

Tekla Structural Designer 2018i

Engineer's Handbooks (Australian Codes)

September 2018

(6.1.06)

Table of Contents

Wind Modeling Handbook.....	1
Application of manual wind loads	1
Simple Wind	1
Overview of Simple Wind	1
Simple Wind application and decomposition.....	2
Limitations of wind decomposition to diaphragms	4
Wind load perpendicular to disconnected diaphragms.....	4
Wind load parallel to disconnected diaphragms	8
Snow Loading Handbook	11
Overview of Snow Loading	11
Roof Panel Types	12
ASCE7 Snow Wizard	12
EC1991 1-4 Snow Wizard	12
Snow Loadcases	13
ASCE7 Snow Loadcases	13
EC1991 1-4 Snow Loadcases	13
Snow Load Decomposition.....	13
Manual application of snow loading.....	13
References.....	13
Stability Requirements Handbook.....	15
Introduction to stability requirements	15
Global second-order ($P-\Delta$) effects.....	15
Member second-order ($P-\delta$) effects.....	16
When must global and member effects be considered?.....	17
Global imperfections	18
Member imperfections	18
Potential for a rigorous second order analysis.....	18
Comparing the Design Codes.....	19
Allowing for global second-order effects	21
Use of Modification Factors	21
Allowing for member second-order effects.....	21
Allowing for global imperfections.....	22
Allowing for member imperfections	22
Wind drift.....	22

Overall displacement.....	24
Analysis Handbook.....	27
Analysis Limitations and Assumptions	27
Linear analysis of structures containing material nonlinearity	27
Nonlinear spring supports.....	27
Tension only, or compression only elements	27
Nonlinear axial spring, or nonlinear torsional spring elements.....	27
Tension only X braces.....	27
Analysis of structures containing geometric nonlinearity	28
Analysis of structures containing curved beams.....	28
Analysis of compound (plated) steel beams and columns	28
Story Shears	28
Member Deflections	28
Torsion load analysis - relative angle of twist.....	29
Vibration analysis - active mass.....	29
Summed Active Mass.....	30
Summed Total Translational Mass.....	30
Translation %.....	30
Participation Translation %.....	30
Vibration analysis - modal mass	30
Modal Mass.....	31
Unstable Structures.....	32
Flat Slab Structures	32
Solver models	32
Solver Model Components.....	32
Working Solver Model	33
Solver Model used for 1st Order Linear	33
3D Analysis model.....	33
Solver Model used for 1st Order Non Linear	35
Solver Model used for 2nd Order Linear	35
Solver Model used for 2nd Order Non Linear	36
Solver Model used for 1st Order Vibration.....	36
Solver Model used for 2nd Order Buckling	36
Solver Model used for Grillage Chasedown	37
Solver Model used for FE Chasedown	38

Solver Model used for Load Decomposition.....	39
Refresh Solver Model	40
Solver elements for 1D members.....	40
Solver Element (1D) Types.....	40
Solver element types applied to members in the different solver models.....	41
Member connectivity.....	42
Rigid offsets in concrete beams and columns	42
Rigid offsets example 1 - concrete column.....	43
Rigid offsets example 2 - concrete beam	45
Rigid zones in concrete beams and columns.....	47
Application of rigid zones.....	47
Rigid zones example 1 - fixed ended beam.....	48
Rigid zones example 2 - pin ended beam	51
Inactive solver elements	53
Solver elements for concrete shear walls.....	53
Meshed concrete shear wall geometry.....	53
Mid-pier concrete shear wall geometry	57
Sub-division of concrete shear walls	59
Concrete shear wall openings and extensions.....	59
Concrete wall openings	59
Limitations of wall openings	59
Analysis model applied to meshed wall panels with openings	59
Alternative model for wall openings.....	60
Concrete wall extensions	61
Use of concrete wall extensions.....	61
Concrete wall extension examples.....	62
Solver elements for bearing walls	66
Solver elements for 2-way spanning concrete slabs.....	69
Releases.....	70
Column releases	70
Wall releases	71
Beam releases.....	71
Brace releases.....	72
Supports.....	72
Support degrees of freedom.....	72

Non linear spring supports	73
Partial fixity of column bases.....	73
Diaphragms and floor meshing	74
Diaphragm types.....	74
Rigid	75
Semi-rigid.....	76
Diaphragm constraint and mesh type configurations	77
Diaphragm option.....	77
Decomposition.....	77
Mesh 2-way Slabs in 3D Analysis	78
Summary of diaphragm constraint and mesh type configurations	78
Other slab properties affecting the solver models.....	79
Mesh parameters	80
Slab Mesh.....	80
Semi-Rigid Mesh	80
Static Analysis and Design Handbook	81
Stages of the Design All (Static) process	81
3D pre-Analysis	82
Application of global imperfections	83
Load reductions.....	83
Generation of pattern loading	84
3D Analysis.....	84
Why chasedown analyses are required.....	84
Grillage Chasedown Analysis	90
Grillage Chasedown	90
Accounting for lateral loading in Chasedown Results	90
FE Chasedown Analysis	91
Member design.....	91
Comparison of solver models used in static design.....	92
Seismic Analysis and Design Handbook	95
Introduction to Seismic Analysis and Design.....	95
Definitions.....	95
Code Spectra.....	95
Site Specific Spectra	95
Base Shear Combination	95

RSA Seismic Combination	95
Static Loadcase	95
RSA Seismic Loadcase	96
RSA Torsion Loadcase	96
Fundamental Period (T)	96
Level Seismic Weight	96
Effective Seismic Weight	96
Seismic Base Shear	96
Square root of Summation of Square (SRSS)	96
Complete Quadratic Combination (CQC)	96
Cross Modal Coefficient	97
Overview	97
Seismic Wizard	98
Vertical and Horizontal Irregularities	99
Torsion	99
Vibration Analysis	100
Equivalent Lateral Force Method	100
Response Spectrum Analysis Method	100
Summary of RSA Seismic Analysis Processes	100
Seismic Drift	101
Limitations of Seismic Design	101
Specific limitations of steel seismic design	102
Seismic Force Resisting Systems	102
Available SFRS types	103
SFRS types included for steel members	103
SFRS types available for concrete members	103
SFRS types excluded	103
Members allowed in the SFRS	103
Assigning members to the SFRS	104
Special Moment Frames - assigning connection types at steel beam ends	104
Validation of the SFRS	104
Auto design of SFRS members	105
Seismic Design Methods	105
Seismic analysis and conventional design	106
ELF seismic analysis and conventional design	106

RSA seismic analysis and conventional design	106
Seismic analysis and seismic design	107
ELF seismic analysis and seismic design	107
RSA seismic analysis and seismic design	108
Steel Design Handbook	111
General design parameters (steel)	111
Material type	111
Autodesign (steel)	111
Design Section Order	112
How do I view the list of sections in a design section order?	112
How do I specify that a section in the list should not be considered for design?	112
How do I sort the listed sections by a different property?	112
How do I specify that a section is non-preferred?	113
How do I reset a design section order back to the original default?	113
How do I create a new Design section order?	113
Size Constraints	114
Gravity only design	114
Design groups (steel)	114
How is the “design using groups option” activated?	115
What happens in the group design process?	115
Instability factor	115
Steel beam design	116
Steel beam design overview	116
Steel beam fabrication	117
Steel beam section	117
Steel beam restraints	118
Steel beam deflection limits	118
Steel beam camber	119
Steel beams in seismic force resisting systems	119
Steel column design	119
Steel column overview	119
Limitations for sloping columns	120
Simple columns	120
Steel column section	120
Steel column restraints	121

Splice and splice offset	122
Steel column web openings.....	123
Steel columns in seismic force resisting systems.....	124
Steel brace design	124
Steel brace overview	124
Input method for A and V Braces	125
Steel brace section	125
Steel brace in compression	125
Steel brace in tension	125
Steel truss design	125
Steel joist design.....	126
Standard types	126
Special Joists	127
Joist Girders.....	127
Joist Analytical Properties	127
Performing steel structure design.....	127
Gravity design.....	127
Full design.....	128
Concrete Design Handbook.....	129
General design parameters (concrete)	129
Gravity or Static Concrete Design?.....	129
Analysis types performed in the Design Concrete process	129
Pre-design considerations	130
Stacks and reinforcement lifts.....	130
Column and wall clear height.....	131
Nominal cover.....	132
Assume cracked.....	132
Reinforcement Parameters.....	132
Design and detailing groups (concrete).....	132
Why use concrete design and detailing groups?	132
What happens in the group design process?	133
Concrete design group requirements	134
Detailing group requirements	134
Group management.....	136
How is grouped design and detailing de-activated for concrete members?.....	137

Concrete beam design	137
Analysis types used for concrete beam design	137
Autodesign (concrete beam)	137
Deflection control	138
Use of beam flanges	139
Longitudinal reinforcement (concrete beam)	142
Bar layers	142
Longitudinal Reinforcement Patterns Library	144
Longitudinal Reinforcement Regions	146
Relationship between Reinforcement Patterns and Design Regions	148
Shear reinforcement	149
Shear Reinforcement Shapes Library	149
Shear Reinforcement Patterns Library	150
Shear Reinforcement Regions	150
Concrete column design	151
Autodesign (concrete column)	151
Section (concrete column)	152
Slenderness (concrete column)	153
Clear height	154
Longitudinal Reinforcement (concrete column)	154
Column design forces	155
Column interaction diagrams	155
Column axial force-moment interaction diagram	155
Column moment Interaction diagram	156
Concrete wall design	159
Autodesign (concrete wall)	159
Slenderness (concrete wall)	160
Clear height	160
Reinforcement (concrete wall)	161
Wall design forces	161
Wall interaction diagrams	161
Wall axial force-moment interaction diagram	162
Wall moment Interaction diagram	163
Concrete slab design	165
Slab types designed by Tekla Structural Designer	165

Two-way spanning slab panel design moments.....	165
Slab reinforcement	165
Slab patch reinforcement.....	166
Slab panel design checks	168
Flat slab deflection checks.....	169
Overview of Concrete Slab Deflection.....	170
Instantaneous Deflection.....	170
Creep Deflection	171
Shrinkage Deflection.....	171
Slab on beam idealized panels	171
Typical flat slab design procedure.....	172
Overall Slab Design Workflow	173
Flat slab design example	173
Split/join panels as necessary and set up Pattern Loading	174
Analyse All (or Design All).....	175
Consider Simple (linear) Deflection.....	175
Select a Level (or sub-model)	178
Add Patches.....	178
Design Panels.....	179
Review/Optimise Panel Design	180
Design Patches	181
Review/Optimise Patch Design.....	181
Add and Run Punching Checks.....	182
Rigorous Deflection Check	182
Create Drawings and Quantity Estimations.....	182
Print Calculations.....	182
Typical slab on beams design procedure	183
Slab on beam design example	183
Set up Pattern Loading	184
Design All	184
Select a Level.....	185
Add Beam and Wall Top Patches	185
Design Panels.....	186
Review/Optimise Panel Design	187
Design Beam and Wall Patches.....	188

Review/Optimise Beam and Wall Patch Design	189
Create Drawings and Quantity Estimations.....	189
Print Calculations.....	189
Performing concrete structure design.....	189
Typical Design Concrete workflow.....	190
Set up Pattern Loading	190
Set all beams columns and walls into autodesign mode.....	191
Review beam and column design groups.....	191
Review beam, column and wall design parameters and reinforcement settings	192
Perform the concrete design	192
Review the design status and ratios.....	193
Create Drawings and Quantity Estimations.....	194
Print Calculations.....	194
Reviewing Design Concrete and refining the design of individual members.....	194
Interactive concrete member design	195
Interactive concrete beam design	195
How do I open the Interactive Beam Design Dialog?	195
Overview of the Interactive Beam Design Dialog	196
How do I change the bar pattern?.....	199
Interactive concrete column design.....	199
How do I open the Interactive Column Design Dialog?.....	199
Overview of the Interactive Column Design Dialog	200
How do I arrange bars in the Interactive Column Design Dialog?	204
How do I define additional column design cases?	205
Interactive concrete wall design.....	206
How do I open the Interactive Wall Design Dialog?.....	206
Overview of the Interactive Wall Design Dialog.....	206
How do I define additional wall design cases?	211
Working with large models	211
Foundation Design Handbook.....	215
Isolated foundation design.....	215
Overview of the isolated foundation analysis model.....	215
Association with member supports.....	215
Analysis types.....	215
Design forces and checks.....	216

Bearing pressure calculations	217
Pad base and strip base design procedures.....	217
Pad base design example.....	218
Apply bases under supported columns	218
Auto-size bases individually for loads carried.....	219
Apply grouping to rationalize pad base sizes	221
Review/Optimise Base Design.....	223
Create Drawings and Quantity Estimations.....	223
Print Calculations.....	224
Mat foundation design.....	224
Features of the mat foundation analysis model.....	224
Analysis Types.....	224
Soil Structure Interaction.....	224
Soil Parameters.....	225
Typical mat foundation design procedure	226
Design the structure before supporting it on the mat.....	228
Create the mat, (either with ground springs, or discreet supports).....	228
Model validation.....	229
Perform the model analysis.....	230
Check foundation Bearing Pressure and Deformations	230
Re-perform member design.....	231
Open an appropriate view in which to design the mat	232
Add Patches.....	232
Design Mats.....	233
Review/Optimise Mat Design	233
Design Patches	234
Review/Optimise Patch Design.....	235
Add and Run Punching Checks.....	235
Create Drawings and Quantity Estimations.....	236
Print Calculations.....	237

Wind Modeling Handbook

This handbook describes two approaches for defining wind loads in *Tekla Structural Designer*.

- The **Wind Model** method is the most comprehensive - requiring 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.
- Alternatively you might choose to **manually apply wind loads** (thus avoiding 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** loads, which are subsequently decomposed to the structure during analysis.



The Wind Model method is not currently available for the AS:1170.2 wind code variant.

Application of manual wind loads

This approach provides a quick means to apply wind to the structure, without requiring you to create a wind model.

In order to manually apply wind loads, you must first create a loadcase for them and set its load type to Wind.

Panel, Member, and Structure loads can then be manually applied in this loadcase as required. You can also apply 'Simple Wind' loads in the same loadcase.



*'Manual' wind loadcases are created from the 'Loadcases' dialog **not** from the 'Wind Loadcases' dialog (which is available for 'Wind Model' wind loadcases only).*

Simple Wind

Overview of Simple Wind

There is a simple process to follow when you want to apply Simple Wind loads. The basic steps are detailed below.

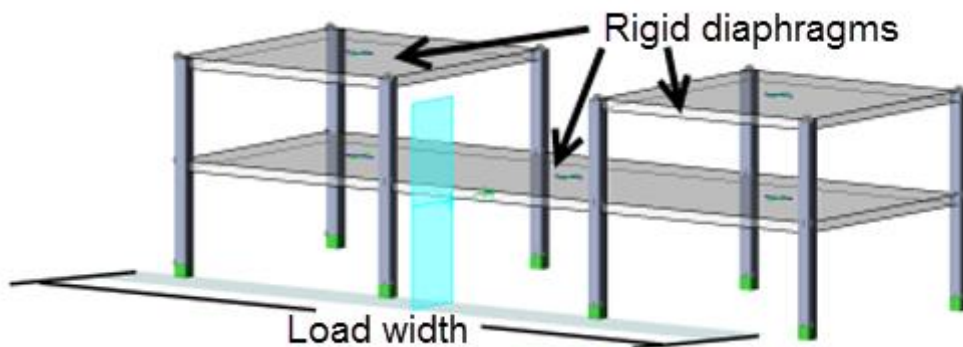
1. Define the structure ensuring that slabs have been created with the diaphragm option set to rigid as opposed to semi-rigid.
2. In order access Simple Wind, you must first create and then select the loadcase into which the Simple Wind loads are to be added.
3. Click **Simple Wind** on the Load ribbon to define the wind loads to be applied in the selected wind loadcase.
4. Combine the wind loadcases with the other loadcases you have defined for your structure to create the design combinations you need to consider.
5. Perform the analysis and design of the structure.

Simple Wind application and decomposition

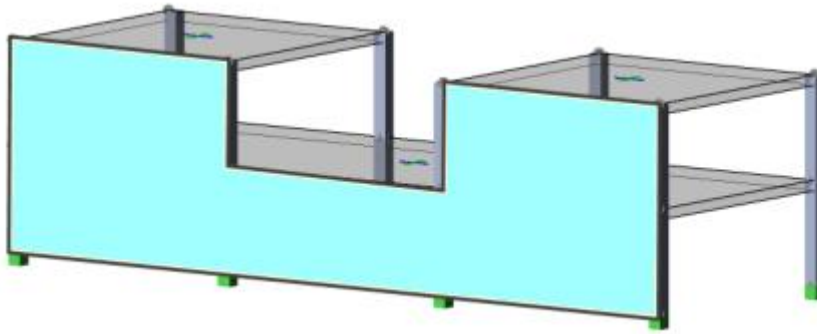
Application

Simple Wind loads require rigid diaphragms to exist within the width of the load.

Each Simple Wind load is applied over a defined width and height onto the structure.



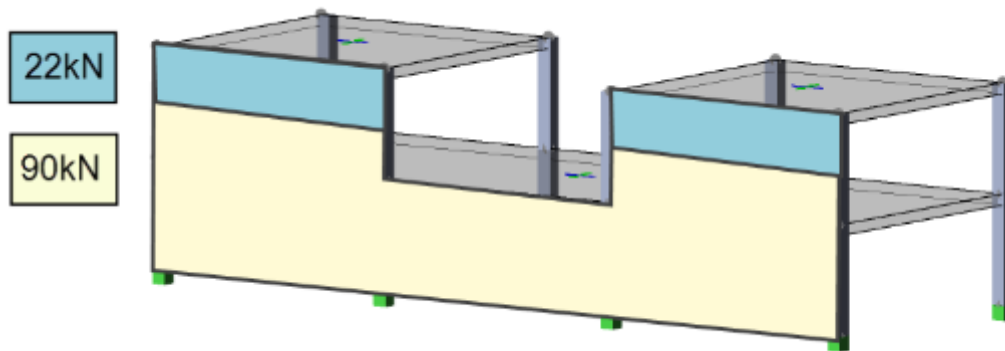
- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation error is generated - this is the case to ensure the loading is distributed as correctly as possible.
- Only the load within the building profile is considered for decomposition.



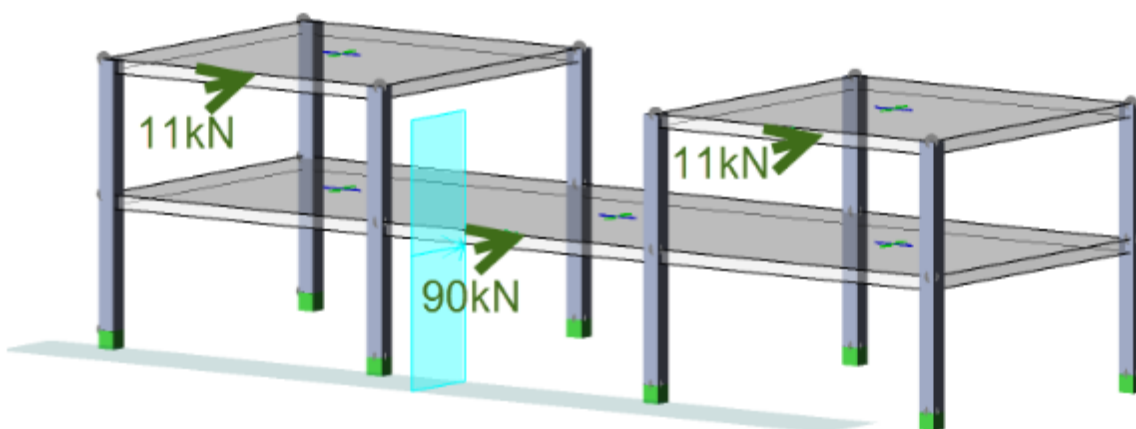
Decomposition

Simple Wind loads are decomposed to point loads on rigid diaphragms only. They are **not** 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.



- The loads are 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.



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.

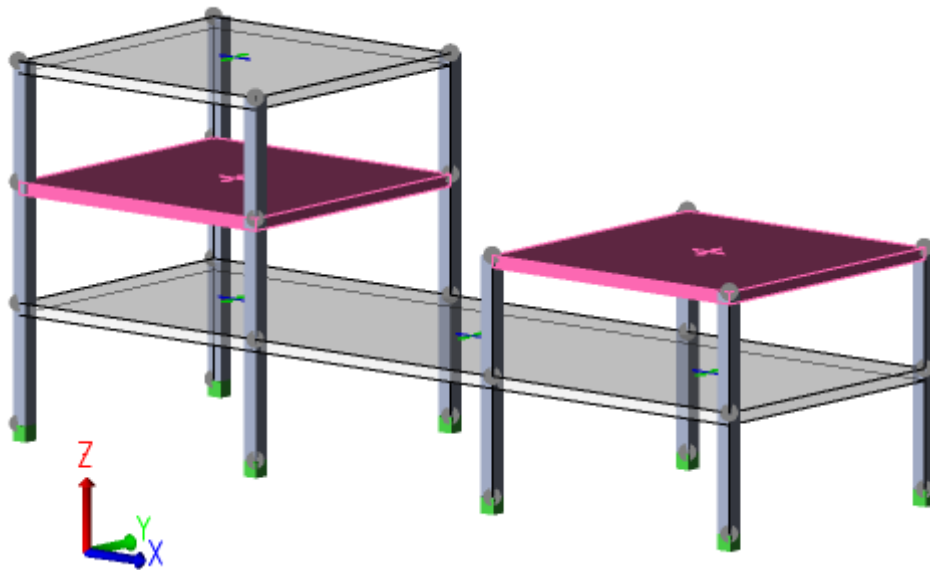
Limitations of wind decomposition to diaphragms

Irrespective of whether loads are input via the 'Wind Model' method, or via 'Simple Wind' loads, certain building shapes need extra consideration when rigid diaphragm load decomposition is applied.

As illustrated by the following examples, buildings containing discreet towers (and thus containing disconnected rigid diaphragms) are a particular concern.

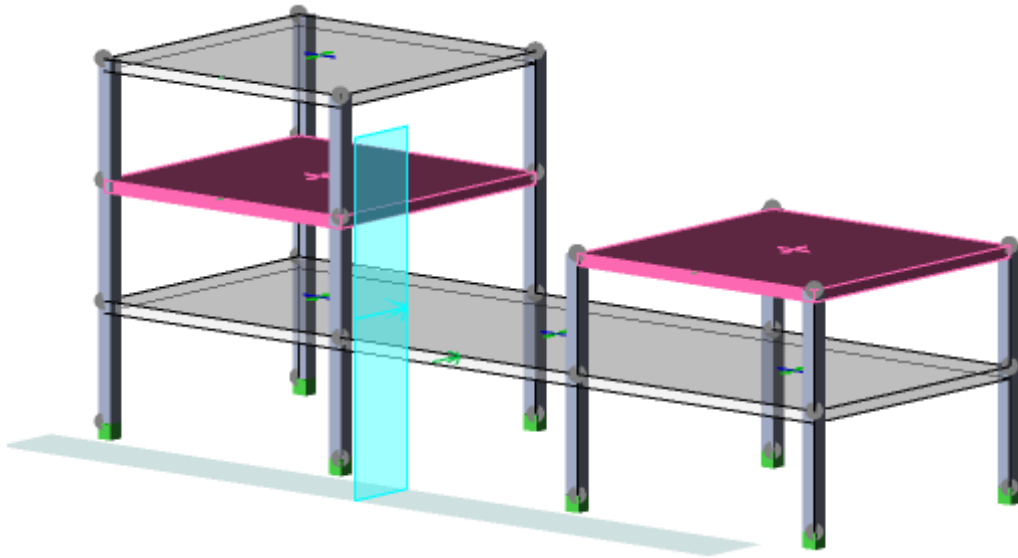
Wind load perpendicular to disconnected diaphragms

In this example wind load is applied in the Global Y direction, perpendicular to the disconnected diaphragms that exist at the second floor level.

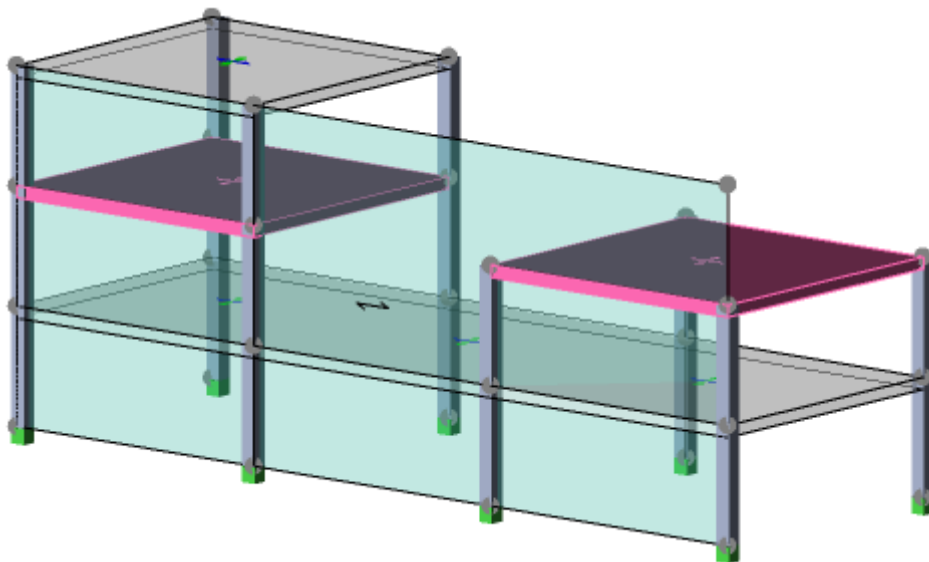


Disconnected diaphragms at second floor level

An issue arises when a wind load is applied in such a way that it has to be decomposed to both diaphragms. Such a load could be applied either via a Simple Wind load, or a wind wall panel:

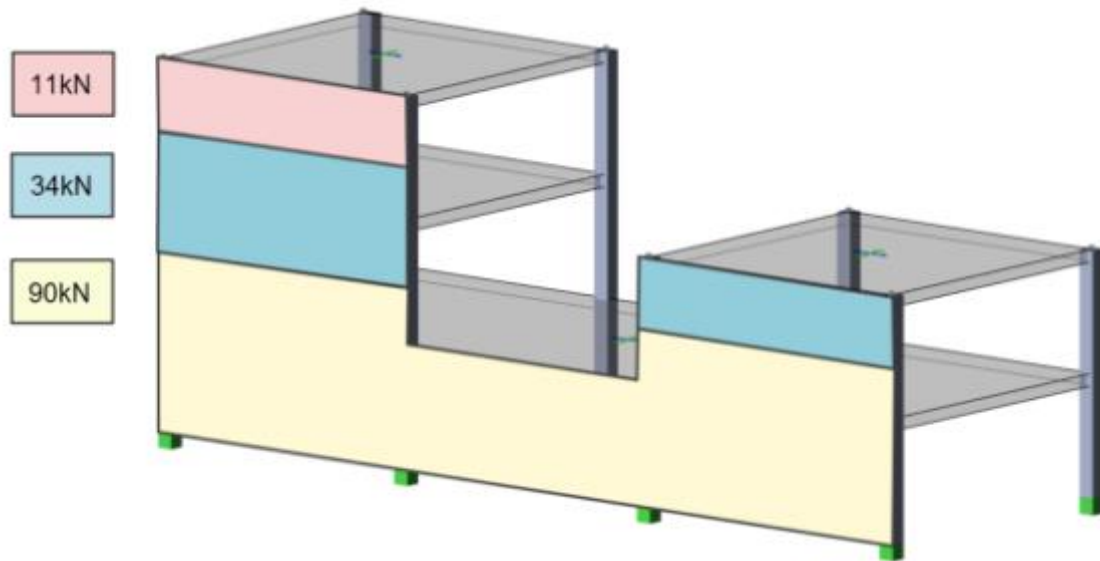


Load applied via a Simple Wind Load



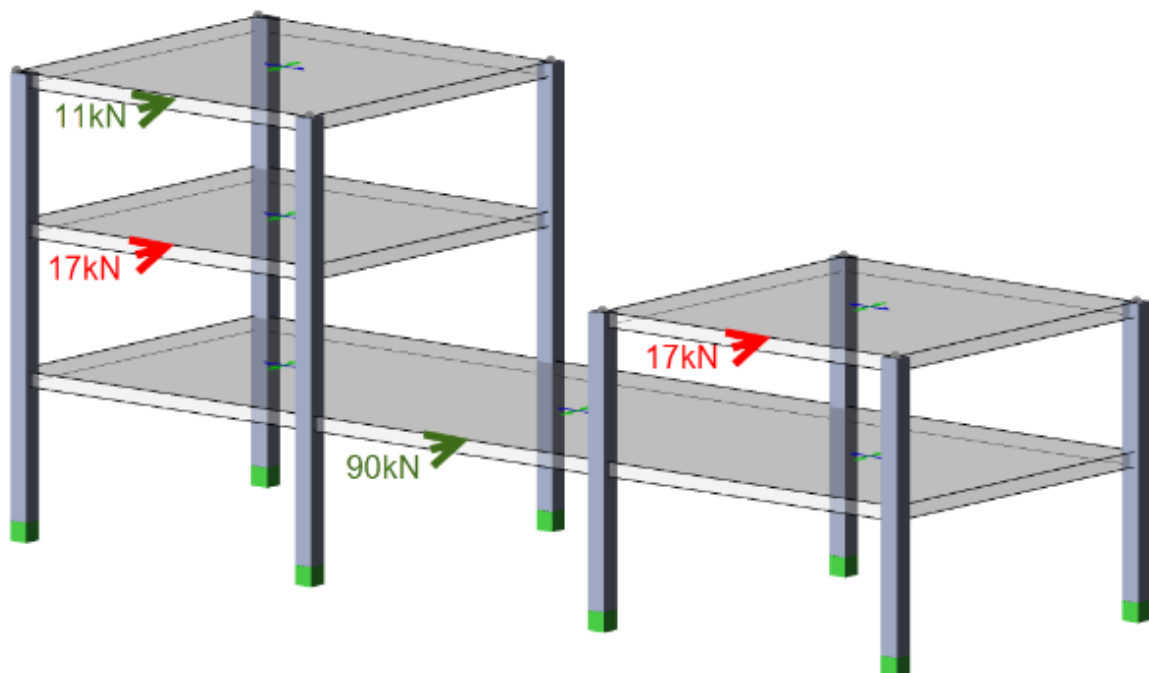
Load applied via a Wind Wall Panel

Irrespective of the method used to apply it, the area load within the building profile is shared between levels prior to decomposition.

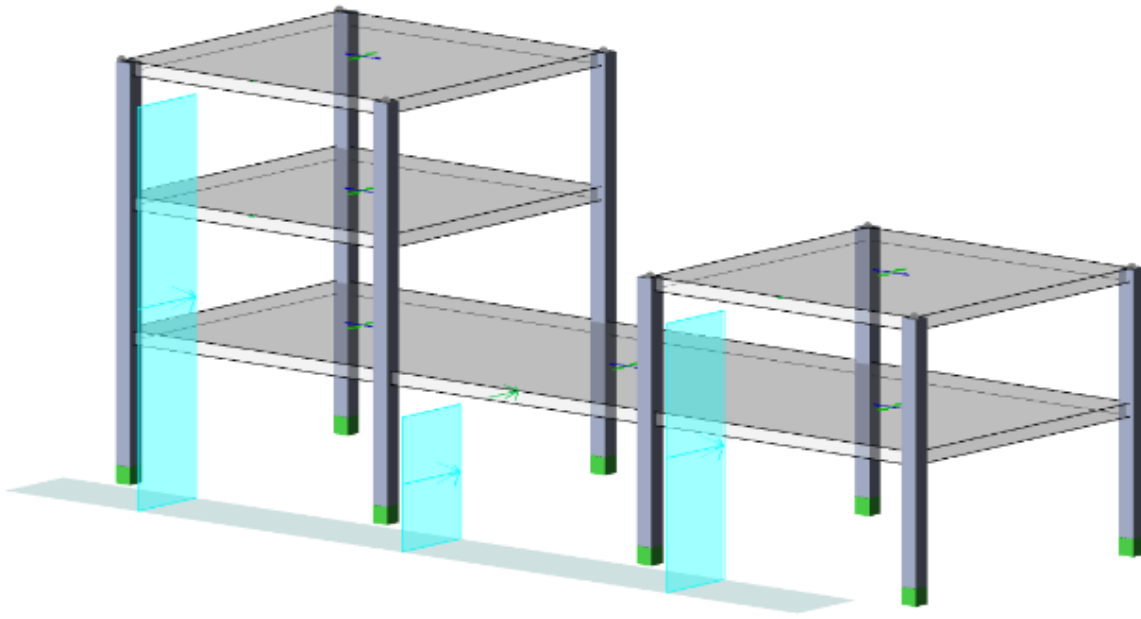


The load is then decomposed to the diaphragms at each level **in proportion to the width of each diaphragm**.

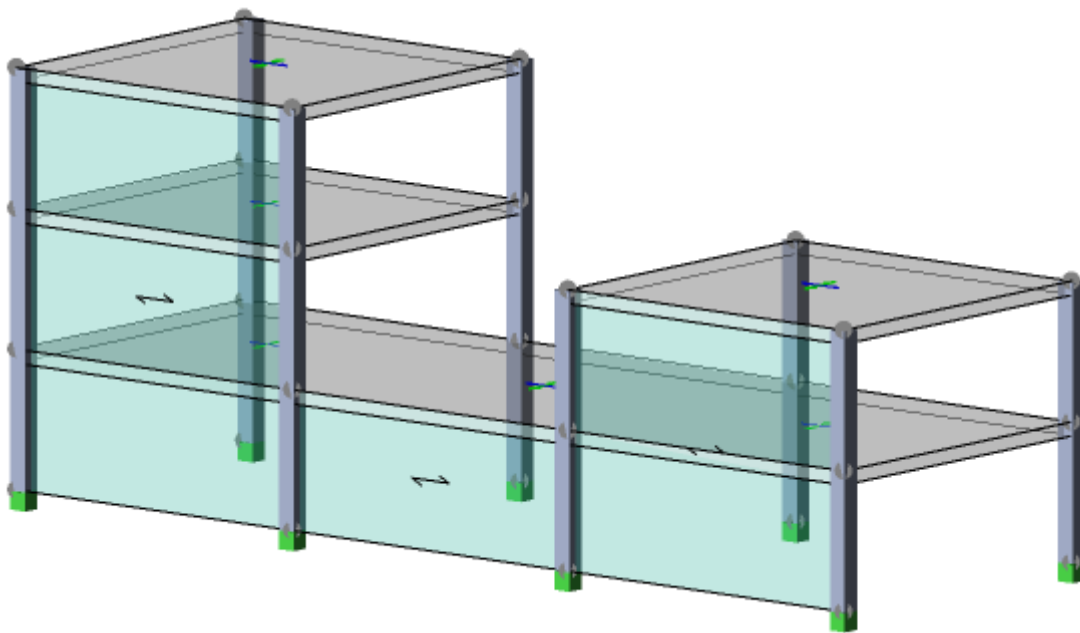
In this case because both diaphragms the second floor level are of equal width, the load is shared equally between them. This is not satisfactory as more of the load should have been applied to the left hand diaphragm in this case.



