** - Opens the Wind Loadcases dialog for defining the details of each wind loadcase
- **Delete Wind** - Deletes the wind model data previously defined using the Wind Wizard
- **Decomposition** (Wind Load) - Decompose the wind loads on to the structure
- **Simple Wind** - Create a Simple Wind Load that will be applied to rigid diaphragms
- **Snow Wizard** - Opens the Snow wizard dialog which is used to automate snow loadcase generation
- **Uniform Snow** - Manually apply Uniform Snow loads
- **Valley Snow** - Manually apply Valley Snow loads
- **Local Drift Snow** - Manually apply Local Drift Snow loads
- **Update Snow Loads** - Update Snow Loads
- **Delete Snow** - Deletes the snow model data previously defined using the Snow Wizard
- **Seismic Wizard** - Opens the Seismic wizard dialog for defining the parameters required for determining the seismic loading and load cases on the structure
- **Horizontal Spectrum** - Display the horizontal spectrum generated by the seismic wizard
- **Delete Seismic** - Delete all of the seismic data entered in the seismic wizard along with the horizontal spectrum, seismic loadcases and seismic load combinations

- **Decomposition** - Decompose loads from 1-way and 2-way slabs onto supporting members.

Panel Loads

- Point Load (page 2194) - Applies a point load to any panel
- Line Load (page 2190) - Applies line load to any panel
- Patch Load (page 2193) - Applies a patch load to any panel
- Polygon Load (page 2194) - Applies a polygonal load to any panel
- Perimeter Load (page 2193) - Applies a load around the external perimeter of all slabs at a given level
- Variable Patch Load (page 2202) - Applies variable patch load to any panel
- Area Load (page 2180) - Applies an area load to any panel. Area loads entirely cover an individual panel
- Variable Area Load (page 2202) - Applies a variable area load to any panel. Area loads entirely cover an individual panel
- Level Load (page 2190) - Applies a level load to all slabs at the current level
- Slab Load (page 2196) - Applies a slab load to all slab panels that constitute an individual slab.

Member Loads

- Full UDL (page 2189) - Applies a full length UDL load to members
- UDL (page 2201) - Applies a UDL load to members
- VDL (page 2202) - Applies a VDL load to members
- Trapezoidal Load (page 2201) - Applies a Trapezoidal load to members
- Point Load (page 2194)- Applies a Point load to members
- Moment Load (page 2191) - Applies a Moment load to members
- Torsion Full UDL (page 2201) - Applies a Torsion Full UDL to members
- Torsion UDL (page 2201) - Applies a Torsion UDL to members
- Torsion VDL (page 2201) - Applies a Torsion VDL to members.

Structure Loads

- Diaphragm Load (page 547) - Applies a load to a rigid or semi-rigid diaphragm
- Diaphragm Table (page 547) - Applies multiple diaphragm loads at multiple levels
- Nodal Load (page 2192) - Applies a load to a node
- Temperature Load (page 2200) - Applies a temperature load (a global rise in temperature) to individual elements/panels, selected elements/panels, or to all elements/panels

- Settlement Load (page 2196) - Applies a Settlement load (a translation or a rotation) to a support.

Analyse tab

- 1st Order Linear (page 2178) - Runs a linear static analysis
- 1st Order Non-Linear (page 2178) - Runs a nonlinear analysis with loading applied in a single step
- 1st Order Vibration (page 2178) - Runs an unstressed vibration analysis to determine the structure's natural frequencies
- 2nd Order Linear (page 2179) - Runs a 2-stage P-Delta analysis
- 2nd Order Non-Linear (page 2179) - Runs a nonlinear analysis with loading applied in a single step
- 2nd Order Buckling (page 2179) - Runs a linear buckling analysis to determine a structure's susceptibility to buckling
- Analyze All (Static) (page 669) - Performs a full series of analyses that would be carried out as part of 'Design All (Static)' but with no design. Includes all relevant analyses. Analysis type (e.g. second-order non-linear) is the same as set in the design settings
- 3D only (Static) (page 669) - Performs same analyses as 'Analyze All (Static)' but excludes chasedowns. This can save time during scheme design, for example while addressing overall stability, sway, drift, wind drift, etc.
- **1st Order RSA Seismic** - Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis
- **2nd Order RSA Seismic** - Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis
- **Model & Results** - Displays the solver model data in spreadsheets for review/editing
- **Wind Tunnel** - Displays the wind tunnel data tab, and wind model data in spreadsheets for running modal analysis and reviewing the results
- **Mesh Slabs** - Slabs are meshed automatically for FE Load Decomposition, FE Chasedown, and also if specified in the 3D Analysis. This command is only used if you need to mesh slabs manually. After the operation the slab mesh is automatically switched on in the scene content.

Design tab

- **Settings** - Opens the Design Settings dialog
- Validate (page 2201) - Validates the model for design issues
- Design Steel (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all steel members in the model based on their individual 'auto-design' setting
- Design Steel (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting
- Design Steel (RSA) (page 2184) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting
- Design Concrete (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all concrete members in the model based on their individual 'auto-design' setting
- Design Concrete (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting
- Design Concrete (RSA) (page 2184) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting
- Design All (Gravity) (page 2184) - Performs an analysis for gravity combinations only, and then designs all members in the model based on their individual 'auto-design' setting
- Design All (Static) (page 2184) - Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting
- Design All (RSA) (page 2185) - Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting
- Design Slabs (page 2185) - Designs two-way concrete slabs for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting
- Patch Column (page 2193) - Creates a localized slab reinforcement patch at a slab-column connection
- Design Patches (page 2185) - Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting
- Punching Check (page 2194) - Creates a punching check associated with the punching object

- Design Punching Shear (page 2185) - Perform a check or design (according to individual autodesign settings) of all punching checks in the model
- Vibration Check (page 2202)- Create a floor vibration check
- Check Floor Vibration (page 2181) - Perform all floor vibration checks in the model
- Check in Tedds (page 2182) - Checks all members where a linked Tedds calculation has been created.

Slab Deflection tab

- **Settings** (Slab Deflection) - Open the slab deflection settings dialog
- **Event Sequences** - Submodel Event Sequences give access to override aspects of the model event sequence on a submodel basis
- **Deflection Checks** - Opens the Slab Deflection Check Catalogue for defining the default check requirements associated with Check Lines (also associated with events)
- **Analyse Current** - Performs a slab deflection analysis for the current sub model
- **Analyse All** - Performs a slab deflection analysis of all submodels regardless of current view/selection
- **Analyse Selected** - Opens dialog allowing analysis of selected submodels.

Foundations tab

- **Settings** - Opens the Design Settings dialog
- **Calalogue** - Add and delete piles from your pile catalogue or edit their properties (e.g. compression and tension capacity)
- Pad Base Column (page 2192) - Create a pad foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist
- Strip Base Wall (page 2198) - Create a strip foundation under a concrete shear wall. These can only be created at the supported level of the wall and are only valid while the associated wall exists
- Pile Cap Column (page 2194) - Create a pile cap foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist
- Design Pad Bases (page 2186) - Design pad bases to all valid combinations based on their individual 'auto-design' settings
- Design Pile Caps (page 2186) - Design pile caps to all valid combinations based on their individual 'auto-design' settings

- Minimum Area (page 2192) - Create a minimum polygon area mat under selected objects (window to select all objects) including a defined overhang dimension
- Rectangular (page 2195) - Create a minimum rectangular area mat under selected objects including a defined overhang dimension (window to select all objects). Set the angle of the rectangle, or use absolute minimal area option
- Strip (page 2198) - Create a strip of mat between selected points with a defined width and end extension
- Area (page 2180) - Create a polygon area mat by selecting perimeter points and including a defined overhang dimension
- Bays (page 2180) - Create a mat by selecting bays bounded by grid lines
- Mat Opening (page 2190) - Create an opening in a mat foundation
- Mat Overhang (page 2196) - Create an overhang at the edge of a mat
- **Mat Split** - Split mats into smaller panels for the purposes of pattern loading and mat design
- **Mat Join** - Join mats to create larger panels for the purposes of pattern loading and mat design
- Pile (page 2193) - Create a pile under a mat foundation.
- Pile Array (page 2193) - Create a pile array under a mat foundation
- Design Mats (page 2186) - Design ground bearing and piled mat foundations for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting
- Design Patches (page 2185) - Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting
- Punching Check (page 2194) - Creates a punching check associated with the punching object
- Design Punching Shear (page 2185) - Check of all punching shear checks in the model.

Report tab

- **Model Report** - Create or edit the chapters and options (content) of model reports
- **Member Report** - Create or edit the chapters and options (content) of member reports
- **Show Report** - Display the selected report style on the screen
- **Filters** - Set up filters to be used when generating a new report
- **Settings** - Change the report appearance

- **Page Setup** - Edit the paper size, orientation and margins
- **Edit Header** - Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers
- **Edit Footer** - Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers
- **Report Index** - Display the report index window to allow you to jump to specific chapters and options (content) within the report.

Draw tab

- **Settings** - Opens the Drawing Settings dialog
- **Drawing Management** - Opens a dialog for the generation and laying out of multiple drawings on to a single drawing sheet. The dialog can also be used to manage drawing revisions
- **Schedule Management** - Opens a dialog for the generation of concrete beam, column and wall schedules on drawing sheets. The dialog can also be used to manage schedule revisions
- **Beam Schedule** - Configure and then create a concrete beam schedule for members in an active 2D view
- **Column Schedule** - Configure and then create a concrete column schedule
- **Wall Schedule** - Configure and then create a concrete wall schedule
- **General Arrangement** - Create and save general arrangement drawing for an active 2D view
- **Beam End Forces** - Create and save beam end forces drawing for an active 2D view for the currently selected loading
- **Column Splice Loads** - Create and save steel column splice loads drawings for an active 2D frame view for the currently selected loading
- **Foundation Reactions** - Create and save support reactions drawings for an active 2D view for the currently selected loading
- **Loading Plan** - Create and save loading information drawings for an active 2D view for the currently selected loading
- **Slab/Mat Detailing** - Create and save slab or mat detailing drawings for an active 2D view
- **Foundation Layout** - Create and save foundation layout drawing for an active 2D view.

Windows tab

- **Structure Window** - Open/close the structure tree window. This lists structural and non-structural components present in the model grouped by commonality in a hierarchical way, and allows for selective control over objects in the structure.

- **Groups Window** - Open/close the groups tree window. Manage groups for general design and reinforced concrete detailing. Review objects assigned user defined attributes.
- **Loading Window** - Open/close the loading tree window. This lists loadcases and combinations in a hirearchical way, and allows to review and delete applied loads and provides information and total load and reaction summations for each load combination.
- **Report Index** - Open/close the report index window. This displays the bookmarks for the active report view which can be used to locate and display a specific section within it, and allows bookmarks to be easily moved around the report.
- **Wind Window** - Open/close the wind model tree window. This lists wind directions and loadcases in a hierachical way, and offers information about wind loadcases and pressure zone calculation status as well as control over wind model directions.
- **Status Window** - Open/close the status tree window. This displays information and statuses of modelling, stages of analyses, design and BIM.
- **Properties Window** - Open/close the properties window. Input, review and edit properties related to ribbon commands, tree view items and individual or multiple objects selected in active views.
- **Scene Content** - Open/close the scene content window. Manage by entity the displayed content in the 2D and 3D scene views.
- **Process Window** - Open/close the process window. This displays in a sequence each step of the analysis and design of the model and their statuses, as well as other useful information about program content.

Review tab

- **Type droplist** - Filters status and ratio displays to show results for Static designs, RSA designs or the Combined designs.

Design

- **Status** - Display member status.
- **Ratio** - Display member utilisation ratio.
- **Depth Ratio** - Display beam span to depth utilisation ratio.

Foundations

- **Status** - Display foundation status.
- **Ratio** - Display foundation utilisation ratio.

Piles

- **Status** - Display pile status.
- **Ratio** - Display pile utilisation ratio.

Slab/Mat Design

- **Status** - Display slab/mat status.
- **Ratio** - Display slab/mat utilisation ratio.
- **Slab/Mat droplist** - Filters the status and ratio displays to show results for the selected category.
- **Status** - Display connection status.
- **Ratio** - Display connection utilisation ratio.

Connections

- **Status** - Display connection status.
- **Ratio** - Display connection utilisation ratio.

Show/Alter State

- **Auto/Check Design** - Each member is color coded to indicate its autodesign setting (On or Off). Clicking on a member toggles its setting.
- **Diaphragm On/Off** - Each slab panel and diaphragm node is color coded to indicate its setting (Excluded or Included). Clicking on a slab or a diaphragm node toggles its setting.
- **Fixed/Pinned** - Each member is color coded to indicate its end fixity setting (N/A, Pinned, Fixed, Moment, Mixed, Cantilever). Clicking on a member toggles its fixity setting between those that are applicable.
- **BIM Status** - Each member is color coded to indicate its BIM status.
- **Slab Reinforcement** - Use property grid to review and edit or rationalize panel or patch reinforcement.
- **Section Material/Grade** - Each steel member is color coded to indicate its section and grade. Clicking on a steel member updates its section and/or grade to match what you have set in the Properties window.
- **Copy Properties** - Copy properties from one member to another. After firstly selecting a parameter in the Properties window you are able to copy it from a designated source member to valid target members.
- **Report Filter** - Only accessible once a Member filter has been defined. Each member is color coded to indicate if it is included in the currently selected filter. Clicking a member toggles its inclusion status.
- **Sub Structures** - Create selected sub-sets of the structure and use for more focused editing, design, review and reporting purposes.
- **Concrete Beam Flanges** - Each concrete beam is color coded to indicate its if flanges are considered and flange widths determined.
- **Column Splices** - Potential splice locations in steel columns are color coded to indicate where splices exist.
- **Property Sets** - Graphically review and modify the property set assigned to the elements by using the property grid. In a typical model you may wish to

apply the same properties to similar model objects. This can be done efficiently using property sets.

- **UDA** - Eraphically review and modify the attribute values assigned to the elements by using the property grid. User defined attributes allow you to save your own data to individual elements. These can be used to select similar elements based upon their UDA group or filter report content based on a UDA.
- **Show/Alter State** - The full range of options is available by selecting this and using the attribute drop down in the property grid.

Design Data

- **Tabular Data** - Open the review data view in which all sorts of design results and quantities information can be reviewed.

1st Order Linear (command)

Runs a linear static analysis.

This analysis type is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: No

1st Order Non-linear (command)

Runs a nonlinear analysis with loading applied in a single step.

This analysis type is suitable for structures where secondary effects are negligible and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: Yes

1st Order Modal (command)

Runs an unstressed modal analysis to determine the structure's natural frequencies.

The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly.

Nonlinearity Included:

- Geometric: No
- Material: No

2nd Order Buckling (command)

Runs a linear buckling analysis to determine a structure's susceptibility to buckling.

The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

2nd Order Linear (command)

Runs a 2-stage P-Delta analysis.

This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

2nd Order Non-linear (command)

Runs a nonlinear analysis with loading applied in a single step.

This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: Yes

3D DXF Import (command)

Import a model from a 3D DXF File.

See also:

- Import data from a 3D DXF file (page 308)

ADAPT Export (command)

Export a model to ADAPT.

See also:

- Export a model to ADAPT (page 335)

Area Load (command)

Applies an area load to any panel. Area loads entirely cover an individual panel.

See also:

- Create area loads (page 543)

Area Mat (command)

Create a polygon area mat by selecting perimeter points and including a defined overhang dimension.

See also:

- Create an area mat (page 843)

Autodesk Revit Export (command)

Export the structural BIM model to Autodesk Revit including synchronization of model data if applicable.

See also:

- Export a model to Autodesk Revit Structure (page 327)

Bays Mat (command)

Create a mat by selecting bays bounded by grid lines.

See also:

- Create a mat within bays (page 843)

Beam (command)

Create a rolled steel, concrete, timber, or cold formed beam.

Beam Lines (command)

Joins existing concrete beams to make a continuous concrete beam.

See also:

- Automatically join all concrete beams (page 505)

Bearing Wall (command)

Creates a wall which resists only axial forces and does not contribute to lateral stability.

Brace (command)

Create a rolled steel, concrete, timber, or cold formed brace .

Cellbeam Export (command)

Export a beam to Westok Cellbeam

See also:

- Export to and import from Westok Cellbeam (page 329)

Cellbeam Import (command)

Import a beam from Westok Cellbeam

See also:

- Export to and import from Westok Cellbeam (page 329)

Check Floor Vibration (command)

Perform all floor vibration checks in the model.

See also:

- Check vibration for all floor vibration check items (page 809)

Check in Tedds (command)

Command	Description
Check in Tedds	Located on the **Design** ribbon tab, this command can be used after any precast or timber members have been initially designed. It is intended for use where a general re-check for all objects is required following changes to the model which require a reanalysis process (e.g. changing the loading applied onto the model). The command finds all members where a linked Tekla Tedds calculation has been created and performs a check design for these members. This command should ideally be used at the end of the design cycle and ensures that the previously designed timber, or precast concrete elements still meet the design criteria of the model. **NOTE** **Check in Tedds** does not perform an analysis before the check - it always uses the latest existing set of analysis results, (even if the analysis status is 'Out of Date').

See also

Timber member design handbook (page 1564)

Precast member design handbook (page 1527)

Cloud Export (command)

Export to the Cloud.

See also:

- Export a model to the cloud (page 341)

Copy Loads (command)

Copy between objects in the current case, or copy from one case to another.

See also:

- Copy loads (page 500)

Column (command)

Create a rolled steel, concrete, timber, or cold formed column.

Copy (command)

Select items then copy.

Selection can be filtered.

See also:

- Copy and rotate objects (page 495)

Column drop (command)

Create a column drop.

See also:

- Create column drops (page 459)

Construction Line (command)

Create a straight construction line between two points.

See also:

- Create and manage architectural grids and grid lines (page 370)

Cores (command)

Assign existing concrete members as part of a concrete core.

Cutting Planes (command)

Move/Activate/Deactivate planes to isolate a smaller visible working area within the model.

See also:

- Manage cutting planes (page 507)

Create Infills (command)

Creates a pattern of infill members in the selected bay.

See also:

- Create infill members (page 509)

Design Steel Gravity (command)

Performs an analysis for gravity combinations only, and then designs all steel members in the model based on their individual 'auto-design' setting.

Design Steel Static (command)

Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting.

Design Steel RSA (command)

Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting.

Design Concrete Gravity (command)

Performs an analysis for gravity combinations only, and then designs all concrete members in the model based on their individual 'auto-design' setting.

Design Concrete Static (command)

Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting.

Design Concrete RSA (command)

Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting.

Design All Gravity (command)

Performs an analysis for gravity combinations only, and then designs all members in the model based on their individual 'auto-design' setting.

Design All Static (command)

Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting.

Design All RSA (command)

Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting.

Design Slabs (command)

Designs two-way concrete slabs for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting.

NOTE **Design Slabs** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design Patches (command)

Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting.

NOTE **Design Patches** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Design or check all patches in the model (page 800)

Design Punching Shear (command)

Depending on how it is accessed, this command will:

- Perform a check or design (according to individual autodesign settings) of all punching checks in the model.
- Perform a check or design (according to individual autodesign settings) of all punching checks in a level.
- Perform a check or design (according to individual autodesign settings) of an individual punching shear check.

NOTE **Design Punching Shear** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design Mats (command)

Design ground bearing and piled mat foundations for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting.

NOTE **Design Mats** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Design or check all mats in the model (page 846)

Design Pad Bases (command)

Design pad bases to all valid combinations based on their individual 'auto-design' settings.

NOTE **Design Pad Bases** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Design or check all pad bases and strip bases (page 840)

Design Pile Caps (command)

Design pile caps to all valid combinations based on their individual 'auto-design' settings.

NOTE **Design Pile Caps** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

- Design or check all pile caps (page 840)

Diaphragm Load (command)

Applies a point load in Building Direction 1 and 2 and a torsion load at any position within a rigid or semi-rigid diaphragm.

NOTE Diaphragm loads can only be applied in 2D views (not 3D)

Property	Description
F, Dir 1	Point load in Building Direction 1
F, Dir 2	Point load in Building Direction 2
Mz	Torsion load about Global Z
X	Distance from origin in Global X
Y	Distance from origin in Global Y

See also:

- Add a diaphragm load in a 2D view (page 547)

Dimension (command)

Create dimensions between appropriate points which can be included on drawings.

Element (command)

Create analysis elements.

Frame (command)

Create a frame when in a 3D view (when a 2D view is displayed, this is inactive). The drop list underneath can be used to delete existing frames.

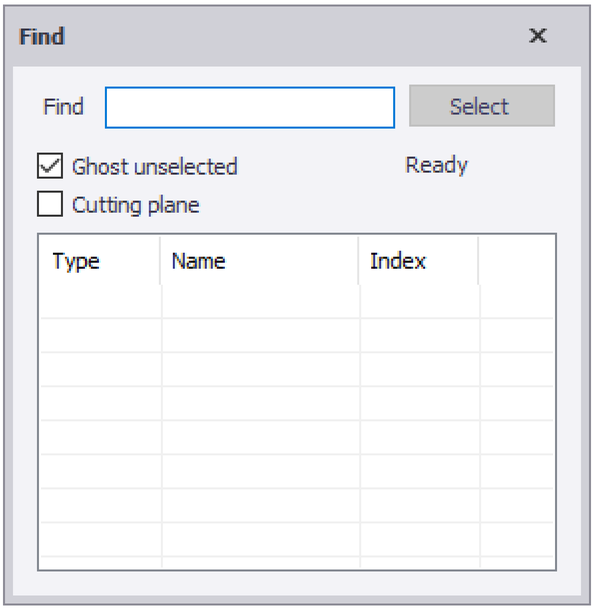
Only those members that lie within the plane of the frame are displayed.

See also:

- Create a frame (page 386)

Find (command)

Find and then select objects in the model by typing a part of their name.



Setting	Description
Find	In the Find box you can type any of the following, in whole, or in part: • An object 'Type' • An object 'Name' • An object 'Index' As you type, any objects that match the criteria are listed in the table below. When the object(s) that you require are shown, click to highlight them as follows: • Single click to highlight a single object • Ctrl+click to select multiple non-sequential rows • Shift+click to select multiple sequential rows
Select	Once the object(s) are highlighted, **Select** becomes available. Click **Select** to zoom in and locate the object in the active view. If the object doesn't exist in the active view a new view containing the object will become active.
Ghost unselected	When clicking **Select**, it can be easy for the object being located to be obscured from view by other objects. By leaving **Ghost**

Setting	Description
	unselected checked this is avoided as all objects, other than those being found are made semi-transparent.
Cutting plane	Having clicked **Select** if you want to make the located object yet easier to see, you can check the **Cutting plane** option. This automatically applies a cutting plane perpendicular to the current point of view which removes everything in the foreground of the object(s) that have been located.

See also:

- Select using Find (page 353)

Free Points (command)

Add, move, or delete points at specific co-ordinates to facilitate creation of other objects.

See also:

- Create and manage free points (page 511)

Full UDL (command)

Applies a full length UDL load to members.

See also:

- Create full-length UDLs (page 544)

Grid Line (command)

Create a straight grid line between two points.

See also:

- Create and manage architectural grids and grid lines (page 370)

IFC Export (command)

Export a physical representation of the structural BIM model in IFC format for use in other products.

See also:

- Export a model to IFC (page 328)

Join (command)

Joins members to make a longer continuous member.

See also:

- Join and split members (page 504)

Joist (command)

Create a joist.

Level Load (command)

Applies a level load to all slabs at the current level.

See also:

- Create level loads (page 543)

Line Load (command)

Applies line load to any panel.

NOTE Point Loads and Line loads can only be applied in 2D views (not 3D)

See also:

- Create line loads (page 539)

Mat Opening (command)

Create an opening in a mat foundation.

Active in 2D views.

See also:

- Create slab or mat openings (page 455)

Measure (command)

Measures the distance between any existing construction points or intersection points.

Measure Angle (command)

Measures angles between existing points in 2D Views.

Merge Planes (command)

Merge two closely spaced planes. Where possible, everything on the source plane is moved to the destination plane.

See also:

- Merge planes (page 511)

Meshed Wall (command)

Create a reinforced concrete shear wall characterized analytically by a mesh of 2D solver elements located along the wall insertion line and extending to all four corners. Non-rectangular and sloping wall geometry is allowed as well as openings.

The type of 2D solver elements used is controlled from the structure properties and can be overriden on a wall by wall basis in the individual wall properties.

Moment Load (command)

Applies a Moment load to members.

See also:

- Create point loads and moment loads (page 545)

Move (command)

Select items then move.

Selection can be filtered.

See also:

- Move and rotate objects (page 495)

Move DXF Shadow (command)

Move the dxf shadow to a new origin.

See also:

- Move the model or the DXF shadow (page 508)

Move Model (command)

Moves the entire model to a new origin.

See also:

- Move the model or the DXF shadow (page 508)

Mid-pier Wall (command)

Create a reinforced concrete shear wall characterized analytically by a single vertical pier object of identical properties as the wall section, located at the centre of the wall and extending from the top to the bottom level.

Mid-pier concrete shear wall restrictions:

- must be rectangular
- must be in a vertical plane
- openings are not allowed

Mirror (command)

Select items then mirror.

Selection can be filtered.

See also:

- Mirror objects to new locations (page 496)

Minimum Area Mat (command)

Create a minimum polygon area mat under selected objects (window to select all objects) including a defined overhang dimension.

See also:

- Create a minimum area or rectangular mat (page 842)

New (command)

FROM ICON: Create a new blank project from scratch.

FROM DROPDOWN: Create a new project based on an existing template.

See also:

- Start a new project (page 243)

Nodal Load (command)

Applies a load to a node.

See also:

- Create nodal loads (page 554)

Pad Base Column (command)

Create a pad foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist.

See also:

- Create pad base columns (page 836)

Patch Column (command)

Creates a localized slab reinforcement patch at a slab-column connection.

Defaults to top surface of slab.

See also:

- Create column patches (page 790)

Patch Load (command)

Applies a patch load to any panel.

NOTE Patch Loads can only be applied in 2D views (not 3D)

See also:

- Create patch loads (page 540)

Perimeter Load (command)

Applies a load around the external perimeter of all slabs at a given level .

See also:

- Create perimeter loads (page 541)

Pile (command)

Create a pile under a mat foundation.

See also:

- Place piles and pile arrays in mats (page 844)

Pile Array (command)

Create a pile array under a mat foundation.

See also:

- Place piles and pile arrays in mats (page 844)

Pile Cap Column (command)

Create a pile cap foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist.

See also:

- Create pile caps (page 838)

Point Load [Member] (command)

Applies a Point load to members.

See also:

- Create point loads and moment loads (page 545)

Point Load [Panel] (command)

Applies a point load to any panel.

NOTE Point Loads and Line loads can only be applied in 2D views (not 3D)

See also:

- Create point loads (page 539)

Polygon Load (command)

Applies a polygonal load to any panel.

NOTE Polygon Loads can only be applied in 2D views (not 3D)

See also:

- Create polygonal loads (page 540)

Punching Check (command)

Creates a punching check associated with the punching object.

Project Wiki (command)

Opens the **Project Wiki** dialog which is used to record miscellaneous properties associated with the project, and to record revisions.

See also:

- Modify project details (page 244)

Portal Frame (command)

Create a portal frame.

To create the frame, you firstly define the two column base positions between which the frame will lie. The base positions are restricted to lie on existing grid points.

All other details of the portal frame are subsequently entered in the **Portal Frame** dialog.

See also:

- Create portal frames (page 467)

Rationalize (command)

Open a dialog which allows you to:

- Remove unused sloped planes and frames
- Remove unused grid lines
- Remove unused construction lines
- Update all grid and construction lines to extend a short distance beyond the point they are required

See also:

- Rationalize the model (page 509)

Rectangular Mat (command)

Create a minimum rectangular area mat under selected objects including a defined overhang dimension (window to select all objects). Set the angle of the rectangle, or use absolute minimal area option.

See also:

- Create a minimum area or rectangular mat (page 842)

Reverse (command)

Reverse Wall Panel Faces (inside becomes outside), or Reverse Beam Directions.

See also:

- Reverse member axes and panel faces (page 506)

Robot Export (command)

Export the analysis model to Autodesk Robot.

See also:

- Export a model to Autodesk Robot Structural Analysis (page 340)

Roof Panel (command)

Create roof panels for distributing loads and/or introducing diaphragm action.

Settlement Load (command)

Applies a Settlement load (a translation or a rotation) to a support.

See also:

- Create settlement loads (page 554)

Slab on Beams (command)

Create a slab on beams.

Slab Load (command)

Applies a slab load to all slab panels that constitute an individual slab.

See also:

- Create slab loads (page 543)

Slab/Mat Opening (command)

Create a rectangular or circular slab/mat opening.

Active in 2D views.

See also:

- Create slab or mat openings (page 455)

Slab/Mat Overhang (command)

Create an overhang at the edge of a slab.

Active in 2D views.

Command Properties	
LengthofBeam	• When checked, the overhang is created by a single click on a supporting beam along the the slab edge.

Command Properties	
	• When unchecked, the overhang is created by clicking two points along the slab edge.
EdgeParallel	• When checked, the overhang is created with a straight edge. • When unchecked, the overhang is created with a curved edge.
Curvature	Specifies the overhang curvature, (when EdgeParallel is unchecked).
Tapered	• When checked, the overhang varies in width. (Width1 to Width2) • When unchecked, the overhang is a constant width (Width1).
Width1	The width at end 1 of the overhang.
Width2	The width at end 2 of the overhang, (when tapered).

See also:

- Add overhangs to existing slab or mat edges (page 457)

Slab/Mat Split (command)

Split slab items/mats into smaller panels for the purposes of pattern loading and slab/mat design.

Active in 2D views.

See also:

- Split and join slabs and mats (page 462)

Slab/Mat Join (command)

Join slabs/mats to create larger panels for the purposes of pattern loading and slab/mat design.

Active in 2D views.

See also:

- Split and join slabs and mats (page 462)

Sloped Plane (command)

Create a sloped plane when in a 3D view (when a 2D view is displayed, this is inactive). The droplist underneath can be used to delete existing sloped planes.

A sloped plane is a 2D View of the model created in a sloped plane. It is defined by selecting 3 existing grid points.

Because only those members that lie within the plane of the slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps.

See also:

- Create a slope (page 387)

Split (command)

Splits existing continuous members.

See also:

- Join and split members (page 504)

STAAD Export (command)

Export the analysis model to STAAD.

See also:

- Export a model to STAAD (page 339)

Strip Base Wall (command)

Create a strip foundation under a concrete shear wall. These can only be created at the supported level of the wall and are only valid while the associated wall exists.

See also:

- Create strip base walls (page 837)

Strip Mat (command)

Create a strip of mat between selected points with a defined width and end extension

See also:

- Create a strip mat (page 843)

Structural BIM Import (command)

Import a model from a Neutral File. (.cxl)

See also:

- Import a project from a Structural BIM Import file (page 301)

Support (command)

Create additional supports underneath existing members.

TCD Export (command)

Export connections to Tekla Connection Designer.

See also:

- Export to Tekla Connection Designer (page 318)

Tekla Structures Export (command)

Export the structural BIM model to Tekla Structures including synchronization of model data if applicable.

See also:

- Export a model to Tekla Structures (page 318)

TEL File Import (command)

Create a design model by importing a file that has been saved in the 'TEL' file format.

See also:

- Import a project from a TEL file (page 302)

TPFD Export (command)

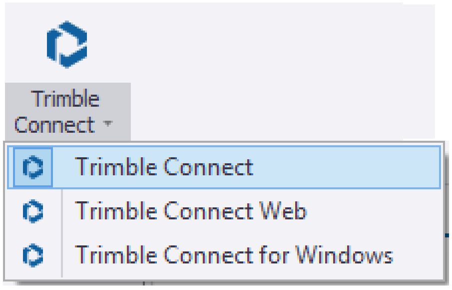
Export portal frames to Tekla Portal Frame Designer.

See also:

- Export to Tekla Portal Frame Designer (page 319)

Trimble Connect (command)

The **Trimble Connect** droplist on the **BIM Integration** tab gives a choice of 3 options:



- **Trimble Connect:** This opens the *Trimble Connect Project Explorer*, a dialog that controls the flow of information between the Tekla Structural Designer model and the Trimble Connect project.

  NOTE Once opened, the Trimble Connect Project Explorer can either be docked, or auto-hidden to a tab on the right of the interface as required.

- **Trimble Connect Web:** This launches the in-browser web app. If the open model is associated with a Trimble Connect project, the project itself is opened, otherwise a default Trimble Connect page is opened.
- **Trimble Connect for Windows:** This launches the Windows application. Again, if the open model is associated with a Trimble Connect project then that project is opened.

NOTE The Tekla Structural Designer model can only be attached to a Trimble Connect project from the Trimble Connect Project Explorer, and not from the web or Windows apps. The model must be attached before any information can move between Tekla Structural Designer and Trimble Connect.

See also:

- Working collaboratively with Trimble Connect (page 310)

Truss (command)

Runs the **Truss Wizard** to define a steel, timber, or cold formed truss

Temperature Load (command)

Create temperature loads on the model.

See also:

- Create temperature loads (page 554)

Torsion Full UDL (command)

Applies a Torsion Full UDL to members.

See also:

- Create full-length torsional UDLs (page 545)

Torsion UDL (command)

Applies a partial length Torsion UDL to members.

See also:

- Create partial-length torsional UDLs and VDLS (page 546)

Torsion VDL (command)

Applies a partial length Torsion VDL to members.

See also:

- Create partial-length torsional UDLs and VDLS (page 546)

Trapezoidal Load (command)

Applies a Trapezoidal load to members.

See also:

- Create trapezoidal loads (page 545)

UDL (command)

Applies a partial length UDL load to members.

See also:

- Create partial-length UDLs or VDLs (page 544)

Validate (command)

Depending on the tab from which it has been run, this command will validate the structure for:

- Physical model and design issues - if run from the **Model** tab, or the Quick Access Toolbar.
- Design issues only - if run from the **Design** tab.

To see the results go to the Project Workspace and select **Status** > **Validation**

See also:

- Validate the model (page 512)

Variable Area Load (command)

Applies a variable area load to any panel. Area loads entirely cover an individual panel.

See also:

- Create variable area loads (page 543)

Variable Patch Load (command)

Applies variable patch load to any panel.

NOTE Variable Patch Loads can only be applied in 2D views (not 3D)

See also:

- Create variable patch loads (page 542)

VDL (command)

Applies a partial length VDL load to members.

See also:

- Create partial-length UDLs or VDLs (page 544)

Vibration Check (command)

Create a floor vibration check.

See also:

- Create floor vibration check items (page 806)

Wall Panel (command)

Create wall panels for distributing loads.

Walk

Walk through 3D views.

Use the arrow keys to move back/forward/left/right.

Use Q/Z to move up/down.

To rotate click and drag the right mouse button.

See also:

- Walk through the model in a 3D view (page 350)

Wall Opening (command)

Creates a rectangular opening in an existing meshed shear wall when in a Frame View. (**Wall Opening** is inactive in 3D Views and Level Views).

See also:

- Create door or window openings (page 432)

Shear Only Wall (command)

Creates a wall which resists only in-plane shear and has no out of plane stiffness or load bearing resistance.

Analyse ribbon

Command	Description
1st Order Linear	- Runs a linear static analysis. This analysis type is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: No

Command	Description
	• Material: No See also: Run 1st order linear analysis (page 664)
1st Order Non-linear	- Runs a nonlinear analysis with loading applied in a single step. This analysis type is suitable for structures where secondary effects are negligible and nonlinear springs/ elements are present. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: No • Material: Yes See also: Run a 1st order non-linear analysis (page 665)
1st Order Modal	Runs an unstressed modal analysis to determine the structure's natural frequencies. The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly. Nonlinearity Included: • Geometric: No • Material: No See also: Run a 1st order modal analysis (page 665)
2nd Order Linear	- Runs a 2-stage P-Delta analysis. This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included:

Command	Description
	• Geometric: Yes • Material: No See also: Run a 2nd order linear analysis (page 666)
2nd Order Non-linear	- Runs a nonlinear analysis with loading applied in a single step. This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: Yes • Material: Yes See also: Run a 2nd order non-linear analysis (page 666)
2nd Order Buckling	- Runs a linear buckling analysis to determine a structure's susceptibility to buckling. The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: Yes • Material: No See also: Run a 2nd order buckling analysis (page 666)
Analyze All (Static)	- Performs a full series of analyses that would be carried out as part of 'Design All (Static)' but with no design. Includes all relevant analyses. Analysis type (e.g. second-order non-

Command	Description
	linear) is the same as set in the design settings. See also: Run Analyze All (Static) (page 669)
3D only (Static)	- Performs same analyses as 'Analyze All (Static)' but excludes chasedowns. This can save time during scheme design, for example while addressing overall stability, sway, drift, wind drift, etc. See also: Run 3D only (Static) (page 669)
1st Order RSA Seismic	- Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis.
2nd Order RSA Seismic	- Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis.
Model & Results	- Displays the analysis model data in spreadsheets for review/editing.
Wind Tunnel	- Displays the wind tunnel data tab, and wind model data in spreadsheets for running modal analysis and reviewing the results.
Mesh Slabs	- Slabs are meshed automatically for FE Load Decomposition, FE Chasedown, and also if specified in the 3D Analysis. This command is only used if you need to mesh slabs manually. After the operation the slab

Command	Description
	mesh is automatically switched on in the scene content.

BIM Integration ribbon

Command	Description
Structural BIM Import	Import a model from a Neutral File. See also: Import a project from a Structural BIM Import file (page 301)
TEL File Import	Import a file that has been saved in the 'TEL' file format. See also: Import a project from a TEL file (page 302)
3D DXF Import	Import a model from a 3D DXF File. See also: Import data from a 3D DXF file (page 308)
Trimble Connect	Access Trimble's cloud-based project collaboration tool. See also: Trimble Connect (command) (page 2199)
Tekla Structures Export	Export the structural BIM model to Tekla Structures. See also: Export a model to Tekla Structures (page 318)
TCD Export	Export connections to Tekla Connection Designer. See also: Export to Tekla Connection Designer (page 318)
TPFD Export	Export portal frames to Tekla Portal Frame Designer. See also: Export to Tekla Portal Frame Designer (page 319)
Autodesk Revit Export	Export the structural BIM model to Autodesk RevitSaves the currently open model to a new name, or to a template. See also: Export a model to Autodesk Revit Structure (page 327)

Command	Description
IFC Export	Export a physical representation of the structural BIM model in IFC format. See also: Export a model to IFC (page 328)
Cellbeam Export	Export a beam to Westok Cellbeam. See also: Export to and import from Westok Cellbeam (page 329)
Cellbeam Import	Import a beam from Westok Cellbeam. See also: Export to and import from Westok Cellbeam (page 329)
FBEAM Export	Export a beam to FBEAM. See also: Export to and import from FBEAM (page 330)
FBEAM Import	Import a beam from FBEAM. See also: Export to and import from FBEAM (page 330)
ADAPT Export	Export a model to ADAPT. See also: Export a model to ADAPT (page 335)
STAAD Export	Export the analysis model to STAAD. See also: Export a model to STAAD (page 339)
Robot Export	Export the analysis model to Autodesk Robot. See also: Export a model to Autodesk Robot Structural Analysis (page 340)
Cloud Export	Export the model to the cloud. See also: Export a model to the cloud (page 341)
One Click LCA	Export the material data to One Click LCA. See also: Export to One Click LCA (page 341)

Design ribbon

Command	Description
Settings	Opens the Design Settings dialog
Validate (page 2201)	Validates the model for design issues.
Design Steel (Gravity) (page 2184)	Performs an analysis for gravity combinations only, and then designs all steel members in the model based on their individual 'auto-design' setting.
Design Steel (Static) (page 2184)	Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting.
Design Steel (RSA) (page 2184)	Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all steel members in the model based on their individual 'auto-design' setting.
Design Concrete (Gravity) (page 2184)	Performs an analysis for gravity combinations only, and then designs all concrete members in the model based on their individual 'auto-design' setting.
Design Concrete (Static) (page 2184)	Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting.
Design Concrete (RSA) (page 2184)	Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all concrete members in the model based on their individual 'auto-design' setting.
Design All (Gravity) (page 2184)	Performs an analysis for gravity combinations only, and then designs all members in the model based on their individual 'auto-design' setting.
Design All (Static) (page 2184)	Performs an analysis of all static combinations (gravity, lateral, and ELF seismic), and then designs all

Command	Description
	members in the model based on their individual 'auto-design' setting.
Design All (RSA) (page 2185)	Performs an analysis of all RSA load combinations (gravity, lateral, and ELF seismic), and then designs all members in the model based on their individual 'auto-design' setting.
Design Slabs (page 2185)	Designs two-way concrete slabs for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting.
Patch Column (page 2193)	Creates a localized slab reinforcement patch at a slab-column connection.
Design Patches (page 2185)	Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting.
Punching Check (page 2194)	Creates a punching check associated with the punching object.
Design Punching Shear (page 2185)	Perform a check or design (according to individual autodesign settings) of all punching checks in the model.
Vibration Check (page 2202)	Create a floor vibration check.
Check Floor Vibration (page 2181)	Perform all floor vibration checks in the model.
Check in Tedds (page 2182)	This command can be used after all precast concrete members have been initially designed. The command is intended for use where a general re-check for all objects is required following changes to the model which require a reanalysis process (e.g. changing the loading applied onto the model). The command finds all members where a linked Tekla Tedds calculation has been created and performs a check design for these members. This command should ideally be used at the end of the design cycle and ensures that the previously designed precast concrete elements still meet the design criteria of the model.

Draw ribbon

Command	Description
Settings	Opens the Drawing Settings dialog
Drawing Management	Opens a dialog for the generation and laying out of multiple drawings on to a single drawing sheet. The dialog can also be used to manage drawing revisions
Schedule Management	Opens a dialog for the generation of concrete beam, column and wall schedules on drawing sheets. The dialog can also be used to manage schedule revisions
Beam Schedule	Configure and then create a concrete beam schedule for members in an active 2D view
Column Schedule	Configure and then create a concrete column schedule
Wall Schedule	Configure and then create a concrete wall schedule
General Arrangement	Create and save general arrangement drawing for an active 2D view
Beam End Forces	Create and save beam end forces drawing for an active 2D view for the currently selected loading
Column Splice Loads	Create and save steel column splice loads drawings for an active 2D frame view for the currently selected loading
Foundation Reactions	Create and save support reactions drawings for an active 2D view for the currently selected loading
Loading Plan	Create and save loading information drawings for an active 2D view for the currently selected loading
Slab/Mat Detailing	Create and save slab or mat detailing drawings for an active 2D view
Foundation Layout	Create and save foundation layout drawing for an active 2D view.

Edit ribbon

Command	Description
Copy	Select items then copy. See also: Copy and rotate objects (page 495)
Copy Loads	Copy between objects in the current case, or copy from one case to another. See also: Copy loads (page 500)
Move	Select items then move. See also: Move and rotate objects (page 495)
Mirror	Select items then mirror. See also: Mirror objects to new locations (page 496)
Delete	Delete Selected Items.
Join	Joins members to make a longer continuous member. See also: Join and split members (page 504)
Split	Splits existing continuous members. See also: Join and split members (page 504)
Reverse	Reverse Wall Panel Faces (inside becomes outside), or Reverse Beam Directions. See also: Reverse member axes and panel faces (page 506)
Beam Lines	Joins existing concrete beams to make a continuous concrete beam. See also: Automatically join all concrete beams (page 505)
Cutting Planes	Move/Activate/Deactivate planes to isolate a smaller visible working area within the model. See also: Manage cutting planes (page 507)

Command	Description
Move Model	Moves the entire model to a new origin. See also: Move the model or the DXF shadow (page 508)
Rationalize	Open a dialog to remove unused grids, planes, frames etc. See also: Rationalize the model (page 509)
Move DXF Shadow	Move the dxf shadow to a new origin. See also: Move the model or the DXF shadow (page 508)
Create Infills	Creates a pattern of infill members in the selected bay. See also: Create infill members (page 509)
Merge Planes	Merge two closely spaced planes. See also: Merge planes (page 511)
Free Points	Add, move, or delete points at specific co-ordinates to facilitate creation of other objects. See also: Create and manage free points (page 511)

Foundations ribbon

Command	Description
Settings	- Opens the Design Settings dialog.
Calalogue	- Add and delete piles from your pile catalogue or edit their properties (e.g. compression and tension capacity).
Pad Base Column	- Create a pad foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist. See also: Create pad base columns (page 836)
Strip Base Wall	- Create a strip foundation under a concrete shear wall. These can only

Command	Description
	be created at the supported level of the wall and are only valid while the associated wall exists. See also: Create strip base walls (page 837)
Pile Cap Column	- Create a pile cap foundation under a column. These can only be created at the column support node and are only valid while both the member and the support exist. See also: Create pile caps (page 838)
Design Pad Bases	- Design pad bases to all valid combinations based on their individual 'auto-design' settings. **NOTE** **Design Pad Bases** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date'). See also: Design or check all pad bases and strip bases (page 840)
Design Pile Caps	- Design pile caps to all valid combinations based on their individual 'auto-design' settings. **NOTE** **Design Pile Caps** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date'). See also:Design or check all pile caps (page 840)
Minimum Area	- Create a minimum polygon area mat under selected objects (window to select all objects) including a defined overhang dimension. See also: Create mats (page 842)
Rectangular	- Create a minimum rectangular area mat under selected objects including a defined overhang dimension

Command	Description
	(window to select all objects). Set the angle of the rectangle, or use absolute minimal area option. See also: Create mats (page 842)
Strip	- Create a strip of mat between selected points with a defined width and end extension
Area	- Create a polygon area mat by selecting perimeter points and including a defined overhang dimension. See also: Create mats (page 842)
Bays	- Create a mat by selecting bays bounded by grid lines. See also: Create mats (page 842)
Mat Opening	- Create an opening in a mat foundation. See also: Create slab or mat openings (page 455)
Mat Overhang	- Create an overhang at the edge of a mat. See also: • Slab/Mat overhang properties (page 2125) • Add overhangs to existing slab or mat edges (page 457)
Mat Split	- Split mats into smaller panels for the purposes of pattern loading and mat design. See also: Split and join slabs and mats (page 462)
Mat Join	- Join mats to create larger panels for the purposes of pattern loading and mat design. See also: Split and join slabs and mats (page 462)
Pile	- Create a pile under a mat foundation. See also: Place an individual pile in a mat (page 844)

Command	Description
Pile Array	- Create a pile array under a mat foundation. See also: Place a pile array in a mat (page 844)
Design Mats	- Design ground bearing and piled mat foundations for out-of-plane bending to all valid combinations based on their individual 'auto-design' setting. **NOTE** **Design Mats** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date'). See also: Design or check all mats in the model (page 846)
Design Patches	- Design slab patches for out-of-plane slab bending to all valid combinations based on their individual 'auto-design' setting. **NOTE** **Design Patches** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date'). See also: Design or check all patches in the model (page 800)
Punching Check	- Creates a punching check associated with the punching object. See also:
Design Punching Shear	- Check of all punching shear checks in the model. **NOTE** **Design Punching Shear** does not perform an analysis before the check - it always uses the existing analysis

Command	Description
	results, (even if the analysis status is 'Out of Date'). See also: Design and check punching shear (page 803)

Home ribbon

Command	Description
New	Create a new blank model. See also: Start a new project (page 243)
(droplist)	Create a new project based on an existing template. See also: Create a new project based on a template (page 246)
Open	Open a Tekla Structural Designer model.
Close	Close the model.
Save	Close the model.
Save As	Saves the currently open model to a new name, or to a template.
Project Wiki	Opens the Project Wiki dialog for recording properties associated with the project, and to record revisions. See also: Modify project details (page 244)
Model Settings	Opens the Model Settings dialog (page 2422) to specify settings for the current project.

Command	Description
Manage Property Sets	Import, Export and Delete property sets. See also: Transfer property sets between models (page 1025)
Manage View Configurations	Opens or deletes saved View Configurations.
Find	Find and select objects by typing a part of their name. See also: Select using Find (page 353)
Walk	Walk through 3D views. See also: Walk through the model in a 3D view (page 350)

Load ribbon

Structure

Command	Description
Loadcases	Opens the **Loading** dialog at the Loadcases page. See also: Manage load cases (page 514)
Load Groups	Opens the **Loading** dialog at the Groups page. See also: Manage load groups (page 516)
Combination	Opens the **Loading** dialog at the Combinations page. See also: Manage load combinations (page 518)
Envelope	Opens the **Loading** dialog at the Envelopes page. See also: Manage envelopes (page 527)
Update Patterns	If load patterns have been applied and the building geometry or loading is subsequently modified, this command should be run to ensure

Command	Description
	the load patterns reflect these changes. See also: Manage load patterns (page 527)

Wind, Snow & Seismic

Command	Description
Wind Wizard	The Wind Wizard dialog is used for defining the information that is required in order to calculate the wind loading on the structure
Update Zones	Recalculates the zoning details
Wind Loadcases	Opens the Wind Loadcases dialog for defining the details of each wind loadcase
Delete Wind	Deletes the wind model data previously defined using the Wind Wizard
Decomposition (Wind Load)	Decompose the wind loads on to the structure
Simple Wind	Create a Simple Wind Load that will be applied to rigid diaphragms
Snow Wizard	Opens the Snow wizard dialog which is used to automate snow loadcase generation
Uniform Snow	Manually apply Uniform Snow loads
Valley Snow	Manually apply Valley Snow loads
Local Drift Snow	Manually apply Local Drift Snow loads
Update Snow Loads	Update Snow Loads
Delete Snow	Deletes the snow model data previously defined using the Snow Wizard
Seismic Wizard	Opens the Seismic wizard dialog for defining the parameters required for determining the seismic loading and load cases on the structure
Horizontal Spectrum	Display the horizontal spectrum generated by the seismic wizard
Delete Seismic	Delete all of the seismic data entered in the seismic wizard along with the horizontal spectrum, seismic

Command	Description
	loadcases and seismic load combinations
Decomposition	Decompose loads from 1-way and 2-way slabs onto supporting members.

Panel Loads

Command	Description
Level Load	Applies a level load to all slabs at the current level. See also: Apply panel loads (page 538)
Slab Load	Applies a slab load to all slab panels that constitute an individual slab. See also: Apply panel loads (page 538)
Area Load	Applies an area load to any panel. Area loads entirely cover an individual panel. See also: Apply panel loads (page 538)
Point Load	Applies a point load to any panel. **Point Load** is inactive in 3D Views. See also: Apply panel loads (page 538)
Line Load	Applies line load to any panel. **Line Load** is inactive in 3D Views. See also: Apply panel loads (page 538)
Patch Load	Applies a patch load to any panel. See also: Apply panel loads (page 538)
Polygon Load	Applies a polygonal load to any panel. See also: Apply panel loads (page 538)
Perimeter Load	If load patterns have been applied and the building geometry or loading is subsequently modified, this command should be run to ensure the load patterns reflect these changes. See also: Apply panel loads (page 538)
Variable Patch Load	Applies variable patch load to any panel. See also: Apply panel loads (page 538)

Command	Description
Variable Area Load	Applies a variable area load to any panel. Area loads entirely cover an individual panel. See also: Apply panel loads (page 538)

Member Loads

Command	Description
Full UDL	Applies a full length UDL load to members. See also: Create full-length UDLs (page 544)
UDL	Applies a UDL load to members. See also: Create partial-length UDLs or VDLs (page 544)
VDL	Applies a VDL load to members. See also: Create partial-length UDLs or VDLs (page 544)
Trapezoidal Load	Applies a Trapezoidal load to members. See also: Create trapezoidal loads (page 545)
Point Load	Applies a Point load to members. See also: Create point loads and moment loads (page 545)
Moment Load	Applies a Moment load to members. See also: Create point loads and moment loads (page 545)
Torsion Full UDL	Applies a Torsion Full UDL to members. See also: Create full-length torsional UDLs (page 545)
Torsion UDL	Applies a Torsion UDL to members. See also: Create partial-length torsional UDLs and VDLS (page 546)
Torsion VDL	Applies a Torsion VDL to members. See also: Create partial-length torsional UDLs and VDLS (page 546)

Structure Loads

Command	Description
Diaphragm Load	- Applies a load to a rigid or semi-rigid diaphragm. See also: Add a diaphragm load in a 2D view (page 547)
Diaphragm Table	- Applies loads to multiple rigid or semi-rigid diaphragms. See also: Add multiple loads using the diaphragm load table (page 547)
Nodal Load	- Applies a load to a node. See also: Create nodal loads (page 554)
Temperature Load	- Applies a temperature load (a global rise in temperature) to individual elements/panels, selected elements/panels, or to all elements/panels. See also: Create temperature loads (page 554)
Settlement Load	- Applies a Settlement load (a translation or a rotation) to a support. See also: Create settlement loads (page 554)

Model ribbon

A list of the commands accessible from the Model tab on the ribbon.

Levels Group

Command	Description
Construction Levels (page 2248)	Opens the Construction Levels dialog (page 2402) allowing you to define the levels required in order to construct your model.
Frame (page 2187)	Create a frame when in a 3D view (when a 2D view is displayed, this is inactive). Only those members that lie

Command	Description
	within the plane of the frame are displayed. The drop list underneath can be used to delete existing frames.
Sloped Plane (page 2197)	Create a sloped plane when in a 3D view (when a 2D view is displayed, this is inactive). A sloped plane is a 2D View of the model created in a sloped plane. It is defined by selecting 3 existing grid points. Because only those members that lie within the plane of the slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps. The droplist underneath can be used to delete existing sloped planes.

Grid & Construction Lines Group

Command	Description	Note
Grid Line (page 2189)	Create a straight grid line between two points. The drop list underneath can be used to create grid lines in other ways.	Inactive in 3D Views.
Construction Line (page 2183)	Create a straight construction line between two points. The drop list underneath can be used to create construction lines in other ways.	Inactive in 3D Views.

Steel Group

Command	Description
Column (page 2183)	Create a rolled steel columnn. The drop list underneath can be used to create columns of a specific fabrication type.
Beam (page 2181)	Create a rolled steel beam. The drop list underneath can be used to create beams of a specific fabrication type.
Brace (page 2181)	Create a rolled steel brace.
Joist (page 2190)	Create a joist.
Truss (page 2200)	Runs the **Truss Wizard** to define a steel truss.
Portal Frame (page 2195)	Create a portal frame. To create the frame, you firstly define the two column base positions between which the frame will lie. The base positions are restricted to lie on existing grid points. All other details of the portal frame are subsequently entered in the **Portal Frame** dialog.

Concrete Group

Command	Description
Column (page 2183)	Create a concrete columnn. The drop list underneath can be used to create columns of a specific fabrication type.
Beam (page 2181)	Create a concrete beam. The drop list underneath can be used to create beams of a specific fabrication type.

Command	Description
Meshed Wall (page 2191)	Create a reinforced concrete shear wall characterized analytically by a mesh of 2D solver elements located along the wall insertion line and extending to all four corners. Non-rectangular and sloping wall geometry is allowed as well as openings. The type of 2D solver elements used is controlled from the structure properties and can be overriden on a wall by wall basis in the individual wall properties.
Mid-pier Wall (page 2192) I	Create a reinforced concrete shear wall characterized analytically by a single vertical pier object of identical properties as the wall section, located at the centre of the wall and extending from the top to the bottom level. Mid-pier concrete shear wall restrictions: • must be rectangular • must be in a vertical plane • openings are not allowed
Wall Opening (page 2203)	Create a rectangular opening in an existing meshed shear wall when in a Frame View. **Wall Opening** is inactive in 3D Views and Level Views.
Cores (page 2183)	Assign existing concrete members as part of a concrete core.

Slabs Group

Command	Description	Note
Slab on Beams (page 2196)	Create a slab.	

Command	Description	Note
Slab Opening (page 2196)	Create a rectangular or circular slab opening.	Inactive in 3D Views.
Slab Overhang (page 2196)	Create an overhang at the edge of a slab.	Inactive in 3D Views.
Column Drop (page 2183)	Create a column drop.	
Slab Split (page 2197)	Split slab items into smaller panels for the purposes of pattern loading and slab design.	Inactive in 3D Views.
Slab Join (page 2197)	Join slabs to create larger panels for the purposes of pattern loading and slab design.	Inactive in 3D Views.

Timber Group

Command	Description
Column (page 2183)	Create a timber columnn. The drop list underneath can be used to create columns of a specific fabrication type.
Beam (page 2181)	Create a timber beam. The drop list underneath can be used to create beams of a specific fabrication type.
Brace (page 2181)	Create a timber brace.
Truss (page 2200)	Runs the **Truss Wizard** to define a timber truss.

Cold Formed Group

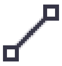
Command	Description
Column (page 2183)	Create a cold formed columnn. The drop list underneath can be used to create columns of a specific fabrication type.
Beam (page 2181)	Create a cold formed beam. The drop list underneath can be used to create beams of a specific fabrication type.
Brace (page 2181)	Create a cold formed brace.
Truss (page 2200)	Runs the **Truss Wizard** to define a cold formed truss.

Walls & Panels Group

Command	Description
Bearing Wall (page 2181)	Creates a wall which resists only axial forces and does not contribute to lateral stability.
Shear Only Wall (page 2203)	Creates a wall which resists only in-plane shear and has no out of plane stiffness or load bearing resistance.
Roof Panel (page 2196)	Create roof panels for distributing loads and/or introducing diaphragm action.
Wall Panel (page 2202)	Create wall panels for distributing loads.

Miscellaneous & Validate Group

Command	Description
Support (page 2199)	Create additional supports underneath existing members

Command	Description
 **Element (page 2187)**	Create analysis elements.
Measure (page 2190)	Measures the distance between any existing construction points or intersection points.
Measure Angle (page 2190)	Measures angles between existing points in 2D Views.
Dimension (page 2187)	Create dimensions between appropriate points which can be included on drawings.
Validate (page 2201)	Validate the structure for physical model issues.

Report ribbon

Command	Description
Select (droplist)	- Select an existing report style in order to either edit the content, or show
Model Report	- Create or edit the chapters and options (content) of model reports
Member Report	- Create or edit the chapters and options (content) of member reports
Show Report	- Display the selected report style on the screen
Filters	- Set up filters to be used when generating a new report
Page display (Group)	- Choose how to display report pages
Settings	- Change the report appearance
Page Setup	- Edit the paper size, orientation and margins
Edit Header	- Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers
Edit Footer	- Open the document headers/footers dialog which can be used to configure the layout and content of the page header and footers

Command	Description
Navigation(Group)	- Use these buttons to navigate the report
Report Index	- Display the report index window to allow you to jump to specific chapters and options (content) within the report.
Export (Group)	- The report can be exported to PDF, Word, Excel or Tedds
Upload Settings	- Specify settings to control the PDF upload to Trimble Connect. See: Upload a model report (page 314)
PDF Upload	- Upload a PDF of the report to Trimble Connect. See: Upload a model report (page 314)

Review ribbon

- **Type droplist** - Filters status and ratio displays to show results for Static designs, RSA designs or the Combined designs.

Design

Command	Description
Status	- Display member status.
Ratio	- Display member utilisation ratio.
Depth Ratio	- Display beam span to depth utilisation ratio.

Foundations

Command	Description
Status	- Display foundation status.
Ratio	- Display foundation utilisation ratio.

Piles

Command	Description
Status	- Display pile status.
Ratio	- Display pile utilisation ratio.

Slab/Mat Design

Command	Description
Status	- Display slab/mat status.
Ratio	- Display slab/mat utilisation ratio.
Slab/Mat droplist	- Filters the status and ratio displays to show results for the selected category.
Status	- Display connection status.
Ratio	- Display connection utilisation ratio.

Connections

Command	Description
Status	- Display connection status.
Ratio	- Display connection utilisation ratio.

Show/Alter State

Command	Description
Auto/Check Design	- Each member is color coded to indicate its autodesign setting (On or Off). Clicking on a member toggles its setting.
Diaphragm On/Off	- Each slab panel and diaphragm node is color coded to indicate its setting (Excluded or Included). Clicking on a slab or a diaphragm node toggles its setting.
Fixed/Pinned	- Each member is color coded to indicate its end fixity setting (N/A, Pinned, Fixed, Moment, Mixed, Cantilever). Clicking on a member toggles its fixity setting between those that are applicable.
BIM Status	- Each member is color coded to indicate its BIM status.
Slab/Foundation Reinforcement	- Use property grid to review and edit or rationalize panel or patch reinforcement, or isolated foundation reinforcement.
Section Material/Grade	- Each steel member is color coded to indicate its section and grade. Clicking on a steel member updates its section

Command	Description
	and/or grade to match what you have set in the Properties window.
Copy Properties	- Copy properties from one member to another. After firstly selecting a parameter in the Properties window you are able to copy it from a designated source member to valid target members.
Report Filter	- Only accessible once a Member filter has been defined. Each member is color coded to indicate if it is included in the currently selected filter. Clicking a member toggles its inclusion status.
Sub Structures	- Create selected sub-sets of the structure and use for more focused editing, design, review and reporting purposes.
Concrete Beam Flanges	- Each concrete beam is color coded to indicate its if flanges are considered and flange widths determined.
Column Splices	- Potential splice locations in steel columns are color coded to indicate where splices exist.
Property Sets	- Graphically review and modify the property set assigned to the elements by using the property grid. In a typical model you may wish to apply the same properties to similar model objects. This can be done efficiently using property sets.
UDA	- Eraphically review and modify the attribute values assigned to the elements by using the property grid. User defined attributes allow you to save your own data to individual elements. These can be used to select similar elements based upon their UDA group or filter report content based on a UDA.
Show/Alter State	- The full range of options is available by selecting this and using the attribute drop down in the property grid.

Design Data

Command	Description
Tabular Data	- Open the review data view in which all sorts of design results and quantities information can be reviewed.

Slab Deflection ribbon

Command	Description
Settings	Open the slab deflection settings dialog
Event Sequences	- Submodel Event Sequences give access to override aspects of the model event sequence on a submodel basis
Deflection Checks	- Opens the Slab Deflection Check Catalogue for defining the default check requirements associated with Check Lines (also associated with events)
Analyse Current	- Performs a slab deflection analysis for the current sub model
Analyse All	- Performs a slab deflection analysis of all submodels regardless of current view/selection
Analyse Selected	- Opens dialog allowing analysis of selected submodels.

Windows ribbon

Command	Description
Structure Window	- Open/close the structure tree window. This lists structural and non-structural components present in the model grouped by commonality in a hierarchical way, and allows for selective control over objects in the structure.

Command	Description
Groups Window	- Open/close the groups tree window. Manage groups for general design and reinforced concrete detailing. Review objects assigned user defined attributes.
Loading Window	- Open/close the loading tree window. This lists loadcases and combinations in a hirearchical way, and allows to review and delete applied loads and provides information and total load and reaction summations for each load combination.
Report Index	- Open/close the report index window. This displays the bookmarks for the active report view which can be used to locate and display a specific section within it, and allows bookmarks to be easily moved around the report.
Wind Window	- Open/close the wind model tree window. This lists wind directions and loadcases in a hierachical way, and offers information about wind loadcases and pressure zone calculation status as well as control over wind model directions.
Status Window	- Open/close the status tree window. This displays information and statuses of modelling, stages of analyses, design and BIM.
Properties Window	- Open/close the properties window. Input, review and edit properties related to ribbon commands, tree view items and individual or multiple objects selected in active views.
Scene Content	- Open/close the scene content window. Manage by entity the displayed content in the 2D and 3D scene views.
Process Window	- Open/close the process window. This displays in a sequence each step of the analysis and design of the model and their statuses, as well as other useful information about program content.

15.3 Project Workspace commands

This section lists the right click context menus commands accessible from the tabs in the **Project Workspace**.

Structure tab - Structure

- Open view (page 2262)
- Open solver view (page 2262)
- Check model (page 2239)
- Design model (page 2250)
- Check model slabs (page 2240)
- Design model slabs (page 2251)
- Check model patches (page 2239)
- Design model patches (page 2250)
- Check punching shear (page 2242)
- Design punching shear (page 2253)

Structure tab - Levels

- Construction Levels (page 2248)
- Open view (page 2262)
- Open solver view (page 2262)
- Select in visible views (page 2262)
- Check plane (page 2240)
- Design plane (page 2252)
- Check plane slabs (page 2241)
- Design plane slabs (page 2253)
- Check plane patches (page 2241)
- Design plane patches (page 2252)
- Check punching shear (page 2242)
- Design punching shear (page 2253)

Structure tab - Frames

Structure tab - Slopes

Structure tab - Architectural Grids

Structure tab - Sub Models

Structure tab - Sub Structures

- Create Sub Structure (page 1031)
- Open view (page 2262)
- Open solver view (page 2262)
- Edit (page 1032)
- Delete (page 1033)
- Rename (page 1033)
- Check sub structure (page 782)
- Design sub structure (page 786)
- Check sub structure slabs (page 797)
- Design sub structure slabs (page 799)
- Create sub structure group (page 1034)
- Check using Tekla Tedds (page 2242)
- Design using Tekla Tedds (page 2255)
- Export to Tekla Tedds (page 324)
- Clear Tekla Tedds Data (page 2247)

Structure tab - Members

- Select in visible views (page 2262)
- Check members (page 2239)
- Design members (page 2249)
- Renumber (page 999)
- Edit (page 360)
- Open view (page 2262)
- Check member (page 2238)
- Design member (page 2248)
- Check using Tekla Tedds (page 2242)
- Design using Tekla Tedds (page 2255)
- Export to Tekla Tedds (page 324)
- Clear Tekla Tedds Data (page 2247)

Structure tab - Walls

- Check walls (page 2247)
- Design walls (page 2261)
- Renumber (page 999)
- Edit (page 360)
- Select in visible views (page 2262)
- Check wall (page 2246)
- Design wall (page 2261)

Structure tab - Cores

- Create new core (page 435)
- Renumber (page 999)
- Open view (page 2262)
- Open solver view (page 2262)
- Edit (page 438)
- Dissociate (page 438)
- Select in visible views (page 2262)
- Check member (page 2238)
- Design member (page 2248)

Structure tab - Slabs

- Renumber (page 999)
- Check panel (page 2240)
- Design panel (page 2251)
- Check Punching Shear (panel) (page 804)
- Design Punching Shear (panel) (page 805)
- Edit (page 438)
- Select in visible views (page 2262)

Structure tab - Isolated Foundations

- Renumber (page 999)

- Select in visible views (page 2262)

Groups tab

- Add Group (page 268)
- Set As Default Group (page 268)
- Rename Group (page 269)
- Remove Group (page 270)
- Split Group (page 268)
- Regroup Members (page 268)
- Check Group (page 270)
- Design Group (page 270)
- Check using Tekla Tedds (page 2242)
- Design using Tekla Tedds (page 2255)
- Export to Tekla Tedds (page 324)
- Clear Tekla Tedds Data (page 2247)
- Generate Detail Drawing... (page 976)

Connections tab

- Update Connections (page 274)
- Edit (page 275)
- Design Connection (page 275)
- Export Connection to Tekla Connection Designer (page 275)
- Export Connection to IDEA StatiCa (page 345)
- Select in visible views (page 2262)

Check member (command)

Command	Description
Check member	Performs a check of the selected member and displays the results.

NOTE **Check member** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check an individual member, wall, or core (page 780)

Check members (command)

Command	Description
Check members	Checks all members in the selected branch of the **Project Workspace**.

NOTE **Check members** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all members (page 781)

Check model (command)

Command	Description
Check model	Checks all members and walls in the model.

NOTE **Check model** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all members and walls (page 781)

Check model patches (command)

Command	Description
Check model patches	Checks all patches in the model.

NOTE **Check model patches** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all patches in the model (page 800)

Check model slabs (command)

Command	Description
Check model slabs	Checks all two-way concrete slabs in the model for out of plane bending.