The workaround varies according to the method of loading, but basically involves replacing the original load with separate loads in each bay:

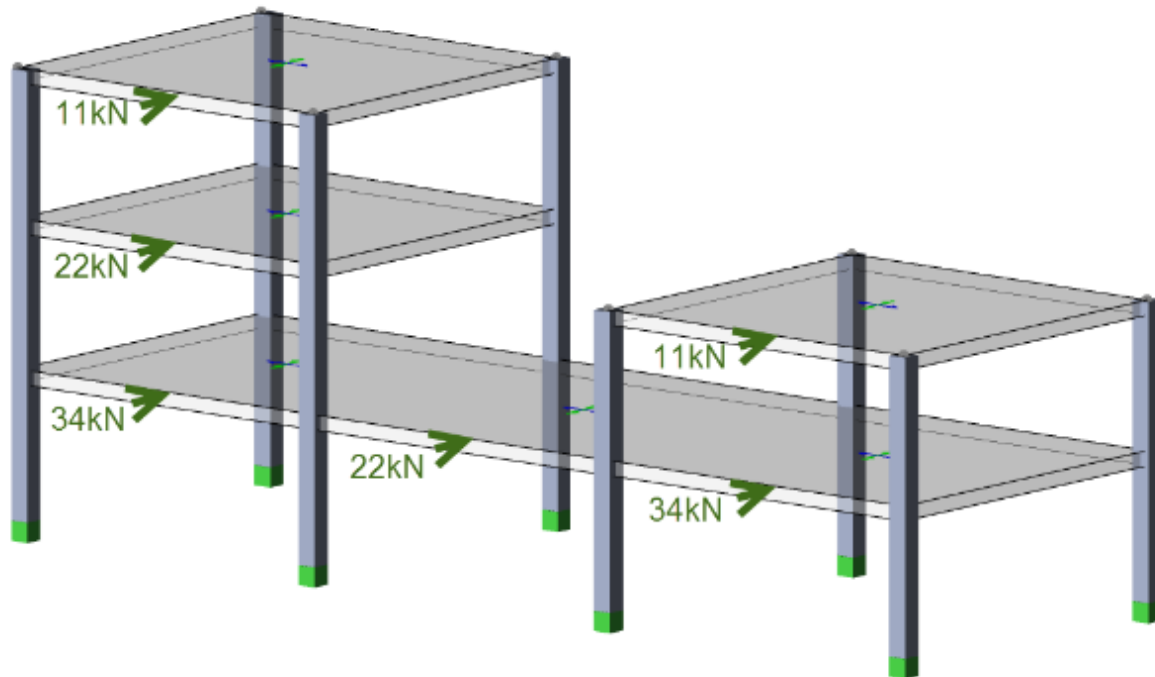


Simple Wind Workaround - Load re-applied as 3 Simple Wind Loads



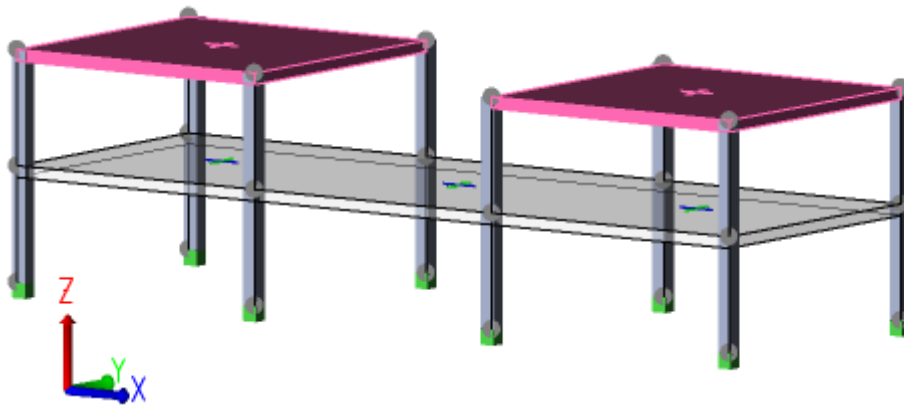
Wind Wall Workaround - Load re-applied via 3 Wind Wall Panels

In both the above cases, the load is then decomposed as originally intended.



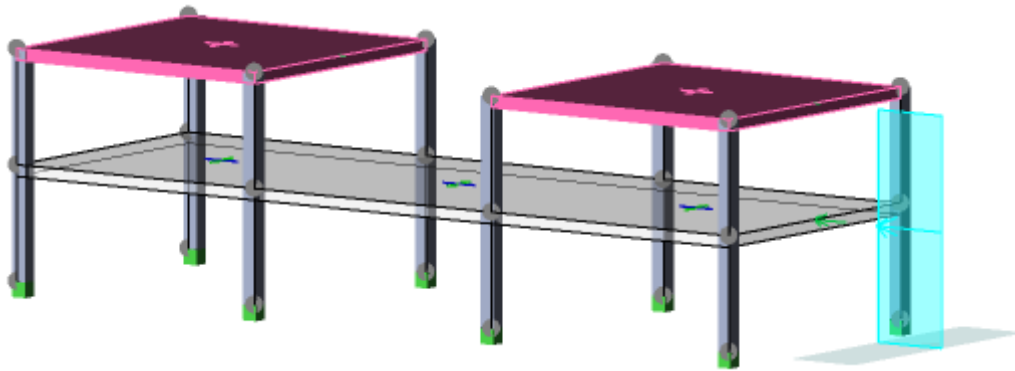
Wind load parallel to disconnected diaphragms

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:

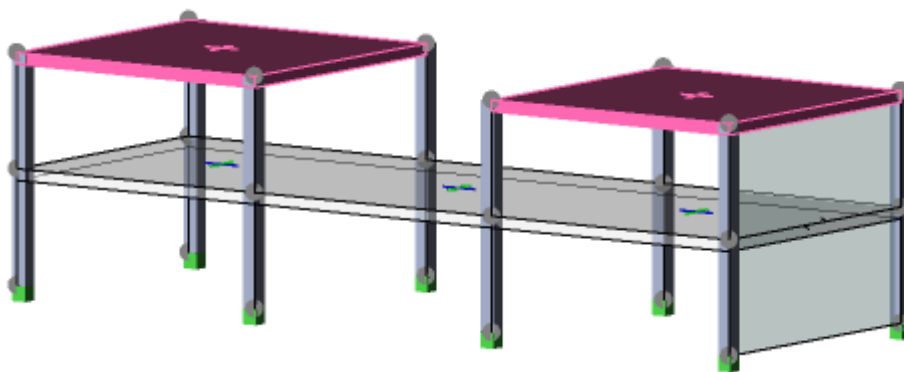


Disconnected diaphragms at second floor level

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:

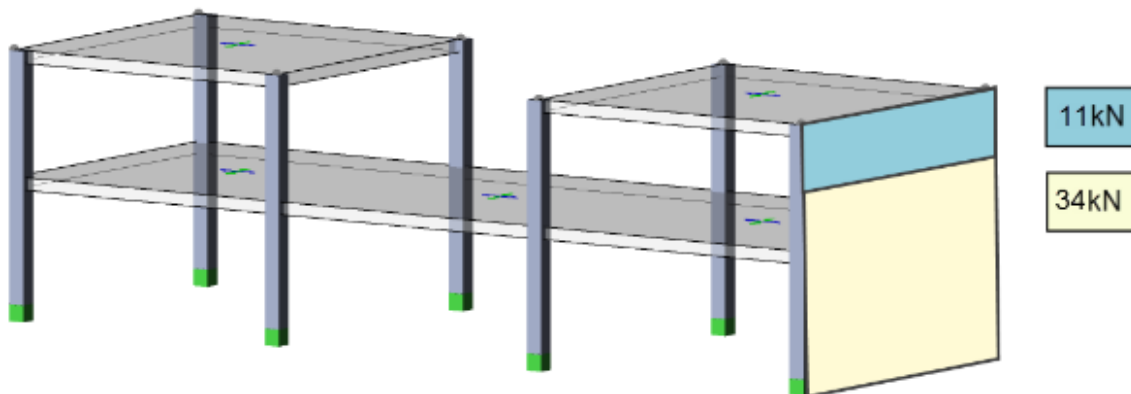


Load applied via a Simple Wind Load

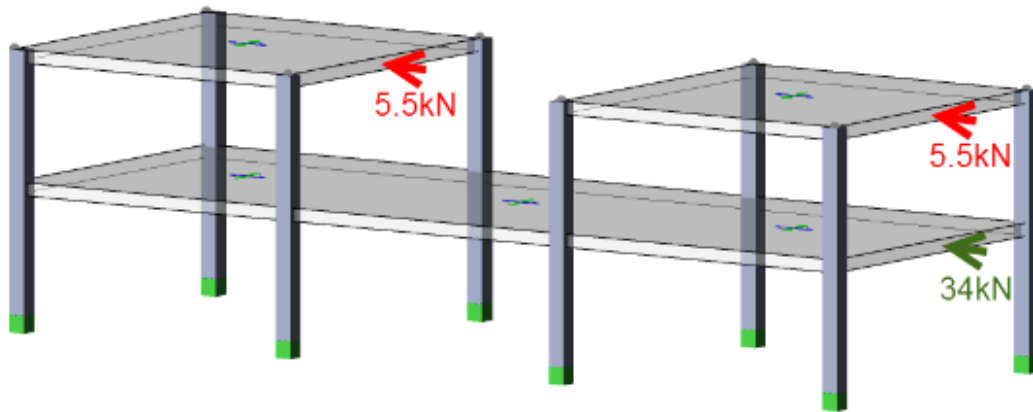


Load applied via a Wind Wall Panel

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 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 instead.

Snow Loading Handbook

This handbook describes the different ways of modelling snow loading in *Tekla Structural Designer*.

- The **Snow Wizard** can be run to input basic snow data and set up the required snow loadcases. Provided that roof panels have been modelled 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.



The Snow Wizard is not currently available for the Indian, Australian, or British Standard snow code variants.

- If you don't run the **Snow Wizard** you will still be able to manually set up snow loadcases from the loading dialog. The 'loadcase type' can be set as Snow, but the load types that can be applied will be restricted to 'Panel', 'Member' and 'Structure' loads. The 'Snow' load type (for creating Uniform Snow, Valley Snow and Local Drift Snow) will not be available.

Overview of Snow Loading

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)

The Snow Wizard

Prior to running the Snow Wizard you should ensure the roofed areas of the building are 'clad' with roof panels.

The Snow Wizard can then be run in order to define the basic snow load factors.

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 load case, 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 required for this method can be summarised as follows:

1. Apply roof panels to structure
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

Roof Panel Types

For the snow loading calculations *Tekla Structural Designer* has to distinguish between monopitch and pitched roofs. This gets determined from the Roof Panel Type that has been assigned as follows:

Roof Type	For Snow Loading this is considered as:
Default	Monopitch
Flat	Monopitch
Monopitch	Monopitch
Duopitch	Pitched
Hip gable	Pitched
Hip Main	Pitched
Mansard	Pitched

ASCE7 Snow Wizard

To access this configuration of the Snow Wizard the Snow Loading Code has to be set to ASCE7.

EC1991 1-4 Snow Wizard



Unless explicitly noted otherwise, all clauses, figures and tables referred to in the topics in this section are from EC1991 1-4.

Snow Loadcases

The snow loadcases that are set up will depend on the head code that is being worked to.



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.

ASCE7 Snow Loadcases

EC1991 1-4 Snow Loadcases

Snow Load Decomposition

The direction of the one way decomposition of the snow loads to roof members is determined by the span direction (i.e. rotation angle) of the roof panel.

Manual application of snow loading

This approach provides an alternative means to apply snow to the structure, without necessarily having to run the Snow Wizard.

In order to manually apply snow loads, you must first create a loadcase and set its load type to Snow.

- Snow loadcases created from the **Loadcases dialog** can have 'Panel', 'Member', and 'Structure' loads applied but cannot have 'Snow' (i.e. Uniform Snow, Valley Snow and Local Drift Snow) loads applied.
- Whether snow loadcases created by running the **Snow Wizard** can have 'Panel', 'Member', 'Structure' and 'Snow' loads applied will vary from loadcase to loadcase.

References

1. **ASCE/SEI 7-10.** *Minimum Design Loads for Buildings and Other Structures.* **ASCE, 2010. ISBN: 978-0-7844-1085-1.**
2. **British Standards Institution (September 2008).** *UK National Annex to Eurocode 1: Actions on structures. NA to BS EN 1991-1-4:2005.*
3. **British Standards Institution.** *Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009.*

Stability Requirements Handbook

Introduction to stability requirements

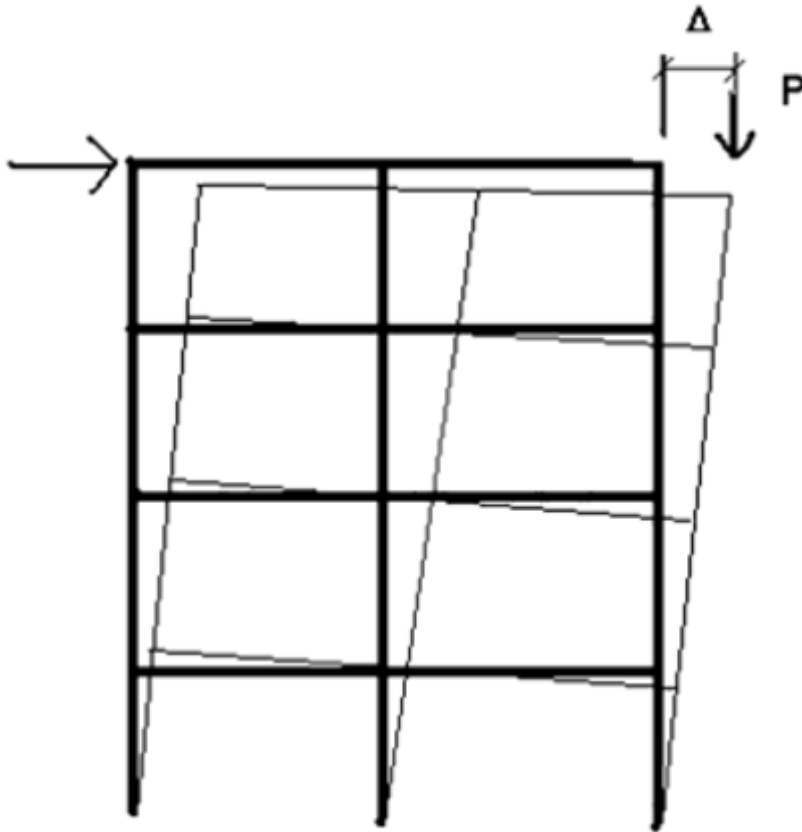
The analysis and design process has to allow for the differences between a theoretical mathematical model of a building and a more realistic representation. For example, buildings are not truly vertical when first built nor do they remain so when subject to load. These are called Stability Requirements and are from four sources:

1. [Global second-order \(P- \$\Delta\$ \) effects](#) to allow for deformation of the structure under load,
2. [Member second-order \(P- \$\delta\$ \) 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.



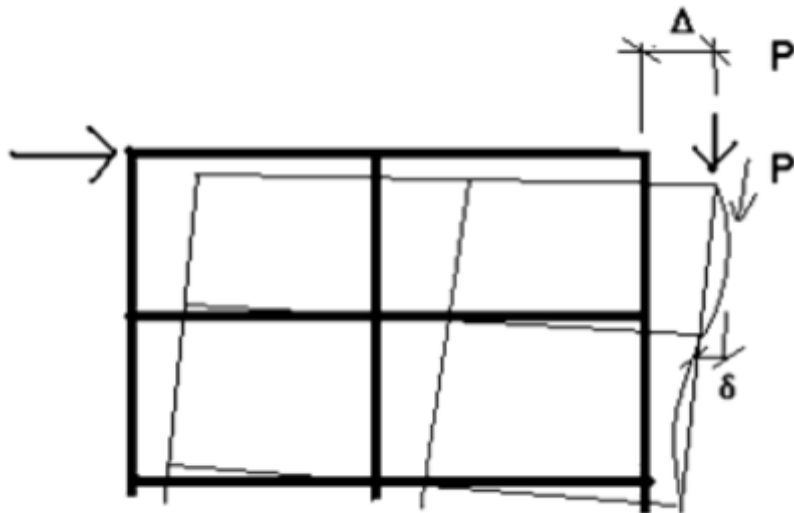
- Provided the deflection is small:
 - Structure is 'Non-sway'
 - Second order effects can be ignored.
- At some level these effects are no longer ignorable:
 - Structure is 'Sway sensitive'
 - And you have to do something to account for the second order effects.



The terminology for 'Non-sway' and 'Sway-sensitive' structures can vary between codes.

Member second-order ($P-\delta$) 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 these effects are considered ignorable.
 - Member is 'Short' or 'Stocky' or 'Non-Slender'



The terminology for this can vary between codes.

- At some level they are no longer ignorable:
 - Member is 'Slender'

In steel structures these effects are intrinsically allowed for in the design equations.

When must global and member 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	1	3
Slender Member	2	4

- 1 – All second order effects can be ignored
- 2 – Global effects can be ignored – member effects must be considered
- 3 – Global effects must be considered - member effects can be ignored

4 – Global effects must be considered - member effects must be considered



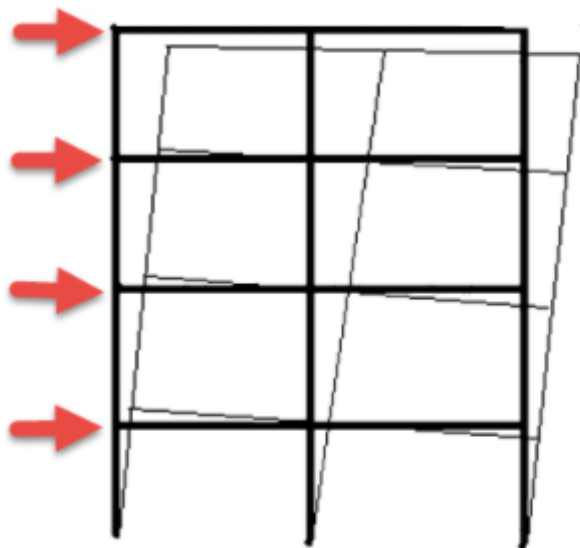
REMEMBER – every structure and every member has 2 directions!

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 Notional Horizontal Forces 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.



Global Imperfections apply regardless of whether the structure is non-sway or sway sensitive.

Member imperfections

Member Imperfections apply regardless of whether members are slender or not. They are dealt with as part of the member design.

Potential for a rigorous second order analysis

'In Theory' – this can look attractively simple...

- A rigorous second order analysis can expose both global and member effects
- Most design codes will say something that allows this analysis route

'In Practice' – It's not quite so simple...

- The analysis method and model need to be up to the task
 - For example the US steel design codes give detailed guidance on this (*Tekla Structural Designer* meets these requirements)
- But for Eurocode design of a concrete structure there are extra requirements, you would need to:
 - Introduce member imperfections to the global analysis.
 - Determine a unique 'nominal stiffness' for every member – an iterative procedure.
 - And there is debate about whether this would actually result in an overestimate of global effects.

Comparing the Design Codes

What all codes ask you to do in one form or another:

- Determine if 2nd order effects are ignorable – if not then don't ignore them!


In practice this tends to be broken down:

- Assess susceptibility to global second order effects (sway sensitivity)
- If sway sensitive do something to introduce global 2nd order effects:
 - There are optional ways of doing this
- For each member consider slenderness
 - If slender add to or amplify design moments.

Comparing the methods of sway sensitivity assessment

Some quite high level observations...

Design Code	Notes on Sway Sensitivity Assessment for 'multi-storey buildings'	
EC3 (steel) (BS 5950 very similar)	Cl 5.2.1 'non-sway' when the elastic critical load factor $\alpha_{cr} \geq 10$ (10% is ignorable)	$\alpha_{cr} = H_{Ed}/V_{Ed} * h/\delta_{HEd}$
ACI 318-11	Cl 10.10.5 'non-sway' when the Stability Index $Q < 0.05$ (5% is ignorable) NOTE - Q is inverse of α_{cr}	$Q = \Sigma P_u \Delta_o / V_{us} l_c$
EC2 (concrete)	Cl 5.8.3.3 and Annex H give several methods. Annex H2 can be re-arranged to match EC3.	$\alpha_{cr} = H_{Ed}/V_{Ed} * h/\delta_{HEd}$

BS8110	Cl 3.8.1.5 - Very Simple – if there are walls then everything is braced.(i.e. it is assumed not sway sensitive).	 Is it true? If there are walls – no matter how insubstantial – there is no significant global (P-Delta) second order effect?
--------	--	--

If the structure is sway sensitive then you need to introduce global second order effects – the codes give optional ways of doing this:

- A P-Delta Analysis
(The simplest option – this is what we suggest you do)
- Some form of Force Amplification method
 - More traditional approach that allows you to continue using 1st order analysis.

Comparing the Force Amplification methods

Although not our recommended route, it is interesting to compare the Force Amplification methods in the different design codes:

Design Code	Notes on Force Amplification Options for 'multi-storey buildings'	
EC3 (steel) (BS 5950 very similar)	Cl 5.2.2 indicates that horizontal loads ('and other possible sway effects') should be increased by a factor as shown.	$1/(1 - 1/\alpha_{cr})$
ACI 318-11	Cl 10.10.7 indicates that the end moments in compression members (due to loading causing sway) should be amplified. NOTE – same amplifier - Q is inverse of α_{cr}	$M_1 = M_{1ns} + \delta_s M_{1s}$ $M_2 = M_{2ns} + \delta_s M_{2s}$ $\delta_s = 1/(1-Q) \geq 1$
EC2 (concrete)	Annex H2 gives an option which can be re-arranged to match EC3.	$F_{HEd} = F_{H0Ed} / (1 - F_{VEd} / F_{V,B})$
BS8110	Cl 3.8.3.7 – an additional moment is added to the end moments. Cl 3.8.3.8 – allows for this to be averaged	$M_{add} = N a_u$

Notes about the amplification methods:

- They all have limitations defined in the code.
- In all cases different amplifiers apply in each direction
- EC applies a single (worst case) amplifier to the entire structure – conservative.
- ACI allows for different amplifiers to be applied in each story height.
- ACI as written suggests amplifying the moments resulting from lateral loading rather than amplifying the lateral loads. This is only applicable to moment frames.
- The EC approach is more general (but with disadvantage that amplified loads impact on foundations)

- The BS approach – may be simple and safe, but it is not amplification and is not realistic.

Bottom Line – these are all simplifications so if you have the option to use a P-Delta analysis why wouldn't you?

Comparing the Member Slenderness Assessment methods

Design Code	Notes on Member Slenderness Assessment	
EC3 (steel) (BS 5950 very similar)	This is inherently dealt with in the buckling resistance calculation	i.e. no classification
ACI 318-11	Cl 10.10.1 – limit below which slenderness effects can be ignored. $k.l_u / r$ = eff length / radius of gyration	$k.l_u / r \leq 34 - 12 (M_1 / M_2) \leq 40$
EC2 (concrete)	Cl 5.8.3.1 – $\lambda_{lim} = 20.A.B.C / \sqrt{n}$ λ = eff length / radius of gyration (But a much more complex limit to calculate)	
BS8110	Cl 3.8.1.3 presents a very simple measurement where the moment profile is ignored	I_{ex}/h and I_{ey}/b are less than 15 (braced)

Allowing for global second-order effects

- [Sway checks](#)

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 member second-order effects

These are dealt with as part of the design of members.

For steel this is incorporated in the design routines for all members (beams, columns, braces).

Similarly for concrete, much of the calculation is carried out as part of the design. However, in order to assess the 'effective length' of the member (columns and walls) the incoming members at the top and bottom of the column stack or wall panel are identified and their properties established.

Allowing for global imperfections

These are typically represented by the application of Notional Loads \ Equivalent Horizontal Forces.

Allowing for member imperfections

These are dealt with as part of the design of members.

Wind drift

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 Options-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 excluding 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 adjusting 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 storey 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.

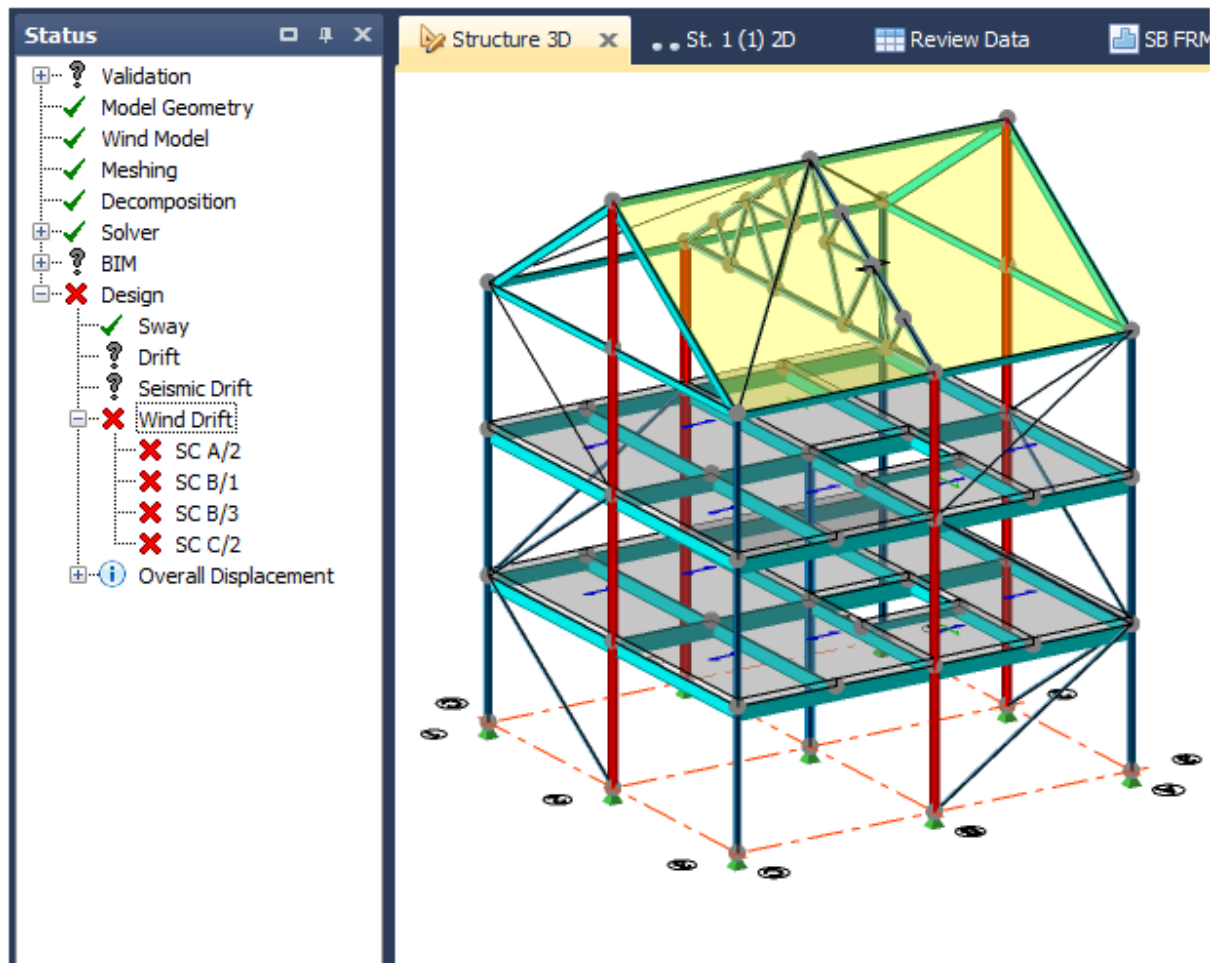


For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

Review of wind drift checks

Any stack failures are flagged in the **Design>Wind Drift** branch of the Project Workspace Status Tree.

- 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..



Wind drift failures remain highlighted in the current view until you press Esc to clear the highlight.

Full details for all columns checked are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the **Analysis>Wind Drift** chapter to your model report.

The shape of the column wind drift displacements can be viewed graphically in the Results View using the Sway Drift and Storey Shear Ribbon group **Wind Drift X** and **Wind Drift Y** buttons when a Wind Load case is selected.

[Sway checks](#)

•

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.

Analysis Handbook

This handbook mainly focuses on the solver models and associated solver elements created by the different types of analysis.

Analysis Limitations and Assumptions

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 vibration, 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 storey 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 behaviour 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:

$$1 / G I_T * \int T(x)$$

Where:

I_T = torsion constant

G = shear modulus of steel

$T(x)$ = function of torsion moment

Vibration analysis - active mass

In a 1st order vibration 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 **Vibration frequencies** table) for all modes for each direction. Design codes stipulate this should be $\geq 90\%$ for seismic analysis usually for two orthogonal lateral directions.

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
				$\Sigma=100$	$\Sigma=100$	$\Sigma=100$			

Vibration analysis - modal mass

After running a 1st order vibration analysis, modal masses for each mode are available in the Vibration 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 vibration 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-016:¹

$$\hat{m}_j = \sum_{i=1}^N \mu_{j,i}^2 m_i$$

where i is each on N points on the structure, having mass m_i at which the mode shape $u_{j,i}$ 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* Vibration Analysis):

$$\Phi_2 = \frac{1}{\sqrt{M_i}} \times 1$$

Hence:

$$\text{Modal mass} = M_i = \frac{1}{\Phi_2^2}$$

1. MPA The Concrete Centre. A Design Guide for Footfall Induced Vibration of Structures CCIP-016 2007

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.


Solver models

If you have performed more than one analysis type on the structure, then (providing the geometry and loading have not changed between runs), each solver model and set of results is retained. You can show results for each analysis type without having to re-analyse.

Changes to mesh size or uniformity do not constitute a change in the geometry. Hence, if different meshes have been applied for each analysis, you can review the different solver models by opening a **Solver View** and then choosing the model required from the right-click menu.

Solver Model Components

The different analysis objects that can be present in a solver model are listed in the table below.

Object	Description
Solver element	<p>A 1D analysis object created between two solver nodes.</p> <ul style="list-style-type: none"> • Releases are applied at the end of a solver element where these have been specified in the physical model • Rigid offsets in concrete beams and columns are applied at the ends of concrete member solver elements where required in order to make connections to other solver elements. • Rigid zones in concrete beams and columns are optionally applied at the ends of solver elements to more accurately model the zone where two concrete members connect. <p> Two solver elements are created from each physical member (column stack, or beam span) so that a solver node exists at mid span/stack for the p-delta analysis.</p>

Solver element 2D	Mesheres of 2D finite elements are created in the solver models where they have been specified for concrete walls and 2 way spanning slabs.
Solver node	<p>Solver nodes are created at defined points in a solver model on the basis of solver element and finite element connectivity.</p> <p>Solver nodes are created at:</p> <ul style="list-style-type: none"> • The ends of solver elements • The corners of finite elements
Rigid diaphragm	By default these are automatically created in one way and two way slabs - in this type of diaphragm all the solver nodes in the plane of the diaphragm are constrained to move together in the plane of the diaphragm.
Semi-rigid diaphragm	Optionally created from in one way and two way slabs - in this type of diaphragm all the solver nodes in the plane of the diaphragm are seed nodes of a quadrilateral/triangular finite element mesh.

Working Solver Model

The **Working Solver Model** shows the model in its form prior to any analysis.

Although 1D elements and solver nodes 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

This solver model is in the form of a [3D Analysis model](#).

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order linear analysis.

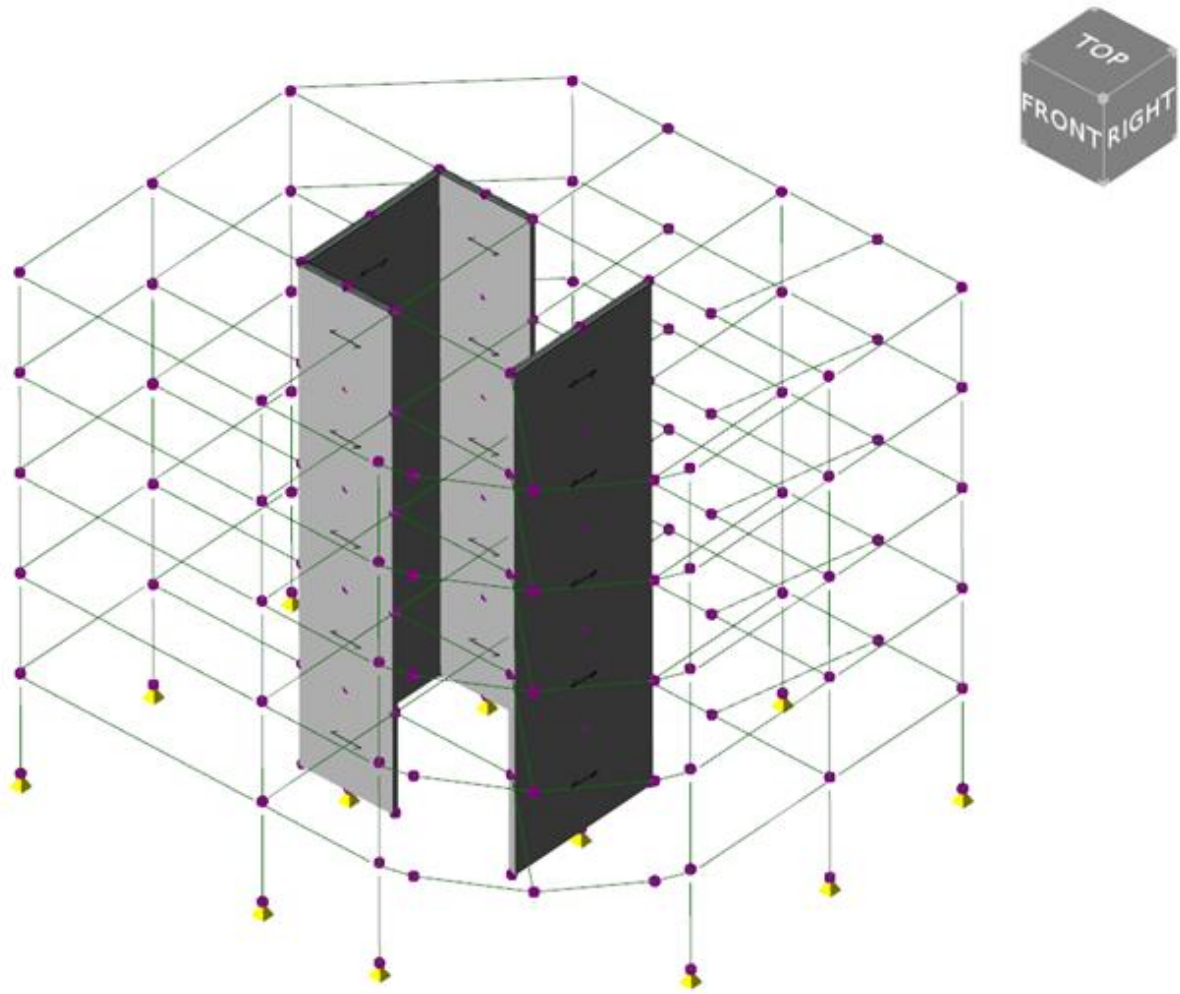
If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.



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.

3D Analysis model



The 3D analysis model consists a mix of 1D analysis elements and FE meshes 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 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 - see: [Why slab load decomposition is required](#)
- 2-way slabs (optionally) can be meshed
 - Recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
- supports are user defined

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

This solver model is in the form of a [3D Analysis model](#).

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order non-linear 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.



*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 Linear

This solver model is in the form of a [3D Analysis model](#).

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 2nd order linear 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.



*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 Non Linear

This solver model is in the form of a [3D Analysis model](#).

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 2nd order non-linear 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.



*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 Vibration

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order vibration 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.



*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 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.

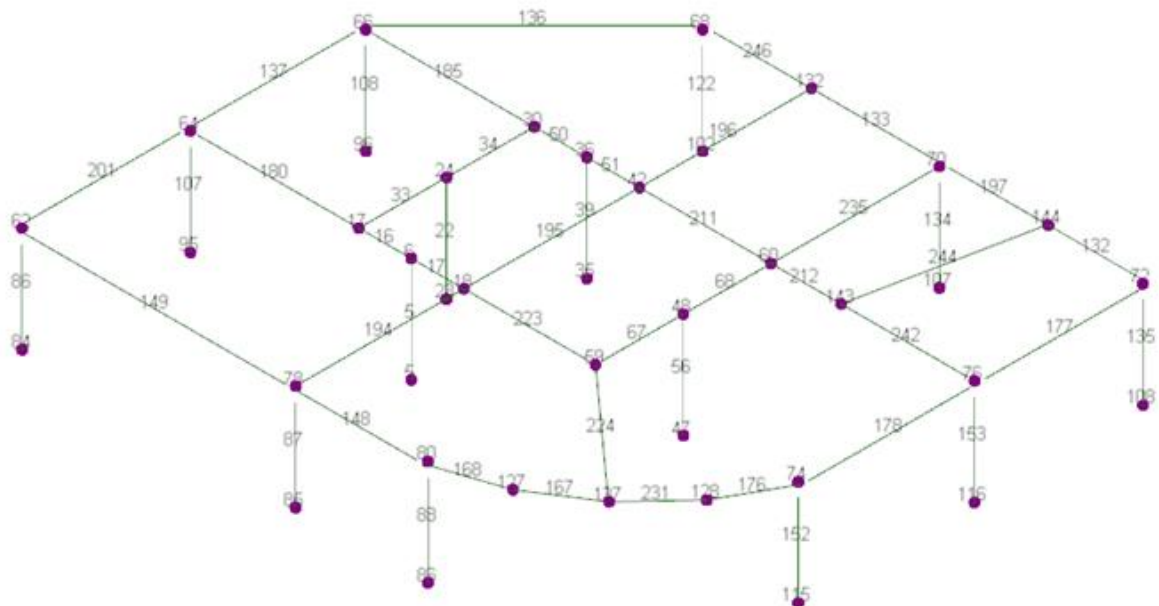


Results are still displayed for the “old” solver model until the working solver model is updated to reflect the changes (by running an analysis).