NOTE **Check model slabs** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all slab items (page 797)

Design model slabs (command) (page 2251)

Check panel (command)

Command	Description
Check panel	Performs a check of the selected panel and displays the results.

NOTE **Check panel** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check an individual slab item (page 796)

Check plane (command)

Command	Description
Check plane	Checks all members in the selected level, slope, or frame.

NOTE **Check plane** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all members in a level, slope, or frame (page 780)

Check plane patches (command)

Command	Description
Check plane patches	Checks all patches in the selected level or slope.

NOTE **Check plane patches** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all patches on a single floor (page 800)

Design model patches (command) (page 2250)

Check plane slabs (command)

Command	Description
Check plane slabs	Checks all slabs in the selected level or slope.

NOTE **Check plane slabs** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check all slab items on a single floor (page 797)

Design plane slabs (command) (page 2253)
Design model patches (command) (page 2250)

Check punching shear (command)

Command	Description
Check punching shear	If run from the **Structure** > **Structure**tab • Checks all punching shear checks in the model. If run from the **Structure** > **Levels** tab • Checks all punching shear checks in the level.

NOTE **Check Punching Shear** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design punching shear (command) (page 2253)

Check selection (command)

Command	Description
Check selection	Checks all members in the selection.

NOTE **Check selection** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design selected members and walls (page 783)

Check using Tekla Tedds

After Design using Tekla Tedds (page 2255) has been performed the following menu options for checking in Tekla Tedds become available for timber and precast members:

- Check using Tekla Tedds> Model
- Check using Tekla Tedds> Member
- Check using Tekla Tedds> Group
- Check using Tekla Tedds> Selection
- Check using Tekla Tedds> <*Substructure name*>

Where to find each check option

Check using Tekla Tedds> Model is displayed if:

- in the Groups Tree you right-click on Groups, Design, or Detailing

Check using Tekla Tedds> Member is displayed if:

- a precast or timber member, which has already been designed is highlighted,

Check using Tekla Tedds> Group is displayed if:

- a precast or timber member, which has already been designed is highlighted,
- group design is active for the highlighted member type,
- there is more than one member in the group to which the highlighted member belongs.

Check using Tekla Tedds> Selection is displayed if:

- one or more precast or timber members, which have already been designed are selected,

Check using Tekla Tedds> <*Substructure name*> is displayed if:

- in the Structure Tree you have right-clicked on a sub structure containing precast or timber members which have already been designed.

Understanding each of the check options

Command	Description
Check using Tekla Tedds > Model	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks all members in the model with the pre-saved design data.

Command	Description
Check using Tekla Tedds > Member	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the member with the pre-saved design data. No other members in the associated Design Group will be checked.
Check using Tekla Tedds > Group	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks each member in the selected group with the pre-saved design data.
Check using Tekla Tedds > Selection	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the selected members with the pre-saved design data.
Check using Tekla Tedds > *<Substructure name>*	This extra option becomes available when working in a sub structure - Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the selected sub structure members with the pre-saved design data.

To check all timber or precast members in the model

NOTE This command is only available for those members that have already been designed.

1. On the **Design** tab, click  Check in Tedds (page 2182).
    - Tekla Structural Designer checks all the existing Tekla Tedds design calculations in the background for the current set of forces.
    - The updated status of the members is shown in a Review View.

NOTE This command is also available in the **Project Workspace** by selecting **Check using Tekla Tedds > Model** from the **Group** branch of the **Group** tab.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check a timber or precast member

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. Hover the mouse pointer over the member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Check using Tekla Tedds > Member**
4. Tekla Structural Designer checks the existing Tekla Tedds precast design calculation in the background for the current set of forces.

At the end of the process the status of the member(s) is shown in a Review View.

NOTE You can also run **Check using Tekla Tedds > Member** by right-clicking the member reference in the Structure Tree.

NOTE **Design using Tekla Tedds > Member** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check a timber or precast member group

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. Hover the mouse pointer over a group member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. In the context menu, select **Check using Tekla Tedds > Group**
   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the group members is shown in a Review View.

To check selected timber or precast members

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. In the active view, drag a box to make your selection.

2. Right-click, then in the context menu, select **Check using Tekla Tedds > Selection**

   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the members is shown in a Review View.

NOTE The **Check using Tekla Tedds > Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check timber or precast members in a sub structure

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Check using Tekla Tedds > *<Sub structure name>***

   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the sub structure members is shown in a Review View.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check wall (command)

Command	Description
Check wall	Performs a check of the selected wall and displays the results.

NOTE **Check wall** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check an individual member, wall, or core (page 780)

Check walls (command)

Command	Description
Check walls	Checks all walls in the selected branch of the Structure tree.

NOTE **Check walls** does not perform an analysis before the check - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

Clear Tekla Tedds Data (command)

Command	Description
Clear Tekla Tedds Data	**Model** Clears all existing data from all the associated Tekla Tedds calculations in the model. **Member** Clears all existing data from the associated Tekla Tedds calculation for the highlighted member. **Group** Clears all existing data from the associated Tekla Tedds calculation for the group to which the highlighted member belongs. **Selection** Clears all existing data from the associated Tekla Tedds calculation for the selected members. **<*Substructure name*>** This extra option becomes available when working in a sub structure - Clears all existing data from the associated Tekla Tedds calculation for the selected sub structure members.

NOTE If you change the Design Settings or section shape, you would need to clear the Tekla Tedds data from existing members in order for the new settings to apply when the members are redesigned.

See also

Timber member design handbook (page 1564)

Precast member design handbook (page 1527)

Construction Levels (command)

To run:	In the **Project Workspace**, select **Structure** > **Levels**, right-click, and then select **Construction Levels** from the context menu.
Usage:	Opens the Construction Levels dialog (page 2402) allowing you to define the levels required in order to construct your model.

Design member (command)

Command	Description

Design member	• *If the member is not in a design group -*
	  • Performs a design of the selected member, then displays the results.
	• *If the member is in a design group (and design using groups is active)*
	  • Performs a design of the highlighted member for it's own individual set of design forces, then displays the results,
	  • Updates all other members of the group to use the same design,
	  • Checks all other members of the group.

NOTE	**Design member** always designs the member irrespective of its autodesign setting.

NOTE	**Design member** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design an individual member, wall, or core (page 783)

Design members (command)

Command	Description

Design members	• *If the member s are not in a design group -*
	    • Performs a check or design (according to individual autodesign settings) of the selected members.
	• *If the members are in design groups (and design using groups is active)*
	    • Performs a check of those members in the selection that are in check mode.
	    • Performs a design of those members in the selection that are in autodesign mode (for the group's set of design forces), and:
	        • Updates all other members of the same group to use the same design, (irrespective of whether they are in the selection).
	        • Checks all other members of the same group (irrespective of whether they are in the selection).

NOTE **Design members** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

NOTE This command requires at least one member in the branch to be in auto-design mode.

See also

Design all members (page 784)

Design all members of a particular section or type (page 785)

Design model (command)

Command	Description
Design model	Performs a check or design (according to individual autodesign settings) of all members and walls in the model according to individual autodesign settings. This command requires at least one member or wall to be in auto-design mode.

NOTE **Design model** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design all members and walls (page 785)

Design model patches (command)

Command	Description
Design model patches	Performs a check or design (according to individual autodesign settings) of all patches in the model according to individual autodesign settings. This command requires at least one patch in the model to be in auto-design mode.

NOTE This command does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design or check all patches in the model (page 800)

Check model patches (command) (page 2239)

Design model slabs (command)

Command	Description
Design model slabs	Performs a check or design (according to individual autodesign settings) of all two-way concrete slabs in the model for out of plane bending. This command requires at least one slab in the model to be in auto-design mode.

NOTE **Design model slabs** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design all slab items (page 798)

Check model slabs (command) (page 2240)

Design panel (command)

Command	Description
Design panel	Performs a check or design (according to individual autodesign settings) of the selected panel according to individual autodesign settings, then displays the results.

NOTE **Design panel** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design an individual slab item (page 798)

Design plane (command)

Command	Description
Design plane	Performs a check or design (according to individual autodesign settings) of all members in the plane according to individual autodesign settings. This command requires at least one member in the model to be in auto-design mode.

NOTE **Design plane** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design all members in a level, slope, or frame (page 784)

Design plane patches (command)

Command	Description
Design plane patches	Performs a check or design (according to individual autodesign settings) of all patches in the plane according to individual autodesign settings. This command requires at least one patch in the model to be in auto-design mode.

NOTE This command does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design all patches on a single floor (page 801)

Check plane patches (command) (page 2241)

Design plane slabs (command)

Command	Description
Design plane slabs	Performs a check or design (according to individual autodesign settings) of all slabs in the plane according to individual autodesign settings. This command requires at least one slab in the plane to be in auto-design mode.

NOTE This command does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design all slab items on a single floor (page 798)

Check plane slabs (command) (page 2241)

Check plane patches (command) (page 2241)

Design punching shear (command)

Command	Description
Design Punching Shear	If run from the **Structure** > **Structure**tab • Performs a check or design (according to individual autodesign settings) of all punching checks in the model. If run from the **Structure** > **Levels** tab • Performs a check or design (according to individual autodesign settings) of all punching checks in the level.

NOTE **Design Punching Shear** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check punching shear (command) (page 2242)

Design selection (command)

Command	Description

Design selection	**For those members in the selection that are not in design groups:**
	• Performs a check or design (according to individual autodesign settings) of the selected members.
	For those members in the selection that are in design groups (and design using groups is active):
	• Performs a check of those members in the selection that are in check mode.
	• Performs a design of those members in the selection that are in autodesign mode (for the group's set of design forces), and:
	    • Updates all other members of the same group to use the same design, (irrespective of whether they are in the selection).
	    • Checks all other members of the same group (irrespective of whether they are in the selection).

NOTE **Design selection** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

NOTE If only **one** member is in the selection, the command operates differently, adopting the logic of the Design member (page 2248) command instead.

See also

Design selected members and walls (page 782)

Design using Tekla Tedds

After analysis has been performed the following menu options for designing in Tekla Tedds become available for timber and precast members:

- Design using Tekla Tedds> Model
- Design using Tekla Tedds> Member
- Design using Tekla Tedds> Group
- Design using Tekla Tedds> Selection
- Design using Tekla Tedds> <*Substructure name*>

Where to find each design option

Design using Tekla Tedds> Model is displayed if:

- in the Groups Tree you right-click on Groups, Design, or Detailing

Design using Tekla Tedds> Member is displayed if:

- a precast or timber member is *highlighted*,

Design using Tekla Tedds> Group is displayed if:

- a precast or timber member is *highlighted*,
- group design is active for the highlighted member type,
- there is more than one member in the group to which the highlighted member belongs

Design using Tekla Tedds> Selection is displayed if:

- one or more precast or timber members are *selected*,

Design using Tekla Tedds> <*Substructure name*> is displayed if:

- in the Structure Tree you have right-clicked on a sub structure containing precast or timber members

Understanding each of the design options

Command	Description
Design using Tekla Tedds > Model	Launches Tekla Tedds and performs interactive designs on all timber and precast members in the model taking into account group design settings.
Design using Tekla Tedds > Member	• *If group design is active* - Launches Tekla Tedds and performs an interactive design on the selected member. Member design uses the

Command	Description
	critical forces in the selected member only. All other members remaining in the associated design group will be check designed for the assigned Tekla Tedds properties and their individual forces. (It should be noted that some members may fail if the critical member was not initially selected for the design process.) Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model. • *If group design is inactive* - Launches Tekla Tedds and performs an interactive design on the selected member. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > Group	Launches Tekla Tedds and performs an interactive design on the selected group. Group design uses the *group* critical forces. This is followed by a check design on each group member for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > Selection	• *If group design is active* - Launches Tekla Tedds and performs an interactive design for each group in the selection, using *group* critical forces. All members in the associated design group (including those that are not in the selection will be check designed for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model both for selected members and members in groups associated with the selection.

Command	Description
	• *If group design is inactive* - Launches Tekla Tedds and performs an interactive design for each member in the selection using the member's individual critical forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > <*Substructure name*>	This extra option becomes available when working in a sub structure. • *If group design is active* - Launches Tekla Tedds and performs an interactive design for each group in the sub structure, using *group* critical forces. All members in the associated design group (including those that are not in the sub structure will be check designed for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model both for sub structure members and members in the same groups but not in the sub structure. • *If group design is inactive* - Launches Tekla Tedds and performs an interactive design for each member in the sub structure using the member's individual critical forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.

To design all timber and precast members

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.

3. In the context menu, select **Design using Tekla Tedds > Model**

   Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the first member to be designed for forces at that section.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
   - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
   - If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
   - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
5. If the member is in a group and group design is active, all other members in the design group are then check designed for the assigned Tekla Tedds properties and their individual forces.
6. Tekla Structural Designer opens the Tekla Tedds design calculation to allow the next member to be designed.
   - Continue in the same way until all the members have been designed.

At the end of the process the status of the member(s) is shown in a Review View.

NOTE **Design using Tekla Tedds > Model** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design a timber or precast member

1. Hover the mouse pointer over the member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Design using Tekla Tedds > Member**

   Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the member to be designed for forces at that section.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
   - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.

- If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
- Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.

5. If the member is in a group and group design is active, all other members in the design group are then check designed for the assigned Tekla Tedds properties and their individual forces.

At the end of the process the status of the member(s) is shown in a Review View.

NOTE You can also run **Design using Tekla Tedds > Member** by right-clicking the member reference in the Structure Tree.

NOTE **Design using Tekla Tedds > Member** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design a timber or precast member group

In grouped design, one member of the group is designed for the worst case forces from all members in the group. The design is then copied to the other group members and each member is checked. The status for each member is then displayed in a Review View.

1. Hover the mouse pointer over a group member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. From the right-click menu, choose **Design using Tekla Tedds > Group** Tekla Structural Designer collects the worst case forces from all members in the group and then opens the Tekla Tedds precast design calculation to allow the first design section (s1) in the selected member to be designed for forces at that section.
4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
    - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
5. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
    - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
6. After the selected group member has been designed, a check is then automatically performed on the remaining group members, resulting in

an individual status and utilisation for each member in the group. The status of the group members is displayed in a Review View.

To design selected timber or precast members

1. In the active view make your selection.
2. Right-click, then in the context menu, select **Design using Tekla Tedds > Selection**
    - Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the first member of the selection to be designed for forces at that section.
3. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
    - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
4. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
    - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
5. Tekla Structural Designer re-opens the Tekla Tedds design calculation to allow the next selected member to be designed.
6. Continue in the same way until all members in the selection have been designed.
7. The status of the members is shown in a Review View.

NOTE The **Design using Tekla Tedds > Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design timber or precast members in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Design using Tekla Tedds > *<Sub structure name>***
4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
    - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.

5. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
    - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
6. Tekla Structural Designer re-opens the Tekla Tedds design calculation to allow the next member in the sub structure to be designed.
7. Continue in the same way until all members in the sub structure have been designed.
8. The status of the members is shown in a Review View.

NOTE The **Design using Tekla Tedds > *<Sub structure name>*** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design wall (command)

Command	Description
Design wall	Performs a check or design (according to individual autodesign settings) of the selected wall according to individual autodesign settings, then displays the results.

NOTE **Design wall** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design an individual member, wall, or core (page 783)

Design walls (command)

Command	Description
Design walls	Performs a check or design (according to individual autodesign settings) of all walls in the selected branch of the Structure tree according to individual autodesign settings. This command requires at least one member in the model to be in auto-design mode.

NOTE **Design walls** does not perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').

Open solver view (command)

The **Open solver view** command is accessible from various branches of the **Project Workspace**Structure tree.

Opens the selected branch of the Structure tree in a Solver View regime, or, if such a view already exists but is not active, it is made the active view.

See also

Open view (command) (page 2262)

Open view (command)

The **Open view** command is accessible from various branches of the **Project Workspace**Structure tree.

Opens the selected branch of the Structure tree in a Structural View regime, or, if such a view already exists but is not active, it is made the active view.

See also

Open solver view (command) (page 2262)

Select in visible views (command)

The **Select in visible views** command is accessible from various branches of the **Project Workspace**Structure tree.

Selects all objects in the active view that are referenced in the selected branch of the Structure tree.

Show references (command)

The **Show references** command is accessible from within the Frames and Slopes branches of the **Project Workspace**Structure tree.

Opens a dialog listing all the objects that are referenced in the selected branch of the Structure tree.

After selecting objects in the dialog, click the **Select** button.

The objects are highlighted in the 3D Structural view and their properties displayed in the **Properties** window.

TIP You can change the color used to highlight the selected objects if required:

From the **Home** tab, click **Settings > Scene > Colors > Selection** and change the **Global - User** color.

Sub Models (command)

The **Sub Models** command is accessible from the Sub Models branch of the **Project Workspace**Structure tree.

Opens the Sub Models dialog (page 2438) allowing you to override the default sub model divisions for the sub models generated during chasedown analysis.

15.4 Settings

Settings for the current project

Several dialogs are provided for controlling the various settings and options in the current project:

- Model Settings dialog (page 2422)
- Analysis Settings dialog (page 2398)
- Design Settings dialog (page 2404)
- Slab Deflection Settings dialog (page 2440)
- Drawing Settings dialog (page 2405)

Settings for future projects

A single dialog is provided for defining all settings and options that are to be applied to new projects:

- Settings dialog (page 2424)

Model Settings

Many (but not all) of the settings for the current project are accessed from the Model Settings dialog (page 2422) on the **Home** tab.

- Design code settings (page 2264)

- Unit settings (page 2265)
- Object reference settings (page 2266)
- Loading settings (page 2268)
- Grouping model settings (page 2269)
- Material list settings (page 2269)
- Beam line settings (page 2270)
- Analysis Model settings (page 2271)
- Validation settings (page 2273)
- Load reduction settings (page 2273)
- EHF settings (page 2274)
- User-defined attribute settings (page 2274)
- Graphics view settings (page 2275)
- Structural BIM settings (page 2276)

See also

Analysis Settings (page 2278)

Design Settings (page 2293)

Slab deflection settings (page 2348)

Drawing settings (page 2352)

Design code settings

The **Design Codes** page allows you to specify the head code and the subsequent design codes that are applied:

- To the **current** project - when accessed from the Model Settings (page 2263)
- To **new** projects - when accessed from the Settings dialog (page 2424)

Button, command, or option	Description
Select the head code list	Allows you to select the head code that you want to apply. When you select a head code in the list, the **Design Codes** table automatically updates. **WARNING** If you change the head code in an existing

Button, command, or option	Description
	project, the following will occur in the model: • Some materials, steel sections, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new head code/unit system. • Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind load cases will need recreating. • Seismic loading (if any) will be deleted. The seismic wizard will need re-running. • All combinations will be deleted.
Design Codes table	Displays the available action codes and resistance codes, which are dependent on the selected head code. The lists within the **Design Codes** table can be used to select between available alternatives.

See also

Define and modify head codes and design codes (page 992)

Unit settings

The **Units** page allows you to specify the units, format and precision that are applied

- To the **current** project - when accessed from the Model Settings (page 2263)
- To **new** projects - when accessed from the Settings dialog (page 2424)

Button, command, or option	Description
System list	Allows you to determine whether metric or US customary units are used.
Table of quantities	Displays each quantity, its current unit and its precision. You can modify quantities by selecting them in the table.
Settings list	Displays and allows you to select the units for the currently selected quantity.
Precision list	Displays and allows you to define the precision of the currently selected quantity.
Use for values lower than	Allow you to define the values for which Tekla Structural Designer applies exponential formats.
or greater than or equal to	

See also

Define and modify units (page 993)

Object reference settings

The **References** page and its subpages allow you to adjust different object reference settings:

- To the **current** project - when accessed from the Model Settings (page 2263)
- To **new** projects - when accessed from the Settings dialog (page 2424)

Button, command, or option	Description
General subpage	
Initial value in levels	Allows you to specify the start number (such as 1, 100, 1000) at each construction level for object

Button, command, or option	Description
	references that contain the **Count** item.
Renumbering Direction	Allows you to specify renumbering directions that control how the member numbering is applied when you use the **Renumber** command.
Ignore letters I & O	Allows you to ignore the letters I and O in grid line names.
Initial number	Allows you to specify the initial number applied to the first grid lines.
Initial letter	Allows you to specify the initial letter applied to the first grid lines.
Naming style list	Allows you to select a desired naming style for grid lines.
Groups subpage **NOTE** This subpage is only displayed in the **Settings** dialog box not in the **Model Settings** dialog box.	
Sub-group Name	Allows you to specify the text used to designate the group labelling. This text forms the stem of the Design Group and Detailing Group names that are displayed on the Groups tab of the Project Workspace. These names are shown in the output reports and drawings when grouped design has been applied.
Formats subpage	
Format table	Lists each object type showing its current reference format and allows you to customize the formats. **NOTE** To customize a object reference format, click the click the **...** button in the **Edit** column
Texts subpage	
Characteristics table	Allows you to specify the text used to designate the characteristic for object references that contain the **Characteristic** item.

Button, command, or option	Description
Materials table	Allows you to specify the text used to designate the material for object references that contain the **Material** item.

See also

Manage object references (page 995)

Loading settings

The **Loading** page and its subpages allow you to adjust certain loading settings:

- In the **current** project - when accessed from the Model Settings (page 2263)
- In **new** projects - when accessed from the Settings dialog (page 2424)

General

Button, command, or option	Description
Use load patterning for steel beams	Allows you to select whether Tekla Structural Designer applies load patterns for steel beams.
Use patterning of eccentricity moments for steel columns	Allows you to select whether Tekla Structural Designer considers patterned eccentricity moments in the design of steel columns.
Pipework Operating & Testing Content Loadcase Type	Allows you to select whether the ancillary loadcases created for lines of pipework are considered as Dead or Live (Imposed).

Line/Area Ancillary Loading

Button, command, or option	Description
Ancillary Type	The loadcase names that will be created when specific ancillary types have been specified.
Dead/Empty Load	The default Dead/Empty Load factors applied for each type.

Button, command, or option	Description
Live (Imposed)/Content Load	The default Live (Imposed)/Content Load factors applied for each type.
Testing Content Load	The default Testing Content Load factors applied for each type.

Grouping model settings

The **Grouping** page in the **Model Settings** dialog box allows you to control the tolerance applied when members are grouped in the current project.

Button, command, or option	Description
Maximum edge length variation	Allows for a tolerance to be applied to the automatic grouping. A member can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.
Maximum length variation	Allows for a length tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.
Maximum height variation	Allows for a height tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its height is within the specified tolerance of the group's (average) height.

Material list settings

The **Material List** page allows you to specify settings affecting material lists:

- To the **current** project - when accessed from the  Model Settings (page 2263)
- To **new** projects - when accessed from the  Settings dialog (page 2424)

Button, command, or option	Description
Ignore openings with area less than	Allows you to specify the size of opening that can be considered small enough to be ignored when

Button, command, or option	Description
	determining the quantity of necessary slab reinforcement.

See also

Apply attribute filters to material lists and reports (page 1030)

Beam line settings

The **Beam Lines** page allows you to control the parameters used for continuous concrete beam formation:

- In the **current** project - when accessed from the Model Settings (page 2263)
- In **new** projects - when accessed from the Settings dialog (page 2424)

NOTE The parameters are only used to control the automatic concrete beam joining that occurs during the design process or when the **Beam Lines** command is run. The parameters are not considered when members are joined manually using the **Join** command.

Button, command, or option	Description
Join pinned beam end	Allows you to control whether joining should occur or not if a pin is defined at the end the last span of the first beam or the start of the first span of the second beam, the fixity at the end in question changing from pinned to continuous once joined. If the beam is subsequently re-split at the same location, the pin gets reinstated.
Limiting join angle in plan	Specifies the limiting angle in plan to be applied for joining beams. Only beams meeting in plan at an angle less than the specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Limiting join angle in elevation	Specifies the limiting angle in elevation to be applied for joining beams. Only beams meeting in elevation at an angle less than the

Button, command, or option	Description
	specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Minimum section overlap	Allows you to apply a tolerance when joining beams if they do not fully overlap in section. You can use it to prevent joining if there is very little physical overlap between the beam cross sections.

See also

Automatically join all concrete beams (page 505)

Analysis Model settings

On the **Analysis Model** page you can adjust certain rigid zone, curved beam, and concrete column centerline settings:

- In the **current** project - when accessed from the Model Settings (page 2263)
- In **new** projects - when accessed from the Settings dialog (page 2424)

Rigid Zones

Design codes allow engineers to assume parts of concrete beams or columns are rigid, leading to more efficient designs.

NOTE Rigid zones should not be confused with rigid offsets, which are used to ensure that the analysis model is properly connected. You can have rigid offsets in the model even if rigid zones are turned off.

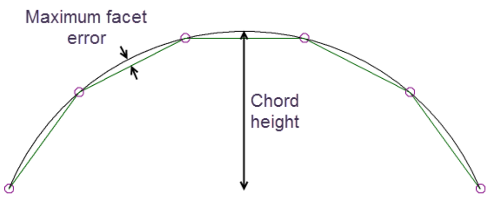
Button, command, or option	Description
Percentage of rigidity	Allows you to specify the extent of the rigid zone created. The option has an effect only if rigid zones are applied (see below).
Rigid zones not applied	Allows you to switch rigid zones on or off. The option affects where releases are applied in the analysis model and

Button, command, or option	Description
	where members start and end for design. When rigid zones are not applied, the design model corresponds to the analysis model, whereas when they are applied, the design model is defined between the ends of the rigid zones.

NOTE There is a significant difference between **Rigid zones not applied** and rigid zones applied with 0% rigidity. The total elastic length of a member will be the same in the two models, but the position of releases and start or end of design members will be different.

Curved Beams

In the solver model, curved beams are replaced by a series of straight line solver elements. The number of solver elements used being controlled by the **Maximum facet error** that has been specified.

Button, command, or option	Description
Maximum facet error	For any given chord height, reducing this value increases the number of solver elements used to represent the curve. 

Column Centerlines

For concrete columns connected to a two-way slab, the column perimeter shape is cut out of the slab 2D element mesh. Stiff analysis ‘rigid link’ elements then connect the column centerline (where the column 1D elements are located) to the 2D mesh nodes. In some circumstances - for example for L-section columns of certain proportions - very short link elements could result, which could cause validation errors or warnings and analysis solution problems.

The following settings are provided in order to move the column centerline a small distance where needed to avoid these problems.

Button, command, or option	Description
Automatically move column centerlines when the smallest rigid link is less than the error limit for 1D element length	This option is initially on by default. The error limit used in this check is specified in Validation settings (page 2273)
Automatically move column centerlines when the smallest rigid link is less than the warning limit for 1D element length	This option is initially off by default. The warning limit used in this check is specified in Validation settings (page 2273)

Validation settings

The **Validation** page of the **Model Settings** dialog box allows you to control the checks that are applied when the current model is validated.

Button, command, or option	Description
Error limit for length	Allows you to control when an error is displayed when very short analysis elements are detected.
Warning limit for length	Allows you to control when a warning is displayed when very short analysis elements are detected.
Error limit for quality	Allows you to control when an error is displayed when poor quality 2D elements are detected. **NOTE** 2D element quality depends on two things: skew and aspect ratio. 0% is bad quality: a squashed triangle tends towards bad quality, whereas 100% is perfect quality, so an equilateral triangle is perfect quality.
Warning limit for quality	Allows you to control when a warning is displayed when poor quality 2D elements are detected.
Check for validation warnings	Allows you to specify which model, analysis, and design validation checks are performed. If a box is cleared, the validation check is not performed.

Load reduction settings

The **Load reductions** page in the **Model Settings** dialog box allows you to control the live load reductions.

Button, command, or option	Description
Reduction percentage column	The cells in this column allow you to specify the reduction that is applied for the number of floors carried.

See also

Activate reductions in live or imposed load cases (page 515)

EHF settings

The **EHF** page in the **Model Settings** dialog boxis used to control the percentages of load used for the EHF calculations when working to Eurocodes. The percentages are material independent, which means that the same percentages are used for concrete and steel. The percentages vary depending on the height of the structure and the number of columns in each direction.

<table>
<tr><th>Button, command, or option</th><th>Description</th></tr>
<tr><td>Height of the structure</td><td>Allows you to specify the effective height of the structure to be used in the EHF calculations.</td></tr>
<tr><td>Set Default</td><td>Sets the height of the structure to the highest construction level.</td></tr>
<tr><td>Number of columns in X direction</td><td rowspan="2">In the Eurocode Cl 5.2(5) the calculation of the reduction factor a_m depends on the number of contributing members, m.

Valid input for m in the X and Y directions is any whole number from 1 to 1000. The default value is 1 which results in a_m = 1.000. If a value of 1000 is entered then a_m would reduce to 0.707.</td></tr>
<tr><td>Number of columns in Y direction</td></tr>
<tr><td>Global initial sway imperfections</td><td>Displays the calculated alpha values and phi percentages for the above input.</td></tr>
</table>

See also

Display notional forces and seismic equivalent lateral forces (page 677)

User-defined attribute settings

The **User Defined Attributes** page allows you to control the attributes that are available:

- In the **current** project - when accessed from the Model Settings (page 2263)
- In **new** projects - when accessed from the Settings dialog (page 2424)

You can add new attribute definitions, delete them, set a type for each attribute, and adjust the acceptable values.

Button, command, or option	Description
Name	Displays and allows you to modify the name of the attribute that will be displayed in the **UDA** section of the **Properties** window.
Type list	Allows you to specify a type for the option. The options are text, number, and file.
Source list	Allows you to restrict the allowable input of the attribute, if necessary. The two options are: • **Custom Value**: Does not restrict the allowable input. • **Value List**: Restricts the allowable input for an attribute to a pre-set list.
Values	Allows you to specify the allowable attribute values when the source is set to **Value List**.
Add	Creates a blank row that allows you to define a new attribute.
Delete	Deletes the currently selected attribute from the table.
Move Up	Move the currently selected attributes up or down in the table and in the **Properties** window.
Move Down	

See also

Create and manage user-defined attributes (page 1026)

Graphics view settings

The **Graphics View Settings** page in the **Model Settings** dialog box allows you to control the display of miscellaneous items in 2D and 3D views in the current project.

Button, command, or option	Description
Do not display values of storey shear below	Allows you to limit the values of storey shear for a new model. This way, you can easily ignore the small values of storey shear that might otherwise detract you from the more important storey shear values. The storey shear values that are less than the limiting value are not displayed in the results view.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Structural BIM settings

The **Structural BIM** page and its subpages allow you to control the structural BIM import and export processes:

- In the **current** project - when accessed from the Model Settings (page 2263)
- In **new** projects - when accessed from the Settings dialog (page 2424)

Button, command, or option	Description
Export --> **Continuous Objects** subpage	
Separate objects for each stack	Allows you to select which columns are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete members as separate

Button, command, or option	Description
	objects, no rebar information is exported.
Separate objects for each span	Allows you to select which beams or other members are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete members as separate objects, no rebar information is exported.
Separate objects for each panel	Allows you to select which wall types are exported as separate objects.
Export --> **Reinforcement Information** subpage	
Concrete columns	Allows you to select whether bar marks are included in the exported concrete column reinforcement information.
Concrete beams	Allows you to select whether bar marks, link leg count, and region length percentage are included in the exported concrete beam reinforcement information.
Export --> **Mappings** subpage	
Materials --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of materials to the default settings.
Decking --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of decking to the default settings.
Import --> **Settings** subpage	
Concrete beams --> **Default "Allow automatic join"**	Allows you to control whether new imported concrete beams are automatically joined or not at the end of the import. **NOTE** If the BIM model contains continuous beams as one piece, you might want to clear the option. After the

Button, command, or option	Description
	import, you can change the properties back, if necessary.
Steel columns --> **Ignore splice offsets in physical member positions**	Allows you to select whether splice offsets are ignored when importing physical member positions.
Import --> **Mappings** subpage	
Materials --> **Reset mappings for 'TEL' File import**	Resets the mappings for TEL file import of materials to the default settings.
Materials --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of materials to the default settings.
Decking --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of decking to the default settings.

See also

Import a project from a Structural BIM Import file (page 301)

Analysis Settings

The **Analysis Settings** page and its subpages allow you to adjust the options applied to the different analyses:

- In the **current** project - when accessed from the Analysis Settings dialog (page 2398) on the **Analyze** tab.
- In **new** projects - when accessed from the Settings dialog (page 2424) on the **Home** tab.

1st order non-linear settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.

Button, command, or option	Description
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1 • **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

2nd order non-linear settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1 • **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be

Button, command, or option	Description
	required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

1st order modal settings

Button, command, or option	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass or number of modes) is not fulfilled, **FEAST** is then automatically used to find higher modes until the stopping criteria is fulfilled. • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are:

Button, command, or option	Description
	• **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used only when the lumped mass model is used.
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are met, the analysis will not look for any more modes.
Modes	Allows you to select one of the following options: • **Automatic number of modes**: If the option is selected, you must specify the mass participation required in each direction. You can optionally specify an initial number of modes that should be close to the actual number required in order to speed up the analysis process. • **Total number of modes**: If **Automatic number of modes** is cleared, you can specify the total number of modes required. The default value is 10.
Jacobi	Allows you to adjust the following settings:

Button, command, or option	Description
	• **Maximum number of sweeps**: A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. • **Sweep tolerance**: At the end of each sweep, values are checked against the previous sweep results. If the difference is less than the tolerance specified here, the result is converged, and the analysis is complete.
Subspace	Allows you to adjust the following settings: • **Maximum number of iterations**: The number of iterations to perform. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance specified here, the result is converged, and the analysis is complete.
FEAST	Allows you to adjust the following settings: • **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be

Button, command, or option	Description
	split, even if the maximum number of modes is greater than that allowed.

2nd order buckling settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 1000.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00001.
Maximum number of sweeps	A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. Default = 50.
Sweep tolerance	At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete. Default = 1.0E-12.
Total number of modes	Allows you to specify the total number of modes required. Default = 10.
Show negative buckling factors	Default = OFF.
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly

Button, command, or option	Description
	finding the lowest frequencies in medium to large models. • **Automatic** (default option): Tekla Structural Designer determines the most appropriate extraction method for the structure.

1st order seismic settings

Button, command, or option	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass, or number of modes) is fulfilled, **FEAST** is then automatically used to find higher modes until the stopping criteria is fulfilled. • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are:

Button, command, or option	Description
	• **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used only when the lumped mass model is used.
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are met, the analysis will not look for any more modes.
Modes	Allows you to modify the following options: • **Initial number of modes**: In order to speed up the analysis process, you can specify an initial number of modes you expect to be required to achieve the required participation. The value specified here should be close to the actual number required because if you enter too few or too many modes, the analysis may take longer. • **Mass participation for RSA**: Allows you to specify the mass participation required in each direction. If this isn't achieved before the stopping criteria apply, the RSA analysis will still be

<table>
<tr><th>Button, command, or option</th><th>Description</th></tr>
<tr><td></td><td>performed, but a warning will be issued.

• Min. Mass participation for RSA: If the minimum participation isn't achieved before the stopping criteria apply, the RSA analysis is not performed.

NOTE The options in Stopping Criteria overrule both the number of modes and mass percentage.</td></tr>
<tr><td>Jacobi</td><td>Allows you to adjust the following settings:

• Maximum number of sweeps: A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed.

• Sweep tolerance: At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete.</td></tr>
<tr><td>Subspace</td><td>Allows you to adjust the following settings:

• Maximum number of iterations: The number of iterations to perform.

• Tolerance: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete.</td></tr>
<tr><td>FEAST</td><td>Allows you to adjust the following settings:</td></tr>
</table>

Button, command, or option	Description
	• **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be split, even if the maximum number of modes is greater than that allowed.
Modal Combination Method	To determine the representative maximum response of interest for a load case, the relevant values for each relevant mode are combined by using the modal combination method specified here. Note that once modes have been combined, the relative signs are lost. The options are: • **Complete Quadratic Combination (CQC)**: Suitable for models where modes are closely spaced or well spaced. • **Square Root of Summation of Squares (SRSS)**: Suitable only for models where modes are well spaced.

Iterative cracked section analysis settings

Button, command, or option	Description
Global Convergence Criteria	Allows you to modify the following options:

Button, command, or option	Description
	• **Maximum number of iterations**: The number of iterations to perform. Default = 200. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00100. • **Relative**: Default: ON. **TIP** To resolve global convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**.
Local Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 500. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.000001. • **Smoothing Parameter**: A property only available for the Eurocode head code. Default = 0.005. **TIP** To resolve local convergence failures:

Button, command, or option	Description
	• Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**. • If you are using the Eurocode head code, increase the value in **Smoothing Parameter**.

Modification factors

Different factors can be applied for each of the different materials in the model in order to adjust the following properties:

- E: Young's modulus
- G: Shear modulus
- $I_{torsion}$: Section inertia about local X
- I_{major}: Section inertia about local Y
- I_{minor} - Section inertia about local Z
- Area: Section area in compression or tension
- $A_{parallel\ to\ minor}$: Section shear area in local Y
- $A_{parallel\ to\ major}$ - Section shear area in local Z
- t: shell thickness (applicable to concrete only)

The factors also vary according to the member types, (and in the case of concrete members whether they are cracked or not).

For concrete members in particular, design codes can require that analysis stiffness adjustment factors are applied since the appropriate properties to use in analysis are load and time dependent.

For various other investigations it is also possible that you will want to apply an adjustment to material properties. One suggested example is the assessment of structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

Although default modification factors for each material are provided in the settings sets to reflect the design code being worked to, you should check that these are appropriate for your particular analysis model.

NOTE	If you make changes to any of the modification factors, analysis must be repeated.

Meshing settings

The meshing options can be adjusted in order to fine-tune the meshes that are produced in slabs and walls. The default settings are generally appropriate, but they could potentially be adjusted if 2D element quality errors are being created during validation.

Composite steel beams settings

Button, command, or option	Description
Inertia used for loading analysis list	Allows you to specify the inertia to be used in the global analysis of the model for a composite steel beam: • Steel beam (default option) • Long term composite • Short term composite **NOTE** • If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. • Relaxation factors are not activated by default as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree. The inertia has different effects on different head codes:

Button, command, or option	Description

	• US head code:
	Long and short term composite inertia only apply to rolled symmetric I sections without web openings. For all other sections, the standard steel beam inertia is used regardless of the analysis option selected.
	• EC head code:
	Long and short term composite inertia only apply to:
	  • Symmetric I rolled sections without web openings
	  • I plated sections without web openings
	  • Westok plated sections without web openings
	  • Fabsec sections without web openings
	For all other sections, the standard steel beam inertia is used regardless of the analysis option selected.
	• BS head code:
	Long and short term composite inertia only apply to:
	  • Symmetric I rolled sections without web openings
	  • Asymmetric I rolled sections without web openings
	  • I plated sections without web openings

Button, command, or option	Description
	• Westok plated sections without web openings • Fabsec sections without web openings For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • AUS and IS head codes: Design of composite beams is not currently supported for the head codes.

Design Settings

The **Design Settings** subpages allow you to control the design options applied:

- to the **current** project - when accessed from the Design Settings dialog (page 2404) on the **Design** tab.
- to **new** projects - when accessed from the Settings dialog (page 2424) on the **Home** tab.

Design Settings contains the following subpages:

- Analysis (page 2294)
- General (page 2294)
- Concrete > Cast-in-place (page 2296)
- Concrete > Precast
- Composite Beams (page 2322)
- Design Forces (page 2323)
- Design Groups (page 2339)
- Autodesign (page 2340)
- Design Warnings (page 2341) (AISC/ASC only)
- Steel Joists (page 2345)
- Sway and Drift Checks (page 2346)
- Fire checks (page 2347)

Design Settings - General and Analysis

General design settings

<table>
<tr><th>Button, command, or option</th><th>Description</th></tr>
<tr><td>Fix column nodes horizontally</td><td>Allows you to automatically apply lateral translational fixed supports to column nodes for the 3D building analysis performed during the gravity design processes.

The option is useful at the early design stage when lateral systems have not been established. The automatic lateral supports can remove unwanted lateral displacements that may prevent the analysis from completing and produce unrealistic forces.

NOTE The option does not apply to:
• The column nodes that are in a rigid floor diaphragm
• An analysis run as a separate process
• The Analyze All (Static) command
• The 3D building analysis performed during any Design (Static) command</td></tr>
</table>

Analysis settings

<table>
<tr><th>Button, command, or option</th><th>Description</th></tr>
<tr><td>Analysis</td><td>Allows you to select if a first or second order 3D building analysis is performed.

TIP For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching</td></tr>
</table>

Button, command, or option	Description
	to a second-order analysis for the final design. In addition, **Solve lateral loadcases in isolation if non-linear analysis is required during design** allows you to try to solver lateral load cases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral load cases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral load cases, you can either run the analysis directly on the **Analyze** tab, or select the option if you know that the analysis does not fail.
Stability coefficient tolerance United States head code (ACI/AISC) head code only	**Stack height /** ratio allows you to adjust the value of deflection that can safely be ignored. Selecting the option prevents any small deflections from causing erroneously high stability coefficients in stability coefficient calculations. **NOTE** If the second order drift is less than the tolerance specified in **Stack height /**, the stability coefficient value is returned as N/A with a note that states that the drift is small enough to be ignored.
Reduced stiffness factor United States head code (ACI/AISC) head code only	This factor is only exposed once the analysis is set to second-order. It is applied to the stiffness (EI and EA) of all steel members in addition to any settings made on Analysis Options, Modification Factors.

Button, command, or option	Description
	For correct design to the AISC Specification using the DAM, it should be set to 0.8. **NOTE** As an alternative to setting the analysis to first-order to explore the reason for any second order analysis failure, it is possible to alter this factor. If you set it to a value of say 10, this will stiffen both the Modulus of elasticity (E) and the shear modulus of elasticity (G) by a factor of 10 in the second order analysis. Although the results will not be able to be used for a valid design, it should now be possible to run the analysis to see which member might fail a design and hence be the cause of the analysis instability. This factor can then be reduced towards 0.8 for further investigation.

Design Settings - Concrete > Cast-in-place

General

Buttons, commands, or options	Description
Limitation on concrete cylinder strength for shear/torsion design (Eurocode only)	
Normal weight concrete limit **Lightweight concrete limit**	For shear/torsion design the cylinder strength and its use in all derived mechanical properties is limited (but 'overridable') for specific national annexes. Default national annex values of these limits for normal weight/lightweight respectively: • UK, Eire, Singapore, Malaysia: 50/50 • Norway: 65/55

Buttons, commands, or options	Description
	• EU, Finland, Sweden: 90/90 (no limit stated) **NOTE** Changing the headcode/settings set does NOT update these values automatically.
Reinforcement anchorage length parameters (used in the calculation of the ultimate bond stress, from which anchorage lengths are determined)	
Plain Bars Bond Quality Modifier	Default value 0.5.
Deformed Bars Bond Quality Modifier	Default value 0.8. Only applies to the Eurocode head code.
Type-1 Bars Bond Quality Modifier	Default value 1.0. Only applies to the Eurocode head code.
Type-2 Bars Bond Quality Modifier	Default value is 1.0. Only applies to the ACI headcode.

Beam

. . . . Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Longitudinal bars	Allows you to adjust the following settings applied to longitudinal bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the longitudinal bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the longitudinal bar design process. • **Minimum side bar size**: Sets the minimum allowable side bar size that can be used in the longitudinal bar design process. • **Minimum top steel clear spacing**: Sets the minimum allowable top steel clear spacing that can be used in the longitudinal bar design process.

Buttons, commands, or options	Description
	• **Minimum bottom steel clear spacing**: Sets the minimum allowable bottom steel clear spacing that can be used in the longitudinal bar design process. • **Maximum tension steel spacing**: Sets the maximum allowable tension steel spacing that can be used in the longitudinal bar design process. • **Maximum compression steel spacing**: Sets the maximum allowable compression steel spacing that can be used in the longitudinal bar design process. • **Use single bars when beam width <=**: Single bars are only permitted in beams whose width is less than the width specified. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design. Only applies to the ACI headcode.
Short span maximum length	Spans smaller than the value set here are treated as short spans. Support bars of short spans are merged with the span bars.

*. . . . **Detailing Settings***

Buttons, commands, or options	Description
Use same size bars in multilayer arrangements	Allows you to use the same bar size in each layer.
Use same number of bars in each layer	Allows you to use the same number of bars in each layer.
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Merge identical longitudinal bars if appropriate	Allows you to merge bars of the same size, number, and position, provided that the total length of the merged bar does not exceed the max allowable bar length. **TIP** The max bar length can be verified on the **Reinforcement** page of the

Buttons, commands, or options	Description
	Materials dialog box. To do so, click an available bar size and then, click **View...** **NOTE** Top bar patterns 1 and 2 do not have any bars that you could merge.
Extend top longitudinal support bars symmetrically	Extends the support bars symmetrically to both spans based on the larger effective span length; but only if the spans vary by less than the percentage specified.
Extend top longitudinal support bars by anchorage length	Adds anchorage lengths to the calculated extension lengths.
Min anchorage length {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter. **NOTE** The option is only available for the Eurocode headcode.
Min anchorage length of plain bars {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter when the rib type is plain. **NOTE** The option is only available for the ACI headcode.
Extend top span bars to end support	Extends the top span bars of the first and last spans to the end supports.
Use 'U' bars at end support if appropriate	Allows you to replace the top and bottom bars at the end support region with 'U' bars under certain conditions. The bars that are joined or merged to create the 'U' bars depend on the top and bottom patterns selected for the beam. The anchorage lengths for the resulting 'U' bars are taken as the lengths required for the pair of bars that made the 'U' bar.
Select same bar size in support region and in the span	Allows you to use the same bar size in the support and span regions.

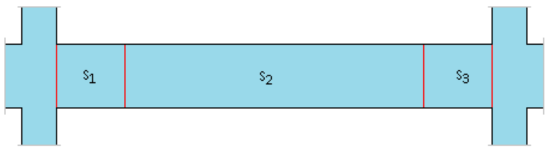
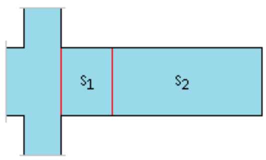
Buttons, commands, or options	Description
Select symmetrical stirrup in support region	Allows you to use the same stirrup arrangement (bar size and spacing) in both supports.

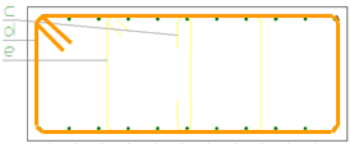
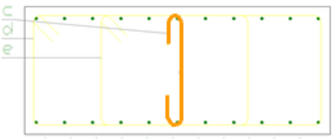
. . . . Top / Bottom Longitudinal Bar Pattern

Buttons, commands, or options	Description
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams. Instead, you should modify the beam properties.
Continuous Span + Cantilever (Backspan) tab	Allows you to define how the selected longitudinal bar pattern is applied to continuous spans and cantilever backspans. An interactive diagram updates to match the changes you make. • Bars: Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs. • Region fields: The regions in which bars are applied are defined as a percentage of the span length. • Bar lapping: Select whether you want to lap the bottom bars in the middle of the support region (inside support) or at the face of the support region (outside support).
Single Span tab	Allows you to define how the selected longitudinal bar pattern is applied to single spans. An interactive diagram is also displayed and updates to match the changes you make.
Cantilever	Allows you to define how the selected longitudinal bar pattern is applied to

Buttons, commands, or options	Description
	cantilevers. An interactive diagram updates to match the changes you make.

. . . . *Stirrup (Link) Settings*

Buttons, commands, or options	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1 , S_2, and S_3 as shown below. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region.  • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the clear span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically. • **Stirrup Type**: Select either open or closed stirrups in the list.
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2 as shown below. 
Minimum bar size	Sets the minimum allowable bar size that can be used in the design process.
Maximum bar size	Sets the maximum allowable bar size that can be used in the design process.
Minimum spacing	Sets the minimum allowable stirrup spacing that can be used in the design process.

Buttons, commands, or options	Description
Maximum spacing	Sets the maximum allowable stirrup spacing that can be used in the design process.
Spacing increment	The designed stirrup spacings are multiples of this value.
Maximum stirrup leg spacing across beam	Allows you to determine if single stirrups, double stirrups, or more are required, depending on the width of the beam.
Use single outside stirrup	Allows you to use a single outside link with additional links added as required (as per stirrup d shown dotted below). 
Accept single leg internal stirrup	Allows the use of single leg internal links (as per stirrup c shown dotted below). 
Optimize stirrup design regions where possible	In this case in the central region S_2, shear reinforcement is provided to meet the minimum of the code requirement or user preference whilst in regions S_1 and S_3, designed shear reinforcement is required. The position and length of region S_2 is determined from considerations of the shear resistance of the concrete cross-section and this then enables the lengths of regions S_1 and S_3 to be determined.

. . . . General Parameters

Buttons, commands, or options	Description
Partial fixity coefficient, β	A coefficient applied to the maximum positive moment in the beam span (excluding support positions) to set a user-defined minimum design moment

Buttons, commands, or options	Description
	for beam support regions. Allowable range is 0.0 to 1.0. Default value is 0.25 **NOTE** The option is only available for the ACI headcode.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Long term deflection period	Allowable range is 3 to 60 months. Default value is 60 months. **NOTE** The option is only available for the ACI headcode.
Time at which brittle finishes are introduced	Allowance range is 1 to 6 months. Default value is 1 month. **NOTE** The option is only available for the ACI headcode.
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Buttons, commands, or options	Description
Tolerance on rectilinearity	The calculation of the effective width is only carried out for concrete beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees. At greater angles you will be prompted to enter the effective width manually.

Column

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the vertical bar design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area.

Buttons, commands, or options	Description
	• **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design.
Tie bars	Allows you to adjust the following settings applied to tie bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the tie bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the tie bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the tie bar design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are.
Multiple Layers	For circular column sections only, if autodesign requires a 2nd layer of reinforcement you can control the **layer spacing**. Autodesign will use a larger spacing if needed - and it is checked in the check results.
Aspect ratio change point	Allows you to specify a value that determines which rectangular sections are considered to have a low aspect ratio and which are considered to have a high aspect ratio.

Buttons, commands, or options	Description
Use double ties for low aspect ratio	Allows you to use double ties where applicable instead of single ties for sections with a low aspect ratio.
Use double ties for high aspect ratio	Allows you to use double ties where applicable instead of single ties for sections with a high aspect ratio.
Use triple ties for low aspect ratio	Allows you to use triple ties where applicable instead of double or single ties for sections with a low aspect ratio.
Use cross ties for low aspect ratio	Allows you to use cross ties where applicable instead of triple, double or single ties for sections with a low aspect ratio.

. . . . Detailing Settings

Buttons, commands, or options	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the column in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties required to use separate regions	Allows you to specify the minimum number of span region ties that are necessary for a span region to be indicated on the detail, and in consequence, standardize the tie spacing on the detail.
Provide ties through full foundation depth	Allows you to draw ties through the full depth of the foundation.
Foundation penetration depth of ties	Allows you to specify the penetration depth of ties into the foundation. **NOTE** The option is only available in the If the **Provide ties through full foundation depth** option is cleared.

Buttons, commands, or options	Description
Provide ties through floor depth for internal walls	For walls restrained by flat slabs: ties are always provided through the floor depth irrespective of this setting. For walls restrained by beam and slab: ties are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, ties are provided up to the soffit of the shallowest beam depth.
Tie bar type	Allows you to select the tie bar type that you want to use.

. . . . General Parameters

Buttons, commands, or options	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Match bar position to stack above	Allows you to have the starting arrangement for longitudinal bars match the arrangement of the bars in the stack above if the section geometry matches.
Increase main bar size preferentially	Allows you to have the corner bars increased in preference to the intermediate bars. All bars start off at the same size (unless the initial bar size is driven by the current arrangement or the stack above), but when the check fails, the corner bars will be increased in size if all bar sizes are the same. Otherwise, the intermediate bars will be increased in size. This means that when the final design is produced, either all bars will be the same size or the corner bars will be one size larger than the intermediate bars. Alternatively, if you require all bar sizes to be the same size, clear the option. This way, all bar sizes are increased together.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the

Buttons, commands, or options	Description
	anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Wall

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the design process.

Buttons, commands, or options	Description
	• **Maximum center spacing**: Sets the maximum value for bar spacing when a wall is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area.
Vertical bars in end zone	Allows you to adjust the following settings applied to vertical bars in the end zone area: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the end zone area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the end zone area.
Horizontal bars	Allows you to set the minimum allowable bar size that can be used in the design process of horizontal bars and the minimum allowable area of reinforcement as a ratio of the end zone area.
Tie/horizontal bars	Allows you to set the minimum and maximum allowable bar sizes that can be

Buttons, commands, or options	Description
	used in the tie/horizontal bar design process and specify the value whose multiples the designed bar spacings are.
Tie/confinement bars	Allows you to set the minimum and maximum allowable bar sizes that can be used in the tie/confinement bar design process.
Confinement bars in end zone	Allows you to set the minimum and maximum allowable bar sizes that can be used in the confinement bar design process.
Substitute loose bars if mesh inadequate	Allows you to use additional loose bars in the end zones when the mesh is inadequate.

*. . . . **Detailing Settings***

Buttons, commands, or options	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the wall in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties required to use separate regions	Allows you to specify the minimum number of span region ties that are necessary for a span region to be indicated on the detail, and in consequence, standardize the tie spacing on the detail.
Provide ties through full foundation depth	Allows you to draw ties through the full depth of the foundation.
Foundation penetration depth of ties	Allows you to specify the penetration depth of ties into the foundation. **NOTE** The option is only available in the If the **Provide ties through full foundation depth** option is cleared.

Buttons, commands, or options	Description
Provide ties through floor depth for internal walls	For walls restrained by flat slabs: ties are always provided through the floor depth irrespective of this setting. For walls restrained by beam and slab: ties are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, ties are provided up to the soffit of the shallowest beam depth.
Tie bar type	Allow you to select the tie bar type that you want to use.
Tie bar type in end zones	

. . . . General Parameters

Buttons, commands, or options	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Design Walls for FE Chasedown analysis results	Allows you to specify that the walls are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Slab

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the slab item auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Maximum bar length	Allows you to set the maximum length of a bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Use mesh where possible	Select to use mesh where possible in bars.
Make bob for top steel of cantilevers	Allows you to apply a bob to the end of the top steel in cantilevers.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.

. . . . *Patches*

Buttons, commands, or options	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

. . . . *General Parameters*

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom,\ lim}$. The full calculation is: $c_{nom,\ lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Allowance for additional detailing bars	Allows you to specify a percentage by which the total mass of bars for slabs or mats shown in material listing reports and in slab or mat detailing drawings is increased.

Punching Shear

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement bars	Allows you to set the minimum and maximum allowable bar sizes that can be welded to the rail in the design process and set the maximum allowable diagonal spacing for radial layouts.
Spacing along rail	Allows you to set the minimum and maximum allowable bar spacing along each rail and specify a value whose multiples the bar spacings are.
Spacing to first reinforcement line	Allows you to set the minimum and maximum spacing from the column face to the first reinforcement line and define a value whose multiples the spacings from the column face are.
Auto-design	Allows you to select whether the auto design method is to minimize the bar size or minimize the number of rails and specify the minimum number of studs per face that will be used when auto-designing from minima.
Diagonal spacing between rails	Allows you to set the maximum diagonal spacing between the outermost studs in the last critical perimeter for circular arrangements.
Spacing between rails in Y direction	Allows you to set the minimum and maximum allowable rail spacing in the Y direction and specify a value whose multiples the bar spacings in the Y direction are.
Spacing between rails in Z direction	Allows you to set the minimum and maximum allowable rail spacing in the Z direction and specify a value whose multiples the rail spacings in the Z direction are.
Spacing from last line to outer perimeter	Allows you to set the maximum spacing from the last reinforcement line to the outer perimeter.

Foundations - General

Buttons, commands, or options	Description
Minimum distance from column/wall face to base/cap edge	Allows you to specify the minimum distance from the column or wall face to the base edge or cap edge.
Use presumed bearing capacity method for pad bases and mats (EN 1997-1 cl.6.5.2.4)	If this option is selected the bearing pressure is checked for serviceability combinations using the presumed bearing resistance specified in the pad base properties (no 'STR' and 'GEO' capacities). **NOTE** The option is only available for the Eurocode head code.

Isolated Foundations

*. . . . **Reinforcement Layout***

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Use mesh where possible	Select to use mesh where possible in bars.

. . . . *Foundation Size*

Buttons, commands, or options	Description
Minimum depth	Allows you to set the minimum allowable foundation depth.
Maximum depth	Allows you to set the maximum allowable foundation depth.
Auto-design depth increment	Allows you to set the depth increment.
Minimum side length	Allows you to set the minimum allowable side length.
Maximum side length / strip width	Allows you to set the maximum side length under columns as a distance.
Maximum side length ratio under columns	Allows you to set the maximum side length ratio under columns.
Rounding increment for footing dimensions	Allows you to specify a value to which the overall footing size is rounded.
Default 3-pile cap shape	Allows you to define whether the default shape is triangular or rectangular when 3 piles are used.

. . . . *General Parameters*

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.

Buttons, commands, or options	Description
(Pad Base tab) **Consider passive resistance of soil for sliding check**	Allows you to control whether the passive resistance of the soil is included in the check of Sliding Resistance.
(Pad Base tab) **Pad base shear reactions at pinned bases**	**Ignorable percentage of passive pressure** This allows you to ignore small shear forces at pinned bases. The default of 1% is typically a negligible load level which would commonly be ignored. If you do not want to ignore any shear forces, set the value to zero.
(Pad Base tab) **FOS for sliding and uplift checks**	Allows you to edit the design safety factor for these checks. **NOTE** The option is only available for the ACI/AISC head codes.
(Pile Cap tab) **Tolerance for pile position variation**	Allows you to specify the allowable deviation value of the pile position from its original plan position. The value is used in the design moment calculations.

. . . . Piles

Buttons, commands, or options	Description
Minimum number of piles for pile caps under columns	Minimum and maximum numbers of piles that apply when a pile cap under a column has been set to auto design the piles.
Maximum number of piles for pile caps under columns	
Minimum spacing option list	Allows you to select whether the minimum spacing check uses a set minimum spacing value or a multiple of the pile width or pile circumference.
Minimum spacing of piles	Allows you to set the minimum spacing value that you can select in the **Minimum spacing option** list.
Minimum pile edge distance	Allows you to set the minimum distance from the pile face to edge of foundation.
Pile auto-design method list	Allows you to select to use the smallest number of high capacity piles or more piles of lowest capacity when a pile cap has been set to auto design.

Buttons, commands, or options	Description
Limit maximum critical section size	When selected, allows you to specify a maximum critical section size as a multiple of the pile dimension.
Use pile capacity for pile punching and shear checks	When selected, allows you to use pile capacity for pile punching and punching shear checks. When unselected, the pile load is used.

Mat Foundations

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the panel auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.

Buttons, commands, or options	Description
Maximum bar length	Allows you to set the maximum length of bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.
Use mesh where possible	Select to use mesh where possible in bars.

. . . . Patches

Buttons, commands, or options	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

. . . . General Parameters

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	Allows you to specify a value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	Allows you to specify a value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$

Buttons, commands, or options	Description
	NOTE The option is only available for the Eurocode headcode.

. . . . Piles

Buttons, commands, or options	Description
Use pile capacity for punching check	When selected, allows you to use pile capacity for the pile punching check. When unselected, the pile load is used.

Design Settings - Concrete > Precast

Beam

. . . . Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to beam reinforcement:

. . . . Top / Bottom Longitudinal Bar Pattern

Buttons, commands, or options	Description
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams. Instead, you should modify the beam properties.
Continuous Span tab	Allows you to define how the selected longitudinal bar pattern is applied to

Buttons, commands, or options	Description
	continuous spans. An interactive diagram updates to match the changes you make. • Bars: Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs. • Region fields: The regions in which bars are applied are defined as a percentage of the span length. **NOTE** The Top and Bottom Region fields dictate how many **design sections** appear in the Tedds Calculation - a different design section is required at each interface between regions.
Pinned Span tab	Allows you to define how the selected longitudinal bar pattern is applied to single pinned spans. An interactive diagram is also displayed and updates to match the changes you make.
Cantilever	Allows you to define how the selected longitudinal bar pattern is applied to cantilevers. An interactive diagram updates to match the changes you make.

. . . . Stirrup (Link) Settings

Buttons, commands, or options	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1 , S_2, and S_3. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region. • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically.

Buttons, commands, or options	Description
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2.

. . . . General Parameters

Buttons, commands, or options	Description
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast beams for lifting forces	Allows you to specify that the beams are also to be designed for lifting forces, (the lifting check being optional in the Tedds calculation).

Column

. . . . Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to column reinforcement:

. . . . General Parameters

Buttons, commands, or options	Description
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast column for lifting forces	Allows you to specify that the columns are also to be designed for a set of lifting forces, (the lifting check being optional in the Tedds calculation).

Design Settings - Composite Beams

Composite beams

Button, command, or option	Description
Tolerance of rectilinearity	The calculation of the effective width of composite beams is only performed if they lie within the tolerance on rectilinearity adjusted here. The default tolerance is 15 degrees. At greater angles, you will be prompted to type the effective width manually.
Update effective width prior to design check	Allows you to determine the effective width each time auto or check design is run. When the box is unselected, the effective width is only set as part of the initial design.

Design Settings - Design Forces

NOTE Values entered in the **Ignorable Forces Below** section:

- **are** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Values entered in the **Minimum Design Forces** section:

- **are** considered when creating **member end force** reports/drawings and **foundation reaction** reports/drawings in Tekla Structural Designer
- **are** considered when performing **connection resistance checks** in Tekla Structural Designer
- **are** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Ignore forces below

NOTE Values entered in the **Ignorable Forces Below** section:

- **are** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Design	
Ignore Forces Below	Allow you to specify appropriate negligible or nominal force levels to prevent 3D analysis from exposing small forces that are normally ignored in design. When the small forces in the 3D analysis are below the specified levels, they are ignored and the design in Tekla Structural Designerproceeds automatically.

Concrete beams (Eurocode only)

Button, command, or option	Description
Torsion force % of concrete resistance	Allows you to limit the maximum torsion force to a % of the concrete resistance.

Minimum design forces

NOTE Values entered in the **Minimum Design Forces** section:

- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

These settings provide flexibility to control minimum design values and rounding increments used in the following:

- Member end force reports/drawings
- Foundation reaction reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Brace Force Axial Force	Click the links to see examples of how the minimum entered here is used in: • Member end force reports/ drawings (page 2325) • Connection resistance checks (page 2334) • Exported connection forces (page 2336)
Beam End Forces Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Member end force reports/ drawings (page 2325) • Connection resistance checks (page 2334) • Exported connection forces (page 2336)
Rounding Increment for Force Rounding Increment for Moment	
Foundation Reactions Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Foundation reaction reports/ drawings (page 2329) • Exported connection forces (page 2336)
Rounding Increment for Force Rounding Increment for Moment	

How Minimum Design Forces are used

The following topics illustrate how Minimum Design Forces are applied in different areas of Tekla Structural Designer.

Member end force reports/drawings

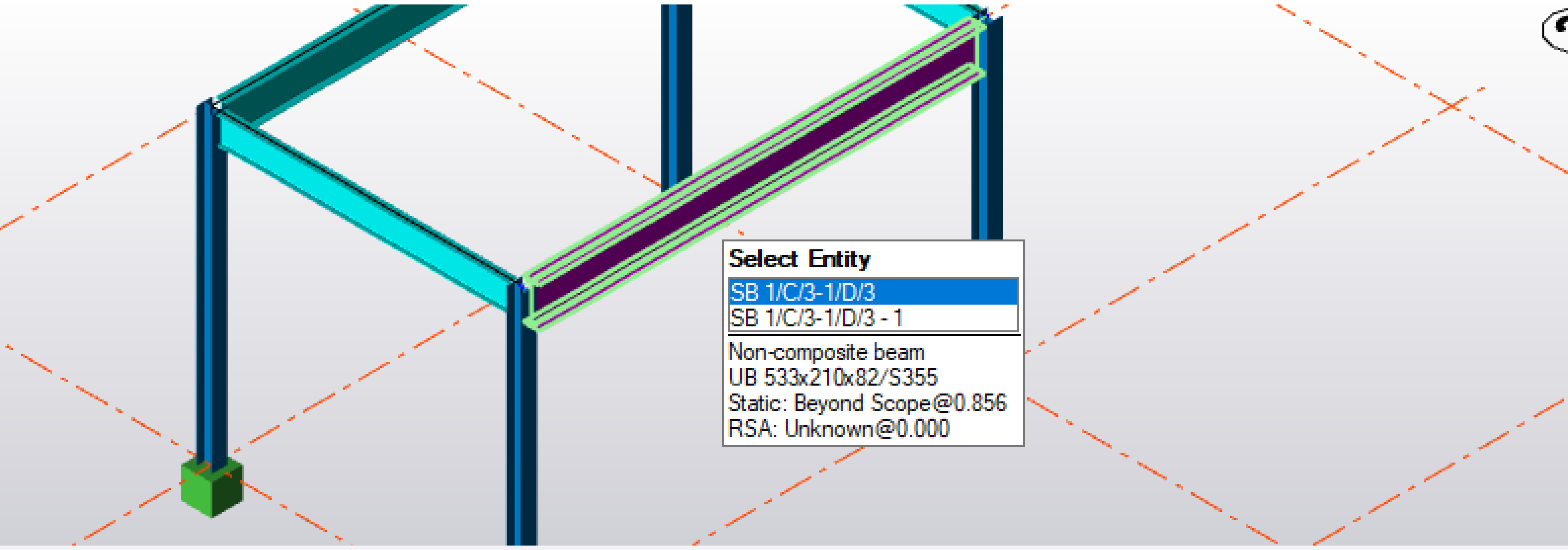
Tekla Structural Designer reports member end forces (Beam end forces/ Bracing forces) based on analysis results.

The ‘Minimum Design Forces’ inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for the reported forces, and also apply rounding increments to them.

NOTE While minimum design values and rounding increments are applied to the member end force reports/drawings, they have no impact on steel and concrete member design in Tekla Structural Designer, or timber and precast design using Tekla Tedds.

Consider a fixed beam with axial force + major/minor shear + torsion + major/minor moment.

Forces/moments are reported in the Beam End Force report.