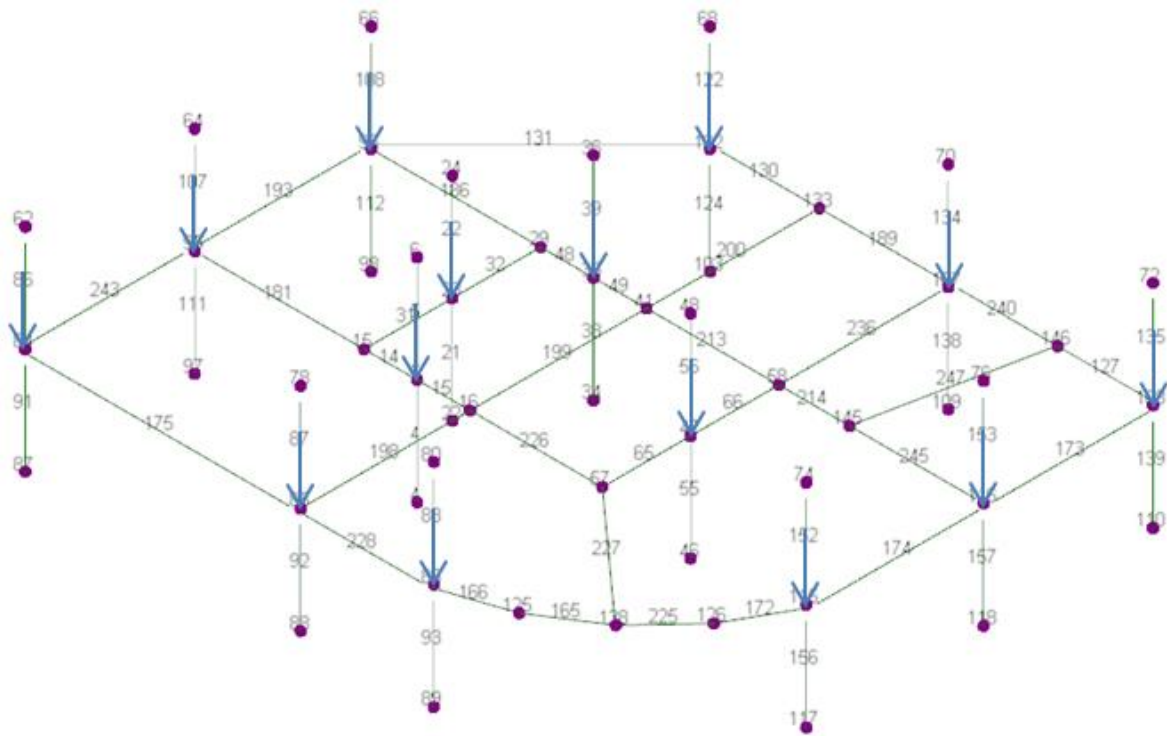
Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

In grillage chasedown a 3D sub model is formed for each floor and the columns connected to it.



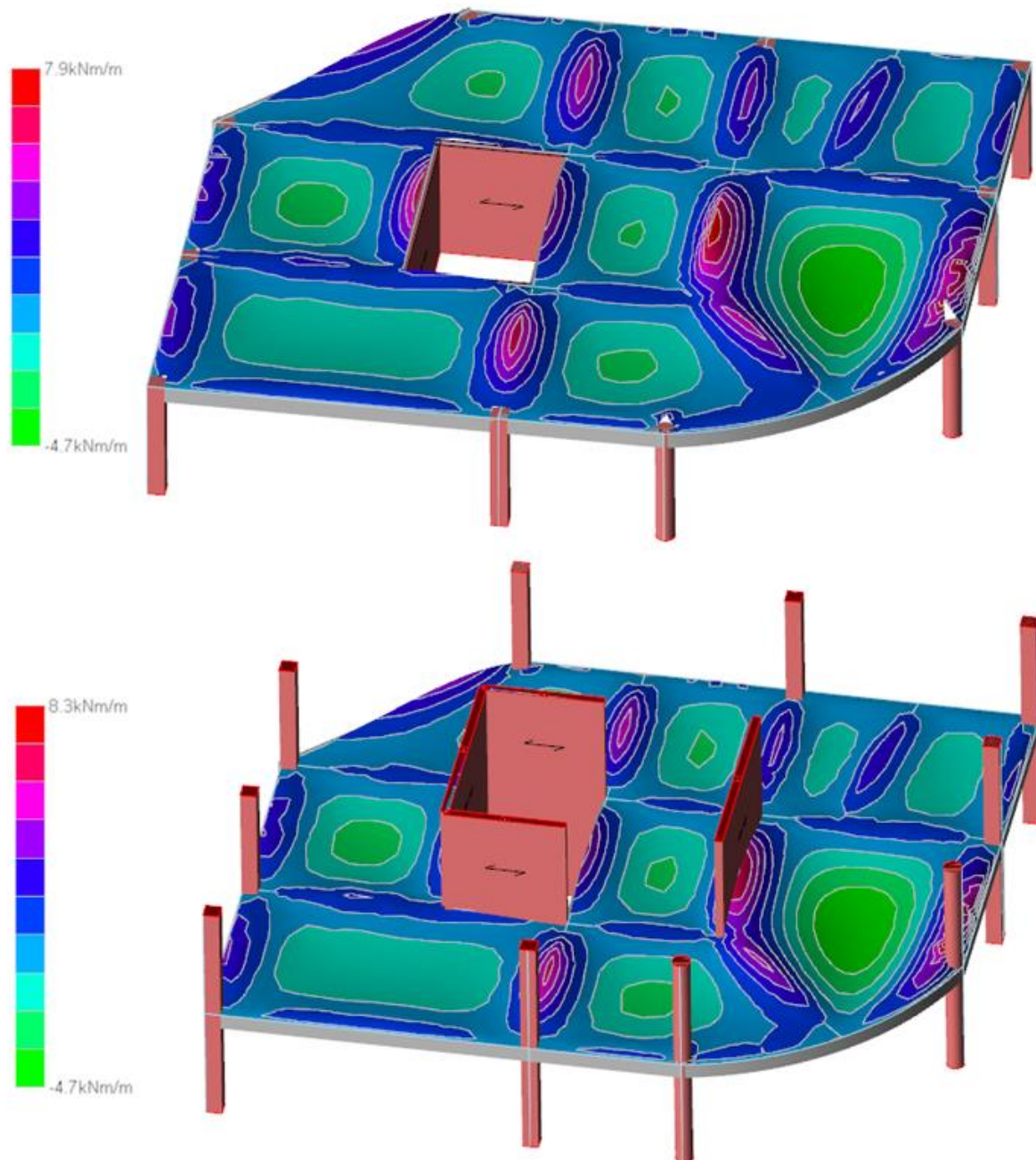
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.



Two-way slabs are only meshed in grillage sub-models at those levels where they have been set as meshed for 3D Analysis. For all other slabs [Why slab load decomposition is required](#) is carried out prior to the analysis.

Solver Model used for FE Chasedown

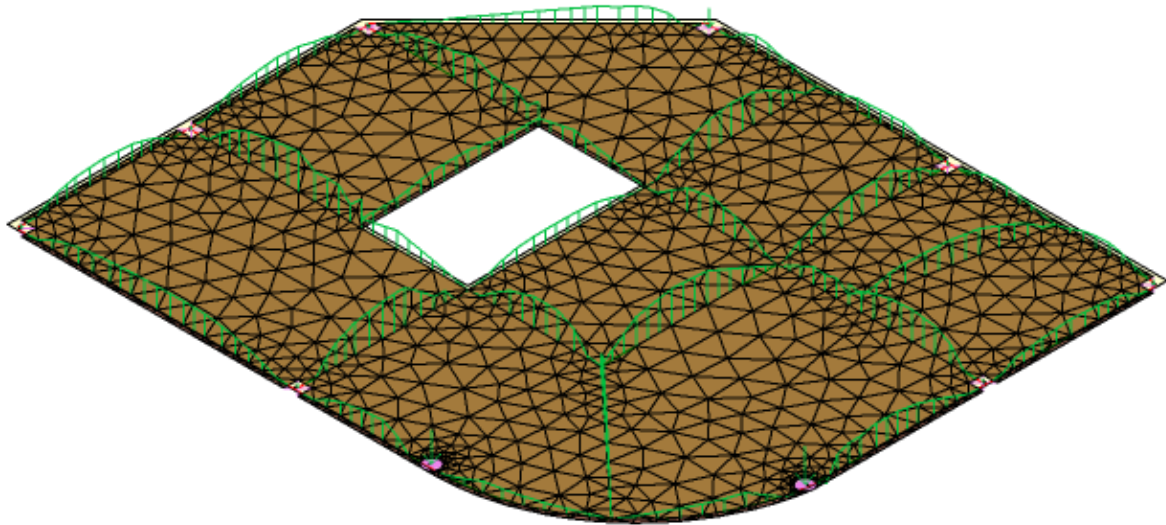
FE chasedown is similar to grillage chasedown, with 3D sub models being formed at each level; the one difference being that in the FE chasedown the two-way slabs are always meshed.



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

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.

Solver elements for 1D members

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.

Truss

An element in any material, with user-defined cross sectional area, and pinned ends (releases not being editable).

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.

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.

Linear axial spring

An element that deflects linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness.

Linear torsional spring

An element that rotates linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness.

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.

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.

Solver element types applied to members in the different solver models

Solver elements are applied to members as follows:

Beam 1D solver elements are 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 1D solver elements are 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 1D solver elements are 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 1D solver elements are 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 1D solver elements are used to model:

- Analysis Element (any material) with element type: Linear axial spring

Linear torsional spring 1D solver elements are used to model:

- Analysis Element (any material) with element type: Linear torsional spring

Non-linear axial spring 1D solver elements are used in non-linear solver models only for:

- Analysis Element (any material) with element type: Non-linear axial spring

Non-linear torsional spring 1D solver elements are in non-linear solver models only for:

- Analysis Element (any material) with element type: Non-linear torsional spring

Member connectivity

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 are then automatically introduced where necessary to connect the solver elements.



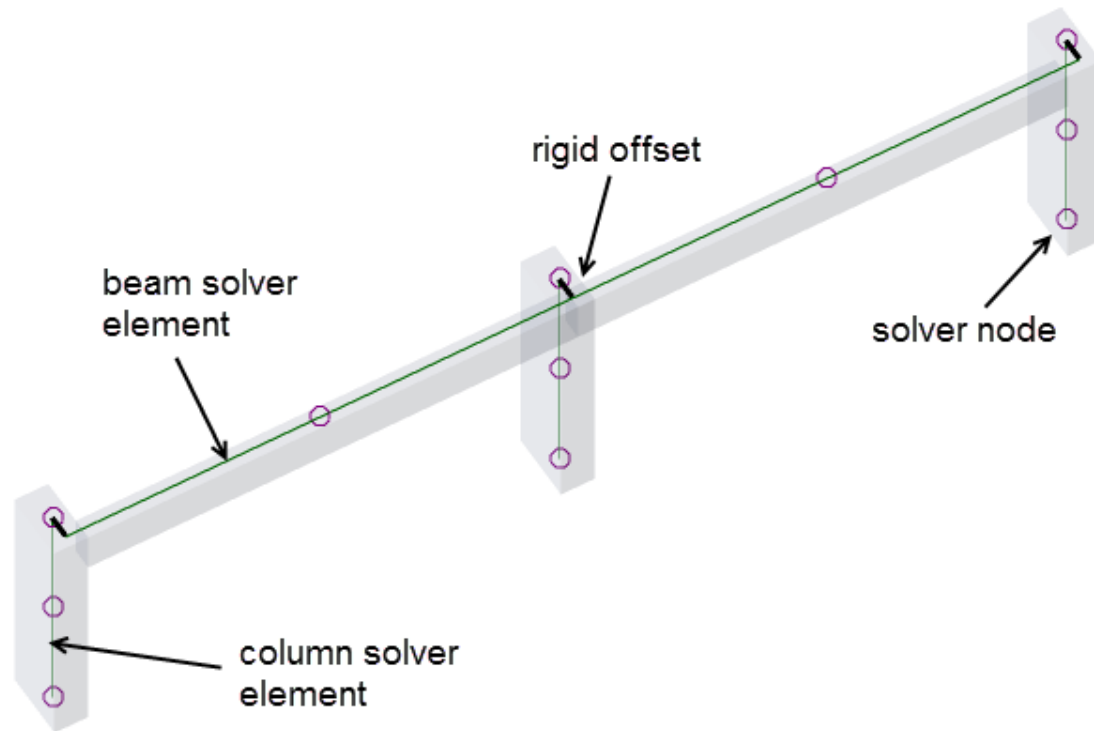
Note that the rules applied to insertion of solver elements for concrete columns are different to those that are applied to concrete beams.

For concrete structures this enables you to simplify the grid layout but then employ offsets to position the members exactly.

Rigid offsets in concrete beams and columns

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.

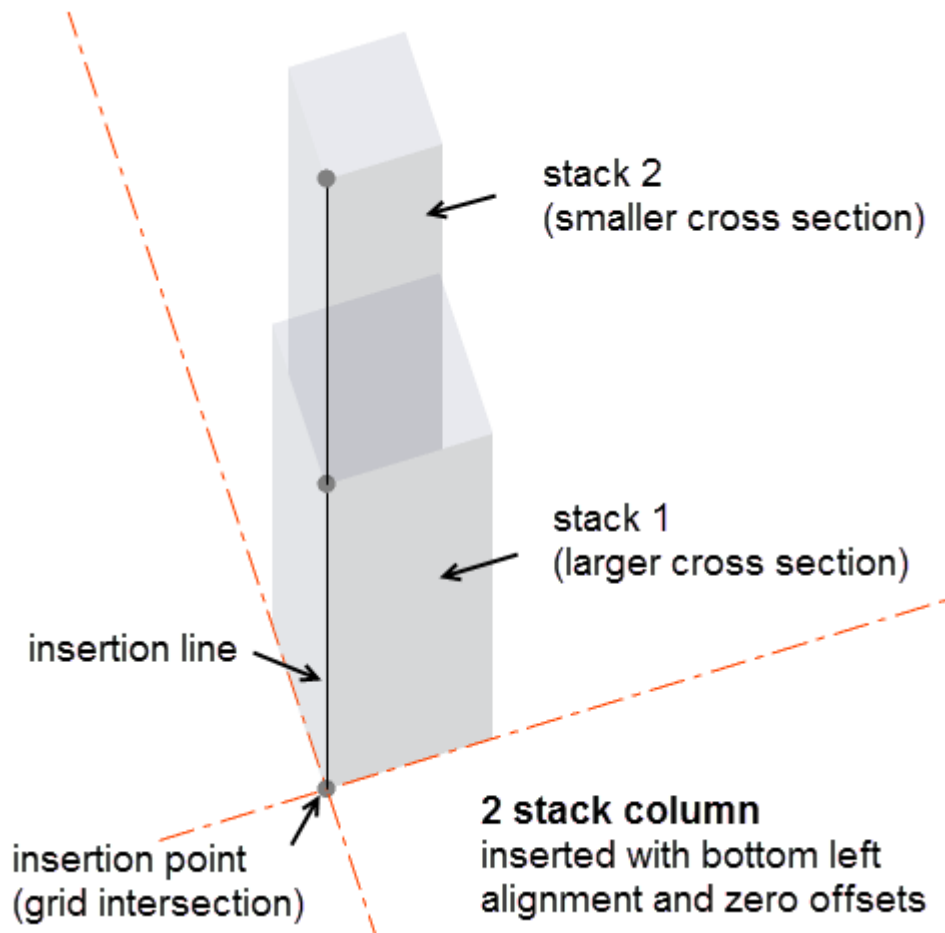
This will be necessary whenever the 1D solver elements are not co-linear. A typical example of this occurs when concrete edge beams are aligned to be flush with the face of the supporting columns, as shown below:



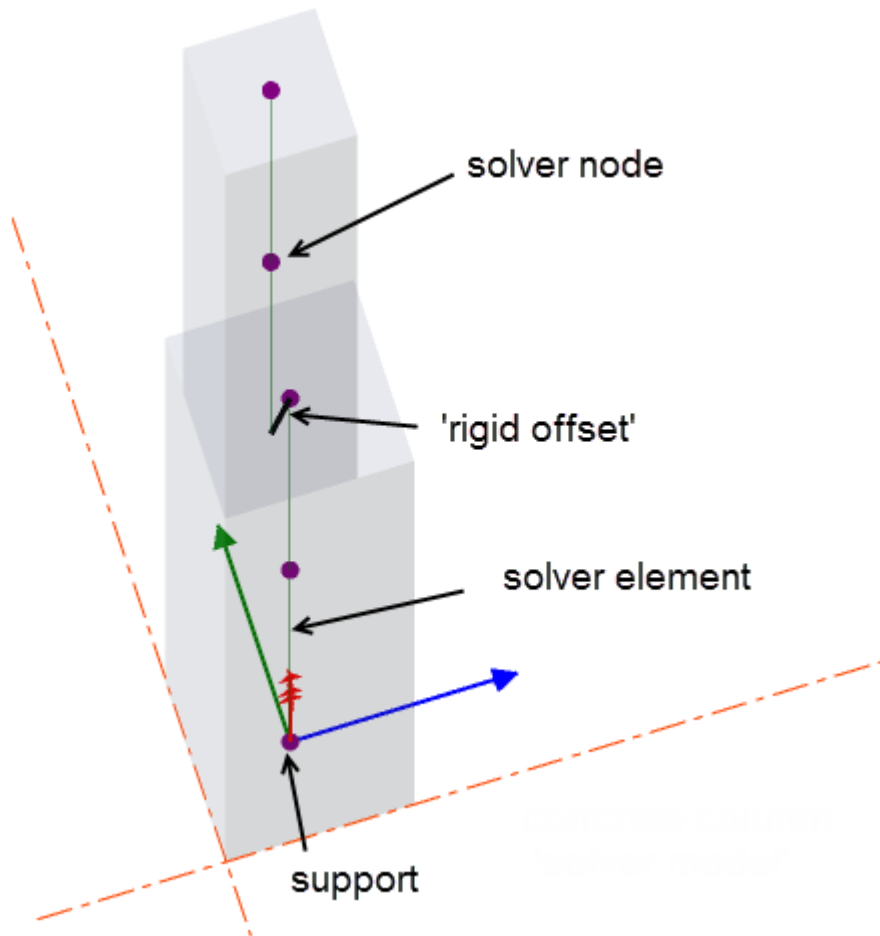
*To see rigid offsets: open a Solver View, and then in Scene Content ensure that **1D Elements > RigidOffsets** is selected.*

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.

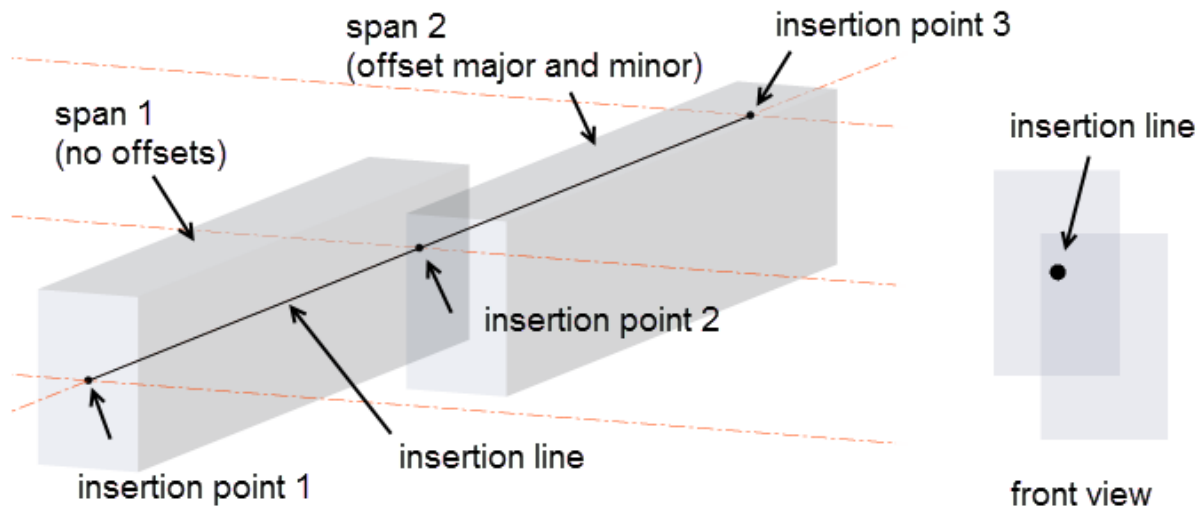


To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset and Solver Nodes> Geometry**.

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.



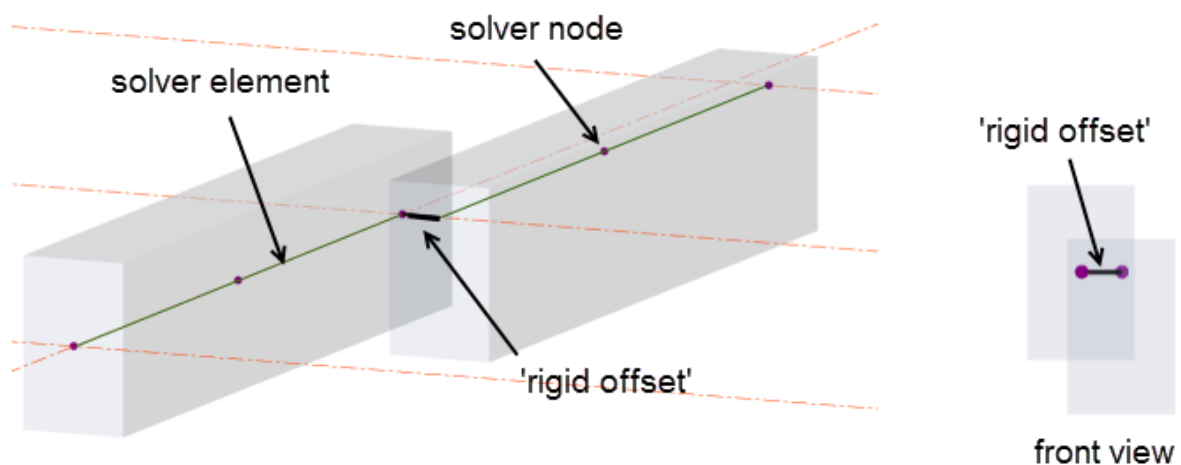
For concrete beams:

- 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 centre 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.





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 in concrete beams and columns

Design codes allow engineers to assume parts of concrete beams / columns are rigid, leading to more efficient designs.

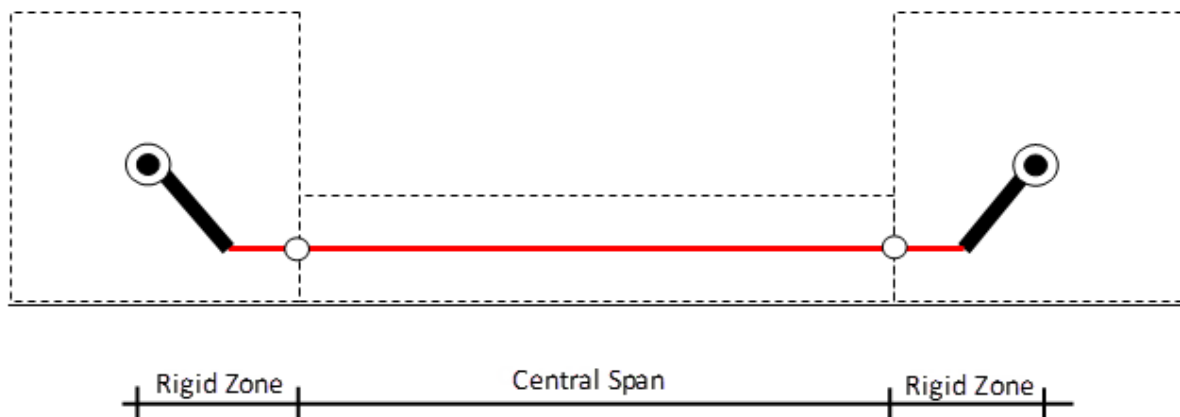
Tekla Structural Designer uses **Rigid Zones** to cater for this where columns and beams are connected and where beams are connected to other beams.



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.

Application of 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.

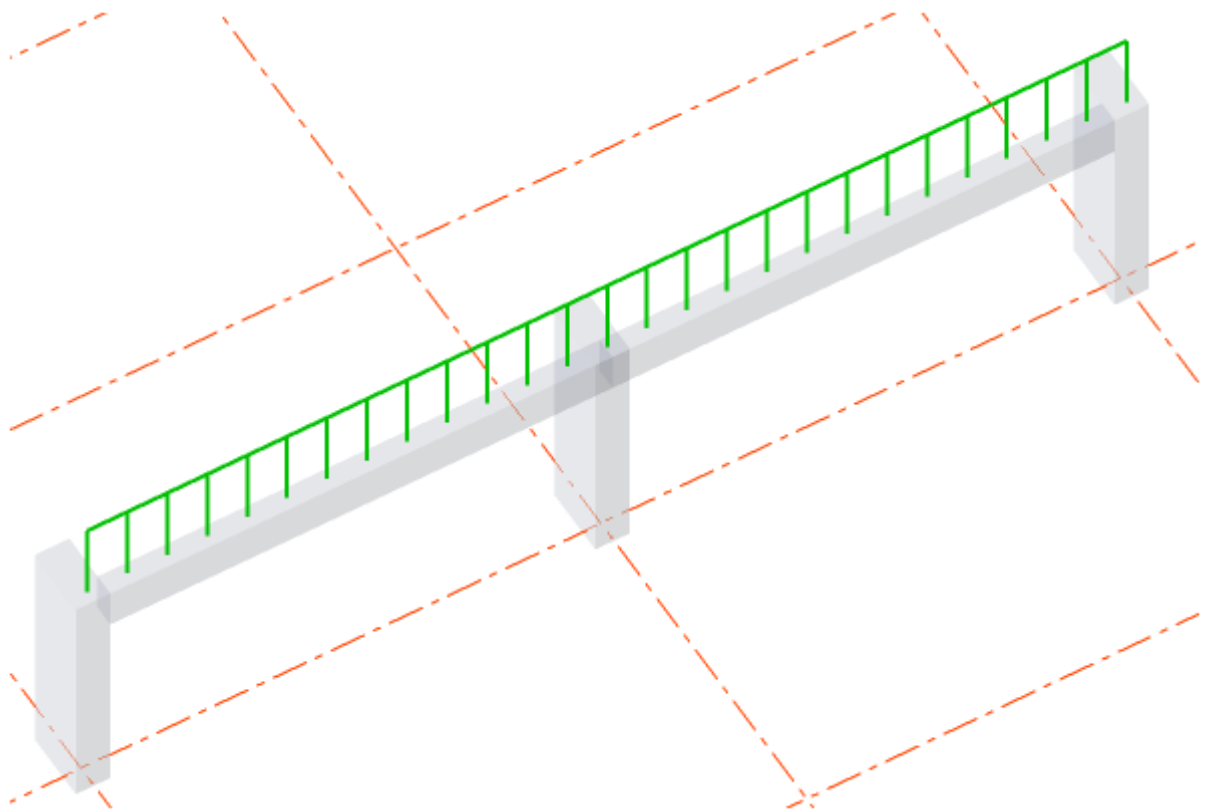
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

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.

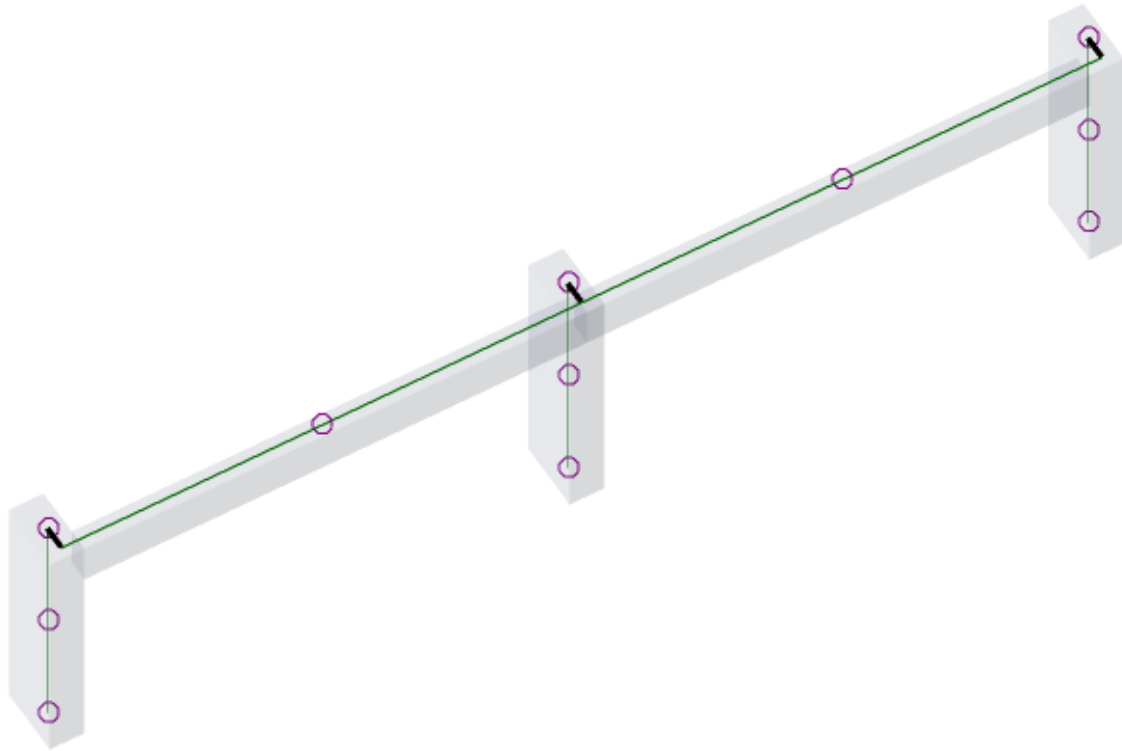
Rigid zones example 1 - fixed ended beam

Consider the following 2 span beam supported on columns and loaded with a udl:

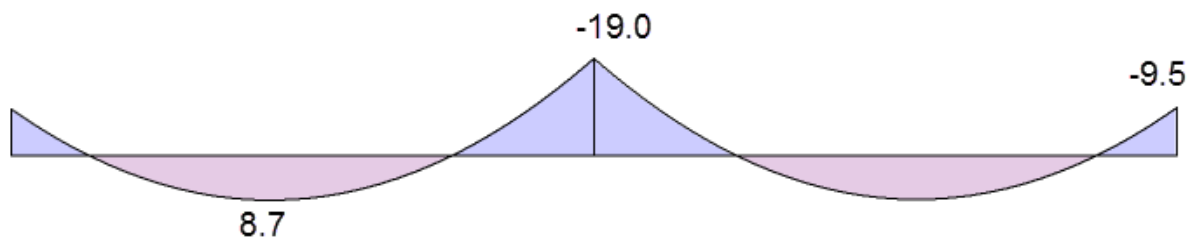


Rigid Zones Not Applied

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.



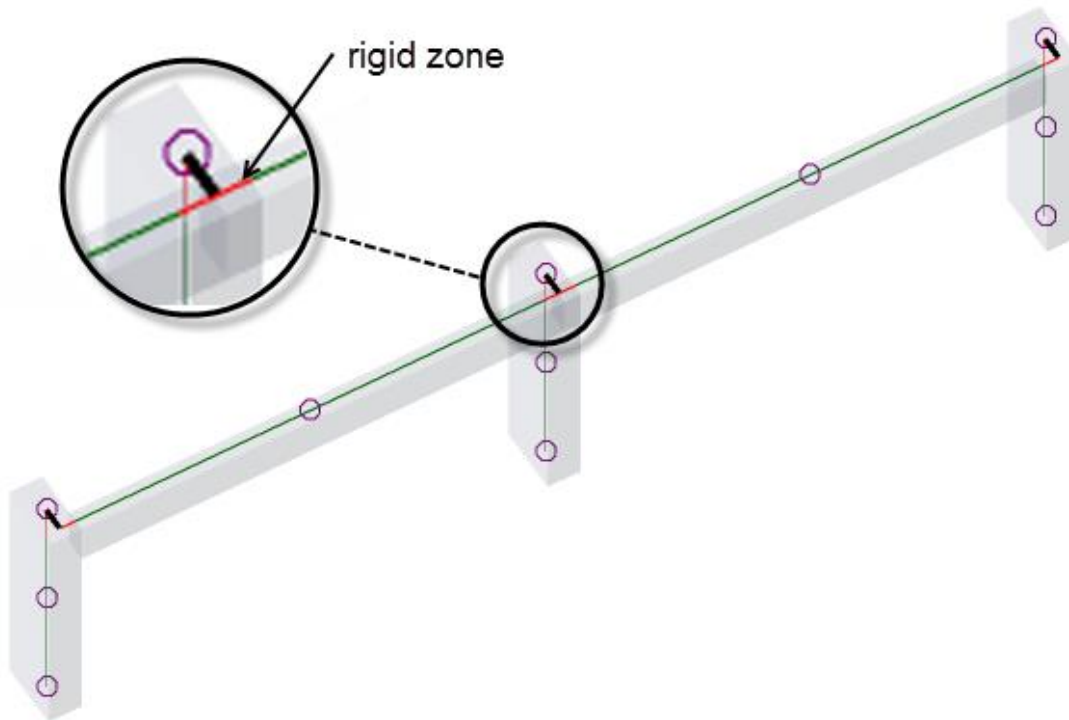
The resulting beam bending moment diagram is as follows:



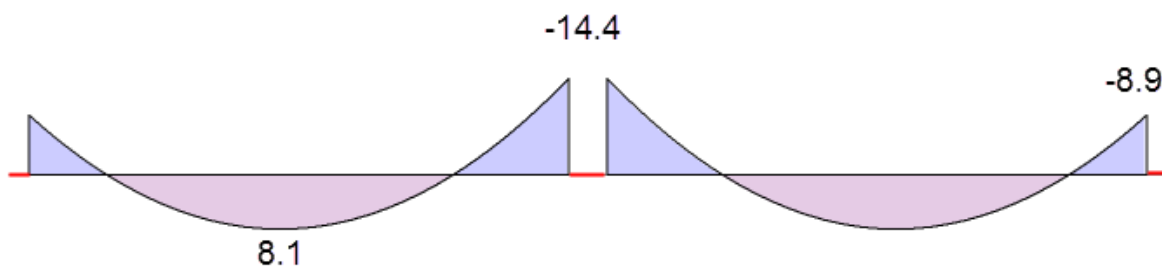
Rigid Zones Applied (default 50%)

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.

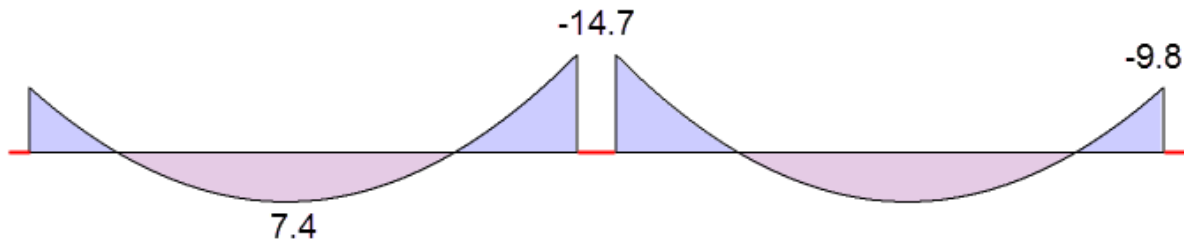


The above diagram was generated from a Results View to illustrate that a “gap” is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

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%)

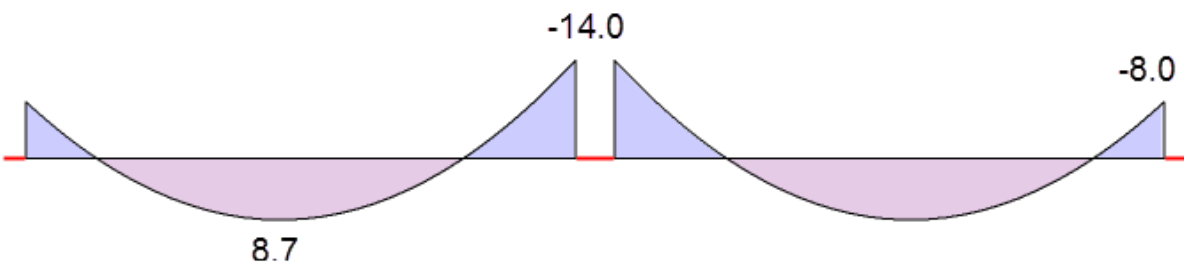
To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:



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).

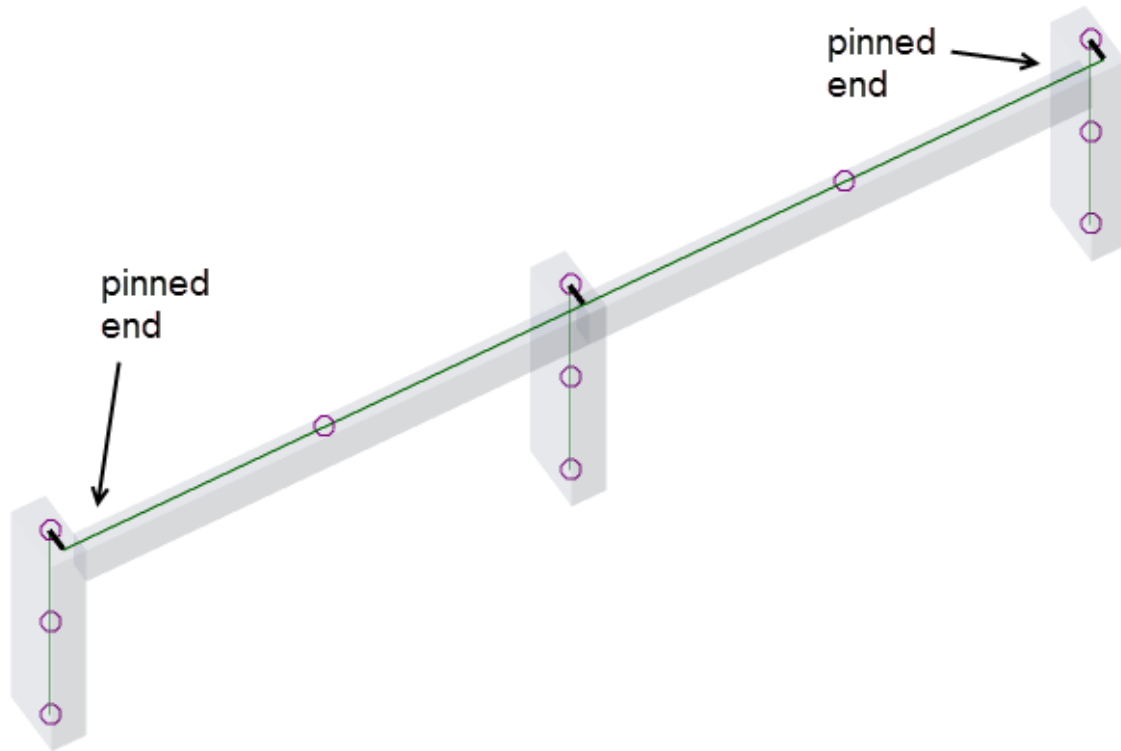
Rigid zones example 2 - pin ended beam

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.

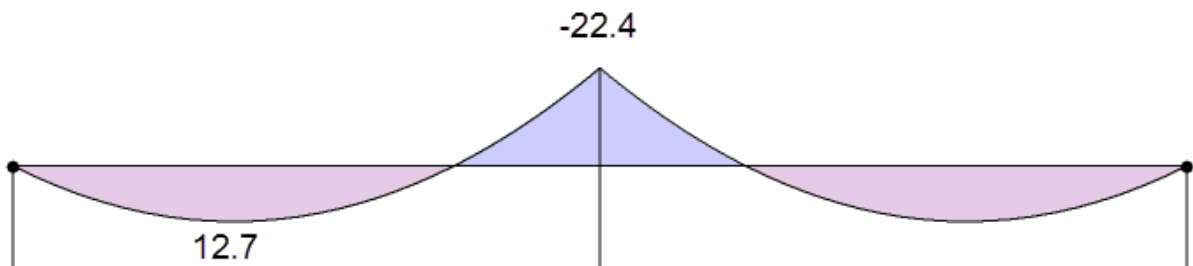
To illustrate this effect the model illustrated in [Rigid zones example 1 - fixed ended beam](#) is modified to have pinned connections introduced at the two remote ends of the beam.

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.



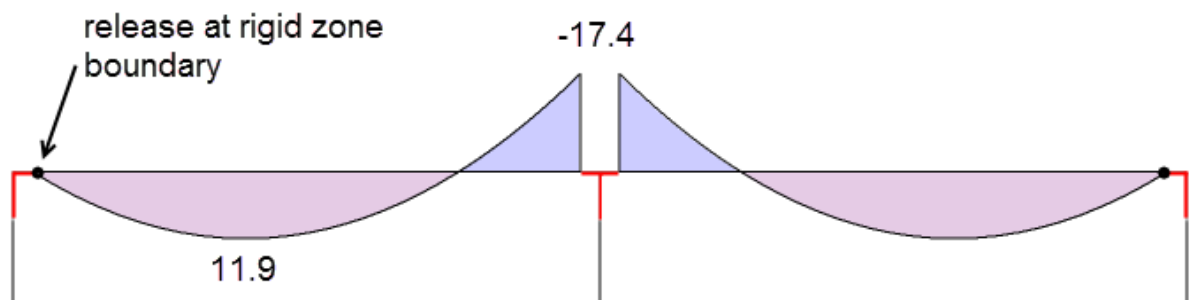
The beam bending moment diagram is as follows:



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.



Inactive solver elements

You do not need to delete a brace or an analysis element in order to investigate the effect of it being taken out of the solver model. Instead you can uncheck the **Active** setting in its member properties.



Only braces and analysis elements can be set to be inactive in this way.

Solver elements for concrete shear walls

Concrete shear walls can either be specified as meshed or mid-pier. Meshed concrete shear walls are defined as quadrilaterals in a single plane that can be vertical or sloping. Mid-pier concrete shear walls must be rectangular in a vertical plane.

Openings are only valid if defined for meshed concrete shear walls.

For both meshed and mid-pier concrete shear walls the alignment and offsets that are 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.

Meshed concrete shear wall geometry

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.

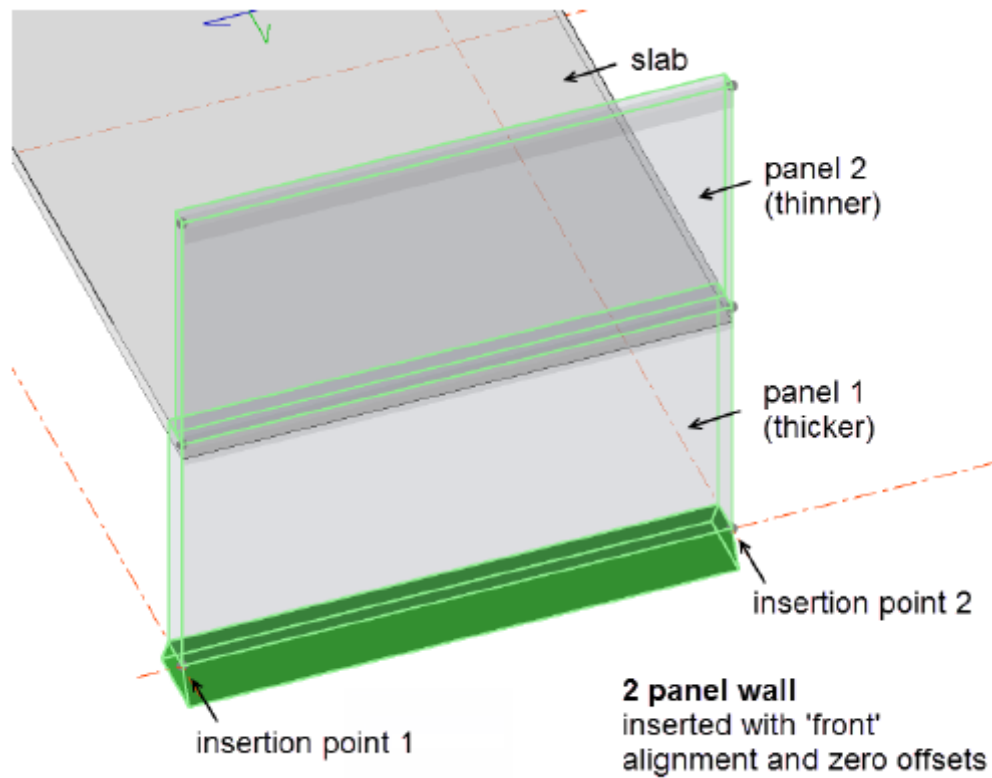
Wall beam elements can also be created where certain properties, (e.g. thickness), differ across panel boundaries.

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](#).

Meshed concrete shear wall example

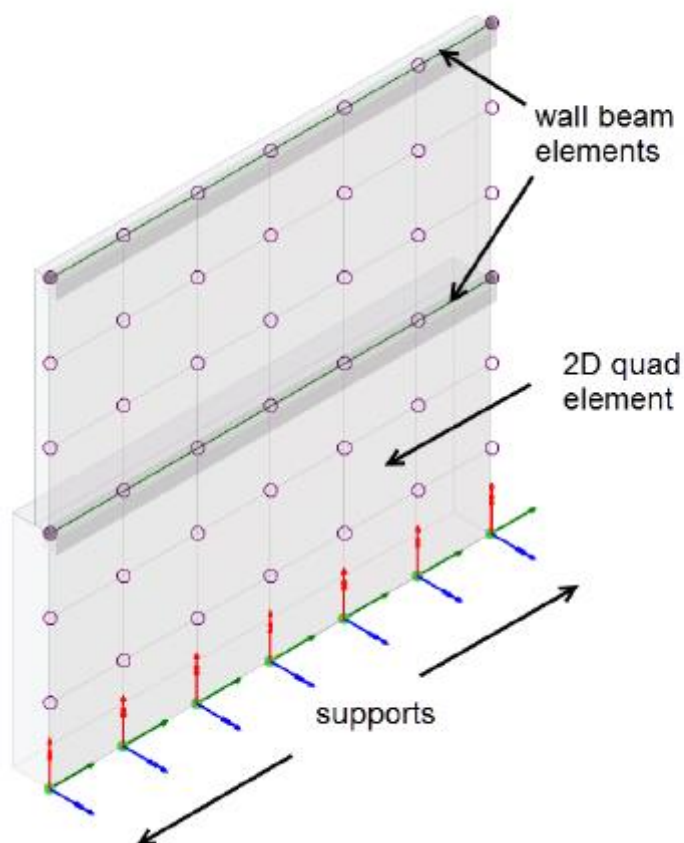
Consider the following two stack wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



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

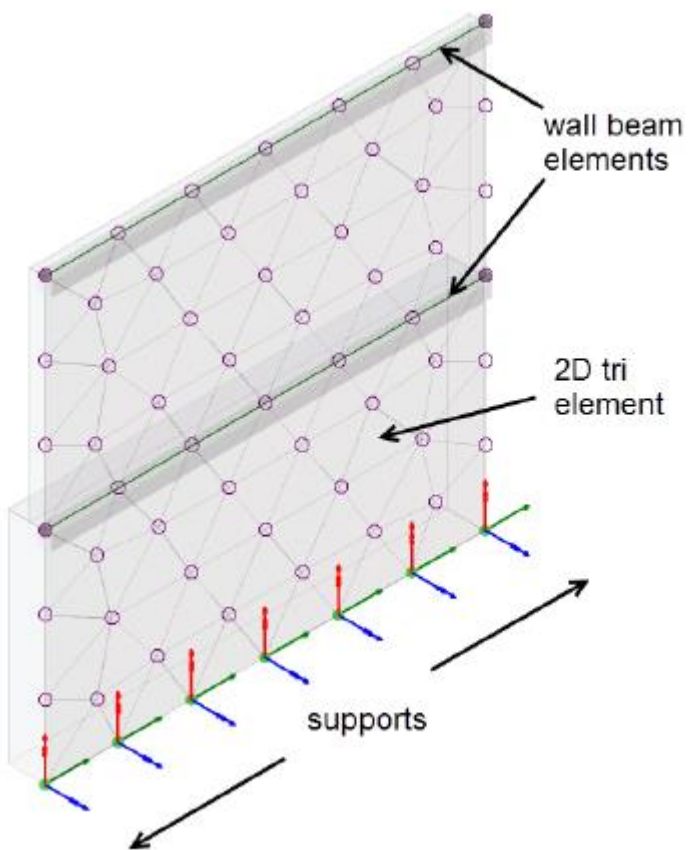
In this two stack example, when the Wall Mesh Type is set to Quad only, solver elements are formed as shown below:



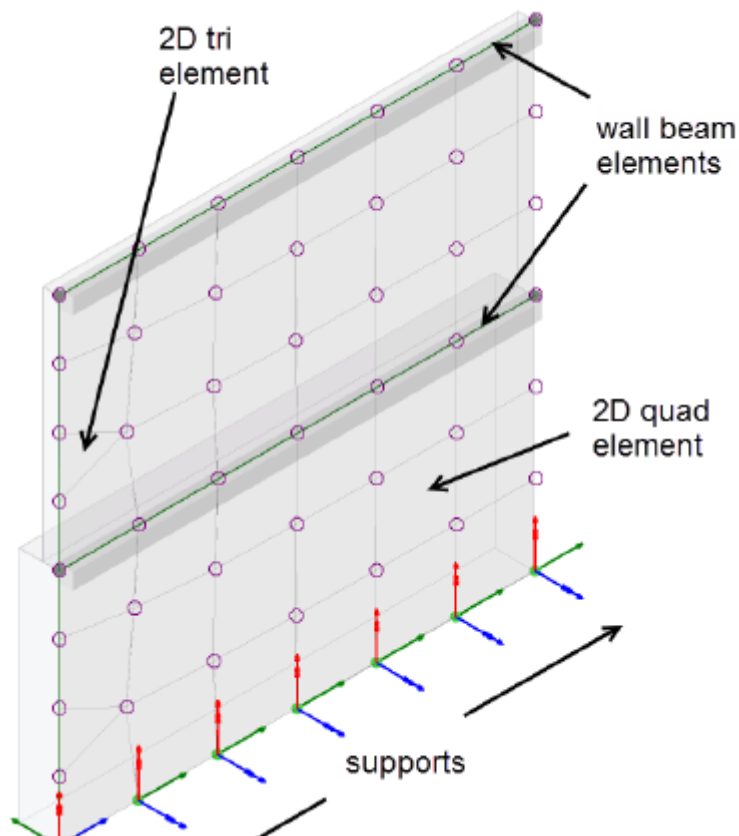
To see solver elements, solver nodes and 2D elements: open a Solver View, and then in Scene Content check **1D Elements> Geometry**, **2D Elements> Geometry** and **Solver Nodes> Geometry**.

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:



Mid-pier concrete shear wall geometry

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.

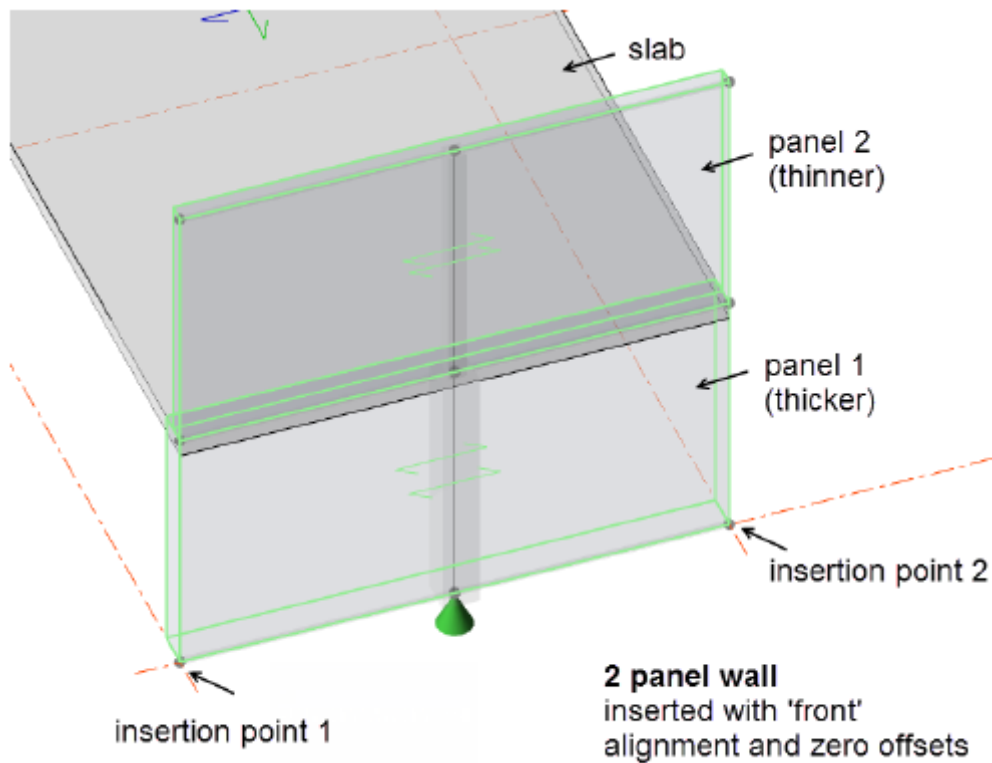


Only horizontal wall beam elements can be generated in mid-pier walls - sloping wall beam elements cannot be generated - this will be indicated by an error in validation.

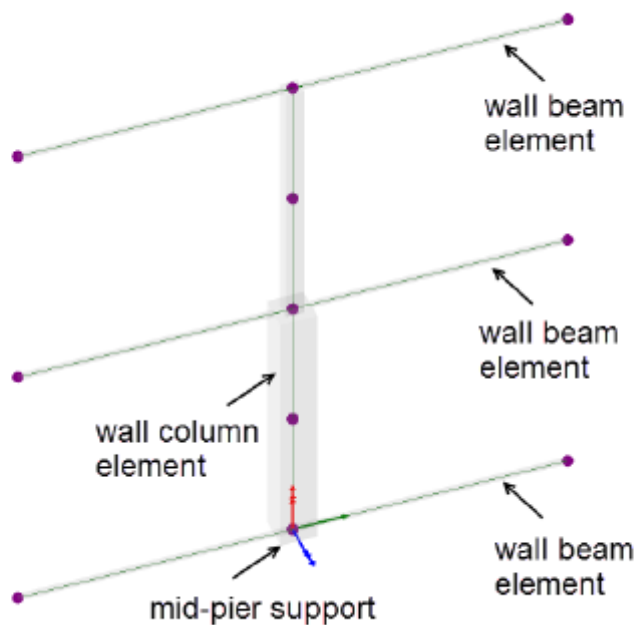
Each mid-pier wall object also has a single vertical wall column element in the middle of the wall, from the top to the bottom level.

Mid-pier concrete shear 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.



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:



To see wall beam elements, wall column elements and solver nodes: open a Solver View, and then in Scene Content select 1D Elements> Geometry and Solver Nodes> Geometry.

Sub-division of concrete shear walls

Each concrete shear 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.

Concrete shear wall openings and extensions

Concrete wall openings

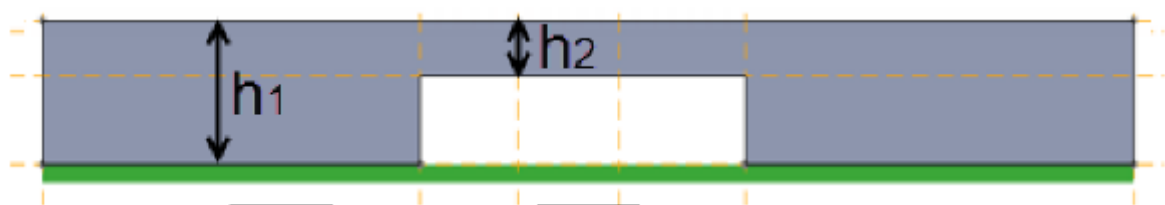
Limitations of wall openings

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. Assuming the wall has been modelled using FE elements, the analysis will still not proceed if you have applied a wind wall panel over the top of the wall. In this situation a "Panel is not surrounded by load carrying members" validation error is displayed. This error can only be cleared by deleting the openings from the affected walls.
3. Given that the analysis has been able to complete; 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](#) may prove to be a better solution.

Analysis model applied to meshed wall panels with openings

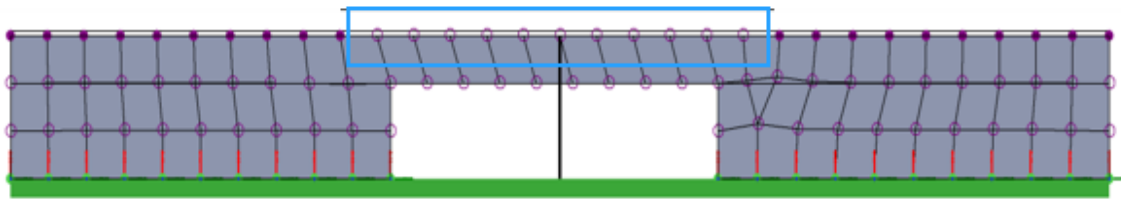
If an opening is introduced in a meshed wall, the properties of the "intel" wall beam directly above the opening are automatically adjusted in order to prevent the panel being unrealistically stiff. The adjustments that are applied are as follows:

- 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

Nodes excluded from diaphragm



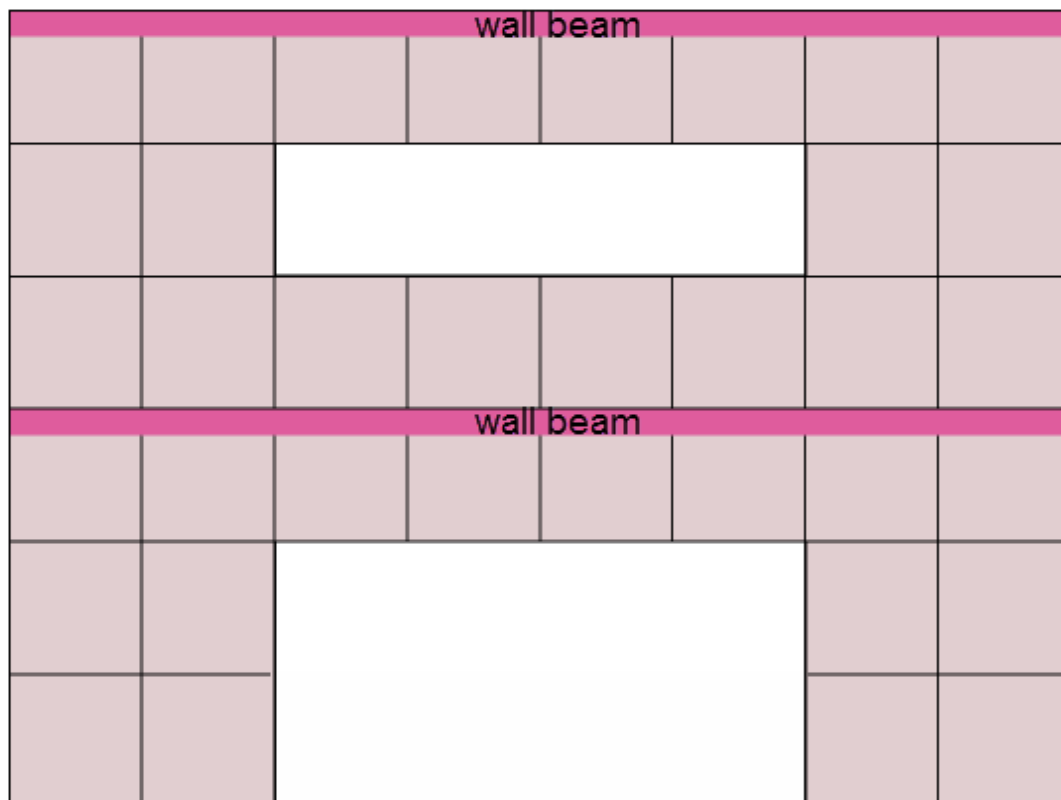
Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

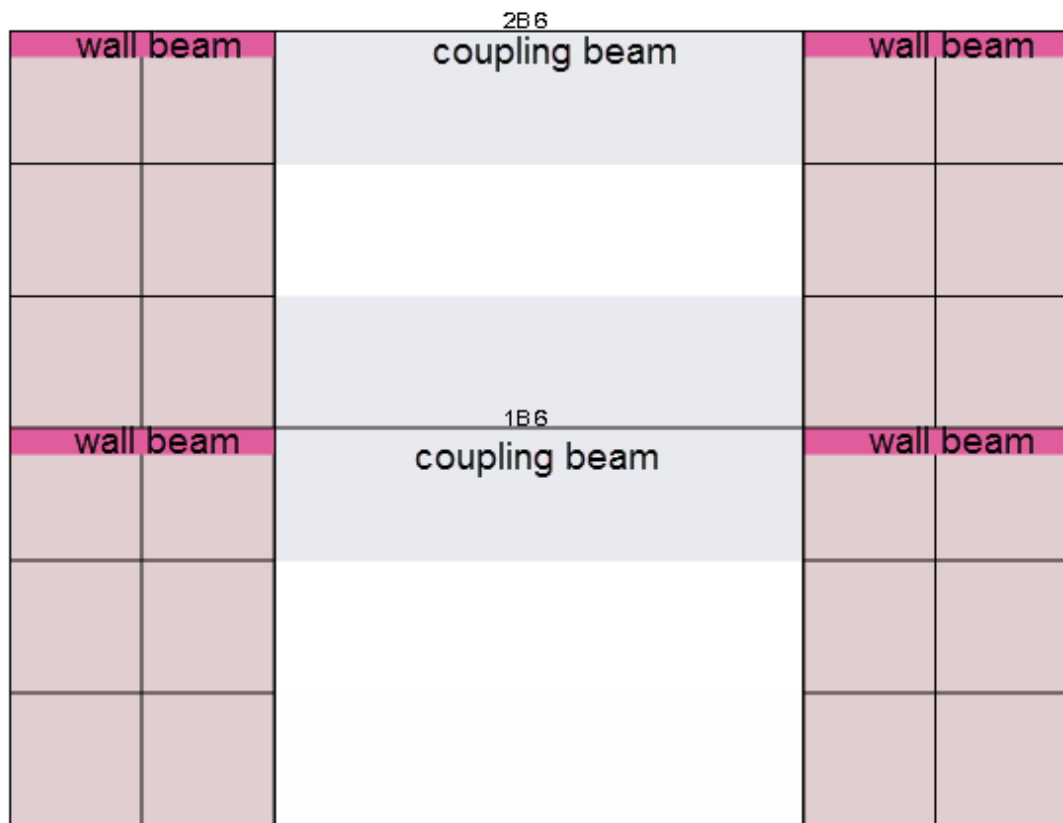
If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam ('coupling beam').

This method can be demonstrated by considering the below example, consisting of a two storey 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.

Concrete wall extensions

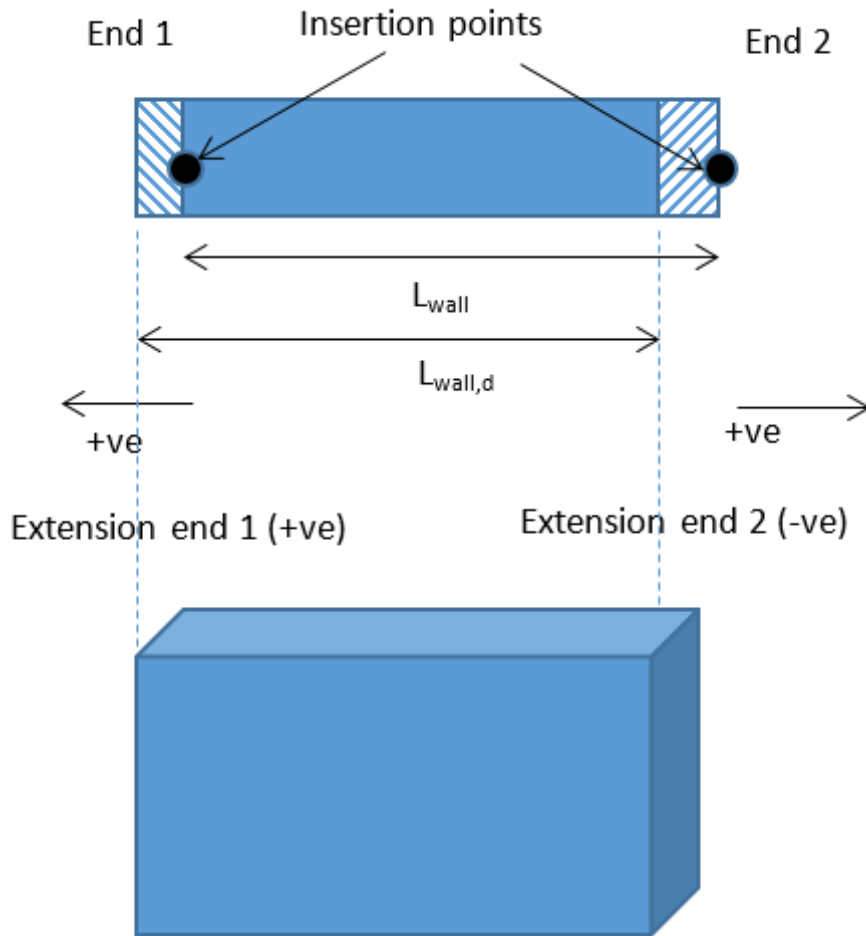
Use of concrete wall extensions

Wall extensions (End 1/End 2) are applied 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.

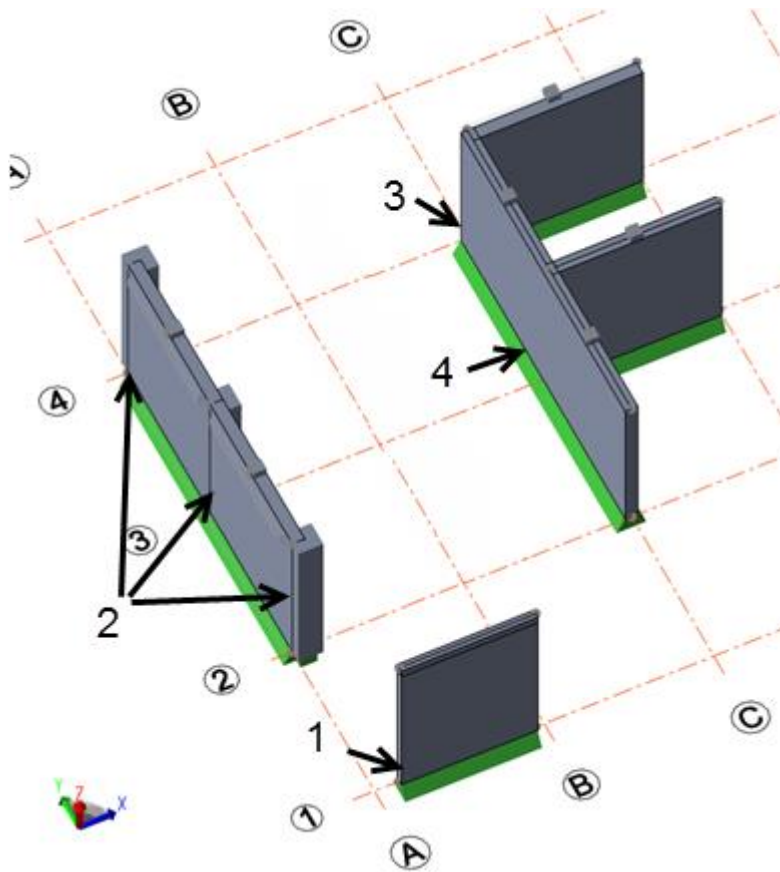
The example below shows the effect of a positive extension at end 1 and a negative extension at end 2.



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 $L_{wall,d}$

Concrete wall extension examples

The view below illustrates some examples where wall extensions can be applied.



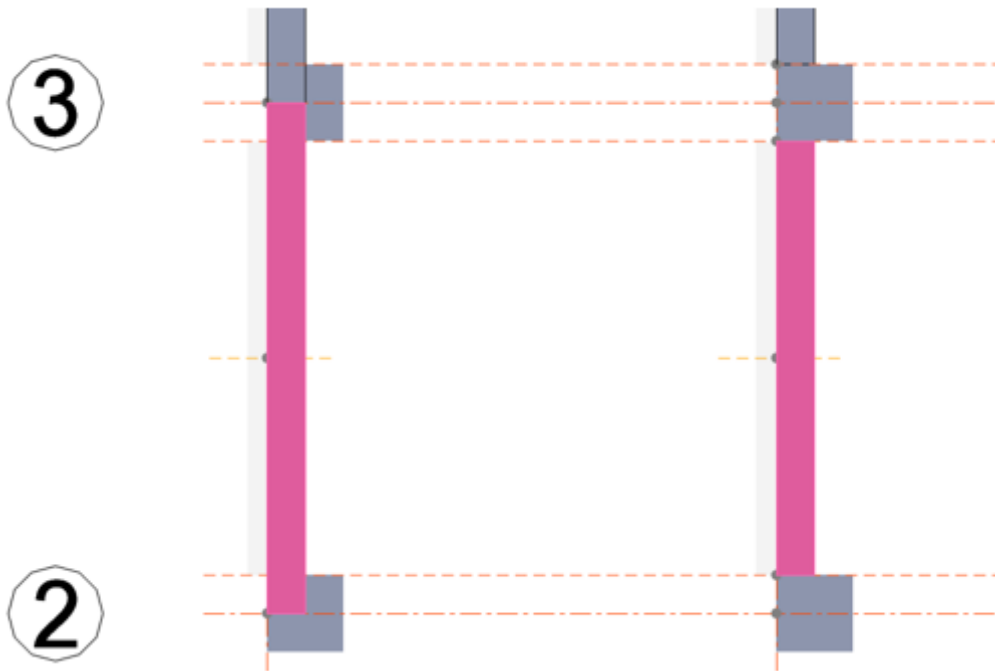
1. Where wall end does not match architectural grid - not created automatically.

Although this case could be catered for by using construction lines, it is both quicker to create and easier to edit by manually applying wall extensions.

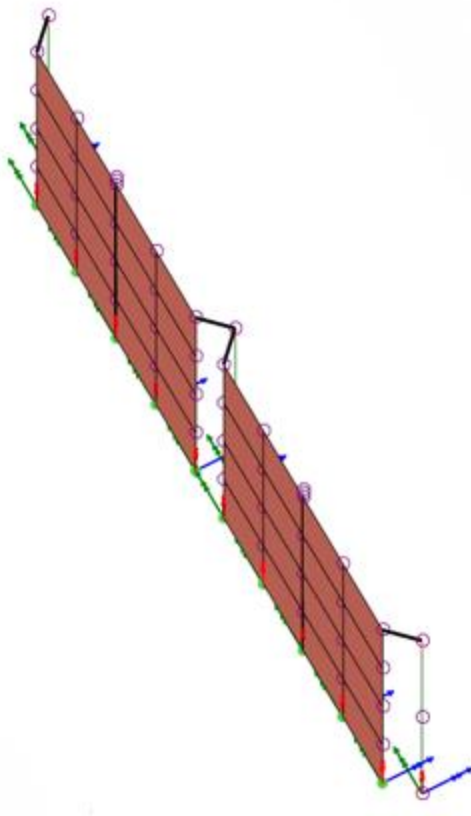
2. Where wall end overlaps a column - a negative extension can be applied automatically.
3. Where two wall ends meet - a negative extension can be applied automatically.
4. Where a wall end meets another wall part way along its length- a negative extension can be applied automatically.