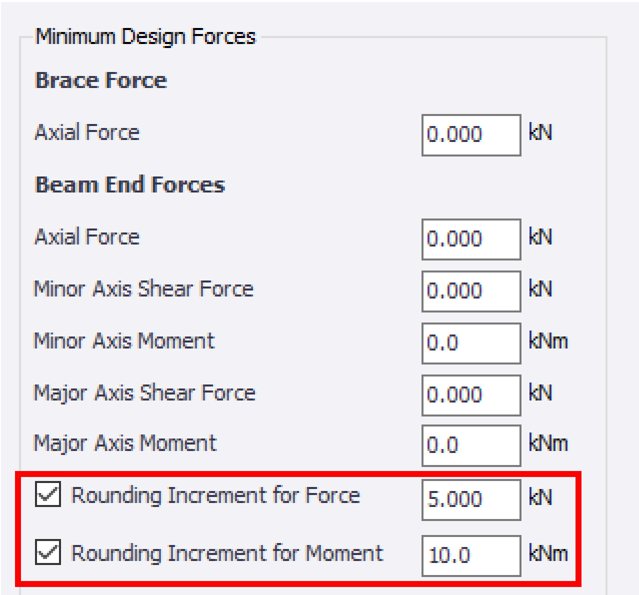


es

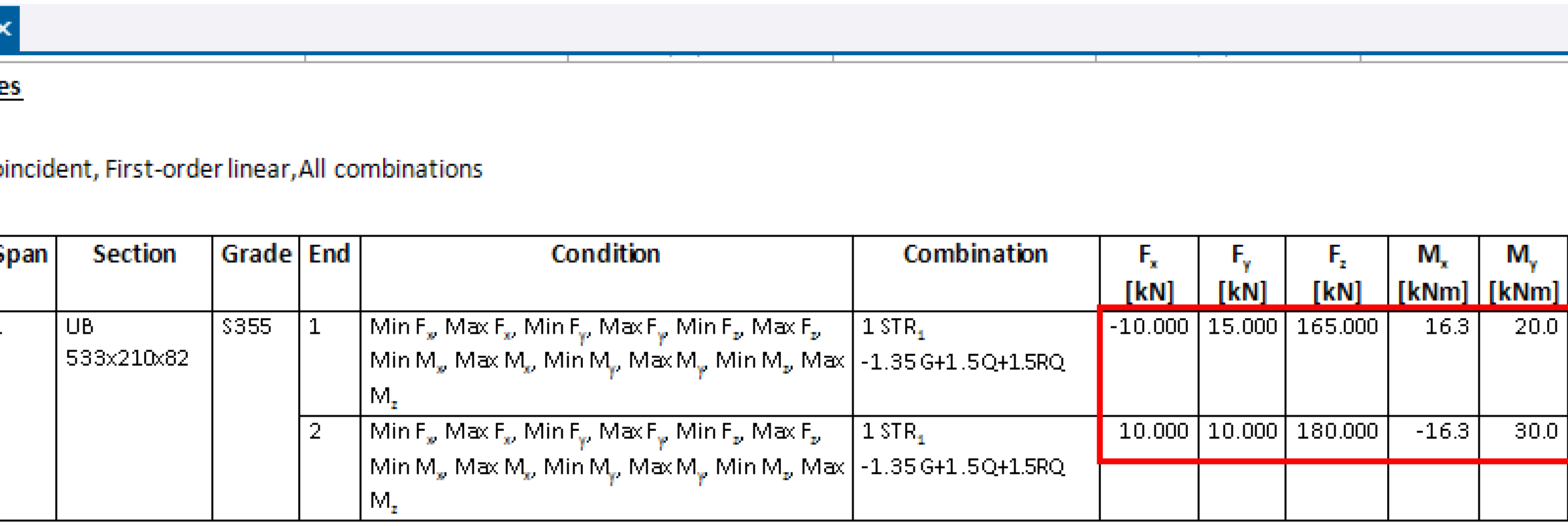
oincident, First-order linear, All combinations

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-6.224	10.438	163.878	16.3	10.4
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	6.224	5.762	175.650	-16.3	24.9

The user wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the ‘Rounding Increment for Force’ and 10kNm in the ‘Rounding Increment for Moment’ under **Beam End Forces**.



The Beam End Forces report displays rounded value of forces and moments.

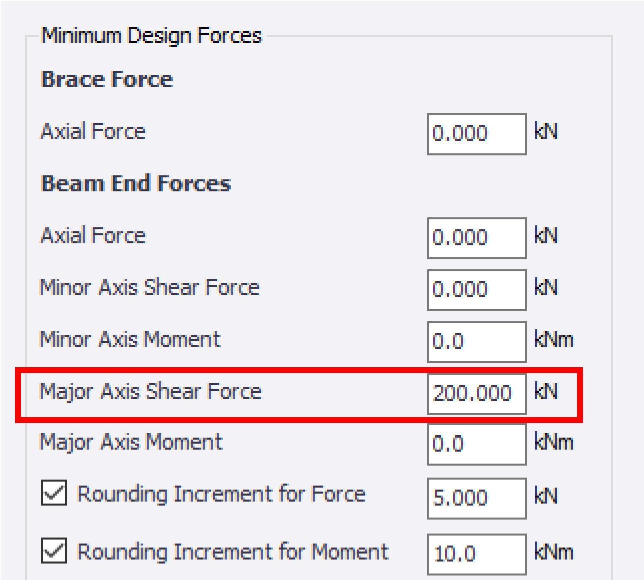

es

oincident, First-order linear, All combinations

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
.	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-10.000	15.000	165.000	16.3	20.0
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	10.000	10.000	180.000	-16.3	30.0

NOTE The rounding increment for moment applies to major axis and minor axis moments, but does not apply to torsional moments (M_x).

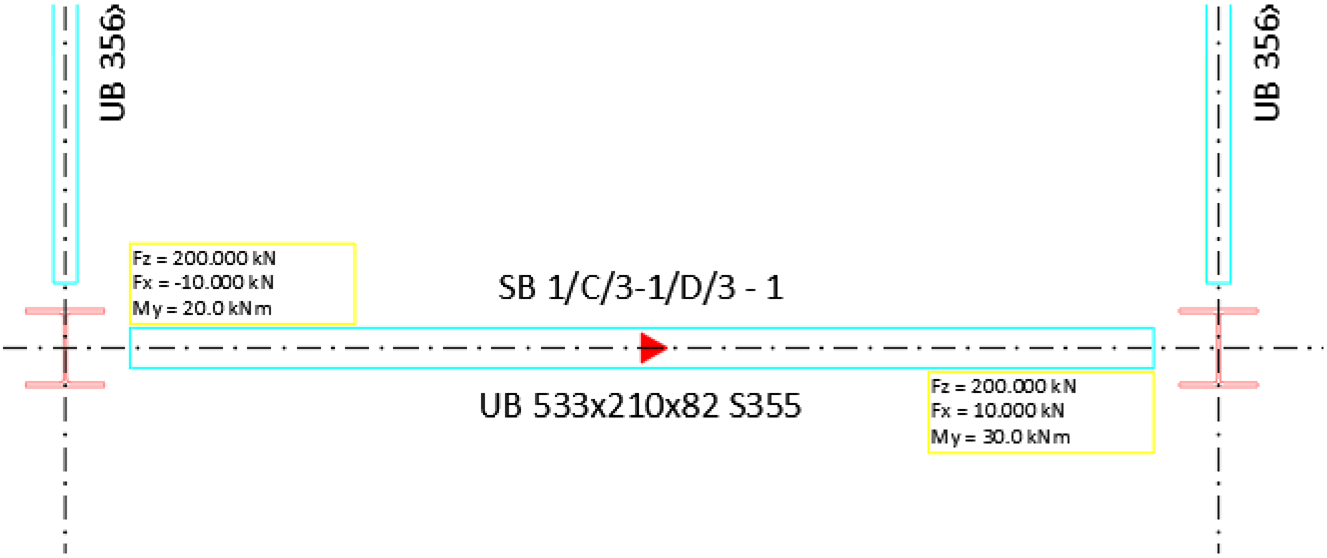
The user might want the forces reported to be not less than minimum values, for example they might enter 200kN in the 'Major Axis Shear Force' under '**Beam End Forces**.



This results in the report being updated as follows.

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-10.000	15.000	200.000	16.3	20.0
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	10.000	10.000	200.000	-16.3	30.0

The changes to the rounding increments and mimimum force are also reflected in the Beam End Forces drawing.



Foundation reaction reports/drawings

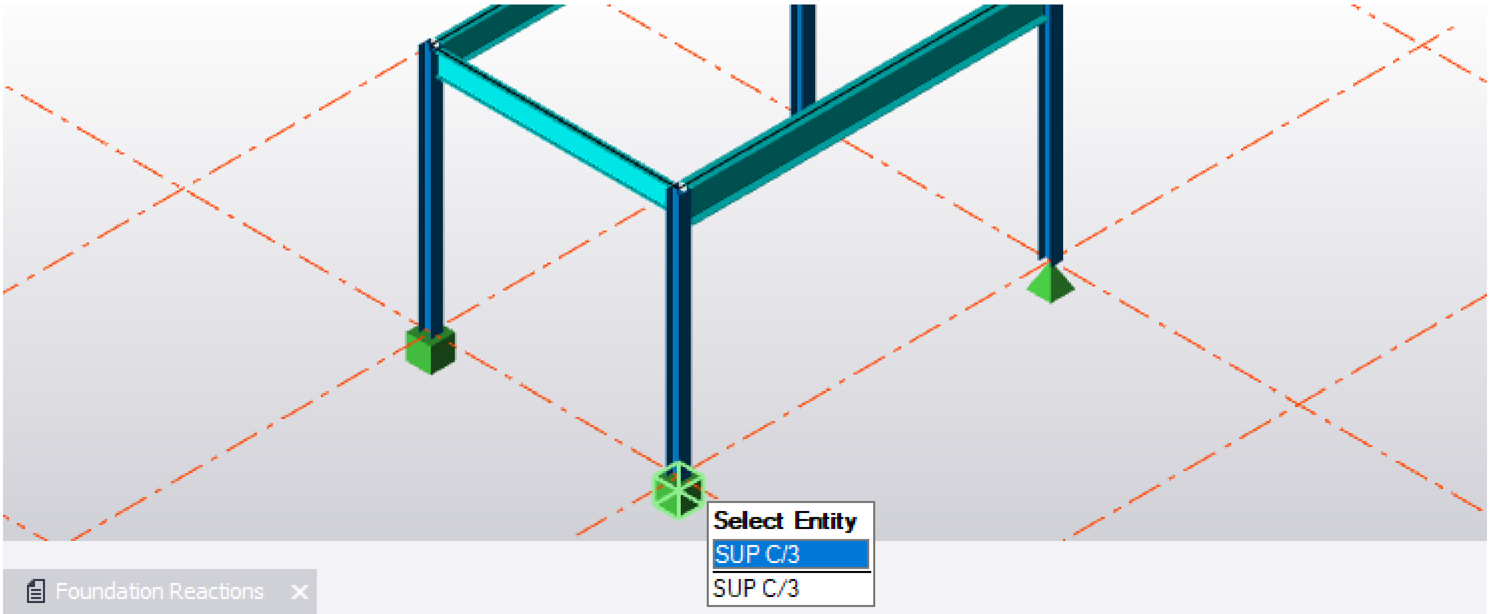
Tekla Structural Designer reports foundation reactions based on analysis results.

The 'Minimum Design Forces' inputs under **Foundation Reactions** allow the user to specify minimum values for the reactions, and also apply rounding increments to them.

NOTE Changed force/moment values are for reporting purposes only.

Consider a fixed end support.

Forces/moments are reported in the Foundation Reactions report.



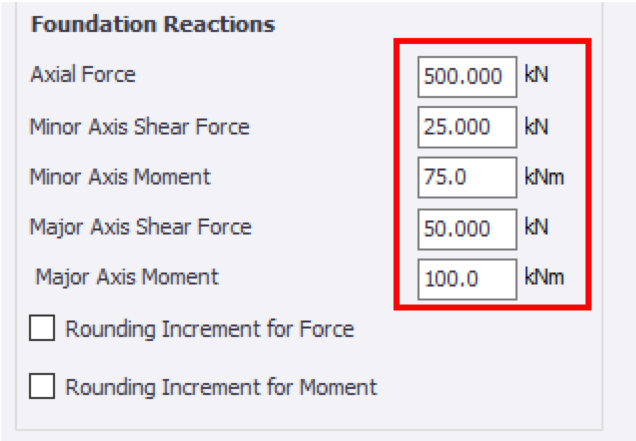
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	147.140	40.500	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	75.824	0.000	0.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	148.911	0.000	0.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	104.596	0.000	0.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	278.195	3.468	-7.276	-30.2	-18.7	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	6.971	0.002	-34.6	0.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	289.967	4.071	-6.224	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	1.691	-0.002	0.0	0.0	0.0

The user wants these reported values to be not less than certain minimum values, so inputs the required minimums under **Foundation Reactions**.



The Foundation Reactions report shows minimum design forces/moments specified by the user.

Foundation Reactions ×

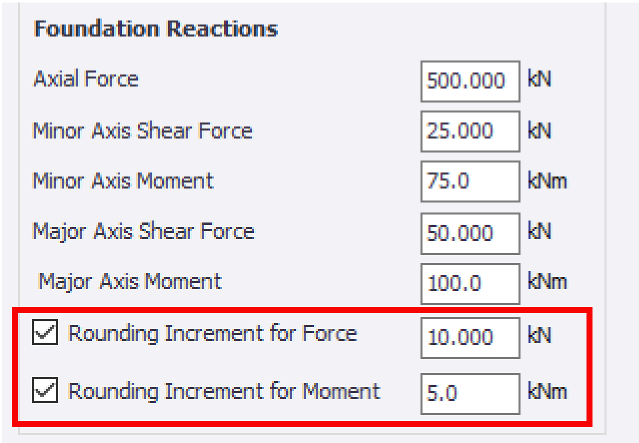
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The user also wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the ‘Rounding Increment for Force’ and 10kNm in the ‘Rounding Increment for Moment’ under **Foundation Reactions**.



When the report is updated, any values exceeding the minimums are rounded up accordingly.

Foundation Reactions ×

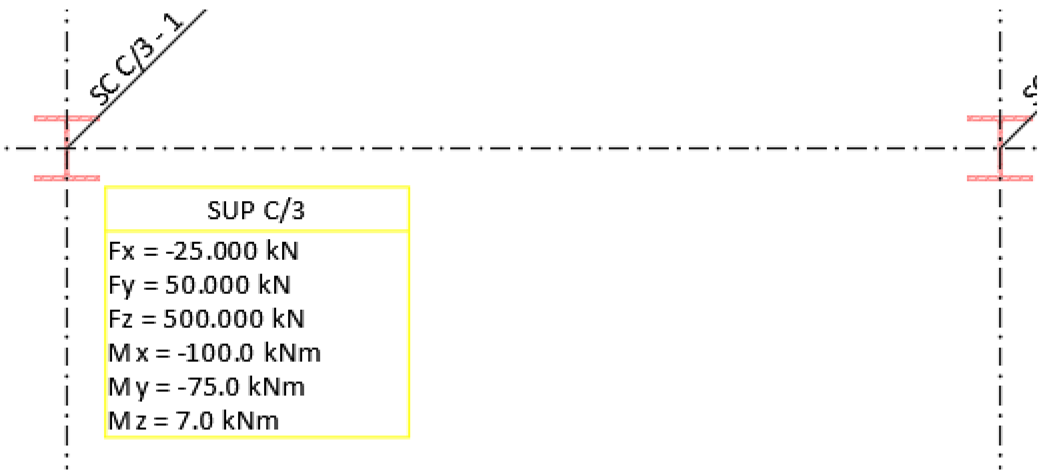
FoundationReactions

FoundationReactions, First-order linear, StrengthFactors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-50.000	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The changes are also reflected in the Foundation Reactions drawing, which now shows the same minimum values of design force and moment.



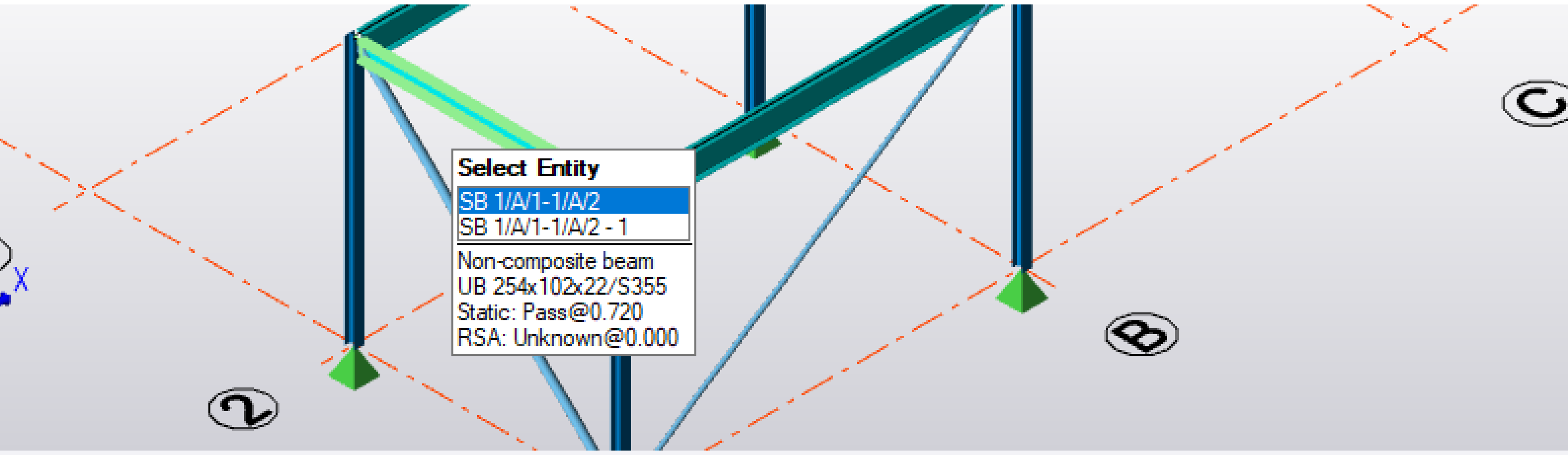
Connection resistance checks

Tekla Structural Designer performs connection resistance checks for simple beams and braces which use design forces from the analysis results.

The ‘Minimum Design Forces’ inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for these forces, and also apply rounding increments to them.

Consider a simple beam connection.

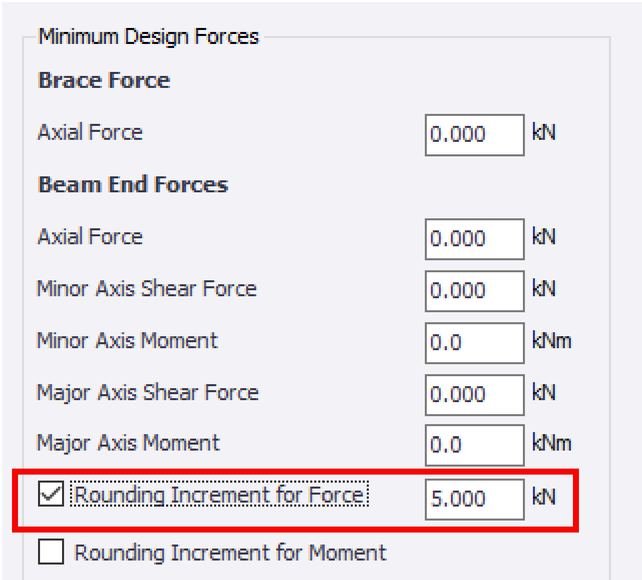
Review Data of Connection Resistance shows the following details.



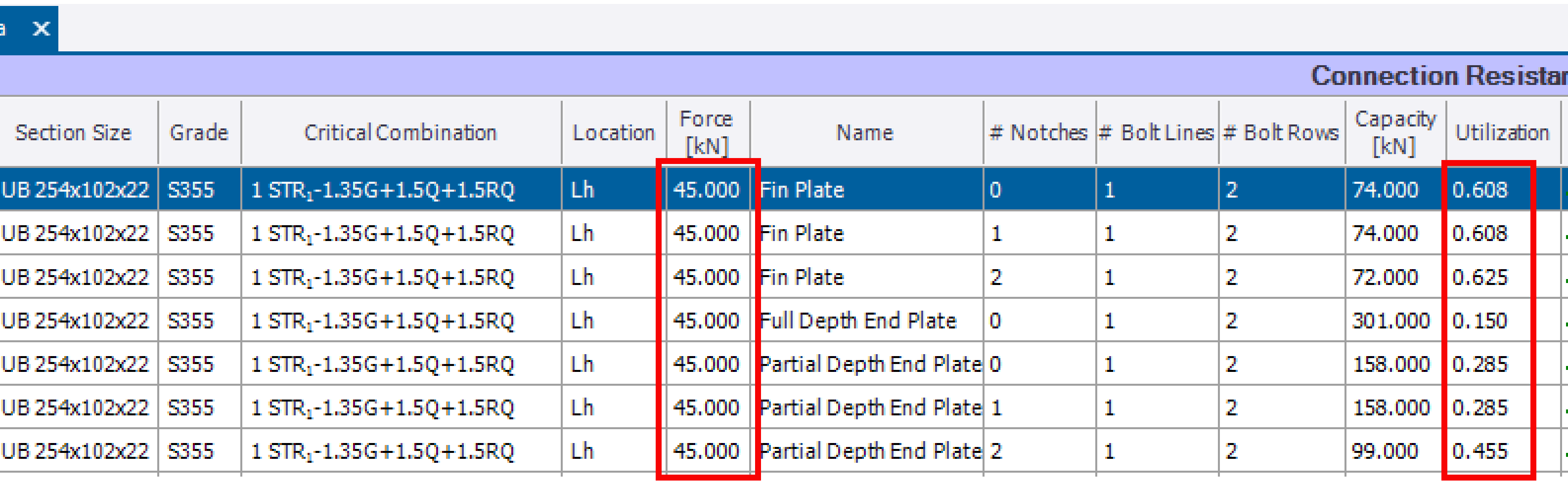
Connection Resistar

Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	0	1	2	74.000	0.575
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	1	1	2	74.000	0.575
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	2	1	2	72.000	0.591
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Full Depth End Plate	0	1	2	301.000	0.141
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	0	1	2	158.000	0.270
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	1	1	2	158.000	0.270
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	2	1	2	99.000	0.430

The user wants the forces rounded to the nearest 5kN so inputs 5kN in the ‘Rounding Increment for Force’ under **Beam End Forces**.

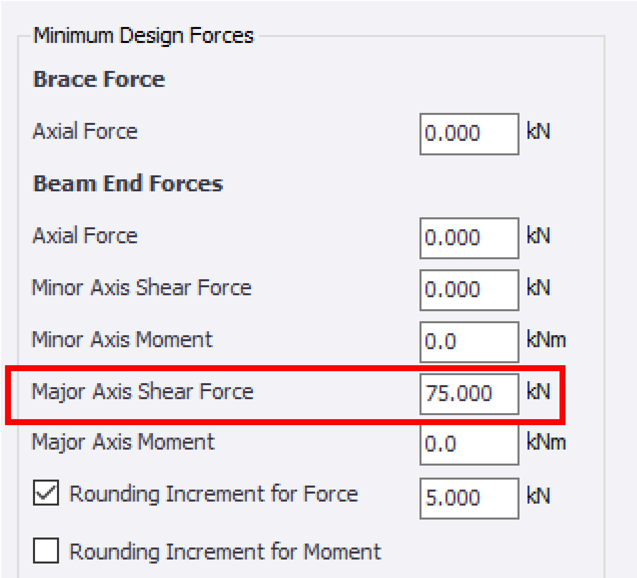


By closing and re-opening the Connection Resistance Review Data, the connection resistance check is updated and now uses the rounded value.


Connection Resistar

Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	0	1	2	74.000	0.608
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	1	1	2	74.000	0.608
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	2	1	2	72.000	0.625
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Full Depth End Plate	0	1	2	301.000	0.150
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	0	1	2	158.000	0.285
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	1	1	2	158.000	0.285
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	2	1	2	99.000	0.455

The user also wants a minimum design force of 75 kN for connection resistance check so inputs 75 kN in the 'Major Axis Shear Force' under **Beam End Forces**.



By closing and re-opening the Connection Resistance Review Data once again, the connection resistance check is updated to use the specified minimum design force.

Connection Resistance

ombination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status	Note
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	0	2	2	92.000	0.815	Pass	
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	1	2	2	92.000	0.815	Pass	Minimum design force us
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	2	2	2	67.000	1.119	Fail	
+1.5Q+1.5RQ	Lh	75.000	Full Depth End Plate	0	1	2	301.000	0.249	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	0	1	2	158.000	0.475	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	1	1	2	158.000	0.475	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	2	1	2	99.000	0.758	Pass	

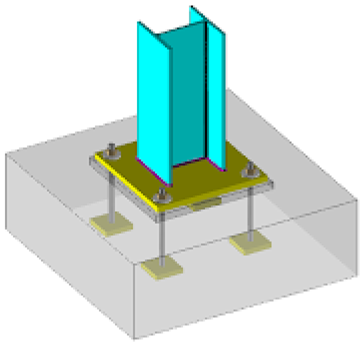
Because the minimum design force is being used this is indicated in the table under the 'Note' column.

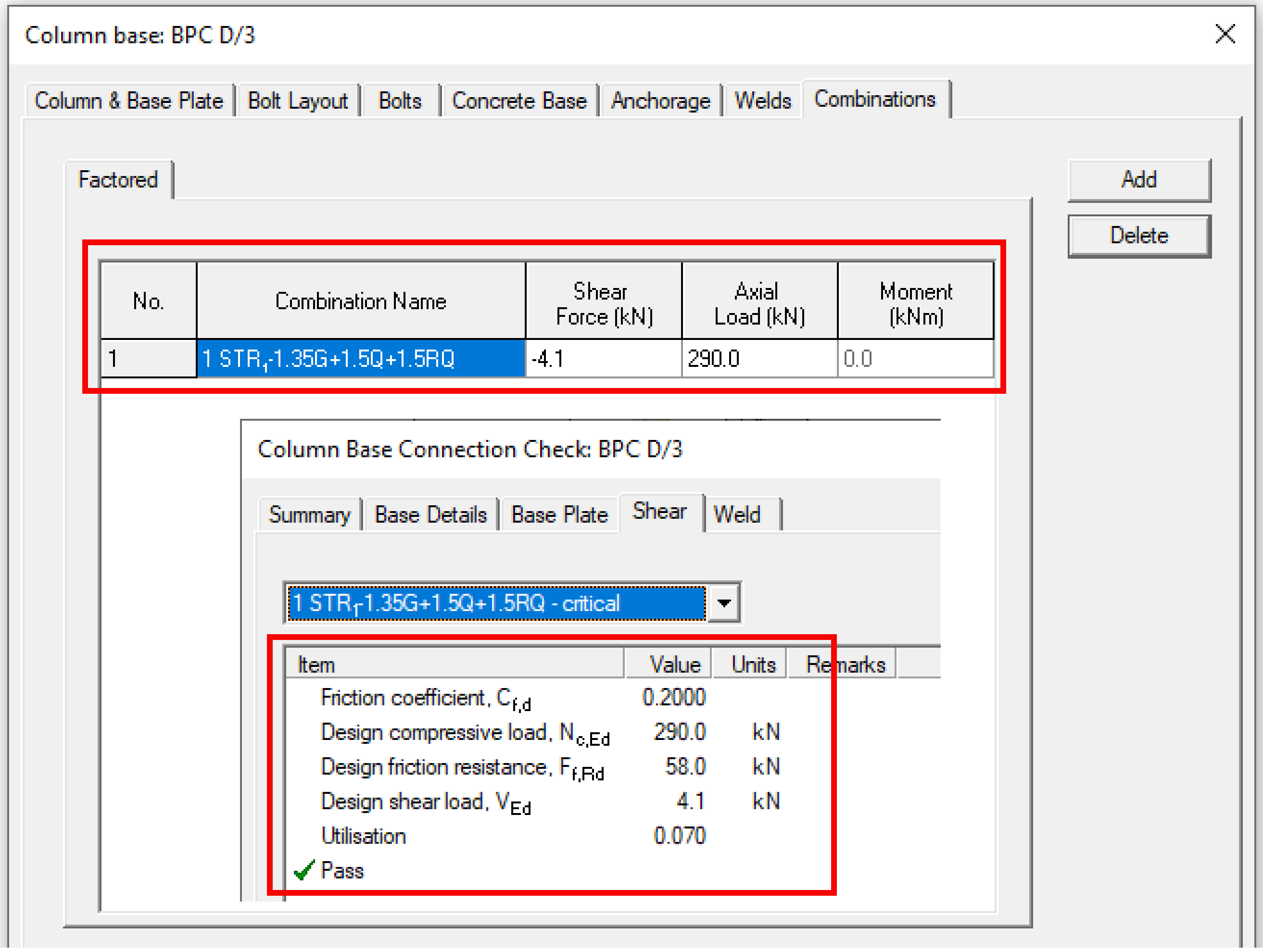
Exported connection forces

Forces exported to Tekla Connection Designer or IDEA StatiCa from Tekla Structural Designer are based on analysis results.

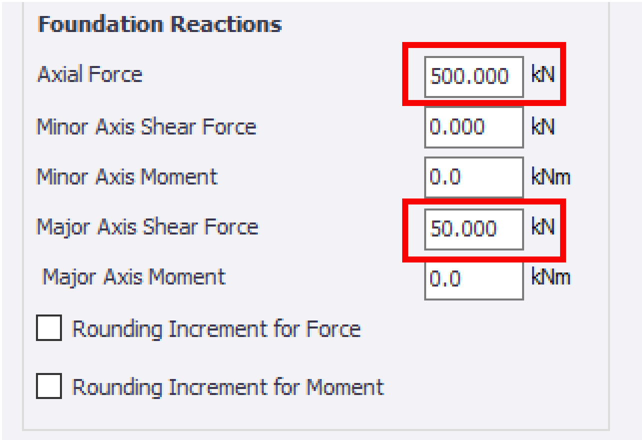
The ‘Minimum Design Forces’ inputs allow the user to specify minimum values for the exported forces, and also apply rounding increments to them.

Consider a base plate connection exported to Tekla Connection Designer and designed for axial load and major shear.





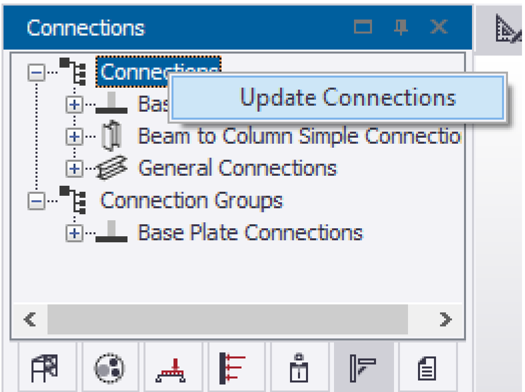
The user wants to design the base plate connection for a minimum shear of 50 kN and for a minimum axial load of 500 kN, so inputs these values under **Foundation Reactions**.



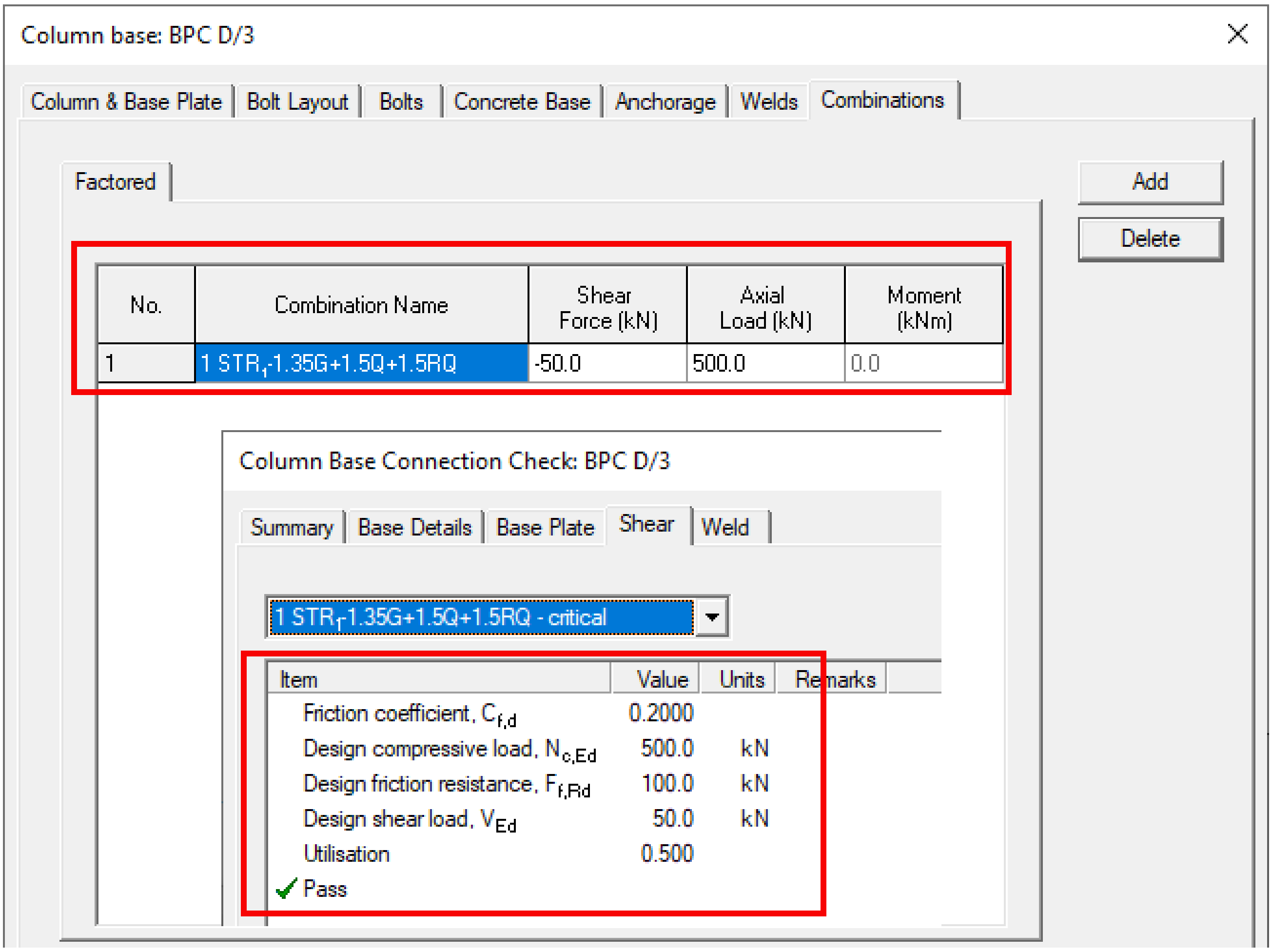
NOTE For other connection types the values under **Brace Forces/Beam End Forces** would be edited instead.

Rounding increments could also be applied if required. These would only apply to forces and moments exceeding the above minimums.

To apply the new minimums to the existing connections, it is necessary to click **Update Connections** from the Project Workspace.



The base plate connection is now designed for the minimum shear of 50 kN and for the minimum axial load of 500 kN specified by the user.



Design Settings - Design Groups and Autodesign

Design groups

Button, command, or option	Description
Members to design using groups	Allows you to select the member types for which you want to apply grouped design. When the members are grouped, only one member in the group is designed. This design is then copied to the remaining members in the group, so that they can be checked. Any failing member in the group is redesigned, and the process is repeated until all

Button, command, or option	Description
	members pass, or a design is not possible.