Wall and column overlap

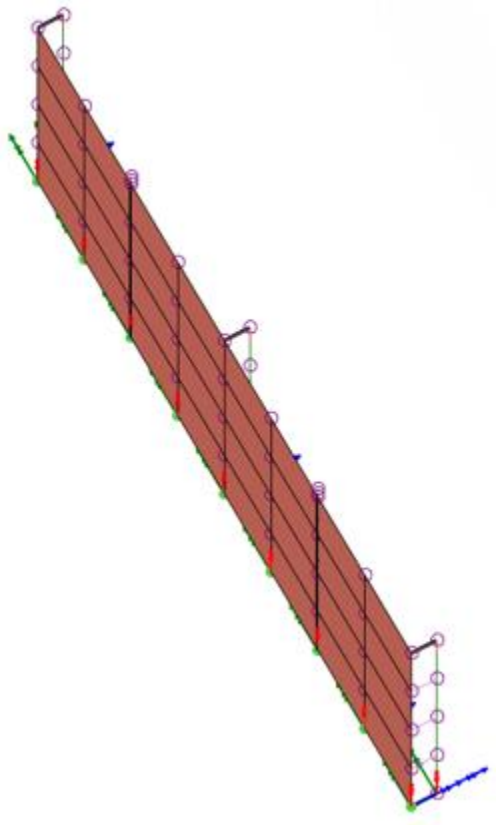
Consider case 2 above, where the wall ends overlap the columns.



If negative extensions are not automatically applied you will see an overlap of the wall with the columns (as shown on the left). Potentially you could attempt to “fix” this by creating extra construction lines or grids on the faces of the columns and then reinsert the wall between the faces. Although this looks better, the analysis model shown below is poor as the wall panels are dis-continuous and poorly connected to the columns.

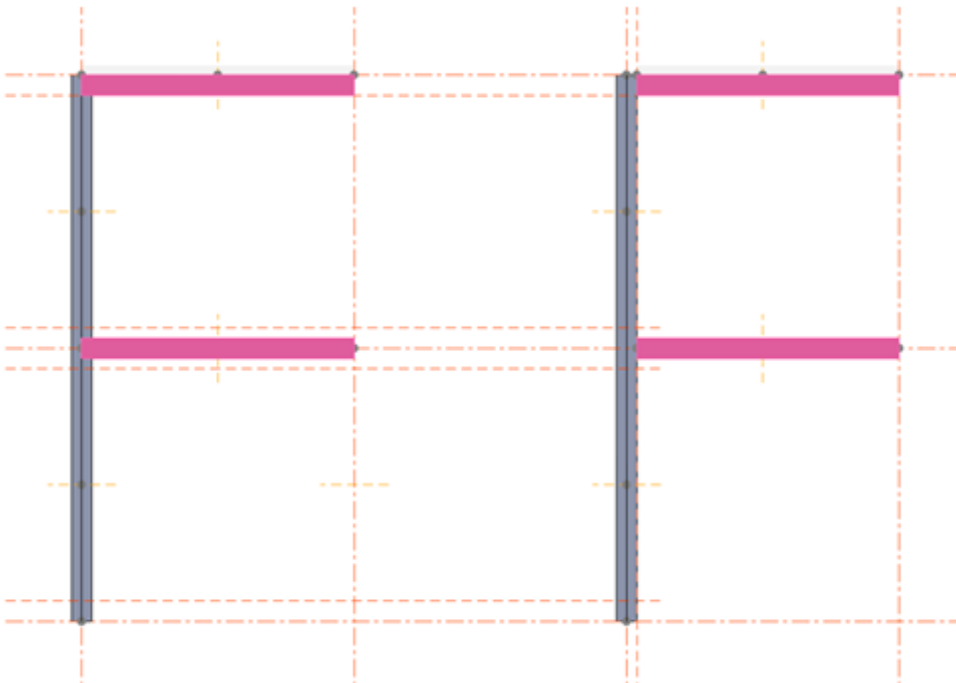


If negative wall extensions are employed instead, the analysis model is much better.



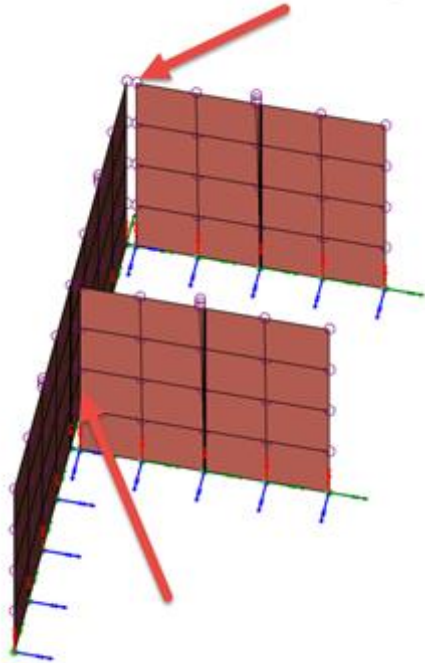
Wall overlaps with another wall

Now consider cases 3 and 4 in the case study, where two walls overlap.

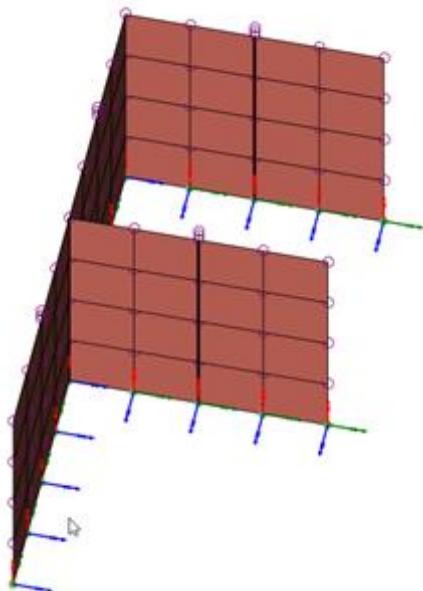


If negative extensions are not automatically applied you will see the overlap of walls (as shown above left). The main problem with this is that from a design point of view the concrete overlaps would result in a duplication of reinforcement in the overlapped areas.

Potentially you could attempt to “fix” this by creating extra construction line or grid on the right hand face of the vertical wall and then reinsert the horizontal walls to this new line (as shown above right). Although this looks better, the analysis model shown below is very poor. The wall panels are completely disconnected from each other, this model will not resist lateral load in anything like the same way.



However by once again employing negative wall extensions, the overlaps are removed from the design whilst still retaining the correct analysis model.

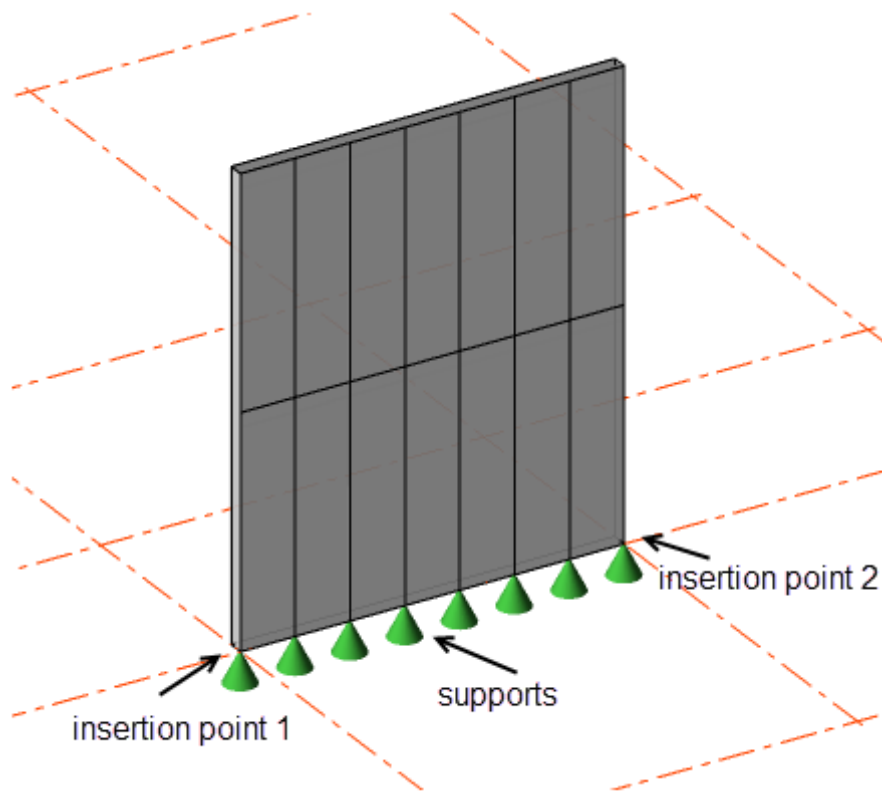


Solver elements for bearing walls

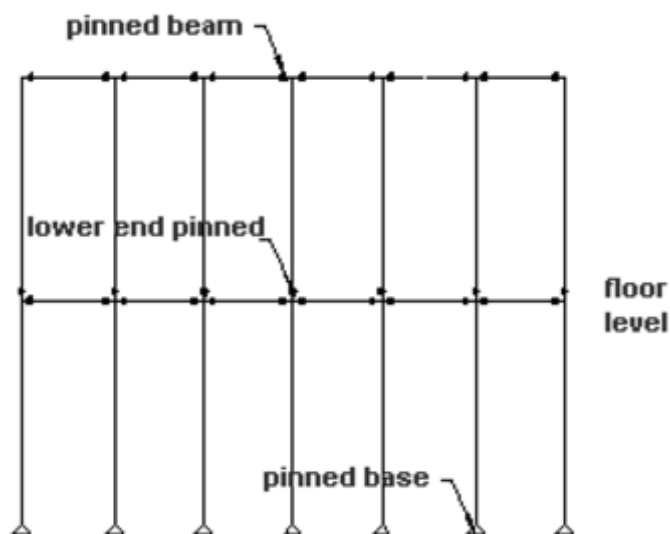
Bearing walls can only be created as rectangular in a vertical plane.

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.

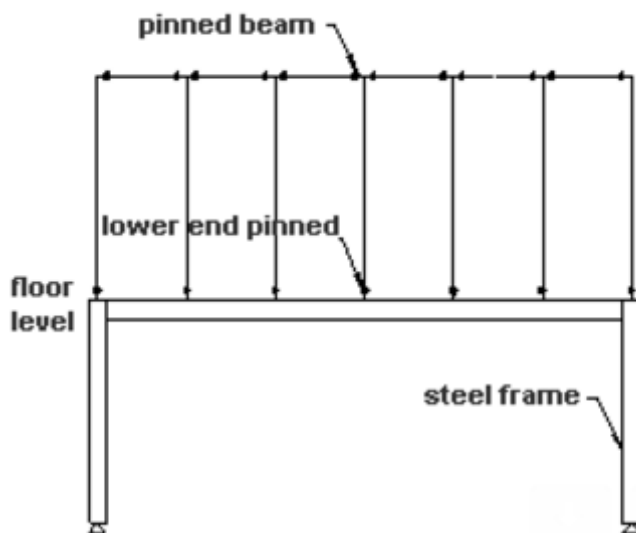


The solver model for this wall is formed using a series of vertical "wall column" and horizontal "wall beam" solver elements. The beams have pinned ends and are placed at the top of the wall spanning between the columns. The next panel above is pinned to the one below and similarly the lower end of a column is pinned to a supporting beam. At the lowest level the column is 'fixed' to a pinned support.

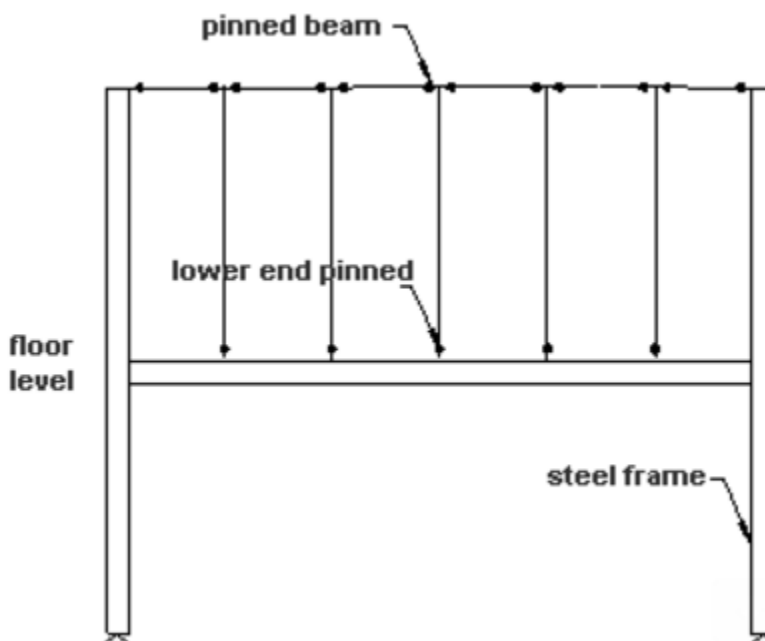


Members supported by the wall either (fortuitously) bear directly on one of the wall columns or on one of the wall beams at the head of the wall. All wall columns and wall beams in an individual bearing wall are given properties automatically by *Tekla Structural Designer*, based on the width of the 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.

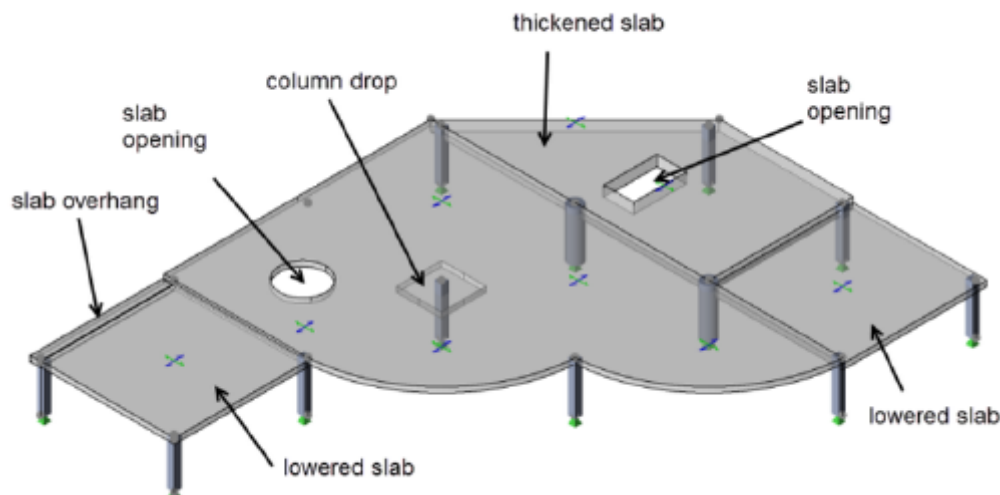
Load transfer in the bearing wall model is not the same as it would be in for example, a masonry wall. A point load applied at the top of a masonry wall would result in a distributed load on any beam supporting the masonry wall, whereas in a bearing wall the supporting beam would be subjected to a pair 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.

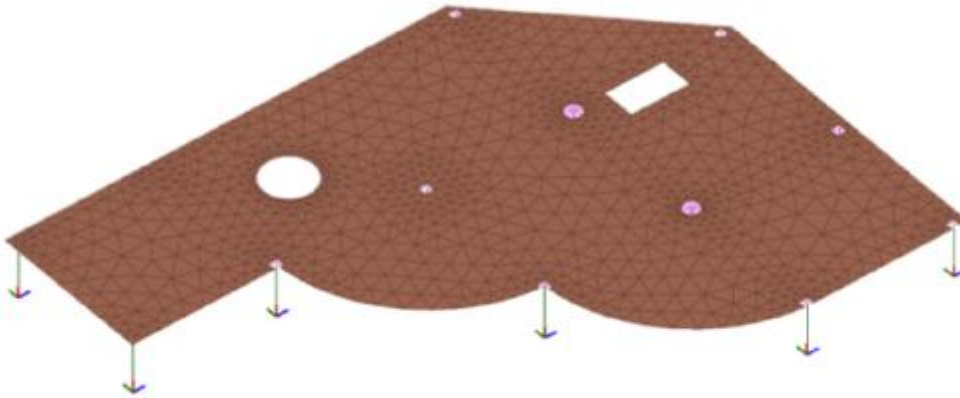
Solver elements for 2-way spanning concrete slabs

For 2-way spanning concrete slabs at a given level, the FE chasedown solver model is formed from 2D solver elements. These 2D elements are always created in the same plane, irrespective of whether slabs have different thicknesses, slabs items been raised/lowered via vertical offsets, or column drops have been applied. 2D solver elements 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.



Beam solver elements and slab meshes can only be offset vertically from one another by being defined in different construction levels.

Releases

End releases are applied by editing member properties in the physical model. They cannot be edited directly in solver views.

Column releases

The fixity at the top and bottom of each column stack can be set as:

- **Free end** - only applicable to the top end of top-most stack and the bottom end of the bottom-most stack
- **Fixed** - in both directions (i.e. encastré, all degrees of freedom fixed)
- **Pinned** - in both directions (i.e. a pinned connection is created between the stack above and the stack below)
- **User defined** - (i.e. fixed in one direction but pinned in the other)

For columns of all materials apart from concrete, in addition to the above release options you are also able to apply a **torsional release** at either end by checking the appropriate box.

For gable posts only, an **axial release** can also be applied at the top of the post.

User Defined

The User defined option (i.e. pinned in one direction but fixed in the other) is not available in the Properties Window and can only be specified as follows:

1. Right-click the column to display the context menu.
2. Choose **Edit**

3. From the Column Property Dialog open the Releases page and choose the stack to edit.
4. Uncheck the My or Mz degree of freedom at the desired end as required.

Wall releases

Walls can be released about the minor axis at the top and bottom of each panel as follows:

- **Fixed** - Encastré, all degrees of freedom fixed.
- **Continuous (incoming members pinned)** - A fully fixed connection is created between the wall panel above and the wall panel below. Incoming members and incoming slabs are pinned to the wall.
- **Pinned** - A pinned connection is created between the wall panel above and the wall panel below.



*The **Pinned** option should be used with caution as it may result in a mechanism during the analysis.*

Beam releases

Releases at the two ends of a beam span can be set as follows:

- **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.)
- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Moment** - Major axis moment connection, and pinned about the minor axis.
- **Fully fixed** - Encastré, all degrees of freedom 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.
- **User defined** - This setting appears if the connection is pinned for major axis bending (My released) but remains fixed for minor axis bending (Mz).

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.

User Defined

The User defined option 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 releases

Braces can only be connected to supports or to the supporting structure via pinned connections. A torsional release can be applied at one end if required. If the brace connects into a beam (e.g. an A brace) an axial end release can be specified at one end to prevent vertical load from the beam being carried by the brace.

An option is provided to include force eccentricity moment.

Supports

Columns and walls have supports automatically placed underneath them, unless they are placed directly upon existing elements which provide a means of support (e.g. transfer beams or transfer slabs).

Supports can also be placed manually at other locations.

If the default supports are inappropriate, they can be changed, simply by box selecting the supports that require editing and then adjusting the support degrees of freedom displayed in the Properties Window.

Supports can be edited in both physical model views and solver model views.

Support degrees of freedom

Each support has six possible degrees of freedom:

- translational (F_x , F_y , and F_z)
- rotational (M_x , M_y , and M_z)

Wall supports and manually placed supports default to fully fixed, whereas column supports default to being rotationally free in M_x & M_y .

The options for a support that is rotationally free in M_x or M_y are:

- Release
- Spring Linear
- Spring Non-linear
- Nominally pinned
- Nominally fixed

The options for a support that is rotationally free in M_z are:

- Release
- Spring Linear
- Spring Non-linear

The options for a support that is translationally free in F_x , F_y , or F_z are:

- Release
- Spring Linear
- Spring Non-linear

Non linear spring supports

For non-linear supports, two spring stiffnesses are required one each for the positive and negative direction of action.

In addition an upper limit should be defined to set a cap on the force or moment that can be supported.

Compression only ground spring

A compression only ground spring would be defined translationally in z only as follows:

Type: Spring Non-Linear

Stiffness -ve: 0

F_{max} -ve (tension): 0

Stiffness +ve: your choice of ground spring stiffness value

F_{max} +ve (compression): your choice of capacity

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 (meshed or mid-pier), or if they are used for any other supports.

The support stiffness is based on the column properties ($E \cdot I / L$)

- E = Young's Modulus of the column
- I = relevant bending stiffness (I_{xx} or I_{yy}) of the column

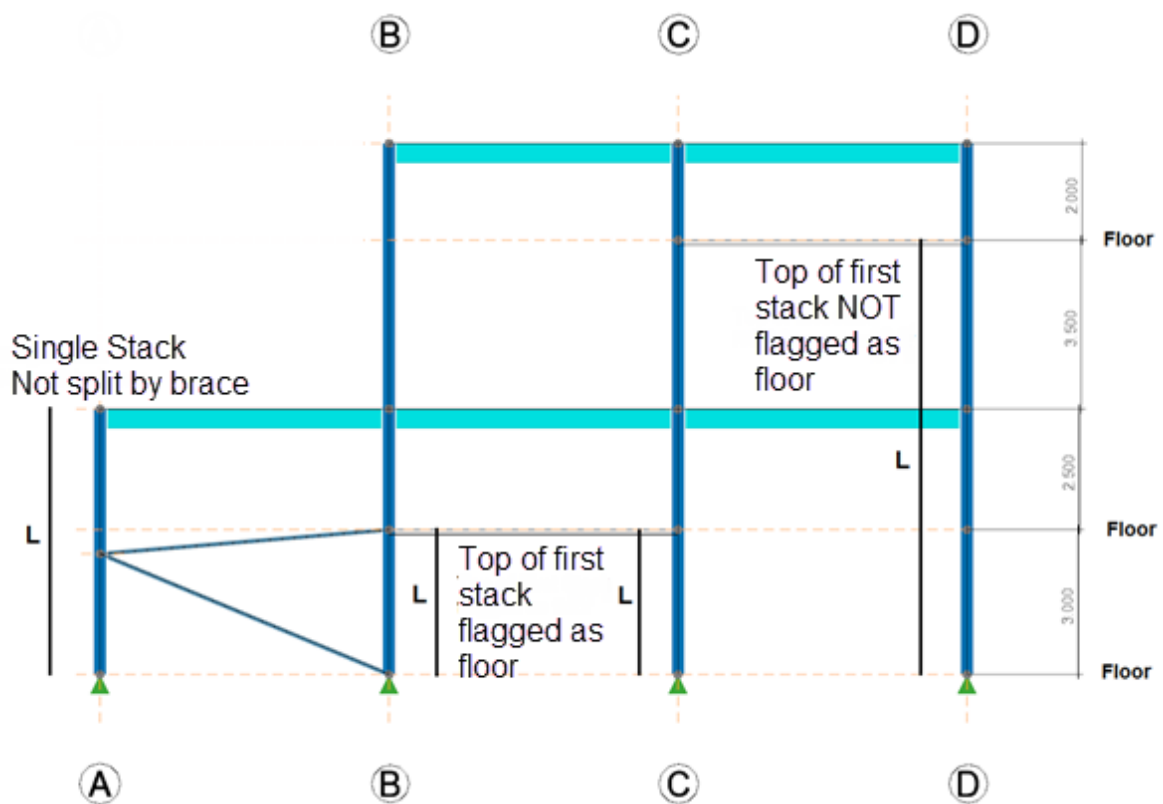
- L = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialogue, i.e. combined length of all the stacks until a floor is found.



Where no Floor has been defined above a support then L is taken as total length of column.

Partial fixity spring stiffness is thus calculated as follows for each of the two bending releases M_x and M_y :

- Nominally pinned (spring stiffness) - $x\% \cdot 4 \cdot E \cdot I / L$ (default $x\% = 10\%$)
- Nominally fixed (spring stiffness) - $x\% \cdot 4 \cdot E \cdot I / L$ (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.

Diaphragms and floor meshing

Diaphragm types

In a typical building lateral resistance is provided at a few discrete points and it is assumed that applied lateral loads will be distributed to the lateral load resisting systems via floor rigid diaphragm action, (or semi-rigid diaphragm action if a more flexible distribution is required).

For slabs the diaphragm type is controlled by the **Diaphragm option** slab property, which can be set as:

- Rigid
- Semi-rigid
- None

For roof panels the diaphragm type is controlled by the **Include in diaphragm** roof property, which when selected creates a semi-rigid diaphragm.

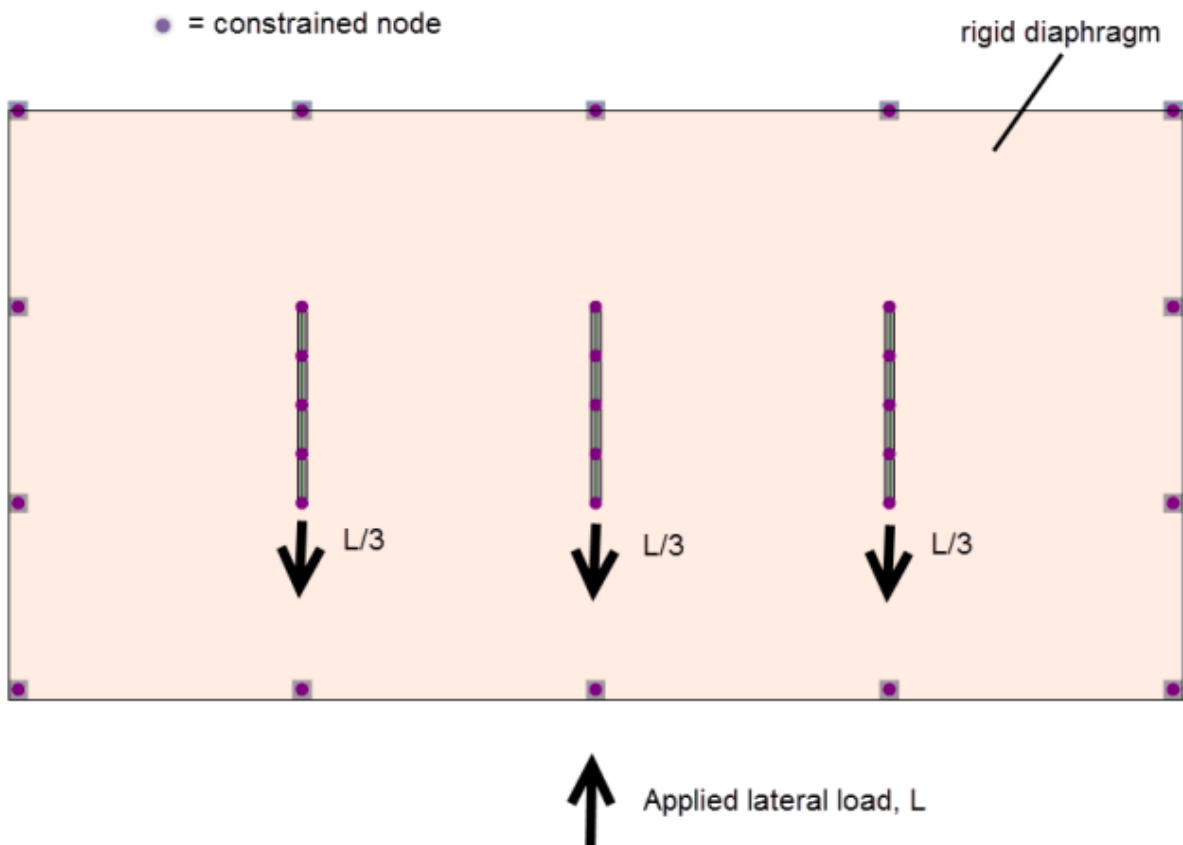


Roof panels cannot be used as rigid diaphragms.

Rigid

In *Tekla Structural Designer* rigid diaphragm action is achieved using nodal constraints in the translational X and Y directions and about Z torsionally.

A nodal constraint maintains exact relative positioning of all nodes that it constrains, i.e. the distance between any two nodes constrained in a rigid diaphragm will never change, therefore no axial load will develop in any member that lies in the plane of the diaphragm between any two constrained nodes



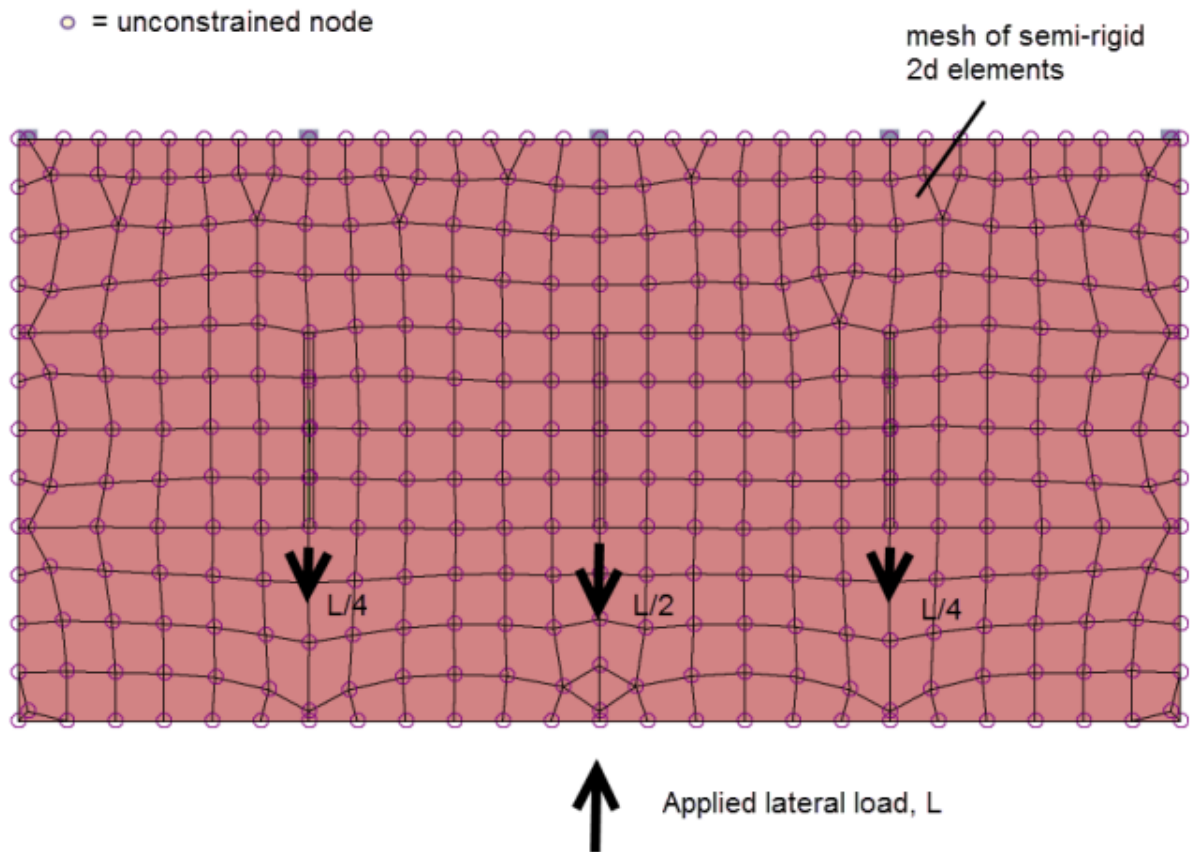
In the Solver View:

- **within the rigid diaphragm boundary:** *solid nodes are constrained; hollow nodes have been manually excluded and are therefore unconstrained.*
- **outside the rigid diaphragm boundary:** *both solid nodes and hollow nodes are unconstrained.*

Semi-rigid

Semi-rigid diaphragm action is achieved using a mesh of semi-rigid 2D elements with user defined properties.

The flexibility of the resulting diaphragm and consequently the distribution of the lateral load into the supports is manually controlled by adjusting the slab property **Divide Stiffness By**.



Diaphragm constraint and mesh type configurations

The slab mesh in a particular solver model is created from 2D elements; either shell or semi-rigid elements are used, depending on the following properties:

- Diaphragm option
- Decomposition
- Mesh 2-way Slabs in 3D analysis

Diaphragm option

Specified at the **Slab** level - this property can be set to:

- Rigid
- Semi-rigid
- None

How this affects the 2D element type used in each solver model is also dependant on the method of decomposition selected.

Decomposition

Specified at the **Slab** level - slabs can be set to either **Two-Way Spanning** or **One-Way Spanning**, although for certain slab types the value is fixed as below:

- 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 in all solver models unless the **Diaphragm option** is set to Semi-rigid, in which case they will be meshed with semi-rigid elements.



Semi-rigid 1-way spanning slabs cannot be designed.

Two-way spanning slabs always adopt a mesh of shell 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 elements - if the **Diaphragm option** is set to Semi-rigid, or,
- meshed with shell elements - if the **Diaphragm option** is set to Rigid.



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 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 summarised 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

Other slab properties affecting the solver models

Rotation Angle

Specified at the **Slab item** level, this property is used for the following where appropriate:

- Span direction for 1-way load decomposition
- To determine the 2D element local axes in the solver model
- Bar direction for Slab on Beam and Flat Slabs.

Include in Diaphragm

Specified at the **Slab item** level, this property is only active if the **Diaphragm option** is Semi-Rigid or Rigid. It has no effect on the shell mesh for 2-way spanning slabs.

- Semi-Rigid - excluded slab items are not meshed
- Rigid - internal nodes not considered in the nodal constraints



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.

Divide stiffness by

Specified at the **Slab** level - this property is only active if the **Diaphragm option** for the Slab is **Semi-Rigid**.

It is applied to the stiffness determined from the material properties and slab thickness in order to adjust semi-rigid diaphragm flexibility.

Mesh parameters

Slab Mesh

In the FE Chasedown and FE Load Decomposition solver models, slabs are meshed using 2D elements. The 2D element shape (triangular or quad) and the degree of mesh uniformity are specified in the [Structure Properties](#). These parameters can be overridden for individual sub-models by setting different values in the respective [Sub Model Properties](#).

Semi-Rigid Mesh

A semi-rigid mesh is created for slabs (both 1-way or 2-way spanning) that have the **Diaphragm Option** set as **Semi-Rigid**, and roofs that have the **Include in Diaphragm** property checked. The meshed elements are included in all solver models, including FE Load Decomposition.

Beam elements are not split by semi-rigid 2D element nodes.

The 2D element shape (triangular or quad) and the degree of mesh uniformity are specified in the [Structure Properties](#). These parameters can be overridden for individual sub-models by setting different values in the respective [Sub Model Properties](#).

Static Analysis and Design Handbook

When you click **Design All (Static)** a lot of things happen automatically, potentially including the creation of multiple different solver models.

This process in *Tekla Structural Designer* is quite unique, but you will find that it is effectively automation of traditional practice.

The following topics explain: the separate stages of the Design All (Static) process, the purpose of the different solver models, and how the solver models relate to the stages involved in traditional practice.




Stages of the Design All (Static) process

The following table lists the order in which processes occur on clicking **Design All (Static)**. (The same basic order would also apply to **Design Steel (Static)** and **Design Concrete (Static)**):



Members can be pre-sized for gravity combinations by running Design (Gravity), but a design that satisfies the code requirements can only be achieved by running Design (Static).

Step	Process	Description
1	Model Validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D pre-Analysis	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D Analysis	A traditional frame analysis of the entire 3D model, with an option to mesh floors.

4	Grillage Chasedown Analysis	<p>Requirements: Only performed if concrete members exist.</p> <p>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.</p> <p> <i>Not performed for:</i> - <i>Design Steel (Gravity), Design Steel (Static)</i></p>
5	FE Chasedown Analysis	<p>Requirements: Only performed if two-way slabs exist.</p> <p>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.</p> <p> <i>Not performed for:</i> - <i>Design Steel (Gravity), Design Steel (Static)</i></p>
6	Sway/Drift and Wind Drift checks	<p>Sway checks and Wind Drift checks are performed for all columns and walls, (apart from any that have been manually excluded from the checks).</p>
7	Member design	<p>Design All (Static) designs all steel and concrete members, and all concrete walls for all static combinations.</p> <p>Design Steel (Static) only designs steel members and concrete walls.</p> <p>Design Concrete (Static) only designs concrete members and walls.</p> <p>Design...(Gravity) designs the same members as above, but for gravity combinations only.</p> <p> Concrete slab design is not performed in any of the above.</p>

3D pre-Analysis

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.

Application 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.

Load reductions

Imposed 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 imposed 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.



Limitation: For head codes other than US, for beam design the reduced forces are only considered in the design of concrete, but not steel beams.

Reductions are only applied to those imposed 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.



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.

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.

Generation of pattern loading

If combinations of pattern load exist then the pattern loading is automatically generated prior to analysis.

3D Analysis

A traditional frame analysis of the complete structure is always the first analysis performed. 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.

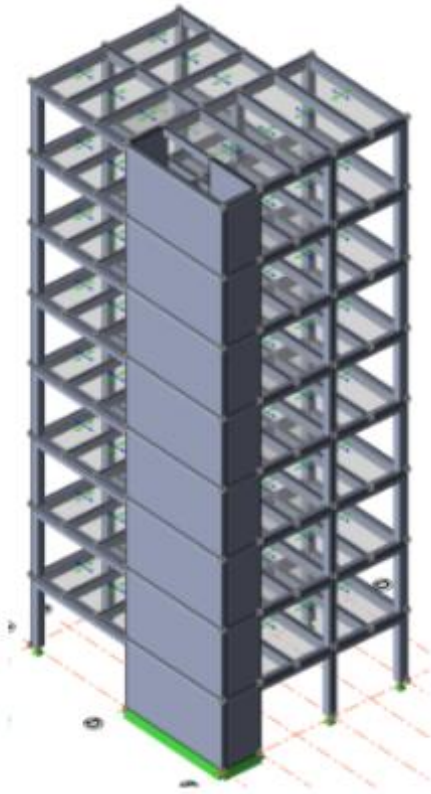
Why chasedown analyses are required

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 – for example...

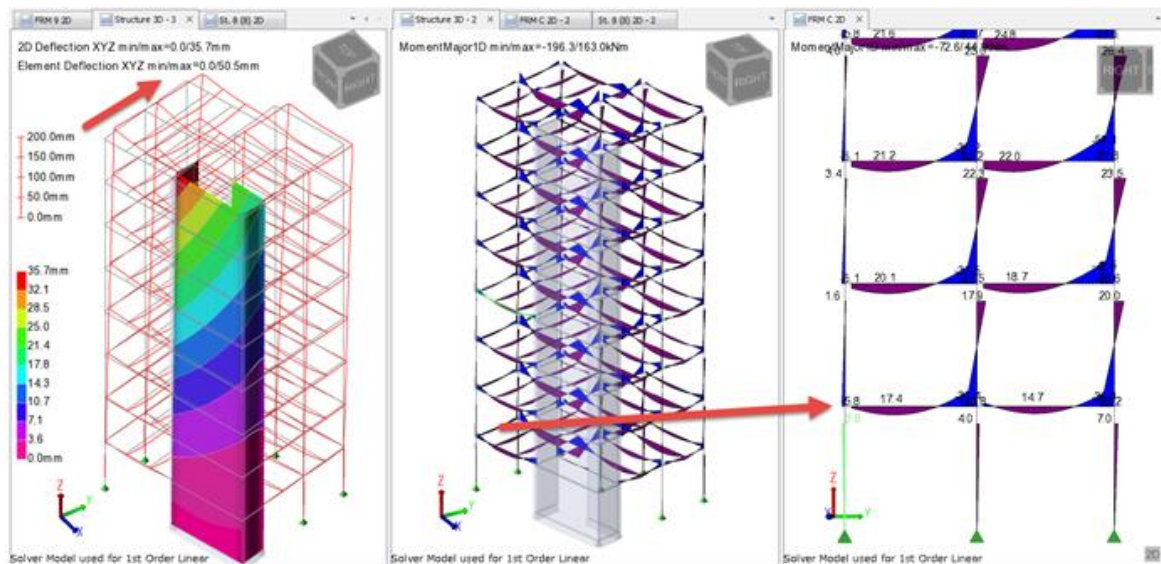
Sway Effects under pure gravity loading

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.

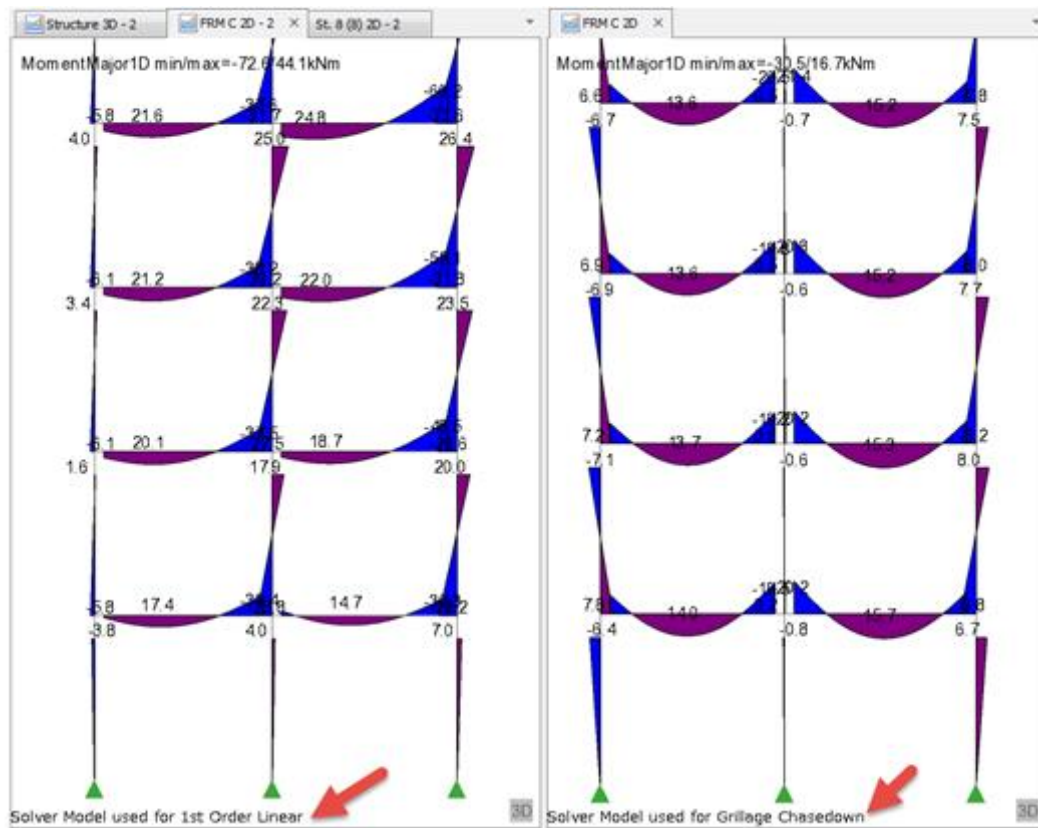


The Identified end frame below is resisting the sway - the design moments shown would be given by any analysis software.

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:

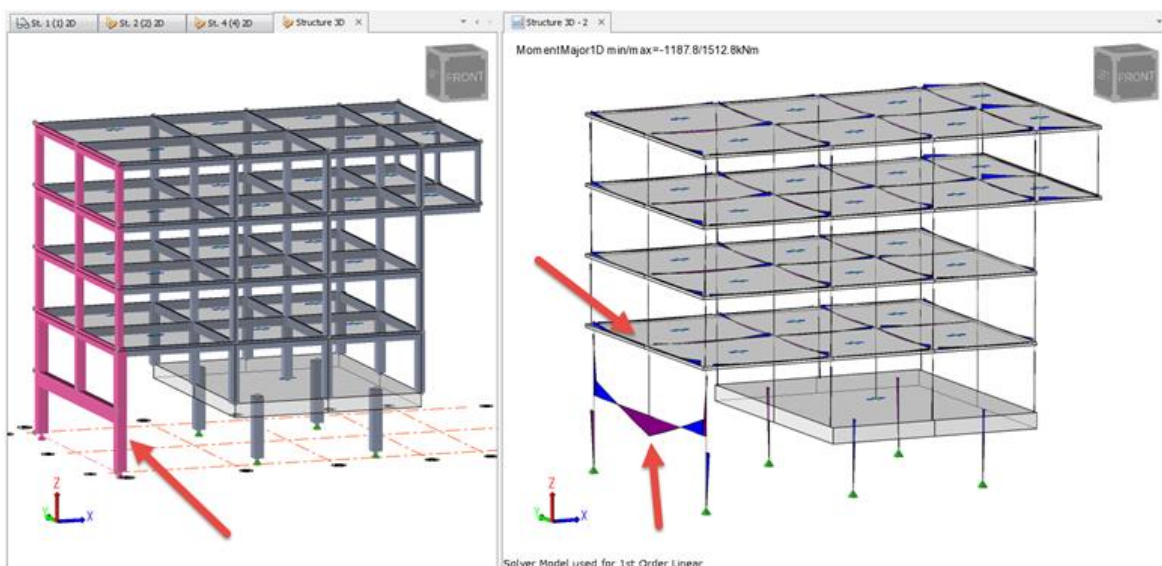
- -ve moment at LH end
- Similar moment profile at every level

Some more examples of why chasedown is wanted....

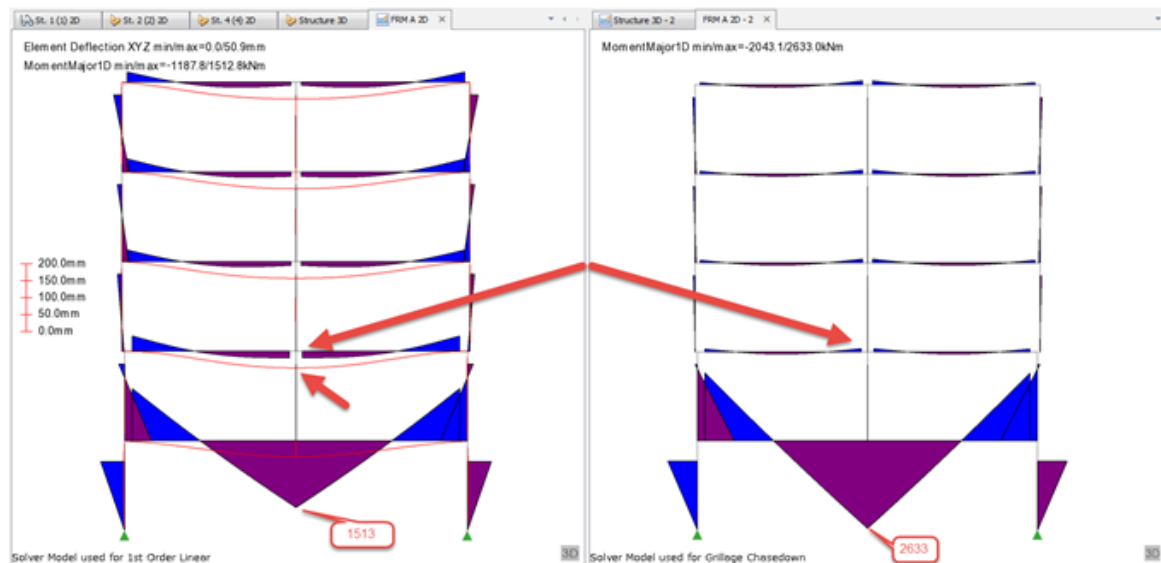
Transfer beam designs

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).

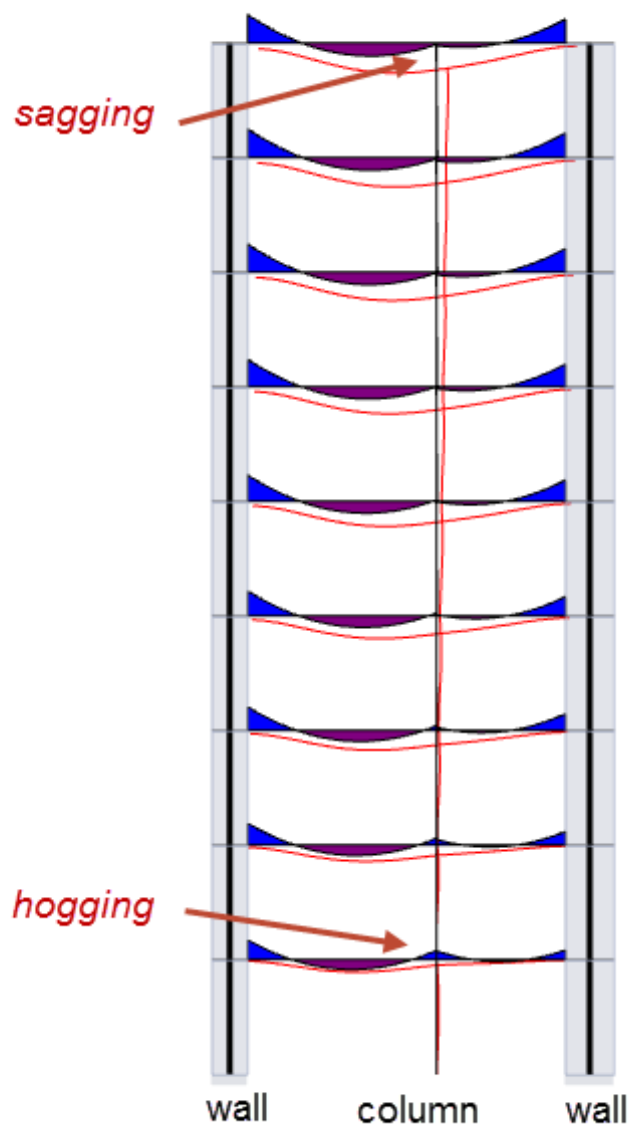


- **3D Analysis** – frame deflects and loads are shared according to stiffness.
- **Grillage chasedown** – loads are collected floor by floor and then applied to the transfer beam – much higher moment in the transfer beam. Once again this is more in line with traditional expectation.

Differential axial deformation

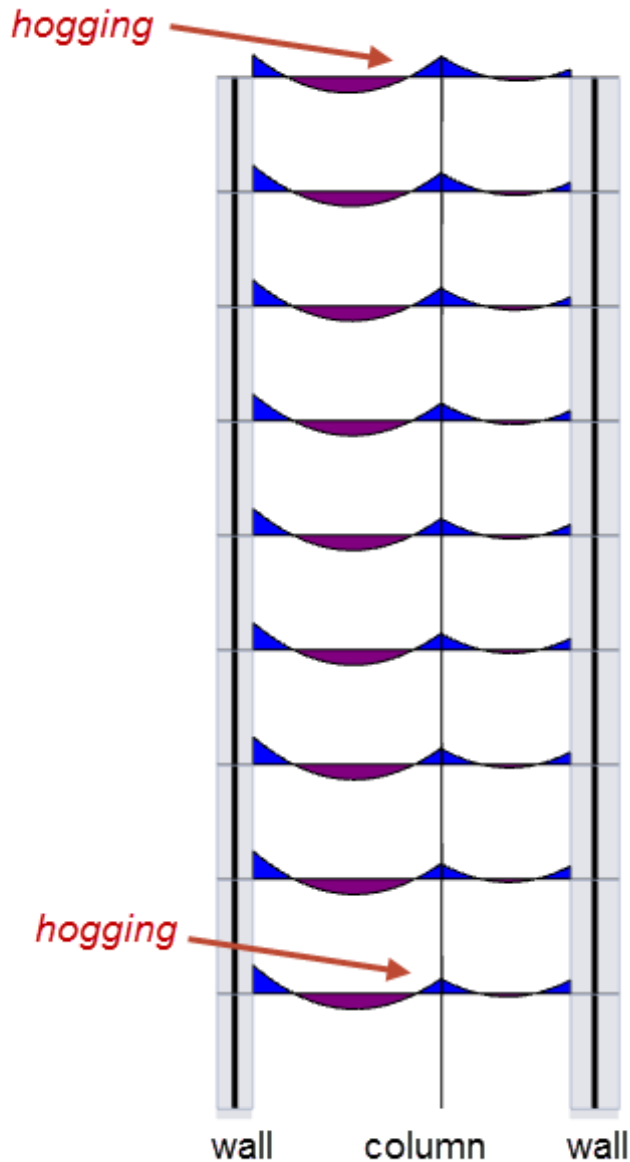
In this case a two span beam sits on walls at each end and a column in the centre.

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.



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.

Grillage Chasedown Analysis

Grillage Chasedown

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](#) emulates a traditional analysis and establishes an alternative second set of design forces for beams, columns and walls.

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 / seismic ELF / 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 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:
 - Subtract the 1st order linear building analysis result multiplied by the relevant load factor
 - 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.

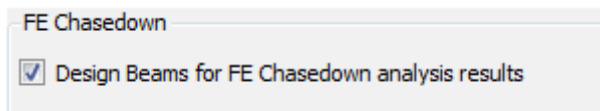


This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral load cases.

FE Chasedown Analysis

The [Solver Model used for FE Chasedown](#) is generated as part of the combined analysis and design process if the model contains flat slabs, or slabs on beams - the results from this analysis being required for the design of these slabs.

The same results can also be used to generate a third set of design forces for the chosen member types, (provided you have chosen in the Design Options dialog to design the concrete beams, columns, or walls for FE Chasedown results).



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.



If duplicate levels have been specified in the Construction Levels dialog separate sub models are created and analysed 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.

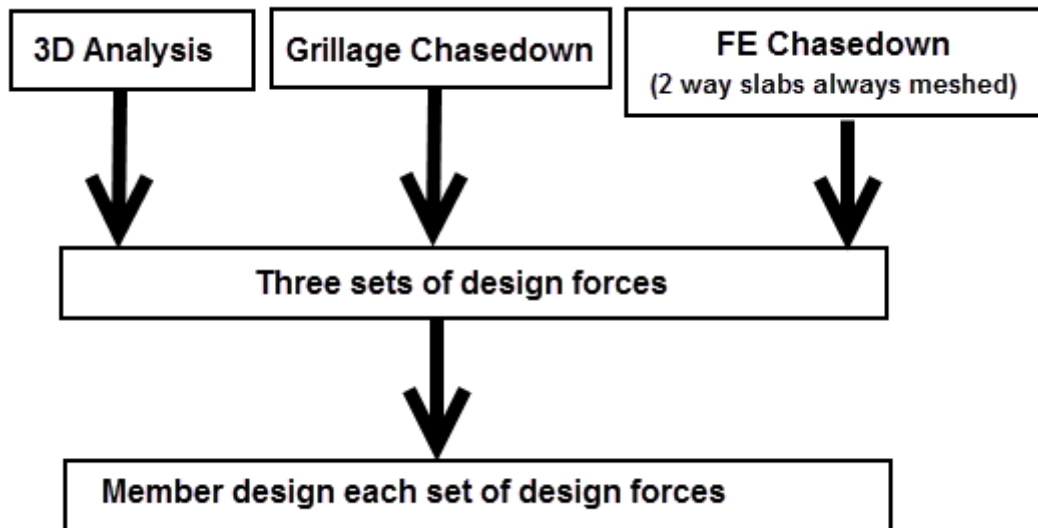
Member design

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 colour coded version of the model is displayed so that the pass/fail status and utilisation ratio of each beam, column and wall can be reviewed graphically. Various other parameters can also be graphically interrogated and/or modified.

Comparison of solver models used in static design

The following table summarises the three analysis models used in the design process:

	3D Analysis	Grillage Chasedown	FE Chasedown
Examples / When useful?	<ul style="list-style-type: none"> • Gravity and Lateral analysis (Notional/Wind/Seismic) 	<ul style="list-style-type: none"> • “Beam & Slab” buildings 	<ul style="list-style-type: none"> • Flat slab and “Beam & Slab” buildings
Special Features	<ul style="list-style-type: none"> • 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 	<ul style="list-style-type: none"> • Mimics traditional design approach (sub-frame analysis) • Pattern loading 	<ul style="list-style-type: none"> • Mimics traditional design approach (isolated floor analysis) • Slab Pattern loading
Benefits	<ul style="list-style-type: none"> • Member Design considers sway and differential axial deformation effects. • Caters for slabs that contribute to the lateral load resisting system 	<ul style="list-style-type: none"> • Member design based on traditional sub frame is considered simultaneously with that for 3D Analysis 	<ul style="list-style-type: none"> • Member design based on traditional sub frame is considered simultaneously with that for 3D Analysis • Irregular slab panel design automatically catered for
Analysis Model	3D model of entire building: <ul style="list-style-type: none"> • either meshed 2-way slabs, • or, slab loads decomposed to beams 	series of 3D sub models: <ul style="list-style-type: none"> • 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: <ul style="list-style-type: none"> • 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	<ul style="list-style-type: none"> • First order • First order - K_{amp} • 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
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

Seismic Analysis and Design Handbook

This handbook describes *Tekla Structural Designer's* seismic analysis and design capability, which is available for ASCE7, UBC 1997 and Eurocode EN1998-1.

Introduction to Seismic Analysis and Design

The below topics provide a simplified outline of how seismic analysis and design forces are determined for a building.

Definitions

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** or the **Seismic Inertia Combination**. This combination is used for Vibration 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 Combination

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 Effective Seismic Weight 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 Vibration 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 Vibration Analysis for the Seismic Inertia Combination.

Level Seismic Weight

For each relevant level, this is the sum of the vertical forces in nodes on that level, for the Seismic Inertia Combination.

Effective Seismic Weight

This is the sum of the level seismic weights for all relevant levels for the Seismic Inertia Combination.

Seismic Base Shear

The base shear is calculated separately for Dir 1 & Dir 2, for the Seismic Inertia Combination.

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 η 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:

The Seismic Inertia 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 of vibration. 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 vibration analysis considering all mode shapes of vibration in the two horizontal directions in which typically 90% of the structure's mass is mobilised.

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 have to resist during an earthquake.



The additional design and detailing requirements of "seismic" design are only supported in Tekla Structural Designer for the ACI/AISC Head Code.

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 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 of vibration 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 elastic design response spectrum to be used for the building.

Additionally the Wizard sets up the Seismic Inertia combination - the combination of loads likely to be acting on the building when the earthquake strikes.

Related topics

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 centre of mass at a level does not align with the position of the centre 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 centre of mass and centre 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



Amplified accidental torsion is beyond scope in the current release of Tekla Structural Designer.

Vibration Analysis

Using the Seismic Inertia combination, a vibration 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

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 Seismic Inertia 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 Analyse toolbar, 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 Options > Analysis.

The process consists of the following steps:

No.	Process	Description
1	Model Validation	Run to detect any design issues which might exist. This is similar to standard model validation but also checks: <ul style="list-style-type: none"> • Seismic Inertia Combination must exist • At least one RSA Seismic Combination must exist including at least one RSA Seismic Loadcase.
2	Vibration Analysis	A 1st Order Vibration 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.

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.

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 Head Code.
- 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 vibration with non-linear element properties - currently the vibration 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 vibration analysis the vibration 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 vibration 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

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)

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) - no seismic design performed.
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Walls

- Special reinforced concrete structural wall
- Intermediate precast structural wall
- 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
- Direction 1 and Direction 2 are mutually exclusive for all members and thus a column, for example, cannot be in more than one SFRS.

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) in

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 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

For those regions categorised as "low seismicity" it is acceptable to assume "ductility class low" applies. Under these conditions the results of a seismic analysis can be fed into "conventional" design.

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.



The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC Head Code.

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:

- Determine building height to the highest level and adjust it if required
- Set the
- Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method)
- 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:

- Determine building height to the highest level and adjust it if required
- Set the
- Select RSA method of analysis
- 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

Seismic analysis and seismic design



The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC Head Code.

The overall modelling, analysis and design process to be followed when seismic design is required depends on the analysis procedure (ELF or RSA) that you have chosen to perform.

The seismic design 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, conventional design 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:

- Determine building height to the highest level and adjust it if required
- Set the
- Select ELF method of analysis (note some vertical or horizontal irregularities (set by the user) can prevent the use of the lateral force method)
- Set up a vibration combination
- Set up the relevant seismic combinations

3. Static Design

Run **Design (Static)** to:

- Conventionally design all members for all non-seismic (gravity and lateral) combinations
- Conventionally design all members for all seismic combinations in the same way as the other combinations.
- 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, conventional design 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:

- Determine building height to the highest level and adjust it if required
- Set the
- Select RSA method of analysis
- Set up a vibration combination
- 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. Vibration Analysis

- At this point it is recommended that you run a 1st order vibration 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 **Design (RSA)** to:

- Conventionally design all members for all RSA seismic combinations in the same way as the other combinations.
- 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

Steel Design Handbook

This handbook provides a general overview of *Tekla Structural Designer* in the context of its application to steel structure design.



Refer to the Reference Guides for details of the specific steel calculations that are performed for each design code.

General design parameters (steel)

A number of design parameters are common to the different steel member types - these are described in the topics below.

Material type

The material types supported depend on the code being designed to:

- **Steel** is the only material permitted if designing to **AISC 360**.
- **Steel** and **Cold formed** materials are both permitted if designing to the **Eurocode** or **BS 5950**.

Autodesign (steel)

The design mode for each member is specified in the member properties.



If a member type has been set to be designed using [Design groups \(steel\)](#), then if at least one member of the group is set to autodesign the whole group will be automatically designed.

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 size.

The following controls can be applied to further limit the sections considered:

- [Size Constraints](#)
- [Steel beam deflection limits](#)

- [Steel beam camber](#) (in the case of beams)

Design Section Order

A **design section order** is only applicable when **Autodesign** 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.

How do I view the list of sections in a design section order?

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.

How do I specify that a section in the list should not be considered for design?

Only checked sections within the list are considered during the design process. Uncheck a section and it will no longer be considered.



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 therefore 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).

How do I sort the listed sections by a different property?

While viewing the list of sections you can:

1. Select a property from the **Sort By** droplist
2. Click **Sort** to re-order by the chosen property.
3. 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**.



Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

How do I 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 Up** or **Move Down** to promote or demote it.



Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

How do I reset a design section order back to the original default?

3. Edit the properties of the member.
4. Click the **Design section order** drop list and select **<New\Edit>...**
5. In the **Select a Section Order** dialog, highlight the section order that you want to reset.
6. Click **Reset**

The highlighted design section order is reset to its default settings.



The Reset button is only displayed for the pre-installed section orders. (User defined section orders can be deleted but not reset.)

How do I 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. Highlight sections in the **Available Sections** list and add, then sort them as required.
7. 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.

Size Constraints

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. 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.

Gravity only design

By checking/unchecking the **Gravity only** option members are set to be:

- Gravity only - designed for gravity combinations and seismic combinations
- Lateral and Gravity - 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 however be required when identifying members as being 'gravity only'.

For example:

- if an inclined braced member connects to a simple/composite 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, simple beams in a sloping roof would also need to be designed for both gravity and lateral load



If a simple, or composite beam is identified to be designed for both gravity and lateral combinations, only the component of the lateral load that acts in the plane of the strong axis of the member is considered. Any axial loads, or loads in the weak axis are ignored. A warning is provided if the ignored loads exceed a preset limit.

Design groups (steel)

Steel members are 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.




A fixed set of rules are used to determine the automatic member grouping: for example beams must be of similar spans, columns must have the same number of stacks etc.

The same rules also constrain manual group editing.

If required grouping can also (optionally) be utilised in order to design steel member types according to their groups.

In order use grouping for this second purpose you should first ensure your groups are configured to only contain those members that you intend to eventually have the same section size applied.

How is the “design using groups option” activated?

1. Click **Design > Options...** ()
2. Click **Design Groups**
3. Select the check box adjacent to each member type for which you want to apply design grouping.

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 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 is adequate for each and if this proves not to be the case, the section is increased as necessary and the revised section 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 utilisations are established.

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 there are many smaller members or sheeting, for example, 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.



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.

Steel beam design

Steel beam design overview

Tekla Structural Designer allows you to analyse and design a structural steel beam or cantilever which may have incoming beams providing restraint and which may or may not be continuously restrained over any length between restraints.

Design is performed 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 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.

Steel non-composite beams that share load with columns form part of a rigid moment resisting frame.

Steel non-composite beams can optionally be set as continuous; in which case all internal connections are considered continuous.

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

The 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 buckling and top and bottom flange for lateral torsional buckling (LTB). It is upon these that the buckling checks are based.

A full range of strength and buckling checks are available. As mentioned above the buckling lengths are based on the restraints along the member. The effective lengths to use in the 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 strut or LTB effective length can take the type “*Continuous*” to indicate that it is continuously restrained over that length.

Each span of a continuous beam can be of different section size, type and grade. The entire beam can be set to automatic design or check design.

Steel beam fabrication

The following fabrication options are available:

Rolled

- A wide range of rolled sections can be designed.
See: [Steel beam section](#)

Plated

Westok cellular

Westok plated

Fabsec

Steel beam section

Tekla Structural Designer will design steel non-composite 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.



Refer to the Steel Reference Guides for details of any design limitations that apply for specific section types.

Steel beam restraints

Lateral and Strut restraints

Lateral and Strut restraints are determined from the incoming members described within the *Tekla Structural Designer*. 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 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).

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.

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.

Only the start and end of the beam are defined in the dialog.

Steel beam deflection limits

It is often found that serviceability criteria control the design of normal composite beams. This is because they are usually designed to be as shallow as possible for a given span.

Deflections 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 imposed 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.



Refer to “Reference Guides> Steel Design - EC3 and EC4> Steel Beam Design to EC3> Torsion” for details of the specific checks performed.

Steel beam 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 Dialog Combinations 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 imposed 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.

Steel beams in seismic force resisting systems



Design of members in seismic force resisting systems is beyond the scope of the current release.

Steel column design

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.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not considered.)

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.

Web openings cannot currently be designed for.

Limitations for sloping columns

The following limitations apply:

- the web of each stack of a sloping column must lie in the same plane,
- sloping general columns are limited to having either their web, or flanges in a vertical plane.
- eccentricity moments are not taken into account in design,
- there is no imposed load reduction.

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.



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 section

Tekla Structural Designer can handle design for an international range of steel I-sections and also rectangular, square and circular hollow sections for many different countries and also for many specific manufacturers.

Steel column restraints

Restraints to flexural and torsional buckling are determined from the incoming members described within the *Tekla Structural Designer* model. The buckling checks are based on these.



Restraints are considered effective on a particular plane providing they are within $\pm 45^\circ$ 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.



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.

Lateral torsional buckling (LTB)

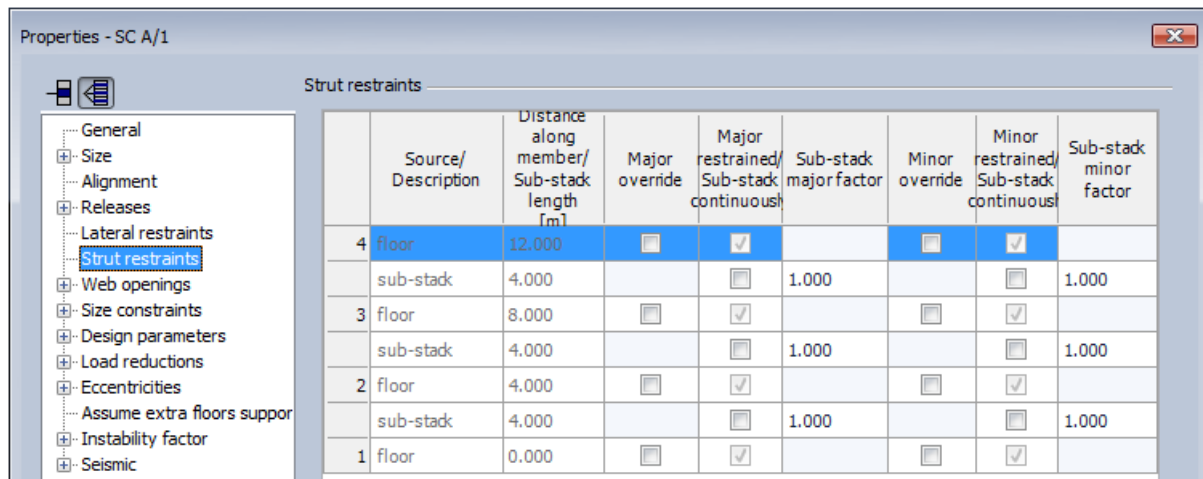
Australian Headcode

Lateral restraints									
	Source/ Description	Distance along member/ Sub-stack length [m]	Face A override	Face A restrained/ Sub-stack continuously	Face C override	Face C restrained/ Sub-stack continuously	Cross-section restraint type	kr	kl
3	floor	3.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Full res...		
	sub-stack	1.500		<input type="checkbox"/>		<input type="checkbox"/>		1.000	1.400
2	member	1.500	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Lat res...		
	sub-stack	1.500		<input type="checkbox"/>		<input type="checkbox"/>		1.000	1.400
1	floor	0.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	Full res...		

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.

Compression/Strut buckling

(Eurocode, British and Australian Head Codes)



	Source/ Description	Distance along member/ Sub-stack length [m]	Major override	Major restrained/ Sub-stack continuous	Sub-stack major factor	Minor override	Minor restrained/ Sub-stack continuous	Sub-stack minor factor
4	floor	12.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	
	sub-stack	4.000		<input type="checkbox"/>	1.000		<input type="checkbox"/>	1.000
3	floor	8.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	
	sub-stack	4.000		<input type="checkbox"/>	1.000		<input type="checkbox"/>	1.000
2	floor	4.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	
	sub-stack	4.000		<input type="checkbox"/>	1.000		<input type="checkbox"/>	1.000
1	floor	0.000	<input type="checkbox"/>	<input checked="" type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	

Members framing into either Face A or C will by default provide restraint against major axis strut buckling. Members framing into either Face B or D will by default provide restraint against minor axis strut buckling. You can remove these default restraints if required by selecting the Major override and/or Minor override checkboxes.

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 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 take account of the $P-\delta$ moment at the position as well as the forces from the analysis.

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.

Column Splice Report

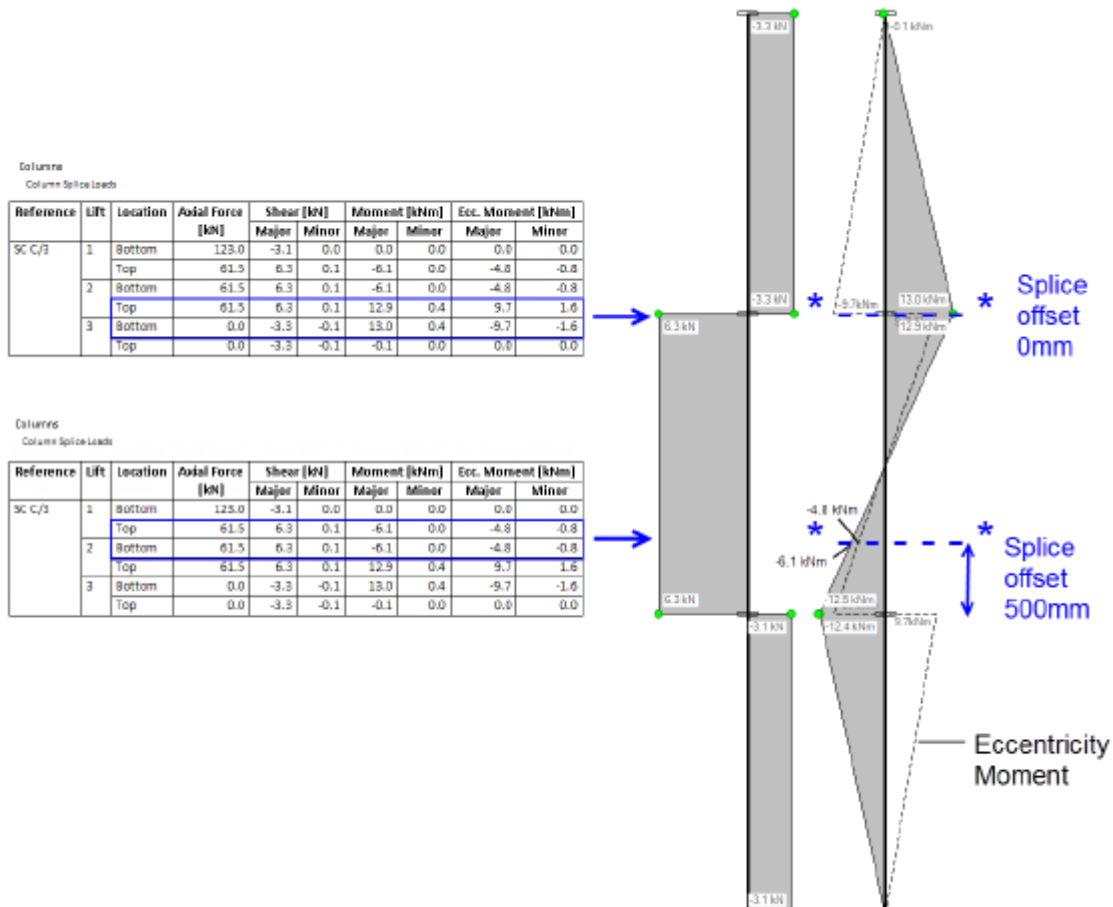
Steel column splice loads can be output to a report if required.