Autodesign settings

Reset Autodesign to off...	Allows you to control what happens to individual steel and concrete member auto design settings and isolated foundation auto design settings at the end of the design process. The options are:
	• **Always**: the auto design setting is automatically cleared at the end of the design process, so that each member is put into check mode.
	• **Never**: the autodesign settings is always retained as it is at the end of the design process.
	• **When check status is at worst**:
	  ◦ **Pass**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass.
	  ◦ **Warning**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass or Warning.
	  ◦ **Fail**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, or Fail.
	  ◦ **Invalid**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail or Invalid.

	• **Beyond Scope**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail, Invalid, or Beyond Scope. **TIP** The most practical use of **When check status is at worst** is to set it to **Pass** and start with all members in auto design mode. At the end of the first design run, passing members would be set to check mode, allowing you to focus on the remaining members still in auto design mode.

Design Settings - Design Warnings

Warnings used as guidance during the design of a structure to US seismic provisions are in effect by default. These can be de-activated as found useful from this page.

Design Warnings (AISC/ASC Head code only)

Button, command, or option	Description
Concrete Seismic Warnings	
Specified concrete grade is too low	• Compliance with ACI 318-08 and ACI 318-11 Section 21.1.4.2, “(..) f'_c, shall not be less than 3000 psi.“ • Applies to normal weight reinforced concrete columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to increase the concrete grade of the member
Specified longitudinal reinforcement grade too high	• Compliance with ACI 318-11 Section 21.1.5.2, “(..)

Button, command, or option	Description
	reinforcement resisting earth-quake induced flexure, axial force, or both, shall comply with ASTM A706, Grade 60.” • Applies to normal weight reinforced columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to use lower longitudinal reinforcement grades
Column at the end of the beam is not in the same seismic frame	• Used to validate if the frame of which the beam is part of is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete beams part of a seismic force resisting system • Applies to both SRFS type and direction • Checks both ends of each beam span • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam at the end of the column is not in the same seismic frame	• Used to validate if the frame of which the column is part off is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete columns part of a seismic force resisting system • Checks the top of each column stack • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam support region size is too small	• Compliance with ACI 318-08 and ACI 318-11 Section 21.3.4.2, “(..) hoops shall be provided over lengths not less than 2h measured

Button, command, or option	Description
	from the face of the supported member toward the midspan” • Applies to normal weight reinforced concrete beams part of Intermediate and Special Moment Frames • Comes into effect if the length of the end shear design regions is smaller than the requirement • Prompts the designer to review the shear design regions - Design options
High seismic to conventional demand on non-SFRS beams **High seismic to conventional demand ratio of non-SFRS columns** **High seismic to conventional demand of non-SFRS walls**	• Used to recognize when a member has a significant contribution to the building’s lateral force resistance but has not been assigned to the SFRS with the required ductility properties • Specifically apply to each member type • The highest non-seismic design ratio from bending and shear design is obtained • The highest seismic design ratio from bending and shear design is obtained • Comes into effect when the ratio between seismic and non-seismic values is higher than a given threshold (default = 1.0) • The maximum seismic/non-seismic ratio threshold that triggers the warning is set through the **Seismic to conventional minimum demand ratio** field described below • Prompts the designer to either include the member in the SFRS or review the structure layout to reduce the contribution of the member to the lateral force resistance

Button, command, or option	Description
Non-SFRS design requirements applicable in SDC D-F	• Compliance with ACI 318-08 and ACI 318-11 Sections 21.13.1, “Requirements of section 21.13 apply to members not designated as part of the seismic-force-resisting system in structures assigned to SDC D, E and F” • Applies to normal weight reinforced concrete beams, column and walls not included in the SFRS • Comes into effect in Seismic Design Category, SDC D or above • Prompts the designer to apply code prescribed requirements on top of current design
Steel Static Warnings	
Additional notional loads required when alpha Pr/Py > 0.5, C2.3	• Compliance with 7.3 (3) of AISC 360-05 and C2.3 of AISC 360-10 requires the stiffness of certain members to be reduced. When the ratio of the required axial compressive force (Pr) and the axial yield strength (Py = Fy Ag) is > 0.5 then this stiffness must be reduced further • As an alternative which is the approach adopted in Tekla Structural Designer, an additional notional load of 0.001 Yi can be applied instead • When the program identifies that the ratio of Pr/Py exceeds 0.5 a warning is given to alert the designer • If this additional notional load has been dealt with by the designer then this warning is no longer relevant and this option allows you to switch off the warning
Concrete Seismic	
Seismic to conventional minimum demand ratio	• Sets the maximum seismic/non-seismic ratio threshold, which is

Button, command, or option	Description
	used to trigger the **High seismic to conventional demand**warnings listed above • Default: 1.000

Design Settings - Steel Joists

Steel joists

NOTE Steel joists are only applicable in the US.

Button, command, or option	Description
Uniform load tolerance	If the calculated percentage load tolerance for steel joists is less than this value, the load is classified as Uniform.
Equivalent load tolerance	If the calculated percentage load tolerance for steel joists is greater than this value, the load is classified as Non-uniform. **NOTE** If 0% equivalent load tolerance is specified, only pure uniform loading is accepted.
Maximum sum of concentrated loads	Allows you to specify a user-defined maximum sum of concentrated loads that limits K, LH and DLH joists. If any concentrated loads exceed the specified maximum sum, the relevant load combination is classified as Non-uniform.
Maximum eccentricity of zero shear	In the equivalent uniform load method, the position of the point of zero shear relative to the center span point of the joist is determined. The option allows you to specify the maximum eccentricity of zero shear value in the equivalent uniform load method. If the position is located outside the value, the load is classified as Non-uniform.

Button, command, or option	Description
Deflection increase to allow for shear	Deflections due to live load are calculated using the following effective inertia value: $I_{effective} = I_{gross} / (1 + R)$, where R = the deflection increase to allow for shear effects (defaulted to 15%) and applied to KCS only. The option allows you to adjust the value of R, (the deflection increase to allow for shear, percentage).

Design Settings - Sway & Drift Checks

Sway & drift check settings

Button, command, or option	Description
Merge short stacks	With this setting off the Sway/Drift, Wind Drift & Seismic Drift checks are performed for all column stacks and wall panels irrespective of their stack lengths. With this setting on you can specify a minimum stack length. **NOTE** The ‘Merge short stacks’ option does not apply to columns/walls where you have elected to merge stacks manually (via the column/wall properties). The following logic is applied to merging stacks when the setting is on: 1. Start at the topmost stack of the column/wall. 2. If the stack length is less than the merge limit then merge with the stack below. 3. Check again to see if then new length exceeds the merge limit, if not then merge again with the next stack below.

Button, command, or option	Description
	4. Repeat step 3 as required until the stack length exceeds the merge limit. 5. For subsequent stacks repeat steps 2 and 3 as required until the length exceeds the merge limit. 6. When the bottommost stack is reached, if this needs to be merged, merge with the stack above. **NOTE** If a single stack column or single panel wall length is less than the merge short stacks limit it is not considered for sway/drift, wind drift, seismic drift checks.
Check wind cases only	Allows you to set that the wind drift check performed on all columns only considers the effect of the wind load cases in each wind combination. Clear the setting to consider the effects of all load cases in wind combinations, such as drift induced by gravity loads.

Design Settings - Fire check

Fire check (Eurocode Head code only)

Button, command, or option	Description
Time interval for critical temperature iteration if 'Unprotected'	default 5 sec
Time interval for critical temperature iteration if 'Protected'	default 30 sec

Design Settings - Timber

The settings allow you to apply your own set of timber design defaults that broadly apply to all members. These are incorporated into Tedds calculations as default variables, which can then be edited within the calculation if required.

Timber (US Headcode)

Button, command, or option	Description
Service condition	NDS: Service condition • Dry (default) • Wet
Temperature range	• up to 100 degF (default) • 100 degF to 125 degF • 125 degF to 150 degF
Is section incised	• yes • no (default) The incisions made when applying a preservative treatment result in a loss of area and section modulus. When you indicate that the section is incised, appropriate incising factors are applied in the Tedds calculation.

Timber (Eurocode)

Button, command, or option	Description
Service class	• 1 (default) • 2 • 3
Individual grade stamp	• yes • no (default) Depending on the National Annex being worked to, the partial factor for material properties can be reduced when timber has been individually marked.

Slab deflection settings

The **Slab Deflection** page and its subpages allow you to control the slab deflection defaults applied:

- In the **current** project - when accessed from the Slab Deflection Settings dialog (page 2440) on the **Slab Deflection** tab.
- In **new** projects - when accessed from the Settings dialog (page 2424) on the **Home** tab.

Slab deflection settings when accessed from the Slab Deflection Check Catalogue

Button, command, or option	Description
New Load Event Defaults subpage	
Start time offset	Allows you to define a default start time offset value for new event load start times. The value of a new event load start time is always the previous event start time + the default start time offset.
Construction load	Allows you to define a default value for new construction loads. **NOTE** The default construction load value depends on the head code that you are using.
New Check Defaults subpage	
Deflection limit, L /	Allows you to define a default deflection limit to new checks added to the When a new check is added to the Slab Deflection Check Catalogue it initially defaults to the deflection limit set here.
Aging, Creep & Shrinkage subpage	
Allowance for shrinkage effects in total deflection	Allows you to define an amplification base factor for shrinkage. **NOTE** The value must be within the range from 0.1 to 0.9.
Aging Coefficient	Allows you to select between 2 creep/aging methods: **User defined** and **Automatic**. **NOTE** The **Aging Coefficient** option is only available for the US head code.

Button, command, or option	Description
Modification Factors subpage	Allows you to adjust the properties used for the different element types in the iterative cracked section analysis.
Iteractive Cracked Section Analysis subpage	
Global Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 200. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00100. • **Relative**: Default: ON. **TIP** To resolve global convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**.
Local Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 500. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.000001. • **Smoothing Parameter**: A property only available for the

Button, command, or option	Description
	Eurocode head code. Default = 0.005. **TIP** To resolve local convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**. • If you are using the Eurocode head code, increase the value in **Smoothing Parameter**.

Slab deflection settings in the Settings dialog

Button, command, or option	Description
New Load Event Defaults subpage	
Start time offset	Allows you to define a default start time offset value for new event load start times. The value of a new event load start time is always the previous event start time + the default start time offset.
Construction load	Allows you to define a default value for new construction loads. **NOTE** The default construction load value depends on the head code that you are using.
New Check Defaults subpage	
Deflection limit, L /	Allows you to define a default deflection limit to new checks added to the When a new check is added to the Slab Deflection Check Catalogue it initially defaults to the deflection limit set here.
Aging, Creep & Shrinkage subpage	

Button, command, or option	Description
Allowance for shrinkage effects in total deflection	Allows you to define an amplification base factor for shrinkage. **NOTE** The value must be within the range from 0.1 to 0.9.
Aging Coefficient	Allows you to select between 2 creep/ aging methods: **User defined** and **Automatic**. **NOTE** The **Aging Coefficient** option is only available for the US head code.
Load Event Sequence subpage	Allows you to modify the global event sequence according to your needs. **TIP** The global event sequence provides the basis for the model event sequence that you can modify, whereas the model event sequence provides the basis for a submodel event sequence that you can modify.

Drawing settings

The **Drawings** page and its subpages allow you to adjust different drawing settings:

- In the **current** project - when accessed from the Drawing Settings dialog (page 2405) on the **Draw** tab.
- In **new** projects - when accessed from the Settings dialog (page 2424) on the **Home** tab.

Export preferences

NOTE The **Export Preferences** subpage is only available when drawing settings are accessed from the Settings dialog (page 2424)

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose export preferences you want to modify.
Minimum Text Block Spacing	Allows you to define a minimum value between independent drawing blocks. **NOTE** If the value is too great, text labels can end up far from the objects to which they refer.
Available Scales list	Displays the existing drawing scales. **NOTE** You can add user-defined scales in the list by typing the scale in the **Scale** field and clicking the **Add** button, or delete them by selecting the scale and clicking the **Remove** button.
Scale	Allows you to create a user-defined drawing scale by typing it in the field.
Name	Allows you to specify a name for a user-defined drawing scale.

Layer configurations

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer configurations you want to modify. **NOTE** Use the subpages of the **Layer Configurations** subpage to modify the required drawing type.
Available Configurations list	Displays the available layer configurations and allows you to select or modify them. You can use the following buttons to modify the available layer configurations: • **Add**: creates an empty layer configuration that you can modify

Setting	Description
	in the **Active Configuration** section. • **Add copy...**: allows you to copy a drawing item to create a new layer configuration. • **Remove**: deletes the selected layer configuration.
Active Configuration	Displays the name and layers in the selected layer configuration.
Name	Displays and allows you to modify the name of the selected layer configuration.
Layers list	Displays and allows you to select or clear the layers that are included in the selected layer configuration. **TIP** If you access the drawing settings from the **Draw** tab and modify them, consider saving the changes to a settings set, so that you can use them in later projects as well.

Layer styles

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer styles you want to modify. **NOTE** Use the subpages of the **Layer Styles** subpage to modify the required drawing type.
Available Styles list	Displays the available layer styles and allows you to select or modify them. You can use the following buttons to modify the available layer styles: • **Add**: creates an empty layer style that you can modify in the **Active Style** section.

Setting	Description
	• **Add copy...**: allows you to copy a drawing item to create a new layer style. • **Remove**: deletes the selected layer style.
Active Style	Displays the name and layers in the selected layer style.
Name	Displays and allows you to modify the name of the selected layer style.
Apply to All...	This button opens a dialog for rapidly applying a single color, report color, line type, font, or font size to all layers in the active layer style.
Layers list	Displays and allows you modify the appearance of layers that are included in the active layer style.
Description	The layer description is fixed and can't be changed.
Name	The layer name as it will appear in the drawing can be edited if required.
Is Merged, **Merged with**	By checking the box you are able to merge two or more layers together. All objects in the merged layer are drawn in the **Merged with** layer, adopting its line type, font and font size.
Color	The colors specified here are used for the different layers in the dxf drawing output
Report Color	**Member Reports** have the facility to include a member drawing along with the calculations and diagrams etc. When a member report is printed the drawings adopt the report colors that have been specified here.
Line Type	Different line types can be specified for the different drawing layers as required.
Font	Different fonts can be specified for the different drawing layers as required.

Setting	Description
Font Size	Different font sizes can be specified for the different drawing layers as required.

Planar drawing options

. . . . General (all drawing variants)

Setting	Description
Hatching	Allows you to select how columns and walls are hatched in planar drawings. You can select or clear the following options: • **Show columns and walls above the level as hatched**: The columns and walls that continue above the current level are hatched. • **Show transfer columns and walls as cross hatched**: Transfer columns and walls are cross hatched.
Force and Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces • Column Splice Loads Allows you to select which force and moment values are displayed in planar drawings. The options are: • **Strength factors**: Displays the strength factors in planar drawings. • **Service factors**: Displays the service factors in planar drawings. **Additionally factor by** allows you to specify a value by which the strength or service factors are factored in the selected drawing category.

Setting	Description
Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces • Column Splice Loads Allows you to select which moment values are displayed in planar drawings. The options are: • **None**: Does not display any moment values. • **Only greater than**: Moment values are only displayed if their value is greater than the value specified here. • **All**: Displays all moment values, even if zero.
Shear Force Values	**NOTE** Foundation Reactions drawing variant only. Allows you to select which shear values are displayed in Foundation Reactions drawings. The options are: • **None**: Does not display any shear values. • **Only greater than**: Shear values are only displayed if their value is greater than the value specified here. • **All**: Displays all shear values, even if zero.
Axial Force Values	**NOTE** Beam End Forces drawing variant only. Allows you to select which axial force values are displayed in planar drawings. The options are: • **None**: Does not display any axial force values.

Setting	Description
	• **Only greater than**: Axial force values are only displayed if their value is greater than the value specified here. • **All**: Displays all axial force values, even if zero.
Display End Forces for	**NOTE** Beam End Forces drawing variant only. Allows you to select the members for which end forces are displayed.
Enveloped Reaction Values	**NOTE** Beam End Forces drawing variant only. **Include coincident forces**: when this option is selected, further options are presented allowing you to choose the max/min values and coincident forces to display when the beam end forces drawing is created for a loading envelope.

. . . . Beams (all drawing variants)

Setting	Description
Grouped Beam Labelling	Allow you to modify the labeling of beams when the beams have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the beam label, or clear the option to use the design group name in the beam label. • **Include the beam name**: Select the option to include the beam name in the beam label for grouped beams.
Concrete Beam Labelling Position	Allows you to set the position of the beam label in relation to the beam in planar drawings. The options are: • **Above**

Setting	Description
	• **Inside** • **Below**
Beam Mark Position	Allows you to adjust the appearance and position of the beam mark in relation to the brace in planar drawings. To display beam marks, select **Show beam mark**. The position options are: • **Above** • **Below**
Beam Attributes Position	Allows you to adjust the appearance and position of beam attributes in relation to the beam in planar drawings. To display beam attributes, select **Show beam attributes**. The position options are: • **Above** • **Below**
Beam Attributes	Allows you to select which beam attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label. • **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list. • **Transverse reinforcement**: Displays transverse reinforcement in planar drawings.

. . . . *Braces (all drawing variants)*

Setting	Description
Brace Mark Position	Allows you to adjust the appearance and position of the brace mark in relation to the brace in planar drawings. To display brace marks, select **Show brace mark**. The position options are: • **Above** • **Below**
Brace Attributes Position	Allows you to adjust the appearance and position of brace attributes in relation to the brace in planar drawings. To display brace attributes, select **Show brace attributes**. The position options are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in planar drawings.

. . . . *Columns (all drawing variants)*

Setting	Description
Grouped Column Labelling	Allow you to modify the labeling of columns when the columns have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the column label, or clear the option to use the design group name in the label. • **Include the column name**: Select the option to include the column name in the column label for grouped columns.
Column Mark Position	Allows you to adjust the appearance and position of the column mark in relation to the brace in planar drawings. To display column marks, select **Show column mark**. The position options are:

Setting	Description
	• **Above** • **Below**

Column Attributes Position	Allows you to adjust the appearance and position of the column attribute in relation to the brace in planar drawings. To display column attributes, select **Show column attributes**. The position options are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

Column Attributes	Allows you to select which column attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options: • **Show column size in parentheses**: Places brackets around the column size in the column label. • **Grade**: Displays the column grade in planar drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

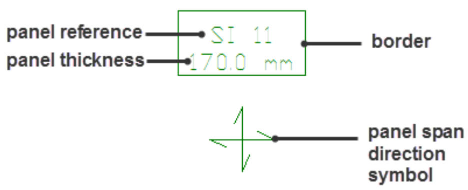
. . . . Walls (all drawing variants)

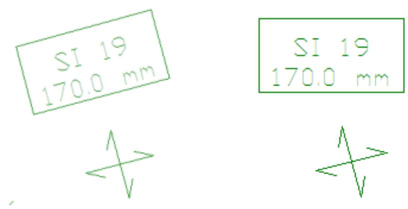
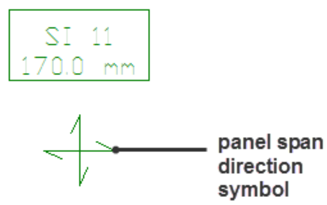
Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in planar drawings. The options are: • **Above** • **Inside** • **Below**

Setting	Description
Show wall size in parentheses	Allows you to select whether brackets are placed around the wall size in the wall label.

Wall Reactions	Allows you to select whether or not to include distributed vertical reaction values of walls in the foundation reactions planar drawing. These are available in two different formats:
	• **Show distributed wall reactions table**: a table with the distributed wall reaction end values is included in the drawing,
	• **Show vertical reaction values**: values are added for each wall in the planar drawing,
	    • **Distributed (Maximum)**: Maximum absolute values of the vertical distributed reaction retaining sign,
	    • **Distributed (Average)**: Average value of the vertical distributed reaction along the length of the bottom analytical chord of the wall,
	    • **Distributed (Extremes)**: Extreme values at each end of the wall bottom analytical chord,
	    • **Total**: Single integrated total vertical reaction of the wall.

. . . . Slabs/Mats (all drawing variants)

Setting	Description
Panel Labelling	Allows you to select which panel properties are included in panel labels and how panel labels are displayed in planar drawings. 

Setting	Description
	You can select or clear the following options: • **Include panel reference**: Allows you to include the panel reference in the panel label. • **Include panel thickness**: Allows you to include the panel thickness in the panel label. • **Include surface offset (if non-zero)**: Allows you to include any surface offset that has been applied to the panel in the panel label. • **Include border around label**: Allows you to add a border around the label. • **Align label to panel reinforcement**: Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 

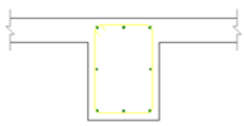
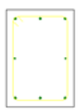
. . . . Loads (Loading Plan drawing variant only)

Setting	Description
Display size	Allows you to adjust the width and height of line, udl, and vdl loads and the size of point loads in planar drawings. The different options that you can adjust are: • **Width of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the width of the hatched rectangle. • **Height of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the height of the hatched rectangle. • **Point load marker size**: Allows you to adjust the size of point loads in planar drawings.
Show dimensions for panel loads	Allows you to include the dimensions of panel loads in planar drawings.
Include dimensions to reference points	Allows you to include the dimensions from panel loads to their reference points in planar drawings.
Show dimensions for member loads	Allows you to include the dimensions of member loads in planar drawings.

Member detail options

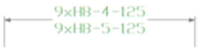
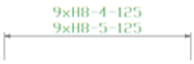
. . . . Beam Detail

Setting	Description
Content subpage	
Grouped Beams tab	
Show number of beams in group	If the beams in the model have been arranged into detailing groups, selecting the option causes the detailing number to be used as the

Setting	Description
	member label instead of the beam reference. If detailing groups have not been used, the beam reference is always used as the member label.
Levels tab	
Show span levels	Allows you to display the span levels on the elevation.
Cross-sections tab	
Spans list	Allows you to select which cross-sections to display. You can decide to display no cross-sections, only the cross-sections for first spans of multi-span beams, or all spans.
Positions list	If cross-sections are displayed, the list allows you to select where they are positioned. You can decide to only position cross-sections for spans, for supports, or for supports and spans.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Show slab lines in section	Select the option to display slab lines in sections, as shown in the first image:  Clear the option to not display slab lines in sections, as shown in the second image: 

Setting	Description
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Show steel bar layer information	Allows you to display steel bar layer information (B1, B2, T1, T2, and so on).
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL).
Anchorage lengths list	Allows you to select whether any required anchorage lengths are dimensioned, not dimensioned, or the dimension is replaced by a standard label (AL).
Axes list	Allows you to select whether axes are not displayed, are displayed above the beam with dimensions, or are displayed below the beam with dimensions.
Additional bottom span bar positioning dimensions	Allows you to dimension any existing optional second span bars from the face of the support.
Support region length	If different stirrup regions have been used along a beam span, the option allows you to dimension the length of the support regions on the elevation.
First and last stirrups	Allows you to add dimensions from the face of the supports to the first and last stirrups.
Support columns and clear spans	Allows you to add dimensions showing the width of each supporting column and the clear beam span between supports.

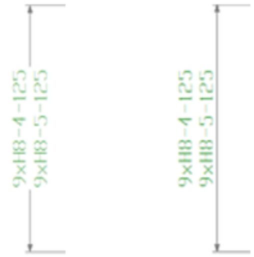
Setting	Description
Beam section	Allows you to dimension the beam depth and width on the cross-section.
Slabs in beam sections	Allows you to dimension the slab depth on the cross-section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in beam detail drawings.
Style subpage	
Beam Labels tab	
Print beam labels below the detail	Allows you to display the beam label centrally below each span. Clear the option to position the label immediately above each span.
Underline beam labels	Allows you to to underline the beam label on the elevation.
Label every span of multi-span beams	Allows you to display a label for each span in multi-span beams.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each beam line	Allows you to restart the section labels for each line when multiple beam lines are displayed on the same drawing sheet.
Add beam name label as prefix to section labels	Allows you to prefix each section label with the beam name.
Place cross-sections underneath the elevation	Allows you to to position cross-sections under the elevation.
Longitudinal Bars tab	
Draw bobs as shifted	Allows you to apply a slight offset to bobbed bars in drawings, so that the bobbed bars do not overlap each other in drawings an therefore, the drawings are easier to read.
Display only a single side bar in detail	Allows you to draw only a single side bar is drawn full length when multiple side bars are required in each face. Clear the option to draw all side bars full length.
Stirrups tab	

Setting	Description
Draw stirrup labels in line	Allows you to display stirrup labels in line on the elevation, as shown in the first image:  Clear the option to display stirrup labels above the line, as shown in the second image: 
Print stirrup labels inside beam	Allows you to display the stirrup labels inside the beam. Clear the option to display the stirrup labels below the beam.
Stirrup label distance from bottom edge [n] % of the beam depth	Allows you to control the vertical position of stirrup labels that are positioned inside the beam.
Dimensions tab	
Lap and anchorage rounding increment	Allows you to control the rounding increment of lap and anchorage dimensions.

. . . . Column Detail

Setting	Description
Content subpage	
Grouped Columns tab	
Show number of columns in group	Allows you to display the number of columns in a column group.
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**

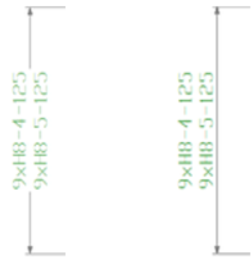
Setting	Description
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.
Connecting elements and clear heights	Allows you to add dimensions connecting elements and clear heights.
Column section	Allows you to add column dimensions to the column section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in column detail drawings.
Style subpage	
Column Labels tab	
Underline column labels	Allows you to to underline the column label on the elevation.
Cross-sections tab	

Setting	Description
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each column line	Allows you to restart the section labels for each line when multiple column lines are displayed on the same drawing sheet.
Add column name label as prefix to section labels	Allows you to prefix each section label with the column name.
Ties tab	
Draw tie labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Dimensions tab	
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

. . . . *Wall Detail*

Setting	Description
Content subpage	
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**

Setting	Description
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.
Wall section	Allows you to add wall dimensions to the wall section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in wall detail drawings.
Style subpage	
Wall Labels tab	
Underline wall labels	Allows you to underline the wall label on the elevation.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Add wall name label as prefix to section labels	Allows you to prefix each section label with the wall name.

Setting	Description
Don't label typical panel bars	Allows you to not label typical bars in cross-sections. Clear the option to label every bar in cross-sections.
Horizontal Bars and Ties tab	
Draw labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Label bars inside panel	Allows you to label bars inside the wall panel.
Dimensions tab	
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

Member schedule options

. . . . General

Setting	Description
NOTE The settings that you see on the **General** tab depend on which drawing category subpage you have selected in the left side pane.	
Item list	Allows you to select an item to view the text label that will applied to it in the concrete beam schedule drawing.
Text	Allows you to modify the text displayed in the beam schedule for the item selected in the **Item** list.
Size Column Format	Allows you to select whether to display the width of the beam or its height first in the size column.
Use single column for size	Allows you to display both the width and height in a single column.

Setting	Description
Use single column for bottom bars	Allows you to display the bottom bars in a single column
Omit top middle bars column	Allows you to omit the top middle bars from the schedule drawing.
Use single column for stirrups	Allows you to display the stirrups in a single column.
Show design group name	Allows you to display only the design group name in the mark column.
Show grouped column number	Allows you to display column groups.
Include starter bars	Allows you to include starter bars in member schedule drawings.
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in member schedule drawings.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **Standard** • **IStructE**
Display bar marks	Allows you to display bar marks in cross-section labels.
Show outline of stack below	Allows you to display the outline of the column stack below.
Dimension column section	Allows you to add column dimensions to the column section.
Dimension levels	Allows you to add dimensions between levels.
Show outline of panel below	Allows you to display the outline of the wall panel below.
Dimension wall section	Allows you to add wall dimensions to the wall section.

. . . . Bar Key

Setting	Description
Reference column	Lists every reference that can potentially appear in the bar bending

Setting	Description
	details table in concrete beam schedule drawings. **TIP** Hover the mouse pointer over a reference to see the bar and its associated note.
	Allows you to use a custom name for the selected reference.
	Allows you to specify the custom name that replaces the original reference in the bar bending details table if the

Slab and mat layout options

. . . . General

Setting	Description
Layout subpage	
General tab	
Show columns and walls above the level as hatched	Allows you to hatch columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.

. . . . Beams

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Beam Labelling position	Allows you to set the position of the beam label in relation to the beam in slab and mat layout drawings. The options are: • **Above** • **Inside**

Setting	Description
	• **Below**
Show beam mark	Allows you to display beam marks in slab and mat layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in slab and mat layout drawings. The position options for beam attributes are: • **Above** • **Below**
Show beam size in parentheses	Allows you to place brackets around the beam size in the beam label.
Grade	Allows you to display the beam grade in slab and mat layout drawings.

. . . . Braces

Setting	Description
Show brace mark	Allows you to display brace marks in slab and mat layout drawings. The position options for brace marks are: • **Above** • **Below**
Show brace attributes	Allows you to display brace attributes in slab and mat layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in slab and mat layout drawings.

. . . . Columns

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in slab and mat layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in slab and mat layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

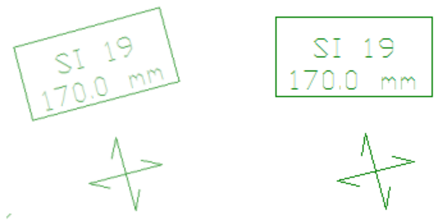
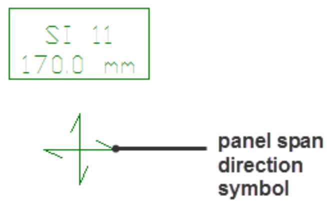
Show column size in parentheses	Allows you to place brackets around the column size in the column label.
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

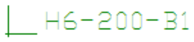
. . . . Walls

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in slab and mat layout drawings. The options are:

Setting	Description
	• **Above** • **Inside** • **Below**
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

. . . . Slabs/Mats

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 

Setting	Description
Extend loose bar panel reinforcement lines across full panel	Allows you to draw bar panel reinforcement across the entire panel. Clear the option to display the loose bar reinforcement as shown in the following image: 
Always show main bar layer for rectangular mesh	If the mesh is not a square mesh, the main bars are normally put on the outer layer of the drawing, and no text is required in the drawing. However, if the bars are not on the outer layer of the drawing, they are indicated by adding B2 (if bottom mesh) or T2 (if top mesh) aligned to the main bar direction. **NOTE** If a square mesh (has the same size and spacing of bars in both directions) is applied, a square mesh symbol is used, and bars are shown equally spaced in both directions. If the mesh is not a square mesh (does not have the same size and spacing of bars in both directions), a rectangular mesh symbol is used. Bars are shown in both directions, but with closer spacing for the more closely spaced bars in the mesh.
Anchorage rounding increment	Allows you to specify the rounding value applied to the anchorage length.

. . . . Patches

Setting	Description
Show patches with no reinforcement	Allows you to display patches that have no reinforcement specified in drawings.

Setting	Description
Draw full anchorage lengths	Allows you to display bars with full anchorage lengths.
Draw curtailed (indicative) anchorage lengths = max()	Allows you to draw curtailed (indicative) anchorage lengths instead of full anchorage lengths and specify their size.

. . . . Punching Shear

Setting	Description
Show punching reinforcement details	Allows you to include a detail to the side of the layout showing the punching shear reinforcement provided.
Don't show area of steel requirement where rails have been designed	Allows you to only display the area of steel requirement for punching shear check items where rails are not provided (such as walls or column drops). Clear the option to display the area of steel requirement for all punching check items.
Hide punching reinforcement on the main layout	Allows you to hide the punching reinforcement on the main layout. The reinforcement is still displayed in the punching check detail drawing, provided that **Show punching reinforcement details** is selected.

Slab and mat punching check detail options

Button, command, or option	Description
Content subpage	
Dimensions tab	
Show column to first stud spacing	Allows you to add a dimension from the column face to the first stud.
Show stud spacing	Allows you to add dimensions displaying the stud spacings along the rail.
Show rail spacing	Allows you to add dimensions displaying the rail spacings.

Button, command, or option	Description
Include schematic showing stud dimensions	Allows you to add a schematic displaying the stud width and height above the rail.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in punching check detail drawings.
Style subpage	
Show column as hatched	Allows you to hatch the column.
Underline punching check label	Allows you to underline the punching check label in punching check detail drawings.

Foundation layout options

. . . . General

Setting	Description
Show columns and walls above the level as hatched	Allows you to hatch the columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.
Show pile location and loading table	Click **Columns** to choose which columns of tabular data are to be included in the table on the drawing. **NOTE** Only the single most critical load condition is reported in the table for each pile. - could be tension or compression - could be a gravity, wind, or seismic combination

. . . . Beams

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.

Setting	Description
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Concrete Beam Labelling Position	Allow you to modify the labeling of beams when the beams have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the label, or clear the option to use the design group name in the label. • **Include the beam name**: Select the option to include the beam name in the label for grouped beams.
Show beam mark	Allows you to display beam marks in foundation layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in foundation layout drawings. The position options for beam attributes are: • **Above** • **Below**
Beam Attributes	Allows you to select which beam attributes are displayed in foundation layout drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label. • **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list.

Setting	Description
	• **Transverse reinforcement**: Displays transverse reinforcement in foundation layout drawings.

. . . . Braces

Setting	Description
Show brace mark	Allows you to display brace marks in foundation layout drawings. The position options for brace marks are: • **Above** • **Below**
Show brace attributes	Allows you to display brace attributes in foundation layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in foundation layout drawings.

. . . . Columns

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in foundation layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in foundation layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:

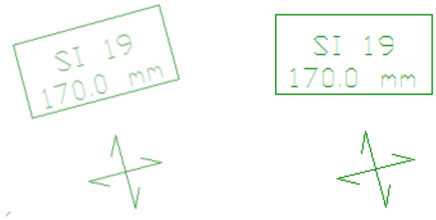
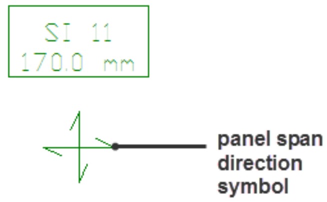
Setting	Description
	• **To the right of the mark** • **Below the mark**
Show column size in parentheses	Allows you to place brackets around the column size in the column label.
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

. . . . Walls

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in foundation layout drawings. The options are: • **Above** • **Inside** • **Below**
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

. . . . Slabs/Mats

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the panel label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the

Setting	Description
	left, and clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 
Show pile location table	Allows you to include a table showing the pile locations in foundation layout drawings.
Show pile reference	Allows you to display the pile references in foundation layout drawings.

. . . . Isolated Foundations

Setting	Description
Show pile type name	Allows you to include the pile type names in the footing attributes displayed for each pile cap in foundation layout drawings.
Show foundation details	Allows you to include pad base or strip base and pile cap details, displaying the designed reinforcement, on the side of the foundation layout drawing.
Show reinforcement quantities table	Allows you to include the reinforcement quantity table for the pad bases and strip bases and pile caps displayed in foundation layout drawings.
Show pad/strip base schedule	Allows you to include the reinforcement schedule for the pad bases or strip bases in foundation layout drawings.
Show pile cap schedule	Allows you to include the reinforcement schedule for the pile caps in foundation layout drawings.

Setting	Description
Show allowable pile capacity	Allows you to include the allowable pile capacity table for the piles under pile caps in foundation layout drawings.
Show overall dimensions	Allows you to include the overall dimensions of the pad bases or strip bases and pile caps in foundation layout drawings.
Show pile spacings	Allows you to include the pile cap pile spacing dimensions in foundation layout drawings.
Include the foundation name	Allows you to include the pad base or strip base or pile cap name in the footing mark for grouped footings in foundation layout drawings.
Labelling Position	Allows you to control where the footing mark and attributes are displayed for isolated foundations. The position options are: • **Above** • **Below**

Isolated Foundation detail options

. . . . Content

Setting	Description
Content subpage	
Grouped Foundations tab	
Show number of foundations in group	Allows you to display the number of bases or pile caps in the group.
Cross-sections tab	
Positions list	Allows you to select how cross-section positions are displayed. You can select to not display cross-section positions at all, display a length-wise section or display cross-sections in both directions.
Pile Labels tab	
Show pile type name	Allows you to include the pile type name in the pile label.
Dimensions tab	
Show overall dimensions	Allows you to include the overall dimensions of the base/pile cap.
Show supported member dimensions	Allows you to include dimensions of the column or wall being supported.

Setting	Description
Show pile spacings	Allows you to include dimensions from center to center of each pile for pile caps.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in isolated foundation detail drawings.

. . . . Style

Setting	Description
General tab	
Show supported member as hatched	Allows you to hatch columns and walls that are supported on the base pile cap in isolated foundation detail drawings.
Underline foundation label	Allows you to underline the foundation label on the detail in isolated foundation detail drawings.
Cross-sections tab	
Section label style list	Allows you to select the label style to be applied to cross-sections in isolated foundation detail drawings.
Restart labels in each foundation	Allows you to restart labeling from the beginning of each section when multiple foundations are included in the same drawing.

See also

Adjust and apply drawing settings (page 968)

Settings set settings

The **Settings Sets** page in the **Settings** dialog box controls the settings sets which are used to create the model/analysis/design settings in new projects, and which can also be used to replace the model/analysis/design settings in existing projects.

Button, command, or option	Description
Available settings sets list	Allows you to select a settings set whose content you can view and modify on the other pages of the **Settings** dialog box.
Add Copy	Creates a new settings set based on the one selected in the **Available settings sets** list. You can then

Button, command, or option	Description
	customize the new settings set on the other pages of the **Settings** dialog box.
>> Active	Makes the settings set that is selected in the **Settings** dialog box the active settings set.
Import...	Allows you to import a settings set from another region. The selected settings set appears in the **Settings** dialog box.
Rename	Allows you to rename the settings set that is selected in the **Available settings sets** list.
Remove	Deletes the settings set that is selected in the **Available settings sets** list.
Open Folder	Opens the folder in which the existing settings sets are located.

See also

Manage settings sets (page 1000)

General settings

The **General** page and its subpages in the **Settings** dialog box allow you to configure the language, captions, autosave settings, confirmation messages, and updates in Tekla Structural Designer.

Button, command, or option	Description
Language subpage	
Select the preferred language list	Allows you to select whether the terms in the interface and the output are in US English or UK English. **NOTE** The language cannot be configured differently in different settings sets.
Appearance page	
Show captions on top	Allows you to select whether you want to show captions on the top or the bottom of the **Project Workspace**.

Button, command, or option	Description
Autosave subpage	
Enabled	Allows you to select whether Tekla Structural Designer creates automatic backups of models. Autosave may be useful, for example, when restarting Tekla Structural Designer after a crash.
Interval	Allows you to determine the interval at which Tekla Structural Designer creates automatic backups of models.
Confirmation subpage	
Confirm on list	Allows you to select the operations that you want to confirm each time.
Update Service subpage	
Enable Update Service	Allows you to select whether you want Tekla Structural Designer to notify you when new product updates or service packs can be installed, so that you can keep Tekla Structural Designer up to date.
Check for updates when starting Tekla Structural Designer	Allows you to select whether Tekla Structural Designer check if new updates are available each time it starts.
Check for non critical updates every [n] days	Allows you to determine the interval at which Tekla Structural Designer checks if new non-critical updates are available.
Yes, I am willing to participate anonymously	Allows you to select whether the Tekla Customer Experience Improvement Program can collect anonymous information on your hardware configuration and your use of Tekla products to improve the customer experience of Tekla products.

Results Viewer settings

The **Results Viewer** page and its subpages in the **Settings** dialog box allow you to configure the appearance of the on-screen results viewer for the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Results Viewer** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
General subpage	
Viewer font --> **Change...**	Allows you to modify the font that is displayed on the left side pane of the results viewer.
Styles subpage	
Report Styles list	Allows you to select the style whose font, color, or other properties you want to modify. The options are: • **Calc Normal**: not used in the current version. • **Table**: controls all the text displayed on the right side pane of the results viewer except for the first row in any table. • **Table Heading**: controls the heading row, or first row, in any table.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.
Line spacing	Allows you to determine the line spacing of the selected style.

Structure default settings

The **Structure Defaults** page in the **Settings** dialog box allows you to configure miscellaneous structure default settings that are applied to the settings set selected in the **Select the settings set to edit** list.

Button, command, or option	Description
Construction Levels --> **Default Type**	Allows you to define the default type for the new constructions levels you create in the **Construction Levels** dialog box.
Grid & Construction Lines --> **Extension length**	Allows you to define the default extension length of grid and construction lines.
Pattern loadcases	Allows you to define whether load patterns are applied to load cases.
Nominal Cover	Allows you to modify the nominal cover of different members. **NOTE** Unlike other settings, changes made to the nominal cover settings are instantly applied to new members.

Section default settings

The **Section Defaults** page in the **Settings** dialog box allows you to specify the default section size for each member type when a new member is create. The default section sizes are applied to the settings set selected in the **Select the settings set to edit** list.

To change the default section size for a member type, click the section size in the **Section** column.

Section order default settings

The **Section Order Defaults** page in the **Settings** dialog box allows you to modify the default section orders for different countries by clicking the desired order in the **Section Order** column.

Solver settings

The **Solver** page in the **Settings** dialog box allows you to specify the solver method that is applied to the settings set selected in the **Select the settings set to edit** list.

Button, command, or option	Description
Global Matrix Storage	Allows you to specify the global matrix storage method. The options are **Compressed Sparse Row** and **Skyline**. **NOTE** We recommend that you select the **Compressed Sparse Row** option because it generally reduces the analysis time, particularly in the case of finite element analysis.

Scene settings

The **Scene** page and its subpages in the **Settings** dialog box allow you to control the color and opacity of each object type in scene views when using the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Scene** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
Graphics subpage	
Driver	**NOTE** The default settings on the **Graphics** subpage are the recommended settings. Do not adjust them unless the Tekla Support Team instructs you to do so.
Antialiasing	
Transparency	
Colors subpage	
Background color list	Allows you to select whether the table below displays and allows you to adjust the colors for a light or dark background.

Button, command, or option	Description
Gradient Background	Unselect this option to apply a uniform white or black background to each window, with no gradient applied.
Reset Colors	Resets all the colors to the default settings.
Color Name	Displays the interface item to which the color applies. **TIP** An arrow is displayed to the left of some of the items, this can be clicked on in order to set the colors for sub-items.
Opacity	Allows you to set the opacity of each color.
Color	Allows you to set the color for each interface item according to your needs.
Fonts subpage	
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the interface item to which the font applies.
Font	Allows you to change the font used for the interface item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	
Contours subpage	
Lower bound [%]	Displays the lower bound of each FE contour in the results view. **TIP** To adjust the size of the lower bound, modify **Size [%]**.
Upper bound [%]	Displays the upper bound of each FE contour in the results view. **TIP** To adjust the size of the upper bound, modify **Size [%]**.

Button, command, or option	Description
Size [%]	Allows you to adjust the size of individual contours according to your needs. Note that the sum of the sizes must equate to 100 %.
Color	Allows you to change the color of a contour.
Split	Divides the contour that is currently selected in the table into two contours that are each half the size of the original contour.
Delete	Deletes the contour that is currently selected in the table.
Reset	Resets the contours to the default configuration that consists of 10 evenly sized contours.
Utilization Ratios subpage	
Minimum value	Allows you to adjust the minimum value of a ratio band. **NOTE** The value of the highest ratio band can be increased above 1.0, if necessary.
Color	Allows you to change the color of a ratio band.
Add	Creates a new ratio band that you can modify to the bottom of the table.
Delete	Deletes the ratio band that is currently selected in the table.
Reset	Resets the ratio bands to the default configuration that consists of 5 evenly sized ratio bands.
Sort	Rearranges the ratio bands in order of the values (from highest to lowest).
View Settings subpage	
Do not display values of storey shear below	Allows you to limit the values of storey shear for a new model. This way, you can easily ignore the small values of storey shear that might otherwise detract you from the more important storey shear values. The storey shear values that are less than

Button, command, or option	Description
	the limiting value are not displayed in the results view.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2 x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Report settings

The **Report** page and its subpages in the **Settings** dialog box allow you to modify the appearance of reports.

NOTE Unlike other settings, changes made to the active settings set on the **Report** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
Styles subpage	
Report Styles list	Allows you to select a style used in reports whose font, color, or other properties you want to modify.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Background** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.
Line spacing	Allows you to determine the line spacing of the selected style.
Page Options subpage	

Button, command, or option	Description
Page margins	Allows you to adjust the margins on report pages.
Margin frame --> **Draw page margin frame**	Allows you to select whether Tekla Structural Designer creates a margin frame to separate the report text from the margins.
Margin frame --> **Color** --> **Change...**	Allows you to change the color of the margin frame.
First page number	Allows you to determine the number from which the page numbers start.
Page number prefix	Allows you to specify a prefix to the page number.
Table Options subpage	
Border Style	Allows you to specify the table border style. You can select to have no table borders, a single line border, or a double line border.
Border Properties --> **Color** --> **Change...**	Allows you to change the table border color.
Border Properties --> **Width**	Allows you to adjust the table border width.
Table width --> **[] percent of page width**	Allows you to specify the width of all tables included in reports.
Use landscape orientation for wide tables	When enabled (default) the report page orientation will automatically change to landscape mode when the table header will not fit in portrait orientation. **NOTE** You can reduce the need for landscape orientation and number of pages by; reducing page margins (default values may be quite wide); increasing the allowable table width from 90 to 95% of the available width
Document Options subpage	
Show document header	Allows you to include the header at the top of each report page.
Show border in document header	Draws a border around each field cell in the header.

Button, command, or option	Description
Show document footer	Allows you to include the header at the top of each report page.
Show border in document footer	Draws a border around each field cell in the footer.
Show document field description in line with the value	Allows you to display the field descriptions in line with the field values in each cell. If you do not select the **Show document field description in line with the value** option, the field values are displayed on a new line.
Underline document field cell	Underlines the cell value of each field.
Image width --> [] percent of page width	Allows you to specify the width of all images included in reports.
Paragraphs --> Spacing	Allows you to specify the paragraph spacing in reports.
Start each item on new page	Allows you to start each report chapter on a new page.
Start each member report on new page	Allows you to start each member report on a new page.
Picture Fonts subpage	
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the report item to which the font applies.
Font	Allows you to change the font used for a report item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	

See also

Adjust and apply report settings (page 953)

Performance settings

The **Performance** page in the **Settings** dialog box allow you to optimise analysis and design performance according to your preference and your PC's capabilities.

Button, command, or option	Description
General	To optimise performance choose one of the following: • **Conserve memory** (default) • **Favour speed** **NOTE** We recommend the 'Favour speed' option is used only on higher specification PC's with multi-core processors and the recommended amount of RAM.
Design	Options are provided to allow you to take advantage of multi-core processing in the design and chase-down phases. • **Use multi-core processors for design** • **Run chase-downs concurrently**

15.5 Dialogs

This section covers some of the important dialogs in Tekla Structural Designer and their different options.

Click the links below to find out more:

- Analysis Settings dialog (page 2398)
- Connection Resistance dialog (page 2399)
- Construction Levels dialog (page 2402)
- Design Settings dialog (page 2404)
- Drawing Settings dialog (page 2405)
- Edit Reinforcement dialog (page 2405)
- Interactive Beam Design dialog (page 1311)
- Interactive Column Design dialog (page 1327)
- Interactive Wall Design dialog (page 1345)
- Load Event Sequences dialog (page 2410)
- Loading dialog (page 531)
- Materials dialog (page 2418)

Analysis Settings dialog

Summary

The **Analysis Settings** dialog and its subpages allow you to adjust the settings applied to the different analyses.

Location

On the **Analyze** tab, click **Settings**.

Content

Button, command or option	Description
1st Order Non-Linear	See 1st order non-linear settings (page 2278)
2nd Order Non-Linear	See 2nd order non-linear settings (page 2279)
1st Order Modal	See 1st order modal settings (page 2281)
2nd Order Buckling	See 2nd order buckling settings (page 2284)
1st Order Seismic	See 1st order seismic settings (page 2285)
Iterative Cracked Section Analysis	See Iterative cracked section analysis settings (page 2288)
Modification Factors	See Modification factors (page 2290)
Meshing	See Meshing settings (page 2291)

Button, command or option	Description	
Composite Steel Beams	See Composite steel beams settings (page 2291)	
OK button	Apply the changes to the current project.	
Cancel button	Cancel the changes.	
Save button	Save the changes back to the active settings set for future use.	
Load button	Revert to the model settings specified in the active settings set.	

Connection Resistance dialog

The **Connection Resistance dialog** displays the pre-defined connection resistances for steel beams to Eurocode and US head codes. It can also be used to add or edit user-defined connection resistances.

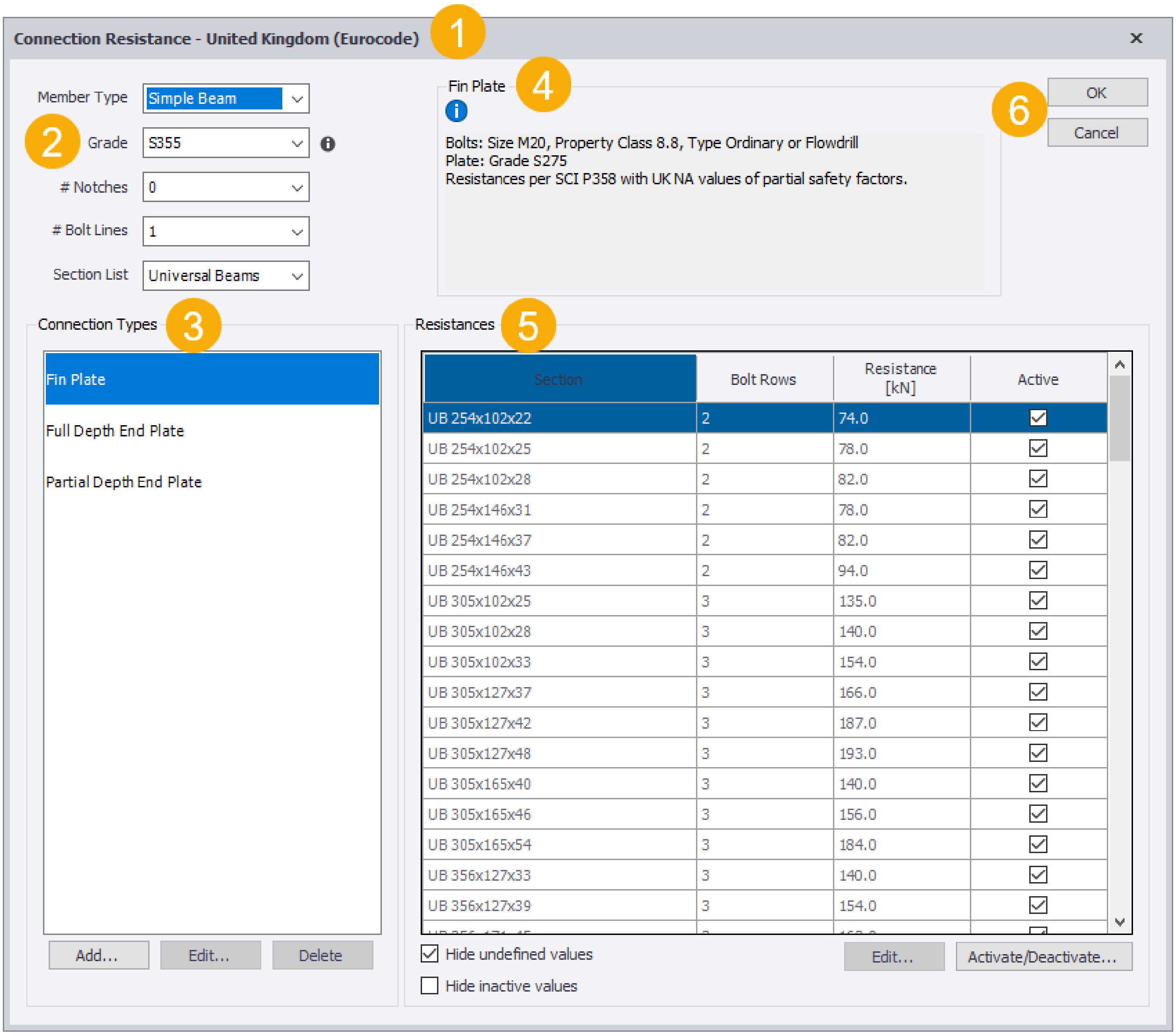
To open the dialog:

1. Click the **Home** tab.
2. Click  **Materials**.

   The **Materials** dialog is displayed.
3. On the **Sections** page, check that the required head code and material are displayed.

   NOTE Both steel and cold formed materials can have resistances defined, unless the head code is US, in which case resistances can be defined for steel only.
4. Click **Connection Resistance**

The dialog content is described below.



1. Title Bar

The title bar displays:

- The currently selected head code.
- For India this also displays LS or WS, and for US displays LRFD or ASD, according to the setting of the currently open model.

2. Filters

The filter settings determine what is displayed elsewhere in the viewer.

The Member Type filter has options for:

- Simple Beam
- Brace

3. Connection Types

Pre-defined and user-defined Connection Types are listed here according to the Filter settings.

- Click the **Add...** button to add a user-defined Connection Type (page 1010).
- Click the **Edit...** button to edit a user-defined Connection Type.
- Click the **Delete** button to delete a user-defined Connection Type.

NOTE The **Edit...** and **Delete** buttons are only active when a *user-defined* Type is selected.

4. Info box

Displays information related to the currently selected Connection Type. For user-defined Connection Types the information displayed here can be added or edited via the Add... and Edit... buttons in the Connection Types area of the viewer.

5. Resistances

- Resistance values are listed in the window according to the Filter settings and the currently selected Connection Type. The resistance values (for user-defined Connection Types) can be edited directly.
- The 'Active' setting means that, when ticked, the particular resistance will be considered in the check. The Active status can be edited directly in this window for individual resistances but is more quickly edited using Activate/ Deactivate. All pre-defined resistances start out Active.
- If 'Hide undefined values' is un-ticked those resistances which have zero value currently assigned will be displayed along with the other sections. This box is ticked by default when opening the viewer.
- If 'Hide inactive values' is ticked those resistances which have 'Active' un-ticked will be hidden in this window. This box is un-ticked by default when opening the viewer.
- Use the Edit... button to edit the bolt row count or resistance or Active status of a user-defined Connection Type. Note that multiples of the same section size can be added through the Edit Resistances dialog, so that the same size with different bolt row counts and resistances can be defined.

  Click here for an example (page 1012) showing the use of Edit... as described above.

- Use the Activate/Deactivate... button to change the Active setting for multiple sections. Set the filters in the Activate/Deactivate dialog, then set either Activate or Deactivate, and then Apply if you want to change further Active settings or Apply & Close to finish.

  Click here for a Eurocode example (page 810) or here for a US example (page 812) showing the use of Activate/Deactivate as described above.

6. OK and Cancel

Button	Description
OK	Commit all changes made in the viewer to the database.
Cancel	Closes the dialog box without saving changes.

See also:

Check simple connection resistance (page 809)

Construction Levels dialog

The **Construction Levels** dialog allows you to define the levels required in order to construct your model.

Choose from:

- On the **Model** tab, click **Construction Levels**.
- In the **Structure** tree, double-click **Levels**.

Content

Button, command or option	Description
Options	
Ref	Allows you to assign a unique reference to a construction level.
Name	Allows you to assign an optional name to further assist identification. For example, "First floor" or "Mezzanine".
Type	Allows you to select the Type from the drop down menu: • **T.O.S.** = Top of steel • **S.S.L.** = Structural slab level

Button, command or option	Description
	• **T.O.F.** = Top of foundation **WARNING** Slabs are modeled above the level when they are set to T.O.S or T.O.F but below the level when they are set to S.S.L
Level	Allows you to assign the height of the construction level above the base level.
Spacing	Allows you to assign the distance of this construction level to the one immediately below.
Source	Allows you to assign the level as Unique or the same as another level. Unique levels can be edited independently, whereas edits to indentical levels are applied at both levels simultaneously.
Slab Th.	Allows you to set the default thickness for any slab created on the level.
Floor	Allows you to assign the construction level as a major level in the building. Floor levels determine the number of sub models that are created for the chasedown analysis. Floor levels are also used to determine items such as your inter story height. There will certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs • K bracing. You require a construction level for the intermediate bracing connection points • Steps in the building floor levels When you define a level that is clearly not a floor, then you should not check the option. If a floor exists only at some locations in the level (e.g. in a building with stepped floor levels), then you should check the option where applicable.
Buttons	
OK	Allows you to save any changes made.
Cancel	Allows you to discard any changes made.
Insert Above	Allows you to insert a new construction level above the currently highlighted level with the same spacing.
Insert Below	Allows you to insert a new construction level below the currently highlighted level with the same spacing.

Button, command or option	Description
New on Top	Allows you to insert multiple construction levels above the uppermost level with variable spacings **NOTE** 3*5 inserts three levels, each at 5m spacing 3,4,5 inserts three levels at spacings of 3,4 and 5m respectively.
New at the Bottom	Allows you to insert multiple construction levels below the lowest level with variable spacings
Delete	Allows you to delete the selected level.

Design Settings dialog

The **Design Settings** dialog applies design settings to the current project.

Location

On the **Design** tab, click **Settings**.

Content

Button, command or option	Description
Analysis	See Analysis (page 2294)
General	See General (page 2294)
Concrete	See Concrete > Cast-in-place (page 2296) and Concrete > Precast
Composite Beams	See Composite Beams (page 2322)
Design Forces	See Design Forces (page 2323)
Design Groups	See Design Groups (page 2339)
Autodesign	See Autodesign (page 2340)
Design Warnings (AISC/ASC only)	See Design Warnings (page 2341)
Steel Joists	See Steel Joists (page 2345)
Sway & Drift Checks	See Sway and Drift Checks (page 2346)

Button, command or option	Description	
Fire check	See Fire check (page 2347)	
OK button	Apply the changes to the current project.	
Cancel button	Cancel the changes.	
Save button	Save the changes back to the active settings set for future use.	
Load button	Revert to the model settings specified in the active settings set.	

Drawing Settings dialog

Location

On the **Draw** tab, click **Settings**.

Content

Button, command or option	Description
General > Default export directory	Specifies the default folder for creation of dxf files
Layer Configuration	See Layer configurations (page 2353)
Layer Styles	See Layer styles (page 2354)
Options > Planar Drawings	See Planar drawing options (page 2356)
Options > Member Details	See Member detail options (page 2364)
Options > Member Schedules	See Member schedule options (page 2372)
Options > Slabs and Mats	See Slab and mat layout options (page 2374)
Options > Foundations	See Foundation layout options (page 2380)
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Edit Reinforcement dialog

Summary

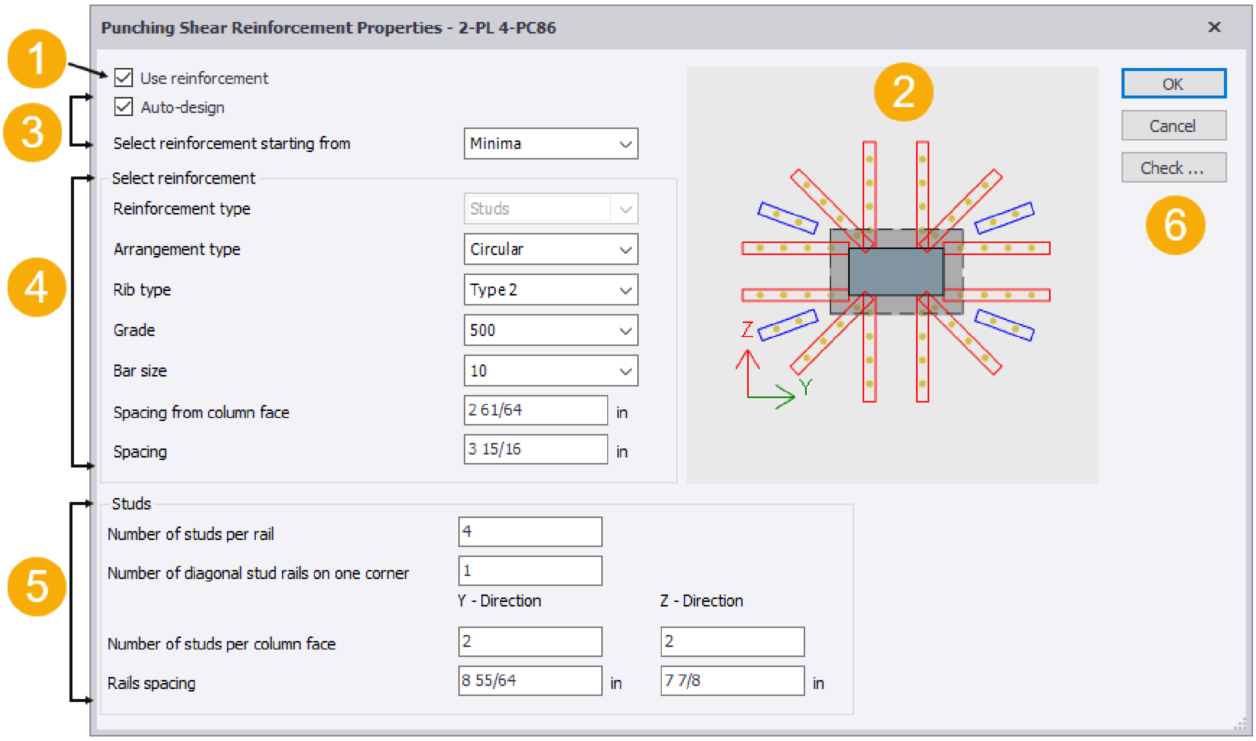
The **Edit Reinforcement** dialog box allows you to modify the reinforcement used in a punching shear check. The graphic in the dialog box previews the reinforcement and updates to match the changes that you make. The graphic also indicates other items that are specific to the check location.

Location

To display the dialog:

1. Right click on an existing punching check item.
2. In the context menu, select **Edit Reinforcement**.

Content



1. Use reinforcement

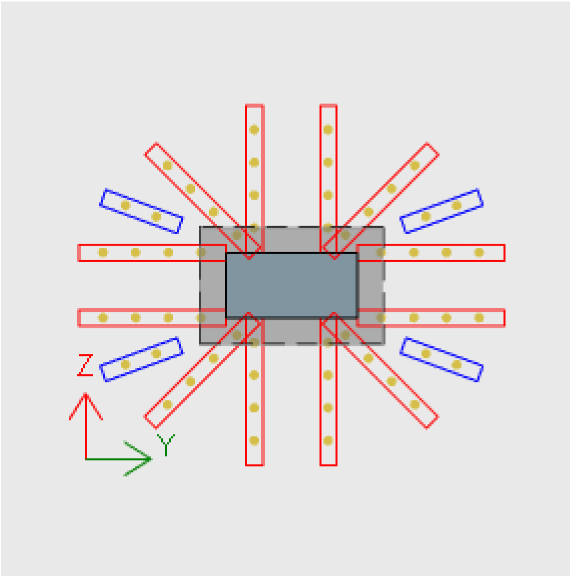
Selecting the **Use reinforcement** option allows you to apply a default punching reinforcement arrangement that can either be checked or used as the starting reinforcement for an auto design.

2. Preview graphic

The components of the preview graphic are as follows:

Stud rail reinforcement

Stud rail reinforcement is only shown if you have selected the **Use reinforcement** option.



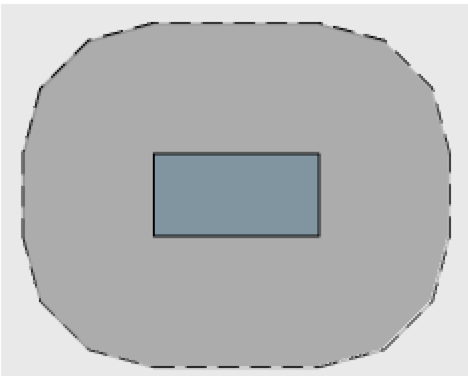
The red design rails signify rails considered in design.

The blue detailing rails are only considered for detailing purposes.

Perimeters

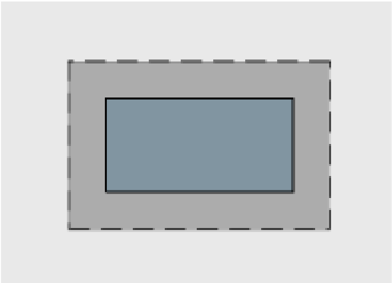
Perimeters are displayed as dashed lines around the column section and will have different shapes and positioning depending upon the head code being worked to.

Control Perimeter (EC)



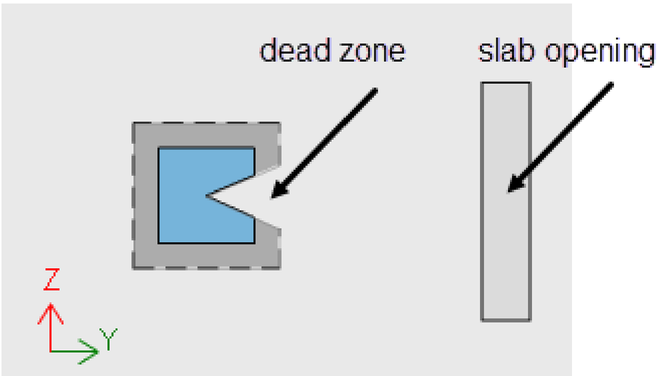
The control perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Critical Perimeter (ACI)



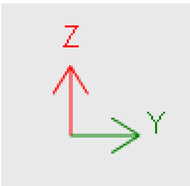
The critical perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Slab opening dead zones



Slab opening dead zones are displayed as interruptions in perimeters and reinforcement.

Punching shear check local axis



Using the YZ system orients the X axis upwards following the right-hand rule. This is also the local axis system for the column elements, so it is easier to relate.

Additional Perimeters

If reinforcement is found to be required, additional perimeters will be displayed representing positions beyond the critical/control perimeter where the reinforcement requirement is checked.

3. Auto-design

If you select the **Auto-design** option, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. You can then select the starting point in the **Select reinforcement starting from** list.

The **Select reinforcement starting from** list allows you to select the staring point for auto-design procedures. The options are:

- **Minima**: removes the current arrangement and starts the reinforcement with the minimum allowed bar size.
- **Current**: auto design starts from the current bar arrangement. The **Current** option is only available if you have selected **Use reinforcement** in the **Properties** window.

4. Select reinforcement parameters

Reinforcement type

In the current release, only stud reinforcement is available.

Arrangement type

Allows you to define whether the reinforcement arrangement is orthogonal or circular.

Rib type

Allows you to specify the reinforcement rib type.

Grade

The reinforcement grades that are available here are set in the **Materials** dialog box.

Bar size

The reinforcement bar sizes that are available here are in the **Materials** dialog box.

Spacing from column face

Defines the spacing of the first bar in each rail from the column face.

NOTE The option is only available if the **Use reinforcement** option has been selected.

Spacing

Defines the spacing between bars along each rail.

NOTE The option is only available if the **Use reinforcement** option has been selected.

5. Studs parameters

Number of studs per rail

Allows you to define the number of studs on each rail.

Number of diagonal stud rails on one corner

Allows you to define the number of stud rails adjacent to each corner of the column.

NOTE The option is only displayed when **Arrangement type** is set to **Circular**.

Number of studs per column face

Allows you to define the number of stud rails adjacent to the column face in the local y and z directions.

Rails spacing

Allows you to define the spacing between rails in the local y and z directions.

6. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.