In this report factored forces are output at the top and bottom of each lift for every column in which a splice has been defined.

The splice offset from the floor level is taken into account when calculating these forces.

Eccentricity moments are also reported. These are determined using the vertical end reactions of incoming beams at each level applied at the connection eccentricities that have been specified in the column properties dialog. Again, the splice offset from the floor level is taken into account when calculating the eccentricity moments.

The below example illustrates the effect of the splice offset. The lower splice has been offset by 500mm, so the (Lift 1 Top and Lift 2 Bottom) loads are reported 500mm above the 1st floor level. The higher splice has not been offset, so the (Lift 1 Top and Lift 2 Bottom) loads are reported at the 2nd floor level.



Steel column web openings



In the current release of the program the design or checking of columns with web openings is "Beyond Scope".

If you need to provide access for services, etc., then you can add openings to a designed column and then check them.

You can define rectangular or circular openings and these can be stiffened on one, or on both sides.

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 column with a stronger web in order to reduce or remove any stiffening requirement.

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. The openings so created are the maximum depth spaced at the minimum centres 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 columns in seismic force resisting systems



Design of members in seismic force resisting systems is beyond the scope of the current release.

Steel brace design

•

Steel brace overview

Tekla Structural Designer allows you to analyse and design a steel member with pinned end connections for axial compression and tension.

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.
- Imposed 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 modelled 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 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 calculations with the result that the calculated are not correct.

Steel brace section

The design of steel braces is carried out for rolled I-sections, C-sections, T-sections, rectangular, square and circular hollow sections, angles, double angles, and flat sections.

Steel brace in compression

Effective length factors are defined for each axis of buckling.

- Effective length factor y-y
- Effective length factor z-z

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

Steel truss design

In *Tekla Structural Designer* although truss members can be defined in any material, design is restricted to steel trusses only.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

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

Internal and side truss members are designed as steel braces (for axial forces only).

See: [Steel brace overview](#)

Top and Bottom Truss Members

Top and bottom truss members are designed as steel beams, with the exception that seismic forces are not considered.

See: [Steel beam design overview](#)

Steel joist design



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.

Steel joists (or bar joists) are simply supported secondary members, which do not support any other members - they only support loaded areas.

- Steel joists can be defined with ends at differing levels.
- They cannot support any other member.

Slab and roof loads are supported by steel joists and loads are distributed to them.

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 - open web, parallel chord steel joists - depths 8" to 30" with spans up to 60ft.
- 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 72" for clear spans up to 144ft.

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. Load diagrams for the relevant joist can be output to forward to the fabricator for designing.

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.

Joist Analytical Properties

Steel joists must be simply supported and cantilever ends cannot be defined. They cannot be released axially.

Only Joist Girders and SP joists are able to support members along their length.

The inertia and area values are taken directly from the Steel Joist Institute tables.

Performing steel structure design



During the design process every steel member in the model will either be designed automatically, or checked - depending on their individual [Autodesign \(steel\)](#) settings. You should therefore ensure the Autodesign property has been set correctly before commencing.

Gravity design

In large models you may prefer to adopt a two-stage design process in which a gravity design is performed in advance of the full design.

The gravity design stage enables you to design or check the simple beams, composite beams and [Gravity only design](#) simple columns for the designated gravity combinations (this will

include the Construction Stage combination). Other members will also be designed or checked for these combinations but the resulting section sizes are less useful and are likely to require increasing in later stages of the analysis/design process. This approach is intended to speed up the design process.

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 therefore decide to reset certain members to auto design e.g. columns and beams in 'moment frames'. In such cases, when the full design is performed member 'pre-sizing' will take place and for members resisting lateral loads this section size will be used if it is larger than that which resulted from the Gravity design.

Full design

All beams, columns and braces are designed or checked for all active combinations. ([Gravity only design](#) members will be designed or checked for the active gravity combinations only).

Full design is initiated by clicking [Design Steel \(Static\)](#), or [Design All \(Static\)](#).

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. (See: [Allowing for global second-order effects](#))

Concrete Design Handbook

This handbook provides a general overview of *Tekla Structural Designer* in the context of its application to concrete structure design.



Refer to the Reference Guides for details of the specific concrete calculations that are performed for each design code.

General design parameters (concrete)

A number of design parameters are common to the different concrete member types - these are described in the topics below.

Gravity or Static Concrete Design?

- [Design Concrete \(Gravity\)](#) can be run (as a precursor to the static design) in order to concentrate on sizing members for the designated gravity combinations only - any lateral stability issues are temporarily ignored.
- [Design Concrete \(Static\)](#) is run to size members for **all** active combinations (gravity and lateral).



For concrete models in Tekla Structural Designer, adopting the above two-stage design approach is often not required. Typically you can forego the (optional) gravity only design and go straight to the (essential) static design.



*Even if **Design Concrete Static** is run, individual beams and columns can still be designated as “Gravity only” if required by selecting the appropriate check box in the member’s properties. Any such beams and columns will then only be designed for gravity and not lateral combinations.*

Analysis types performed in the Design Concrete process

To generate the sets of design forces required for the beam, column, and wall design; up to three separate analyses are automatically performed:

3D Analysis

This analysis type is always performed.



Depending on the code being designed to, there are different methods of 3D Analysis. The available choices are specified via Design Options> Analysis.

Grillage chasedown analysis

This analysis type is also always performed.

FE chasedown analysis

In addition, concrete beams, columns, and walls can (optionally) be designed for a third set of design forces established from an FE chasedown analysis. This set of forces is activated in the Design Options, (via the General Parameters for each member type).

Pre-design considerations

For a concrete structure, the following settings and options in particular should be considered before running the design:

1. **Grouping** - decide if you want to make use of [Design and detailing groups \(concrete\)](#).
2. **Concrete Design Options** - check the concrete [Design Options](#) are appropriate for your design.
3. **Member properties and Autodesign settings** - review the design related properties that have been assigned to individual members.

In particular review the **Autodesign** settings as these control whether the reinforcement in each member will be designed or checked.



If a member is in a group, then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Stacks and reinforcement lifts

Columns and walls are only split into separate stacks at those construction levels where an element or slab is attached to the column/wall.

Reinforcement is designed by stack and longitudinal reinforcement is constant throughout a stack.

A reinforcement “lift” is defined as the column or wall height between two levels anywhere in the building throughout which the cross-section and the reinforcement arrangement is constant. For the cross-section to be constant, all aspects of the shape, dimensions and

rotation must be identical. Reinforcement changes in one stack in a reinforcement lift would apply to all stacks in the reinforcement lift.



For the current release, reinforcement lifts are restricted to contain only one stack.

Column and wall clear height

The clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) 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 unrestrained length may be different in each direction.

When determining the unrestrained 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 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.

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 unrestrained length includes the stack beyond this restraint, and the same rules apply for finding the end of the unrestrained 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 unrestrained length ends at the end of the column.

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.

Nominal cover

The nominal cover for each member is specified in the member properties.

For beams and columns

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.

For walls

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.

Assume cracked

This setting is specified for each member under the **Design control** heading in the member properties.

Assuming concrete sections are cracked has a direct affect on the analysis; smaller [Use of Modification Factors](#) are applied to cracked sections causing an increase in deflection.

Indirectly the design can also be affected because the sway sensitivity calculations are also influenced by this assumption.



The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Reinforcement Parameters

The reinforcement parameters common to concrete members are specified in **Design Options > Concrete > Reinforcement Parameters**.

Design and detailing groups (concrete)

Why use concrete design and detailing groups?

Concrete beams and columns and isolated foundations are automatically put into groups for two reasons:

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.



A fixed set of rules are used to determine the automatic member grouping: 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.

For design and detailing purposes - to reduce the processing time and also 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.



Grouped design and detailing is optional and can be deactivated if required.

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 utilisations are established.

Concrete design group requirements

Concrete member design groups are formed according to the following rules:

Concrete beam design groups

- 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 design groups

- 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 design groups

- 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 design groups

- 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:

Concrete beam detailing groups

- 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 detailing groups

- A detailing group may be associated with only one parent design group,
- A column may only be in one detailing group,
- 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 detailing groups

- 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 detailing groups

- 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 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.



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 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**.




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].

How is grouped design and detailing de-activated for concrete members?

1. Click **Design > Options...** ()
2. Click **Design Groups**
3. Clear the check box adjacent to each concrete member type for which you want to de-activate design grouping.

Concrete beam design

Analysis types used for concrete beam design

Concrete beams are designed for a set of design forces obtained from the **3D Analysis** plus a second set of design forces obtained from **Grillage Chasedown Analysis**.

In addition, the beams can (optionally) be designed for a third set of design forces established from **FE Chasedown Analysis**.

Autodesign (concrete beam)

The design mode for each beam is specified in its properties.



If concrete beams have been set to be designed using [Design and detailing groups \(concrete\)](#), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

- 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 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.

Select bars starting from

This option only appears if **Autodesign** is selected. It sets the autodesign start point for both longitudinal bars and .

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.



*When a beam 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.)*

Rationalisation 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 .

This information is then “rationalised” 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 rationalisation 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 is not rationalised. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

Deflection control

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 (E_{cj}) 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. 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.

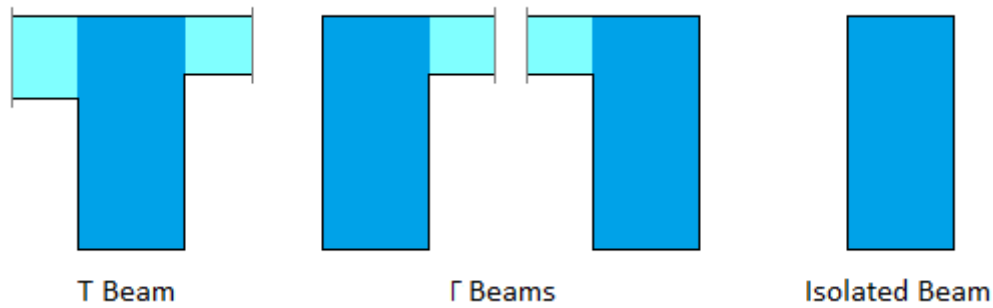
2. Consider flanges

- Checking this beam property (located under the **Design Control** heading) can assist in satisfying the deflection check.

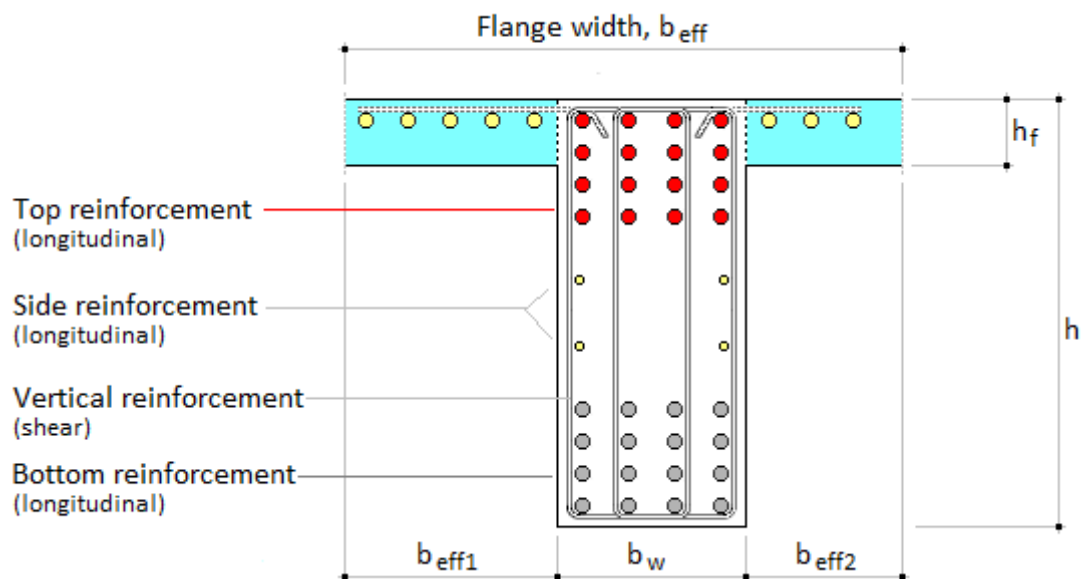
Use of beam flanges

A beam element or beam line is initially created in the model with a rectangular cross-section.

The beam properties can then be edited to take account of flanges arising from adjoining slabs, making the following beam shapes possible:



These shapes have common features which are shown in the figure below:



where

h = overall depth including the depth of the slab

h_f = depth of slab

b_w = width of beam

b_{eff1} = flange width side 1

b_{eff2} = flange width side 2

b_{eff} = flange width

= $b_{eff1} + b_w + b_{eff2}$

Use of Flanged Beams

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 "L" 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 \geq 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, $b_{eff,lr}$ 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.



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.

Adjacent Beams not Parallel

For beams that are not parallel, the effective width of the flange will vary along the length of the beam and the value used in element design calculations is the minimum width that occurs in the distance between the points of zero moment i.e. the previously calculated L_0 length.

Holes/Openings in Calculated Effective Flange Width

The presence of holes or openings in slabs can have an impact on the effective width of the slab used in the element design; indeed, in some circumstances it may mean that the beam cannot be designed as flanged.

However, as it is difficult to identify holes or openings in slabs that are within the calculated effective flange width - such openings are ignored in the automatic calculation of the effective flange width. Where such holes or openings exist you should therefore manually adjust the flange width to take account of them. This is achieved using the **Allowance for openings left/right** parameters in the beam properties.

User defined effective flange width and depth

By checking the **User defined flange left/right** properties it is possible to manually override the calculated effective flange widths and depths if required. When a user defined flange width has been defined, the user specified allowance for openings is not applicable.

Summary of Flange Modelling and Design Choices

The combinations of beam flange modelling and design choices can be summarized as follows:

Consider flanges	Include flanges in analysis	User defined flanges	Outcome
off	N/A	N/A	The beam is analysed and designed as a rectangular beam.
on	off	off	The beam is analysed as rectangular, effective flanges (as defined by the head code) are used in the design.
on	off	on	The beam is analysed as rectangular; user defined flanges are used in the design.
on	on	off	Effective flanges (as defined by the head code) are used in both the analysis and design.
on	on	on	User defined flanges are used in both the analysis and design.



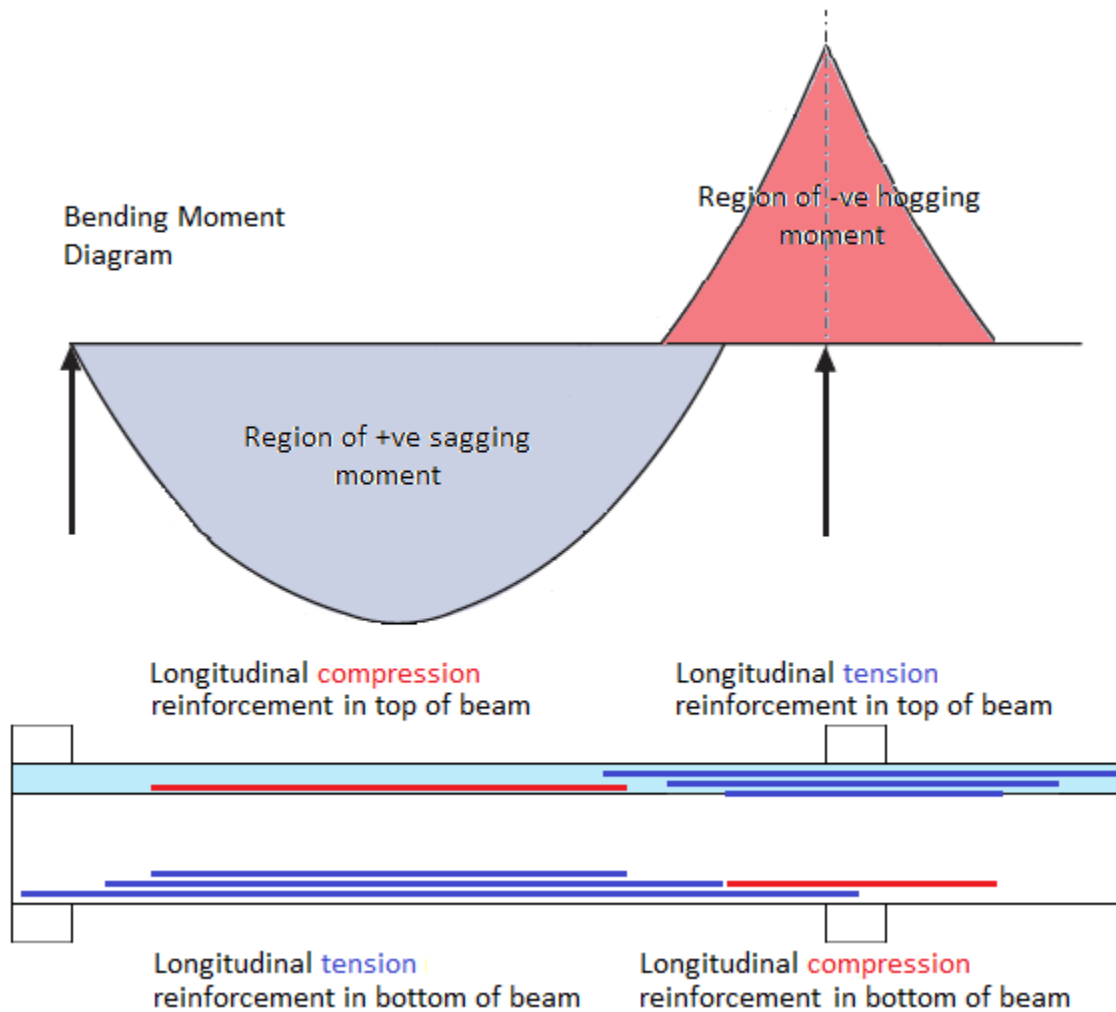
Beams that are curved in plan can have flanges considered, but for such beams the flanges must always be user defined.

Longitudinal reinforcement (concrete beam)

In order to determine the design forces for the bending checks user defined longitudinal reinforcement regions must first be established.

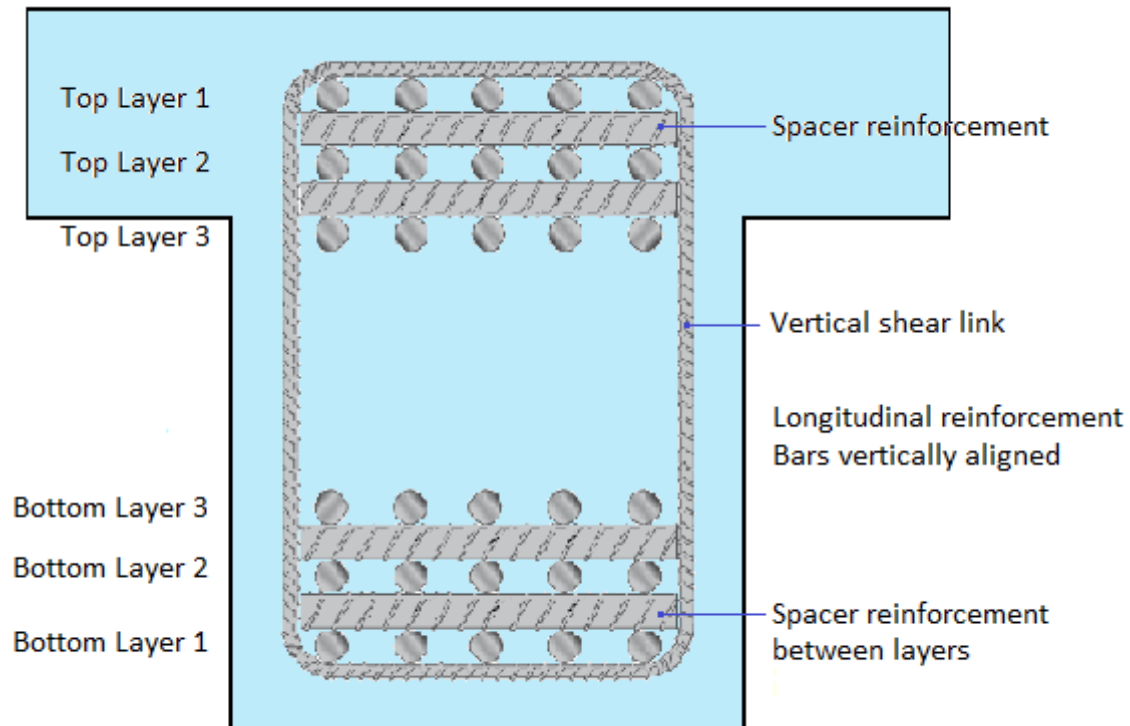
Bar layers

Designed longitudinal reinforcement is positioned in the top and bottom of the beam and can be tension reinforcement or compression reinforcement.



The longitudinal reinforcement in the top and bottom of a beam can consist of 1 to " n_L " parallel layers with the layer nearest to the top or bottom surface of the beam being Layer 1.

The number and diameter of bars in each layer can vary but bars in different layers must be vertically aligned. This is to ensure that there is adequate space to allow the concrete to be poured and properly compacted around the bars.

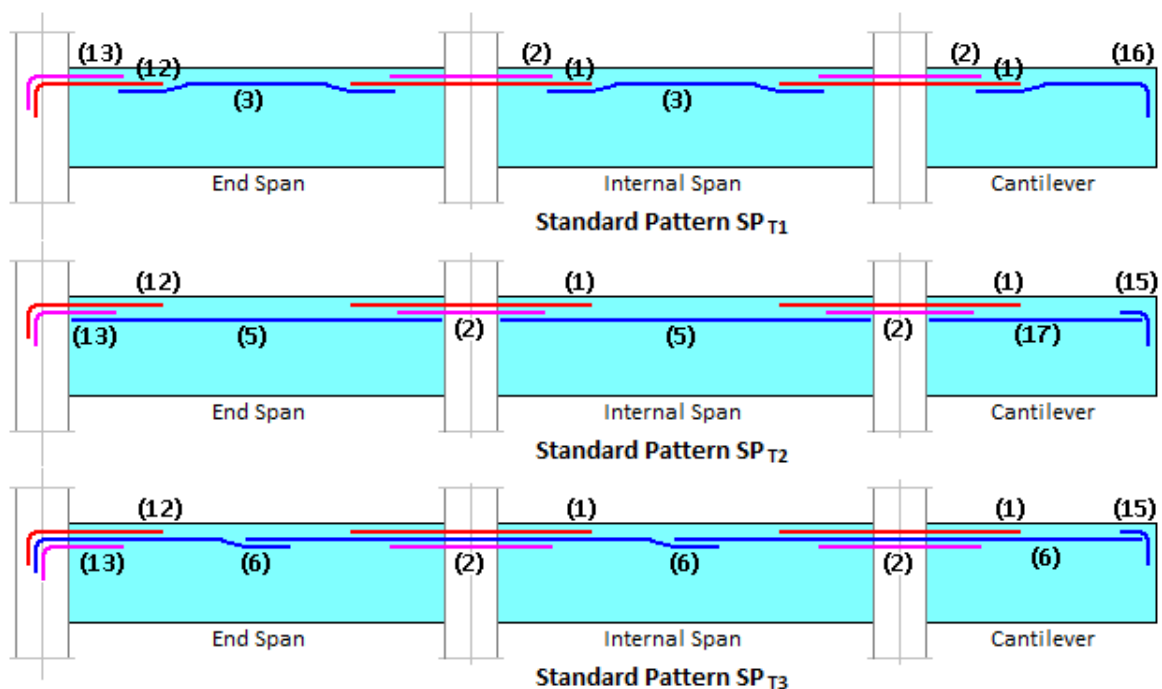


Longitudinal reinforcement in the side of the beam is only provided in beams with a depth greater than a certain value as follows:

Longitudinal Reinforcement Patterns Library

There are three Standard Patterns for top reinforcement, SP_{T1} , SP_{T2} and SP_{T3} and two Standard Patterns for bottom reinforcement, SP_{B1} & SP_{B2} as illustrated in the figures below.

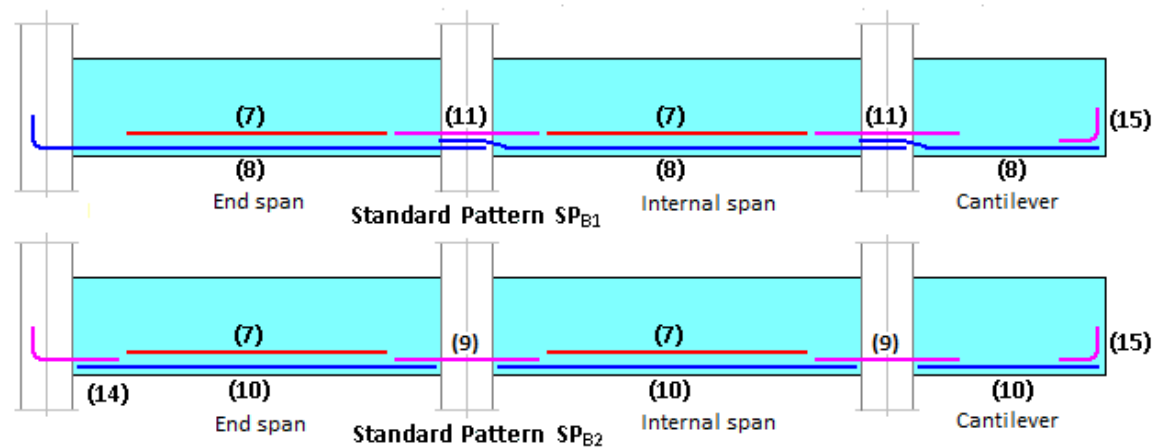
Standard Patterns of Top Reinforcement



The bars used in the Standard Top Patterns are:

- (1) Straight bar extending to approximately 25% 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 centre span to centre 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



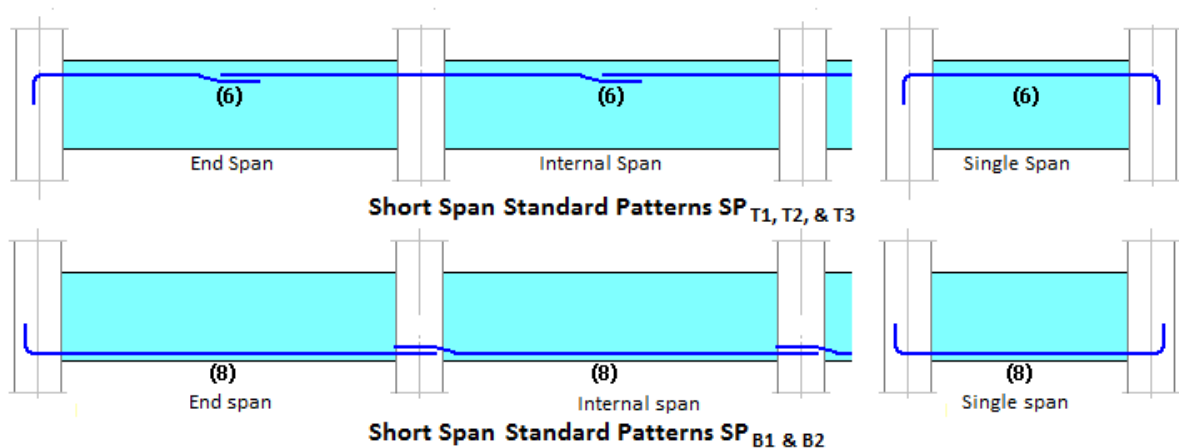
The bars used in the Standard Bottom Patterns are:

- (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



Longitudinal Reinforcement Regions

Design Check Regions for Bending

When considering the longitudinal steel in the top and bottom of the beam, the design checks are performed in a specified number of regions that are symmetrically placed about the centre of the beam. The regions are specified as user defined proportions of the clear span of the beam, expressed as a percentage; the number of regions being initially governed by the choice of longitudinal bar pattern.

Top Regions

Three standard patterns are available for defining the top regions:

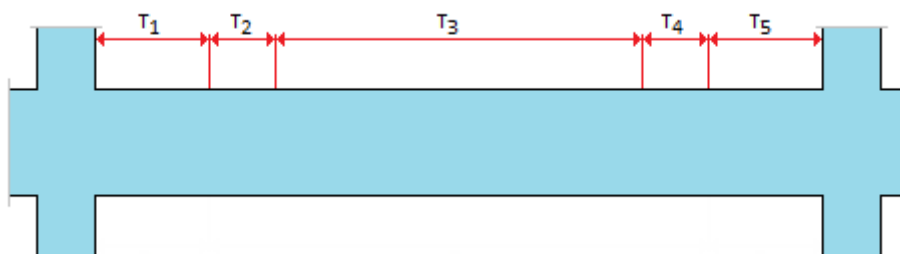
Standard Top 1 – maximum of 3 regions

Standard Top 2 – maximum of 5 regions

Standard Top 3 – maximum of 6 regions

To provide flexibility, once a pattern has been selected you are able to vary the percentage region widths, with the range of each being $0\% \leq T_i \leq 100\%$ and with $\sum T_i = 100\%$.

For example the Standard Top 2 pattern initially consists of 5 regions, T_1 , T_2 , T_3 , T_4 , and T_5 :



By varying the percentage region widths a number of possibilities can be catered for:

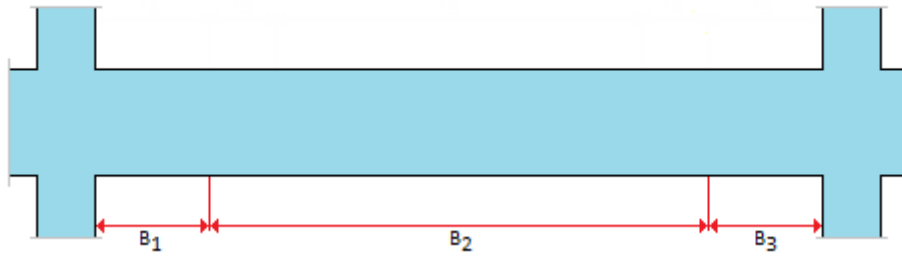
1 Region $T_1=T_5 = 0\%$; $T_2=T_4= 0\%$; $T_3= 100\%$

3 Regions $T_1=T_5 = 0\%$; $T_2=T_4 = 100\%-T_3\%$; $0\% < T_3 < 100\%$

5 Regions $T_1=T_5 > 0\%$; $T_2=T_4 > 0\%$; $T_3 > 0\%$.

In each top region, the maximum negative bending moment within the region is determined for design purposes.

Bottom Regions



Two standard patterns are available for defining the bottom regions:

Standard Bottom 1 – maximum of 3 regions

Standard Bottom 2 – maximum of 3 regions

Similar to the top patterns, once a pattern has been selected you are able to vary the percentage region widths, with the range of each being $0\% \leq B_i \leq 100\%$ and with $\sum B_i = 100\%$.

This enables the following single region, or three possibilities:

1 Region $B_1=B_3 = 0\%$; $B_2 = 100\%$

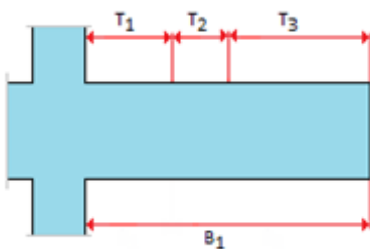
3 Regions $B_1=B_3 = \text{RANGE}(0\%-25\%)$; $B_2 = \text{RANGE}(50\%-100\%)$

In each bottom region, the maximum positive bending moment within the region is determined for design purposes.

Regions for Cantilevers

The standard patterns for cantilevers are edited and applied in the same way as the standard patterns for continuous spans. Up to 3 regions can be defined for the top, but only a single region exists for the bottom.

These regions are illustrated below



The design value of the bending moment used for the design in a region is the maximum factored bending moment arising in the region under consideration.

Relationship between Reinforcement Patterns and Design Regions

There is a close link between the reinforcement patterns and the design regions. After selecting a Standard Reinforcement Pattern, you can then choose the length of each design region. The number of regions adopted will dictate the bars that are used to reinforce the beam and likewise, the selection or de-selection of particular bars will dictate the design regions used.

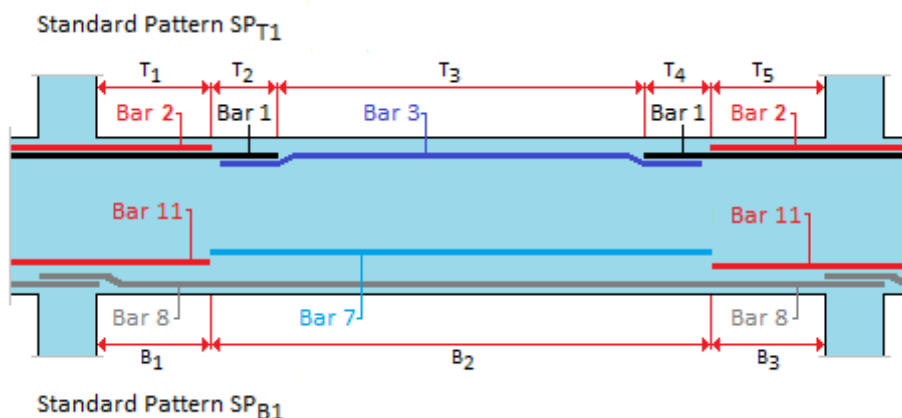
The selection process of Standard Reinforcement Patterns and Design Regions is:

1. Choose a Standard Reinforcement Pattern for the top reinforcement (from the **Standard Pattern Setup** droplist)
2. Select the bars that are to be used
3. Set the length of the resulting design regions
4. Repeat the selection process for the bottom reinforcement.

The bar selection and design region lengths are inextricably linked. If a bar is not selected then the design region has zero length.

It is important that the bar and design region selection is carried out in an orderly manner and that the selections are made in the correct order.

This relationship between bars and design regions is best illustrated using Standard Patterns SP_{T1} and SP_{B1} as an example (for an internal span) as shown in the diagram below.



Considering the top reinforcement first, if the user opts to de-select Bar 2 then design regions T_4 and T_5 will be zero length and the user will then select a length for T_1 .

Likewise for the bottom reinforcement, de-selecting Bars 7 and 11 will set design regions B_2 and B_3 to zero length.

If all the available bars are selected in this example then the bars used to provide the area of reinforcement required by the design in each design region will be;

Design Region T_1 (& T_5) : Bar 1 + Bar 2

Design Region T_2 (& T_4) : Bar 1

Design Region T_3 : Bar 3




Design Region B_1 (& B_3) : Bar 8 + Bar 11

Design Region B_2 : Bar 7 + Bar 8

The above approach is extended for all the Standard Patterns.

Shear reinforcement




Shear Reinforcement Shapes Library



Vertical shear reinforcement is provided in the form of which can be single or multiple with 1 () or 2 ( or ) vertical legs.

The common basic shapes of bars used for the purposes of providing shear reinforcement in beams are shown in the table below.

In the current release, only the shapes listed in the table are available for selection.

Shear Reinforcement Typical Shapes

BS8666 Shape Code	Link/Stirrup Shape	Description
51		Closed Link/Stirrup
47		Open Link/Stirrup
21		Top Closer Link/Stirrup

99		Single Leg Link/Stirrup
63		Torsion Link/Stirrup

Shear Reinforcement Patterns Library

There are three Standard Patterns for shear reinforcement, *Closed*, *Open* and *Torsion*. However, you are initially only offered a choice of 2 patterns, Closed or Open.