See also

Create punching shear checks (page 801)

Design and check punching shear (page 803)

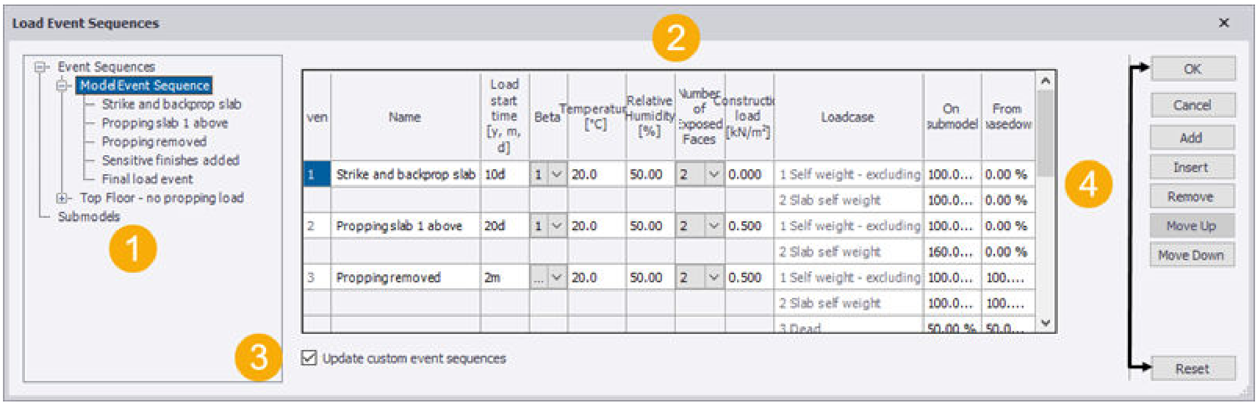
Load Event Sequences dialog

The **Load Event Sequences dialog** is used to define all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

To display the dialog:

- On the **Slab Deflection** toolbar, click **Event Sequences**

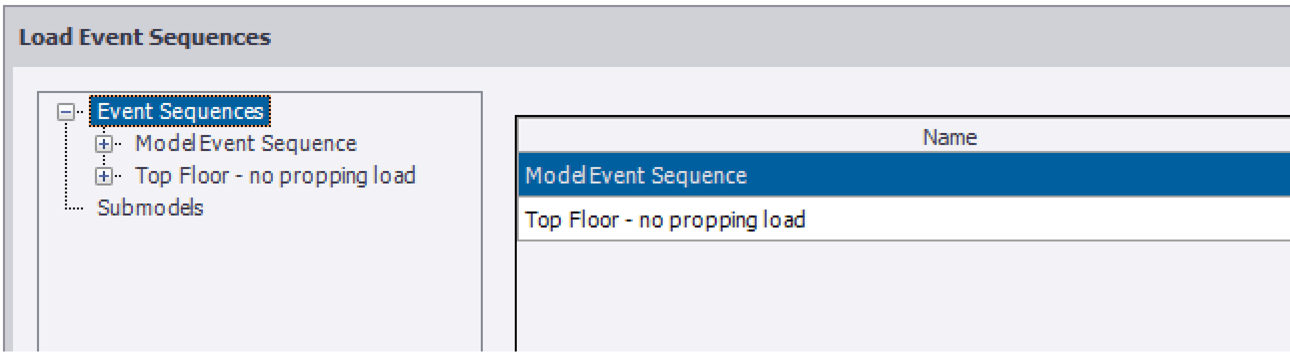
The dialog content is described below.



1. Event sequences and submodels pane

Event Sequences

Select **Event Sequences** to show a summary of the model event sequence and any custom event sequences that have been defined.

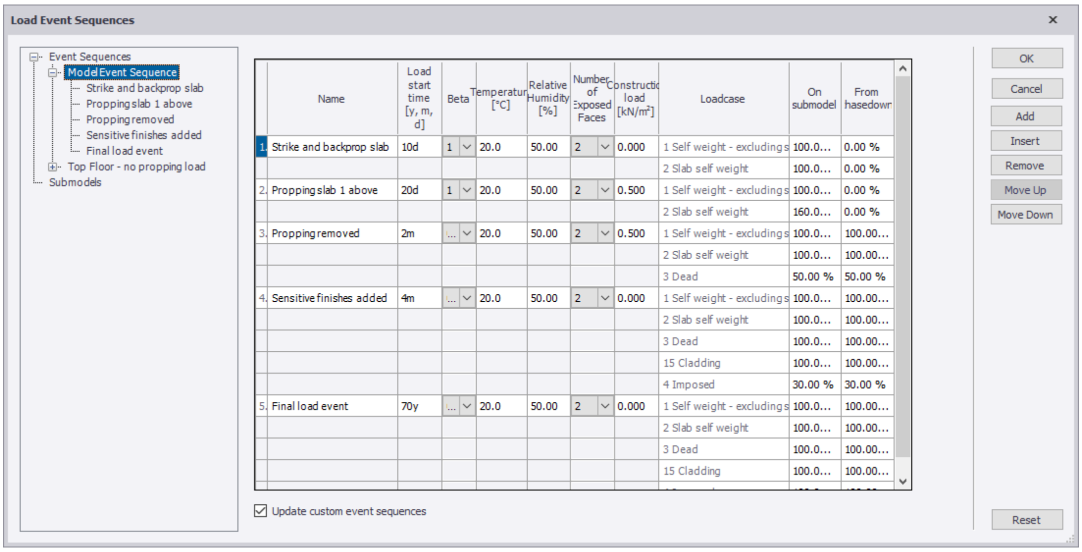


When this page is active you can:

- **Add** a Custom Load Event Sequence.
- **Remove** a selected Custom Load Event Sequence.
- Review which Event Sequences are used in submodels

Model Event Sequence

Select **Model Event Sequence** to display the various event sequence parameters for editing.

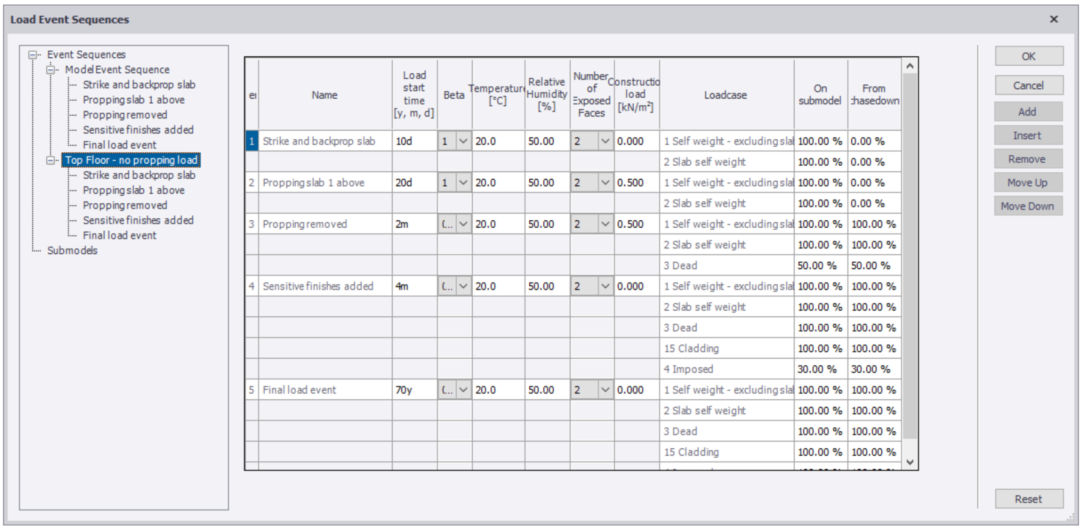


When this page is active you can:

- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the Model Event Sequence to the default Event Sequence in the active Settings Set

Custom Event Sequences

Pages for custom event sequences are only displayed if they have been added from the **Event Sequences** page.



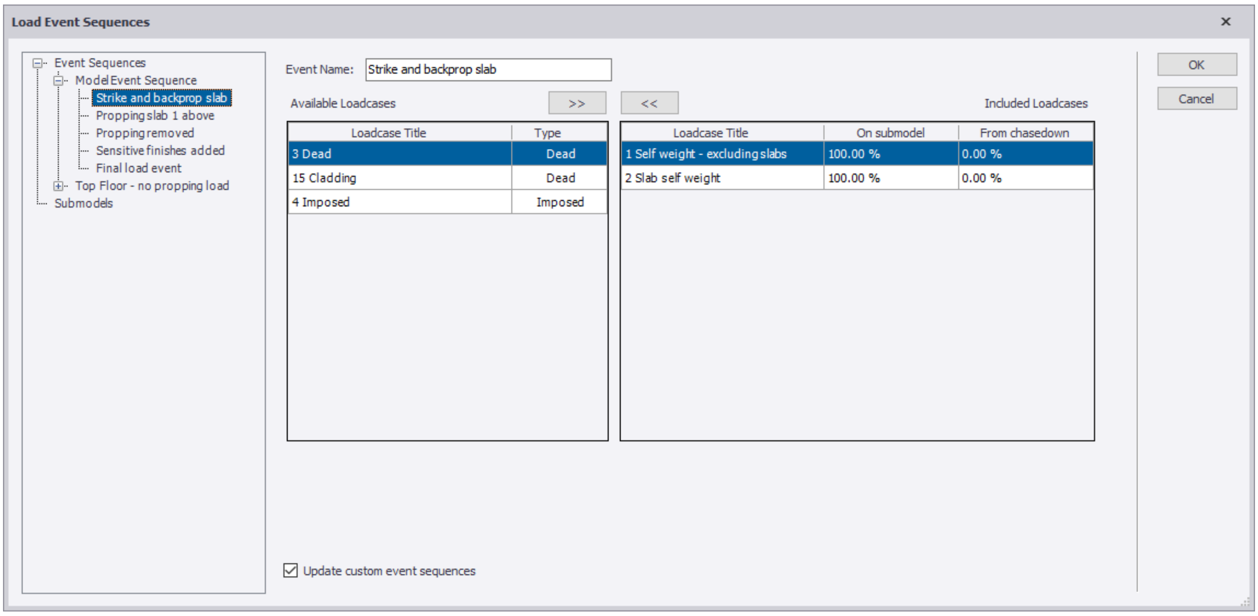
When a custom event sequence page is active you can:

- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the custom event sequence to be the same as the Model Event Sequence

NOTE Model Event Sequences and Custom Event Sequences do not behave differently - they are both just Event Sequences.

Load Event sub-pages

Load Event sub-pages are displayed under the Model Event Sequence, and any custom Event Sequences pages. Each Load Event has a separate sub-page which is used to define the loadcases included in the Load Event.



The [>>] and [<<] buttons are used to add or remove loadcases from the Included Loadcases list.

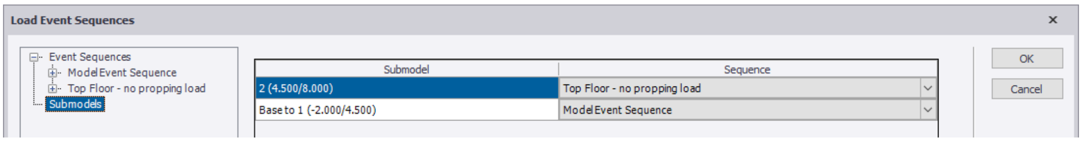
On submodel specifies the percentage of each included loadcase on the submodel.

From chasedown specifies the percentage of each included loadcase from the chasedown.

NOTE For propping loadcases - “From Chasedown” should be 0%.

Submodels

Select **Submodels** to assign a custom event sequence to a submodel if required.



- With this row selected you can edit the section size and grade for all stacks simultaneously.

2a. Event sequence parameters table (Eurocode)

:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructior load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place
- **Beta** - a coefficient to take account of the influence of the duration of loading. (See: EC2 Clause 7.4.3) This coefficient is used to account for the fact that tension stiffening effects reduce over time.

  Tension stiffening is the phenomenon that when the concrete is not fully cracked, there is still concrete in the tension zone that transfers some tensile forces, so the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness). Since this effect reduces over time, to model loads with a longer duration, you want to model a lower stiffness, which means setting beta = 0.5.

  In Tekla Structural Designer, Beta defaults to 1.0 where the start event time is ≤ 30 days and 0.5 if >30 days, but may be changed for any event.

  Tension stiffening reduces over time because of increased cracking and bond failure between the steel reinforcement and concrete.

These phenomena cannot be reversed, so reduced tension stiffening cannot be recovered.

For this reason, if Beta is set to 0.5, and then in a later event set to 1.0, a warning flag is shown.

- **Temperature** - Used in the calculation of the Composite Modulus. The effective age of concrete is adjusted to account for the defined temperature.
- **Relative Humidity** - Used in the calculation of creep.
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the load cases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

2b. Event sequence parameters table (ACI)

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Start event	10d	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
2	Finishes added	3m	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	33.00 %	33.00 %
3	Final load event	5y	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	100.00 %	100.00 %
4	Live load only	5y	2.350	0.800	0	0.0	3 Live	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place
- **Ultimate Creep Coefficient**

- **Aging Coefficient**
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the load cases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

3. Update custom event sequences

If you select the **Update custom event sequences** option, changes made to the Event in the Model Event Sequence are also replicated in custom event sequences.

4. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Add	Depends on the page selected in the Event sequences and submodels pane.
Insert	Depends on the page selected in the Event sequences and submodels pane.
Remove	Depends on the page selected in the Event sequences and submodels pane.
Move Up	Depends on the page selected in the Event sequences and submodels pane.
Move Down	Depends on the page selected in the Event sequences and submodels pane.
Reset	Depends on the page selected in the Event sequences and submodels pane.

Materials dialog

The **Materials** dialog box allows you to view material database properties and manage material databases. The databases contain an extensive range of sections, materials, reinforcement, decking and connectors for each head code and country.

NOTE Although the Materials dialog can be used to view the properties for any of the head codes, only the properties for the currently selected head code in **Model Settings** can be applied to the model.

To display the dialog:

1. Click the **Home** tab.
2. Click  **Materials**.

The dialog content is described below.

Sections settings

The **Sections** page of the **Materials** dialog box allows you to view the available steel, cold formed, cold rolled, and timber sections for each head code. You can also add new user-defined sections.

Field or button	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Unit System	Allows you to select which units are used for sections.
Head Code	Allows you to select the head code whose material database properties you want to view.
Material	Allows you to select the material whose sections you want to view.
Manage Sections	Allows you to add, modify, and delete user-defined sections.
Manage Section Orders	Allows you to adjust section orders according to your needs.
Connection Resistance	Allows you to add connection resistance information for a section.
Steel Joists	Allows you to add steel joist information for a section.
Plate Dimensions	Allows you to adjust plate widths and thicknesses.

Material settings

The **Material** page of the **Materials** dialog box allows you to view properties of each grade of each material for each head code. You can also add new user-defined sections.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Material Type	Allows you to select the material type whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined material grades to the **Available Grades** list.
View...	Allows you to view the properties of pre-defined material grades.
Delete	Allows you to permanently delete user-defined material grades.
>> Default	Allows you to make the currently highlighted material grade a default material grade, so that it appears in the **Default Grade** field.

Reinforcement settings

The **Reinforcement** page of the **Materials** dialog box allows you to view the properties of reinforcement classes and bar sizes.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Country	Allows you to select the country whose material database properties you want to view.

Button or field	Description
Type	Allows you to select the type whose material database properties you want to view.
Rib Type	Allows you to select the rib type whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined reinforcement classes or bar sizes to the appropriate list.
View...	Allows you to view the properties of pre-defined reinforcement classes or bar sizes.
Delete	Allows you to permanently delete user-defined reinforcement classes or bar sizes.
>> Default	Allows you to make the currently highlighted reinforcement class a default class, so that it appears in the **Default Class** field.

Decking settings

The **Decking** page of the **Materials** dialog box contains different sub pages: **Metal Decking** and **Precast Concrete Decking**. The **Metal Decking** page allows you to view the properties of different profiles and gauges, and the **Precast Concrete Decking** page allows you to view the properties of different precast concrete planks and depths. The sub pages contain the same commands.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined profiles, gauges, planks, or depths to the appropriate list.
View...	Allows you to view the properties of pre-defined profiles, gauges, planks, or depths.

Button or field	Description
Delete	Allows you to permanently delete user-defined profiles, gauges, planks, or depths.
>> Default	Allows you to make the currently highlighted profile, gauge, plank, or depth a default option, so that it appears in the appropriate **Default** field.

Shear Connectors settings

The **Shear Connectors** page of the **Materials** dialog box allows you to view the properties of different connectors.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined sources or connectors to the appropriate list.
View...	Allows you to view the properties of pre-defined sources or connectors.
Delete	Allows you to permanently delete user-defined sources or connectors.
>> Def. Metal	Allows you to make the currently highlighted option the default option for metal, so that it appears in the **Default for Metal** field.
>> Def. Concrete	Allows you to make the currently highlighted option the default option for concrete, so that it appears in the **Default for Concrete** field.

Model settings

The **Model** page of the **Materials** dialog box allows you to both update the material databases with new properties from the model and update material properties in the model with new values from the material databases.

Button, field, or column	Description
In Database	Uses **?** to display if there are inconsistencies between the material data in the model and the material databases.
Add to Database	Allows you to update the material databases with values from the model.
Update from Database	Allows you to update the material properties in the model with values from the material databases.
Show only objects not saved in the database	Hides the model properties that are consistent with the material databases.

Model Settings dialog

The majority of settings for the current project are accessed from the **Model Settings** dialog.

Location

On the **Home** tab, click **Model Settings**.

Content

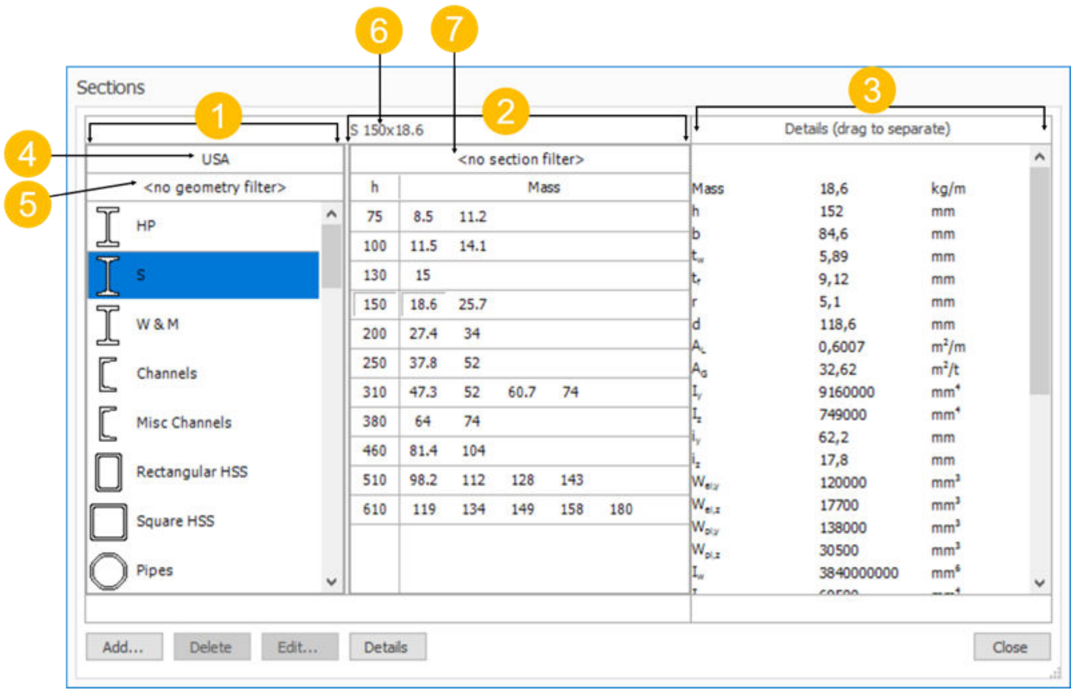
Button, Command or Option	Description
Design Codes	See Design code settings (page 2264)
Units	See Unit settings (page 2265)
References	See Object reference settings (page 2266)
Loading	See Loading settings (page 2268)
Grouping	See Grouping model settings (page 2269)
Material List	See Material list settings (page 2269)
Beam Lines	See Beam line settings (page 2270)
Analysis Model	See Analysis Model settings (page 2271)
Validation	See Validation settings (page 2273)

Button, Command or Option	Description
Load reductions	See Load reduction settings (page 2273)
EHF	See EHF settings (page 2274)
User Defined Attributes	See User-defined attribute settings (page 2274)
Graphics View Settings	See Graphics view settings (page 2275)
Structural BIM	See Structural BIM settings (page 2276)
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Sections dialog

The **Sections** dialog box is used to select sections to apply to the model and also to manage the sections stored in the material database.

The dialog content is described below.



1. **Page** pane: views the different section geometries.

2. **Item** pane: views the different sizes of the selected section when you select a section geometry **Page** pane. Any user-defined sections are marked with *.
3. **Details** pane: views the details of the selected section when you click the **Details** button.
4. **Country** droplist: allows you to display the sections of each country.
5. **Geometry filter**: allows you to filter sections according to their geometry.
6. Selected section: displays the details of the currently selected section.
7. **Section filter**: allows you to filter sections according to their details.

You can also use the following buttons to manage the sections:

- **Add...**: allows you to add a user-defined section to the material database.
- **Delete**: allows you to delete a user-defined section.
- **Edit...**: allows you to modify the properties of a user-defined section.
- **Details**: opens the **Details** pane and displays the properties of the currently selected section.

Settings dialog

The **Settings** dialog allows you to manage defaults, collectively referred to as a settings set, that are used in future projects. In addition, you can manage general and display settings that are applied instantly to the current work session.

Location

On the **Home** tab, click **Settings**.

Content

List, Page, or Button	Description
Select the settings set to edit list, **Available settings sets** list	Allows you to select the settings set to view or modify.
Settings Sets	See Settings set settings (page 2386)
General	See General settings (page 2387)
Results Viewer	See Results Viewer settings (page 2388)
Report	See Report settings (page 2394)
Units	See Unit settings (page 2265)

List, Page, or Button	Description
Design Codes	See Design code settings (page 2264)
Design Settings	See Design Settings (page 2293)
Analysis Settings	See Analysis Settings (page 2278)
Loading	See Loading settings (page 2268)
Structure Defaults	See Structure default settings (page 2389)
Section Order Defaults	See Section order default settings (page 2390)
References	See Object reference settings (page 2266)
Drawings	See Drawing settings (page 2352)
Material List	See Material list settings (page 2269)
Beam Lines	See Beam line settings (page 2270)
Analysis Model	See Analysis Model settings (page 2271)
Solver	See Solver settings (page 2390)
User Defined Attributes	See User-defined attribute settings (page 2274)
Scene	See Scene settings (page 2391)
Structural BIM	See Structural BIM settings (page 2276)
Slab Deflection	See Slab deflection settings (page 2348)
Performance	See Performance settings (page 2396)
OK button	Apply the changes.
Cancel button	Cancel the changes.

Slab Deflection Check Catalogue

The **Slab Deflection Check Catalogue** is used to define the deflection checks that are applied to check lines.

Location

On the **Slab Deflection** tab, click **Deflection Checks**.

Content

Each deflection check has a unique name and can either be defined as a total or instantaneous check for a specific event, or a differential check between load events. A deflection limit is set and you can specify if the check is to be applied to each new check line as it is defined.

Field or button	Description
Name	Allows you to modify the check name.

Field or button	Description
Type	Allows you to select the deflection check type, see Display slab deflection analysis results (page 937) .
Start Event	Allows you to select the start event for a differential check.
Event	Allows you to select the event to which the check applies.
Deflection Limit	Allows you to specify the deflection check limit.
Use in new Check Lines	Check the box only to be automatically apply the check to new check lines as they are created.
Add	Allows you to add a new row in the table for defining a new check.
Remove	Removes the selected check from the table.

Snow wizard (Eurocode)

Summary

You can use the **Snow wizard...** to define sufficient site information to calculate the snow loadcases.

Location

1. Ensure the head code is set as Eurocode with the required National Annex.
2. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

Plain Eurocode, Ireland and Sweden National Annex

Button, command or option	Description
Page 1 - Basic data (Plain Eurocode, Ireland and Sweden National Annex)	The following basic data is required:
Exposure Coefficient	User defined value. (Default 1.0)
Thermal Coefficient	User defined value. (Default 1.0)

Button, command or option	Description
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)
Snow Density	Define the density of snow. (Default 2.0 kN/m^2)
Snow Load	User defined value. (No default value)
Next	Click to proceed to the next page of the **Snow wizard...**
Page 2 - Snow load cases (Plain Eurocode, Ireland and Sweden National Annex)	On this page you specify the design situations to be considered.
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.

Button, command or option	Description	
Loadcase Type (Plain Eurocode and Ireland National Annex)	Each loadcase has a loadcase type which can be: • Snow • Snow > 1000m • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1	
Loadcase Type (Sweden National Annex)	Each loadcase has a loadcase type which can be: • Snow • Snow S_k > 2 kN/m^2 • Snow S_k > 3 kN/m^2 • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1.	
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2	
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.	

UK National Annex

Button, command or option	Description	
Page 1 - Basic data (UK National Annex)	The following basic data is required:	

Button, command or option	Description	
Exposure Coefficient	User defined value. (Default 1.0)	
Thermal Coefficient	User defined value. (Default 1.0)	
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)	
Snow Density	Define the density of snow. (Default 2.0 kN/m^2)	
Snow Load		
Zone Number, Z	Zone Number. (Default 2.0 kN/m^2)	
Altitude, A	Altitude. (Default 1.0)	
Characteristic Ground Snow Load, s_k	Characteristic ground snow load calculated as $s_k = (0.15 + (0.1 * Z + 0.05)) + ((A - 100)/525)$	
Next	Click to proceed to the next page of the **Snow wizard...**	
Page 2 - Snow load cases (UK National Annex)	On this page you specify the design situations to be considered.	
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.	
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically	

Button, command or option	Description
	Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.
Loadcase Type	Each loadcase has a loadcase type which can be: • Snow • Snow > 1000m • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1.
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.

Finland National Annex

Button, command or option	Description
Page 1 - Basic data (Finland National Annex)	The following basic data is required:

Button, command or option	Description	
Topography	Options are: • Windswept • Normal (Default) • Sheltered	
Exposure Coefficient	Value determined from Topography.	
Thermal Coefficient	User defined value. (Default 1.0)	
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)	
Snow Weight Density	Define the density of snow. (Default 2.0 kN/m^2)	
Characteristic Ground Snow Load	User defined value. (No default value)	
Next	Click to proceed to the next page of the **Snow wizard...**	
Page 2 - Snow load cases (Finland National Annex)	On this page you specify the design situations to be considered.	
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.	
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically	

Button, command or option	Description	
	Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.	
Loadcase Type	Each loadcase has a loadcase type which can be: • Snow • Snow, S_k > 2.75 kN/m^2 • Snow Drift • Ice Refer to EC 1992-1-3 Annex A, Table A1.	
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2	
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.	

Norway National Annex

Button, command or option	Description	
Page 1 - Basic data (Norway	The following basic data is required:	

Button, command or option	Description
National Annex)	
Topography	Options are: • Windswept • Normal (Default) • Sheltered
Exposure Coefficient	Value determined from Topography.
Thermal Coefficient	User defined value. (Default 1.0)
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)
Snow Weight Density	Define the density of snow. (Default 2.0 kN/m^2)
Altitude, H	User defined value.
Basic Reference Altitude	User defined value.
Basic Snow Load	User defined value.
????	User defined value.
????	User defined value.
Characteristic Ground Snow Load	Automatically determined from the above values.
Next	Click to proceed to the next page of the **Snow wizard...**
Page 2 - Snow load cases (Norway National Annex)	On this page you specify the design situations to be considered.
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.
Apply	Options are: • Checked If checked the loadcase will be created.

<table>
<tr><th>Button, command or option</th><th>Description</th><th></th></tr>
<tr><td></td><td>Unchecked
If unchecked the loadcase will not be created.
NOTE Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3)
Loads in these cases are generated automatically
Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4)
You must create the loads in these cases manually
NOTE The EC/NA recommends which loadcases to apply. </td><td></td></tr>
<tr><td>Loadcase Type</td><td>Each loadcase has a loadcase type which can be:
Snow
Snow > 1000m
Snow Drift
Refer to EC 1992-1-3 Annex A, Table A1.</td><td></td></tr>
<tr><td>Number of wind directions to be considered for drifted snow</td><td>You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name).
For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created:
Snow Load - Case A - 2) Drifted 1
Snow Load - Case A - 2) Drifted 2</td><td></td></tr>
<tr><td>Finish</td><td>Completes the Snow wizard... and sets up the loadcases based upon the input.</td><td></td></tr>
</table>

Snow wizard (ASCE7)

Summary

You can use the **Snow wizard...** to define sufficient site information to calculate the snow loadcases.

Location

1. Ensure the snow loading code is set as ASCE7.
2. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

Content

<table>
<tr><th>Button, command or option</th><th>Description</th><th></th></tr>
<tr><td>Page 1 - Basic data (ASCE7 Snow wizard)</td><td>The following basic data is required:</td><td></td></tr>
<tr><td>Ground snow load</td><td>(No default)</td><td></td></tr>
<tr><td>Terrain Category</td><td>Options are:
B (urban, suburban, wooded) - Default
C (open terrain scattered obstructions),
D (flat unobstructed areas), Above treeline, Alaska no trees</td><td></td></tr>
<tr><td>Exposure</td><td>Options are:
Fully exposed
Partially exposed</td><td></td></tr>
<tr><td>Exposure factor, C_e</td><td>C_e is determined from the following table:
<table>
<tr><th></th><th>Fully Exposed</th><th>Fully Exposed</th><th>Sheltered</th></tr>
<tr><td>B</td><td>0.9</td><td>1</td><td>1.2</td></tr>
<tr><td>C</td><td>0.9</td><td>1</td><td>1.1</td></tr>
<tr><td>D</td><td>0.8</td><td>0.9</td><td>1</td></tr>
</table></td><td></td></tr>
</table>

<table>
<tr><th>Button, command or option</th><th>Description</th></tr>
<tr><td></td><td>
<table>
<tr><th></th><th>Fully Exposed</th><th>Fully Exposed</th><th>Sheltered</th></tr>
<tr><td>Above treeline</td><td>0.7</td><td>0.8</td><td>0</td></tr>
<tr><td>Alaska</td><td>0.7</td><td>0.8</td><td>0</td></tr>
</table>
</td></tr>
<tr><td>Thermal Condition</td><td>Option are:

All except below
Structures kept just above freezing
Unheated open air structures
Structures kept below freezing
Continuously heated greenhouses

</td></tr>
<tr><td>Thermal factor, C_t</td><td>C_t is determined from the following table:
<table>
<tr><td>A</td><td>1</td></tr>
<tr><td>B</td><td>1.1</td></tr>
<tr><td>C</td><td>1.2</td></tr>
<tr><td>D</td><td>1.3</td></tr>
<tr><td>E</td><td>0.85</td></tr>
</table>
</td></tr>
<tr><td>Risk Category</td><td>Option are:

I
II
III
IV

</td></tr>
<tr><td>Snow Importance factor, I_s</td><td>I_s is determined from the following table:
<table>
<tr><td>I</td><td>0.8</td></tr>
<tr><td>II</td><td>1</td></tr>
<tr><td>III</td><td>1.1</td></tr>
<tr><td>IV</td><td>1.2</td></tr>
</table>
</td></tr>
</table>

Button, command or option	Description
Flat Roof Snow load, , p_f	p_f is determined from the following: $p_f = 0.7 * C_e * C_t * I_s * p_g$
Next	Takes you to the next page.
Page 2 - Snow load cases (AISC Snow wizard)	This page is used to specify the loadcases as follows:
Minimum Snow Load	Option are: • Yes • No
Balanced Snow Load	Option are: • Yes • No
Unbalanced Snow Load	Option are: • No • 1 • 2 • 3 • 4 You can select between 1 and 4 cases to generate (1 Unbalanced..., 2 Unbalanced, etc.)
Draft Snow Load	Option are: • No • 1 • 2 • 3 • 4 You can select between 1 and 4 cases to generate (1 Unbalanced..., 2 Unbalanced, etc.)
Rain on Snow Surcharge	Option are: • Yes • No

Button, command or option	Description	
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.	

Sub Models dialog

The **Sub Models** dialog box allows you to split the structure into a continuous series of sub models, working from the top of the building down to, and including the foundations.

Location

In the **Structure** tree, double-click **Sub Models**.

Content

Button, Command or Option	Description	
Options		
Level	Before any analysis has been performed, only two levels are displayed, one at a distance (2m) above the highest construction level and a second at the same distance below the lowest level. These cannot be changed. Hence, at this point, there is a single sub-model comprising the whole structure.	
Active	Only active levels act to divide the structure into sub-models. The top and bottom levels must always remain active, as there must always be at least one sub model. Once intermediate levels have been inserted, you can choose to inactivate them if required. When this occurs the sub-models immediately above and below the level in question are merged into a single sub-model.	
Auto Generate	If any adjustments to the cutting planes or cutting plane levels, or any of them are set as inactive you must also clear Auto-Generate. Otherwise the changes you	

Button, Command or Option	Description	
	have made will be lost the next time the sub models are generated.	
Buttons		
OK	Allows you to save any changes made.	
Cancel	Allows you to discard any changes made.	
Insert Above	Allows you to insert a new sub model between the selected level and the one above. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.	
Insert Below	Allows you to insert a new sub model between the selected level and the one below. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.	
Delete		
Generate	Allows you to auto generate default sub models or not • Checked Auto Generate creates default sub models for every level specified as a floor in the Construction Levels dialog. It is optional as default sub models are automatically generated for you when you run the analysis, provided the Auto Generate box on the dialog is checked. You may choose to do this if you want to review the sub models prior to the first run of the analysis. • Unchecked Does not Auto Generate sub models prior to the first run of the analysis. **NOTE** The Generate button can also be used to revert back to the default sub models at any time.	

See also

Manage sub models (page 661)

Create sub models (page 662)

Slab Deflection Settings dialog

The **Slab Deflection Settings** dialog and its subpages allow you to adjust the settings applied to the different analyses.

Location

On the **Slab Deflection** tab, click **Settings**.

Content

Button, command or option	Description
• New Load Event Defaults • New Check Defaults • Aging, Creep & Shrinkage • Modification Factors • Iterative Cracked Section Analysis	See: Slab deflection settings
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Index

1

2

3

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

R

S

T

U

V

W

Z