The Standard Patterns for shear reinforcement are:

Closed

Closed (shape code 51) with additional double leg (shape code 51) or single leg (shape code 99) if required by the design. The design dictates the number of vertical legs required. You can choose if single leg are acceptable.

Open

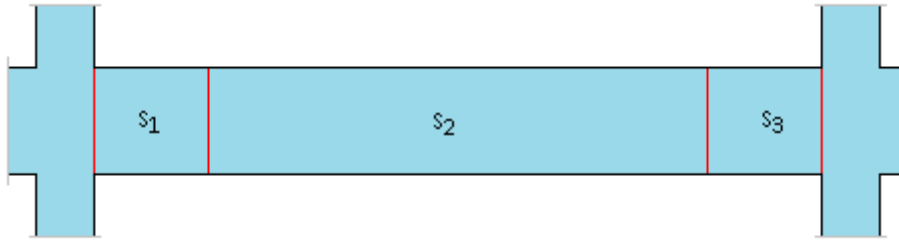
Open with top closers (shape codes 47 and 21) with additional double leg (shape code 47) or single leg (shape code 99) if required by the design. The design dictates the number of vertical legs required. You can choose if single leg are acceptable.

Torsion

Torsion (shape code 63) for the outer with closed (shape code 51) or single leg (shape code 99) as internal if required by the design. Note that this is not a user option but is determined by the design.

Shear Reinforcement Regions

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 and analysis types, V_{zi} , is determined and this maximum value used to determine the shear reinforcement required in that region.



The lengths of the shear regions are subject to user selection and may be either:

Optimised

This option is only valid when the maximum positive shear from all combinations and analysis types occurs at one end of a beam and the maximum negative shear from all combinations and analysis types occurs at the other end of the beam. If this situation does not exist then this option is not allowed and the "Fixed Proportions" method will be used.

In this case in the central region S_2 , shear reinforcement is provided to meet the maximum of the minimum code requirement or minimum 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.

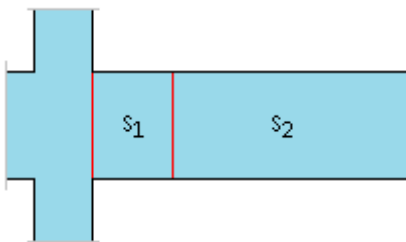
In this method, region S_2 is defined as being that part of the beam in which the minimum amount of shear reinforcement is acceptable.

Or

Fixed proportions

In this case the regions are defined as fixed proportions of the clear span [face to face length] of the beam expressed as a percentage $S_1\%$, $S_2\%$ and $S_3\%$ with the default values for S_1 and S_3 being $\text{MAX}(0.25 \cdot L, 2 \cdot h)$ and that for S_2 being $(L - S_1 - S_3)$.

In cantilevers, the regions are as shown below.



In all cases, the range of each region is $0\% \leq S_i \leq 100\%$ and $\sum S_i = 100\%$.

Concrete column design

Autodesign (concrete column)

The design mode for each column is specified in its properties.



If concrete columns have been set to be designed using [Design and detailing groups \(concrete\)](#), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

- 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 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 only appears if **Autodesign** is selected. It sets the autodesign start point for both longitudinal bars and .

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.



*When a column 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.)*

Section (concrete column)

Shape

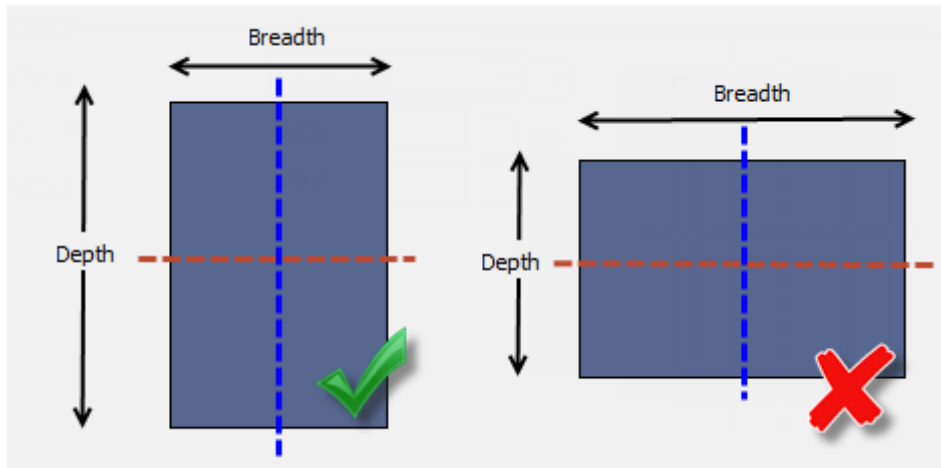
The following column shapes are allowed:

- Rectangular
- Circular
- L-shape
- T-shape
- C-shape
- Elbow with uniform thickness
- Trapezium

- Parallelogram
- Regular polygon (between 3 and 8 sides)

Breadth and Depth

When defining a section with two axes of symmetry (e.g. a rectangular section) you should ensure that the longer dimension is input as the depth and the shorter dimension as the breadth (as shown below left).



A rotation can then be applied if required, in order to orientate the column correctly in the model.

When the section is designed the “major axis” calculations will then relate to bending about the strong axis and the “minor axis” calculations to bending about the weak axis.

If the breadth and depth has been transposed during input (as shown above right), “major axis” would then relate to bending about the weak axis and “minor axis” would relate to bending about the strong axis.

Holes

Rectangular or circular holes can be placed in rectangular and circular columns when the section is being defined, they can’t be placed in other column shapes.

Reinforcement link arrangements such as double links, triple links and cross-links are not designed intelligently to account for holes.

Slenderness (concrete column)

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.

Clear height

The clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) 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 unrestrained length may be different in each direction.

When determining the unrestrained 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.

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 unrestrained length includes the stack beyond this restraint, and the same rules apply for finding the end of the unrestrained 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 unrestrained length ends at the end of the column.

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.

Longitudinal Reinforcement (concrete column)

Reinforcement bars are located in the faces of the section. For straight-edged sections, a principal bar may be located at each link corner, and intermediate bars may be located between these principal bars.

You can select a bar size for principal bars and a different bar size for intermediate bars. Similarly, auto-design methods can create designs where there is either one or two different bar sizes.

In the design process, principal bars are treated as individual bars and intermediate bars are treated as groups of bars, with each group existing between two principal bars.

All principal bar locations have a number for the bar referencing which is consistent for all columns of the same section type. Intermediate bar groups are referenced using the references of the principal bars at each end of the group.

Column design forces

Six design forces (axial, torsion, major and minor moment, major and minor shear) are determined at the top and bottom of each stack for each combination for one or more sets of analysis results.

Intermediate loads are allowed. The moments due to imperfections and slenderness effects are added appropriately, and the result checked against the minimum design moment as appropriate.

Column interaction diagrams

To visually observe the utilisation 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".

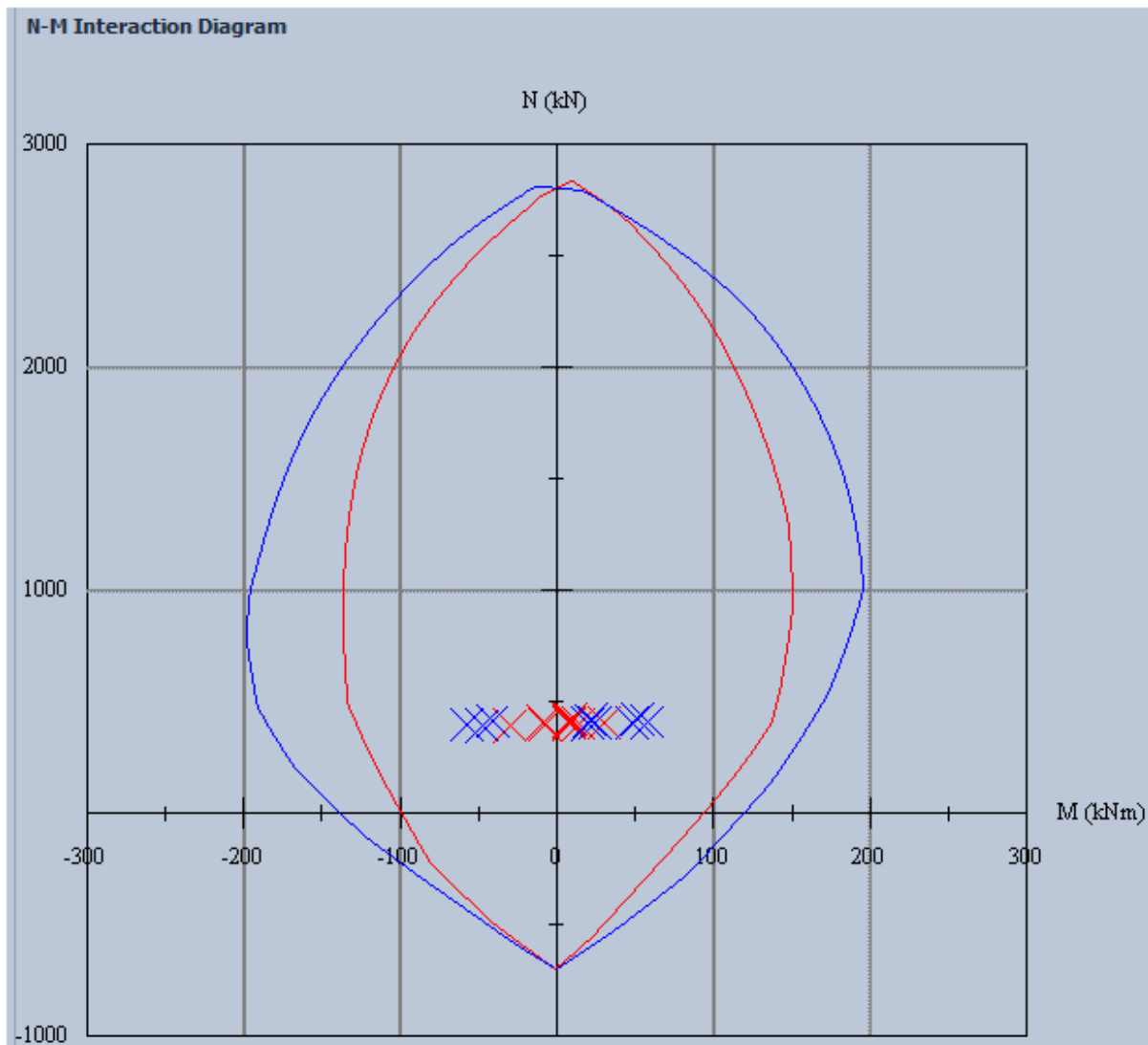
When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

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.

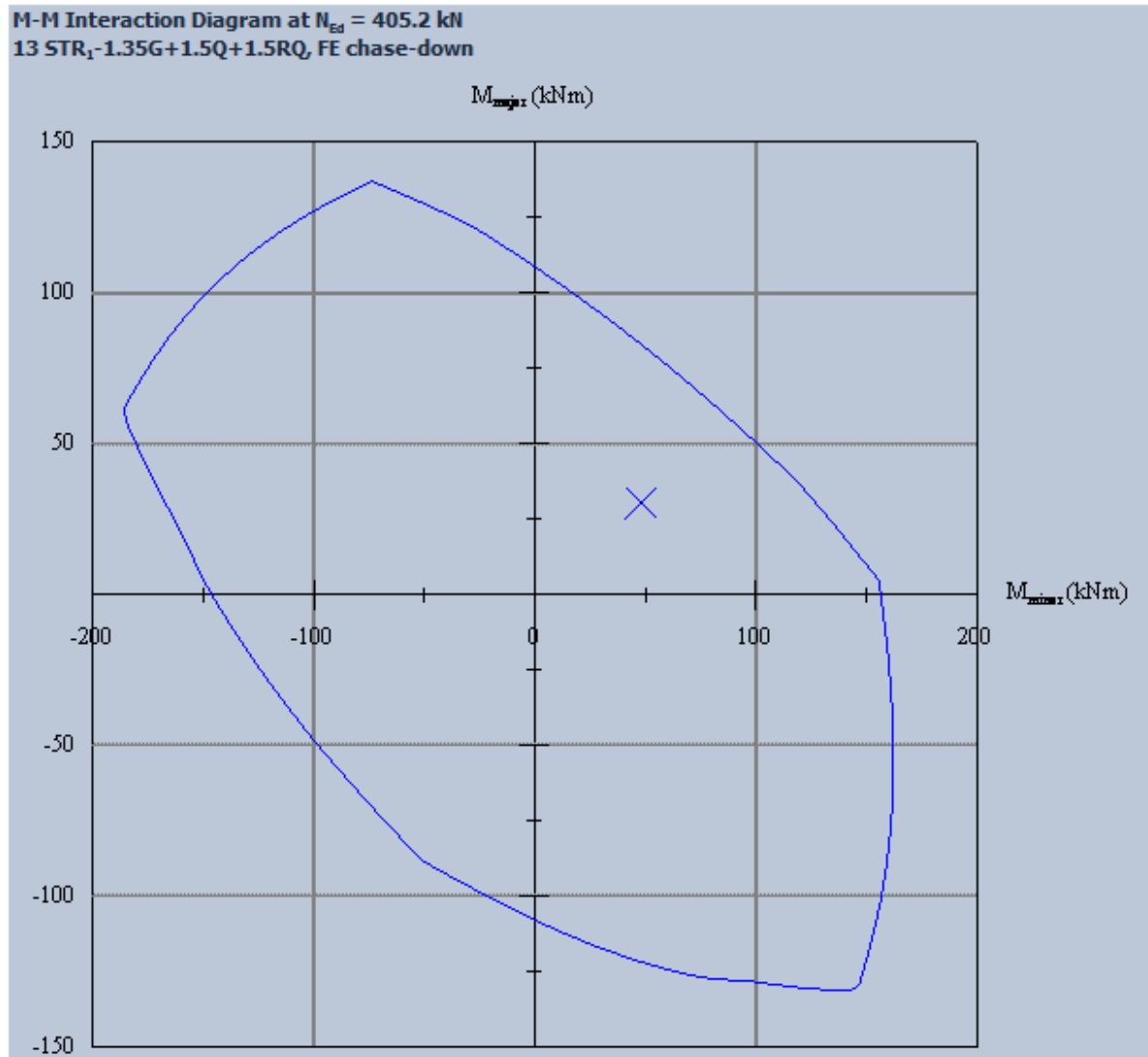


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

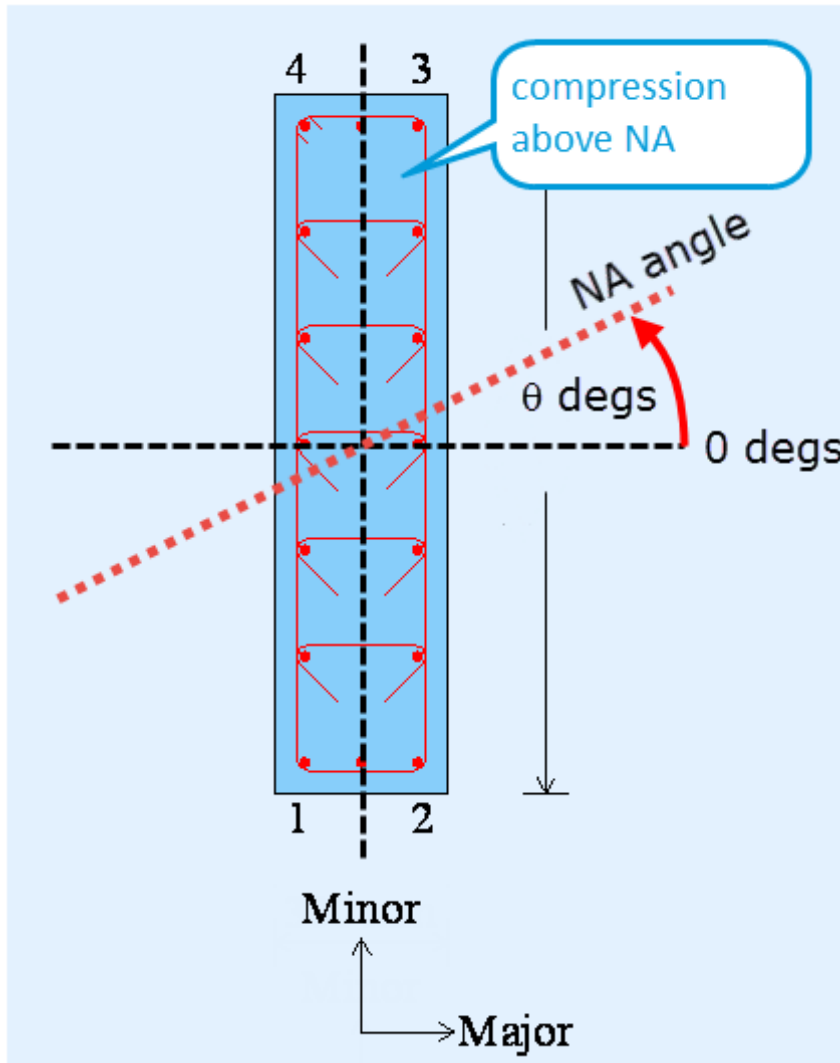
Column moment Interaction diagram

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.

Convention used for neutral axis rotation.



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.

Concrete wall design

Autodesign (concrete wall)

The design mode for each wall is specified in its properties.

- When **Autodesign** is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and . This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible 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 only appears if **Autodesign** is selected. It sets the autodesign start point for both longitudinal bars and .

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.



*When a wall is in check mode, it can still be autodesigned "on the fly" by choosing **Design Wall** from the right-click menu. In this case it uses the **Select bars starting from** option currently assigned to the wall for autodesign. (This property is hidden when in check mode, but can be confirmed if required by switching the wall back to*

Autodesign mode.)

Slenderness (concrete wall)

The significant parameter within the slenderness criteria is the choice of how the wall is contributing to the stability of the structure.

- In-plane (major) direction, a wall is usually considered to be a bracing member.
- Out-of-plane (minor) direction, a wall is usually 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.

Clear height

The clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) 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 unrestrained length may be different in each direction.

When determining the unrestrained 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 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.

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 unrestrained length includes the stack beyond this restraint, and the same rules apply for finding the end of the unrestrained 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 unrestrained length ends at the end of the column.

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.

Reinforcement (concrete wall)

The wall properties: **Reinforcement layers**, **Form** and **Include end zones** 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

Wall design forces

Six design forces (axial, torsion, major and minor moment, major and minor shear) are determined at the top and bottom of each stack for each combination for one or more sets of analysis results.

Intermediate loads are allowed. The moments due to imperfections and slenderness effects are added appropriately, and the result checked against the minimum design moment as appropriate.

Walls can be loaded laterally, but are always considered to span vertically.

Horizontal moments - that is M_x out of plane moments about a vertical axis - that may develop in a meshed wall are ignored in the design.

Wall interaction diagrams

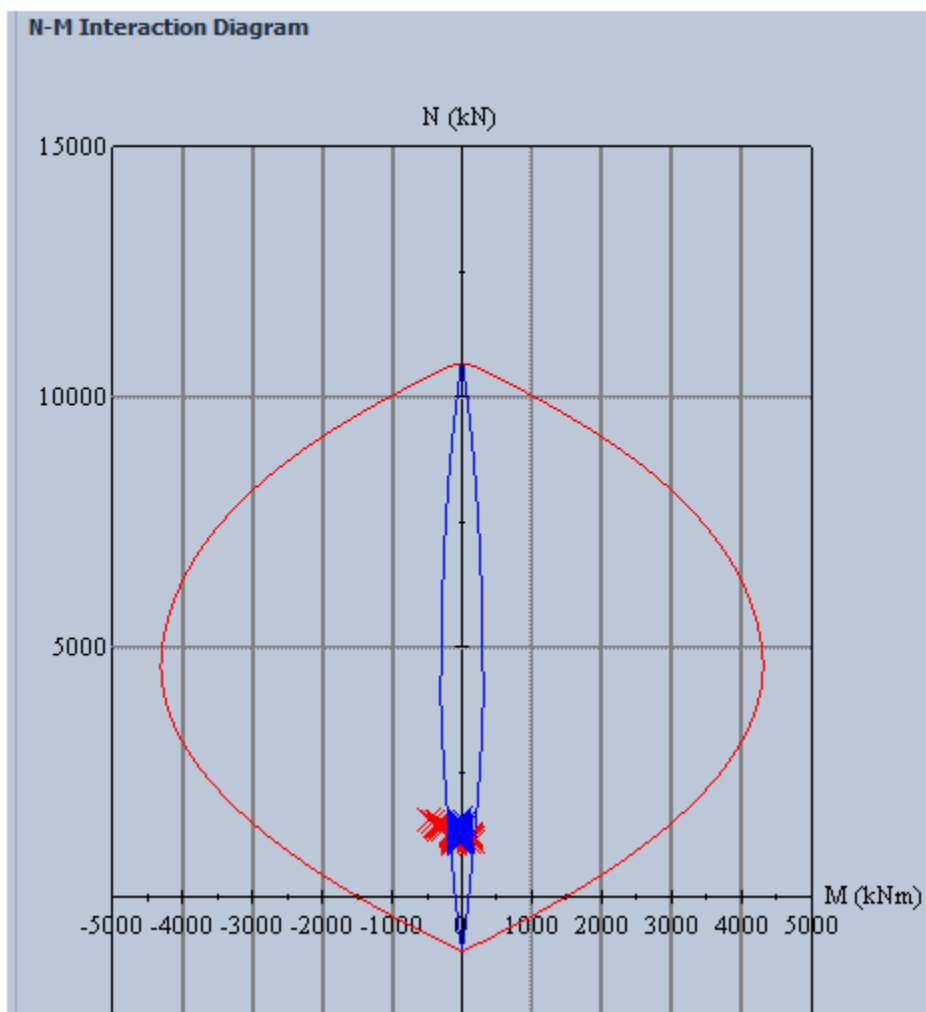
To visually observe the utilisation of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

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.

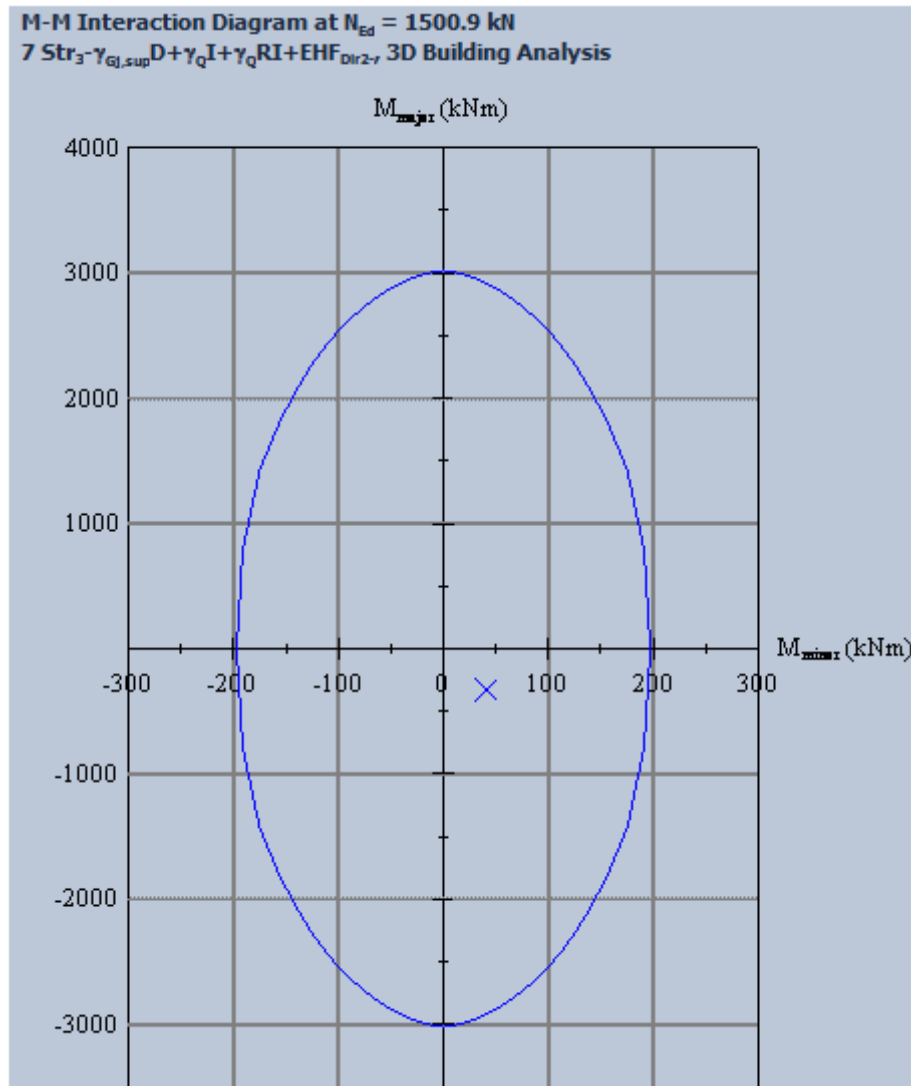
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive 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.

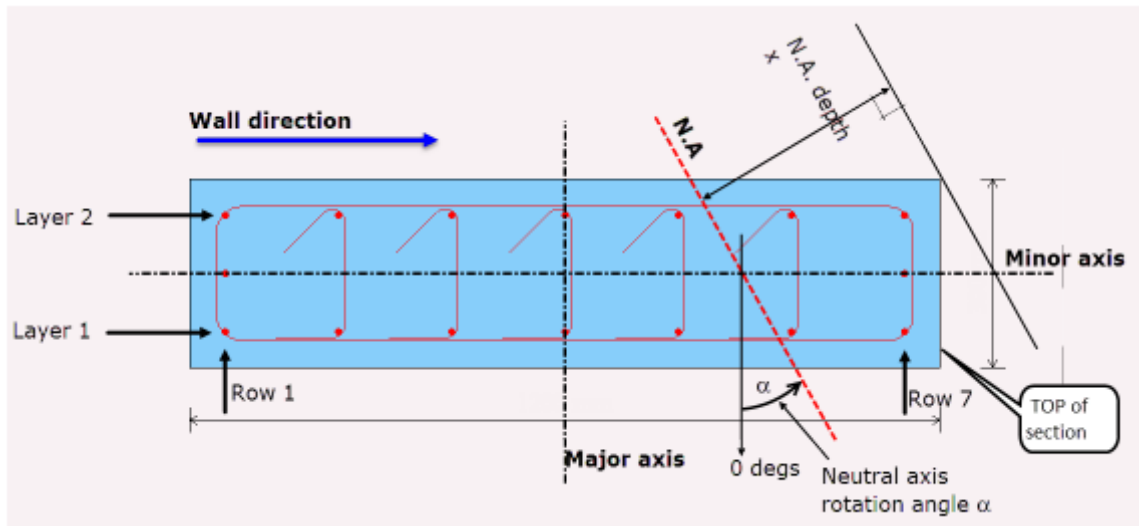
Wall moment Interaction diagram

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.

Convention used for neutral axis rotation



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.

Concrete slab design

Slab types designed by Tekla Structural Designer

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.

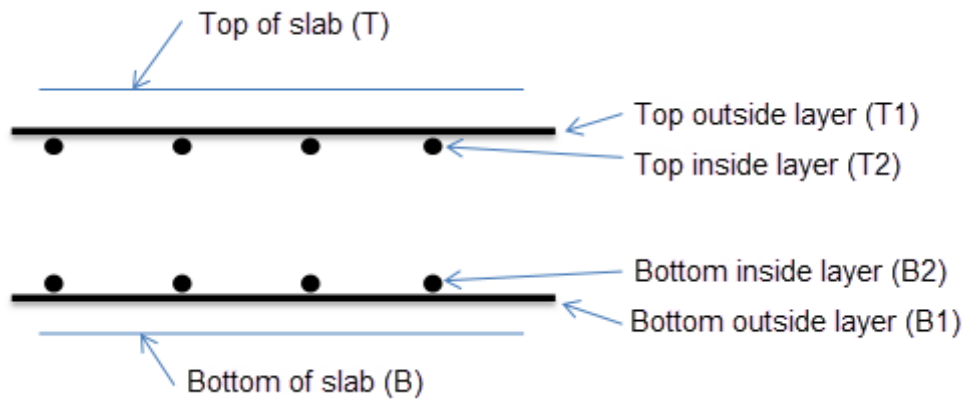
Two-way spanning slab panel design moments

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).

Slab 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.

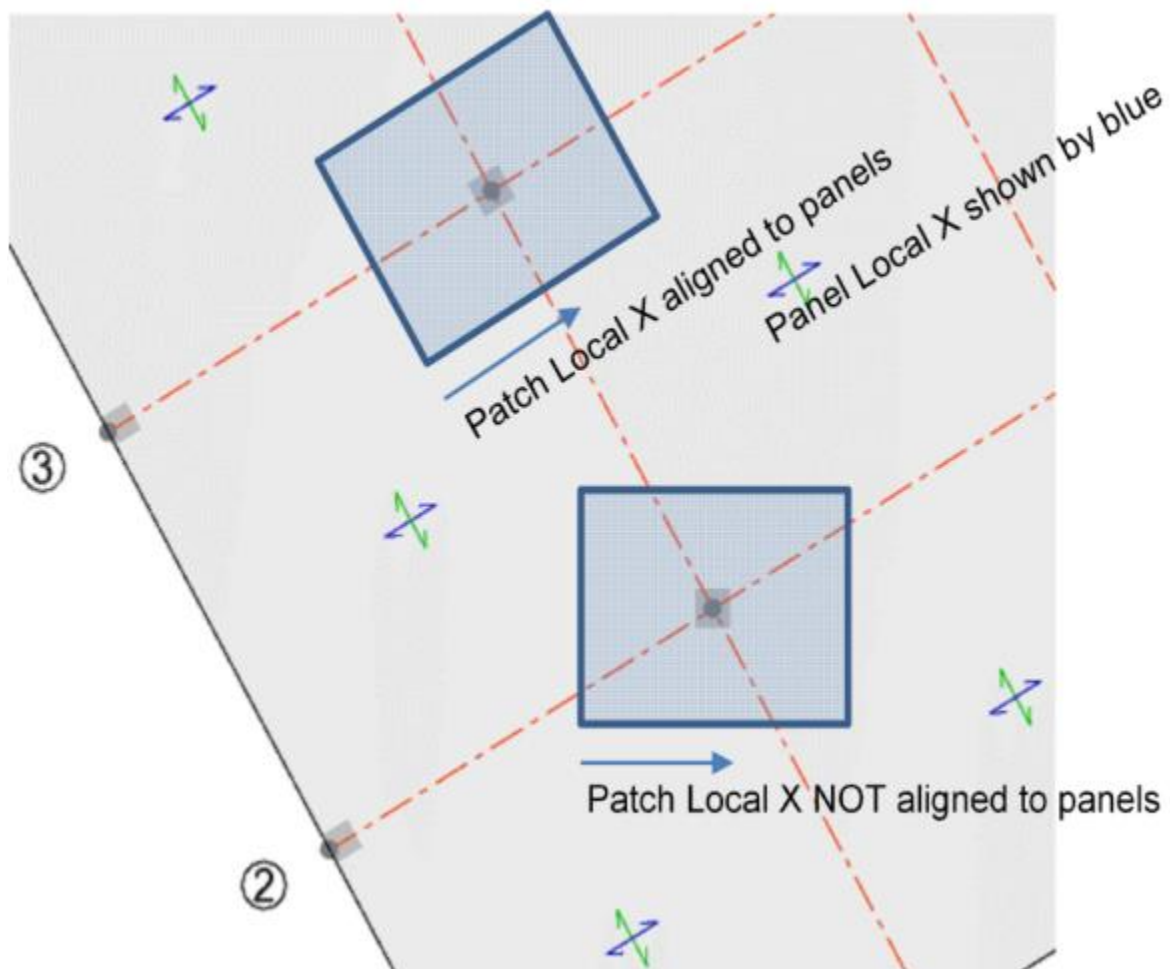
Note that patches may overlap on the plan view, and there is no restriction on this, even patches relating to the same layer of reinforcement are allowed to overlap. This situation is handled conservatively during design by simply ignoring the overlap.

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:

- a. Patch reinforcement type = NONE
then the panel reinforcement is used for all layers
- b. 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.
- c. 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**.

Slab panel design checks

For each slab panel item the basic design checks performed are:

1. For each layer of reinforcement:
 - a. Moment Capacity Checks
In all areas where no relevant patches exist, is $A_{s_prov} > A_{s_reqd}$?
 - b. Limiting Parameter Checks
Are other limiting parameters such as min reinforcement area and spacing limits satisfied?
2. For "Beam and Slab" panels - are Span / Effective Depth Checks satisfied?



For Flat Slab panels - Span / Effective Depth Checks are not performed automatically. In practice flat slab models are often irregular, so engineering judgement is required when assessing the points between which spans should be checked.

Autodesign (concrete slab)

The design mode for each slab item is specified in its properties.

- When **Autodesign** is selected an iterative procedure is used to select the bars/mesh.
- 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 only appears if **Autodesign** is selected. It sets the autodesign start point for bars/mesh.

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.



*When a slab item 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 slab panel for autodesign. (This property is hidden when in check mode, but can be confirmed if required by switching the slab item back to Autodesign mode.)*

Flat slab deflection checks

Provided that you are designing to either the Eurocode and ACI/AISC Head Codes **iterative cracked section analysis** is available for the purpose of checking slab deflections. This approach is covered in detail in the [Slab Deflection Handbook](#).

Alternatively a “Deemed-to-Satisfy” approach can also be applied (which might be necessary if for example you are designing to a different Head Code). In this less sophisticated method flat slab deflections are assessed manually by reviewing the 2D deflection contours resulting from the **FE chasedown analysis**. (Corresponding deflections for the **3D analysis** will only be available if you have elected to mesh 2-way slabs in the analysis.)

In the “Deemed-to-Satisfy” approach deflections can then be assessed manually between appropriate points in the slab in order to check the slab thickness is sufficient.



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.



Similarly, engineering judgement is required to ensure appropriate stiffness adjustments have been applied prior to analysis.

The deflections obtained from the analysis will depend on the applied stiffness adjustments. These are specified from the Analyse ribbon by selecting **Options > Modification Factors > Concrete**.

The default stiffness adjustments are dependent on the design code.

To help you to assess whether the default adjustments will produce a suitable deflection estimate, the different effects influencing slab deflection are discussed in more detail below.

Overview of Concrete Slab Deflection

Total concrete slab deflection can be broken down into 3 main parts each of which is often estimated as accounting for around 1/3 of the total deflection:

- Instantaneous Deflection
expected to be up to or around 1/3.
- Creep
expected to be > 1/3.
- Shrinkage
expected to be < 1/3.

Various texts may be consulted for opinion on the variation in these proportions and other aspects of deflection estimation:

- Ref 1: Concrete Society Publication TR58 - Deflection of Concrete Slabs and Beams.
- Ref 2: Concrete Society Publication TR64 - Guide to the Design and Construction of Reinforced Flat slabs.

The key point is that they are all of significant importance. Now consider each of these headings in a little more detail.

Instantaneous Deflection

A common rough estimate is that the overall effect of cracking along the length of a beam will on average double the uncracked deflection. Bear in mind that the theoretical degree of cracking varies along the length of the beam. Therefore if you calculate deflection ignoring cracking and creep and shrinkage then the value you have is around 1/6 of the total. In essence this is the basis of the accepted practice/guidance that a multiplier of between 4 and 6 should be applied to deflection calculations based on a short term E value. The 4 to 6

range recognises the different loading types and uncertainties involved, offices are typically assumed to be closer to the 4 and storage situations are assumed to be closer to 6.

Creep Deflection

The deflection of a concrete slab subjected to a constant loading will increase over time. The rate of this increase is dependent on a number of factors, but to accurately assess creep you definitely need to know something about the proposed sequence and timing of loading.

Shrinkage Deflection

If you reinforce a concrete section asymmetrically then as the concrete shrinks the shrinkage will be resisted more on the side that has more reinforcement and so the shortening of that side will be less. A curvature is induced in the member with the more heavily reinforced side being on the outside of the curve. This curvature will naturally add to the overall deflection.

Theoretical solutions for this exist but so far as we know no one has found/applied a way of doing this in the context of a 3 dimensional FE floor analysis. Shrinkage deflection is therefore always added as an adjustment factor.

Slab on beam idealized panels

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.

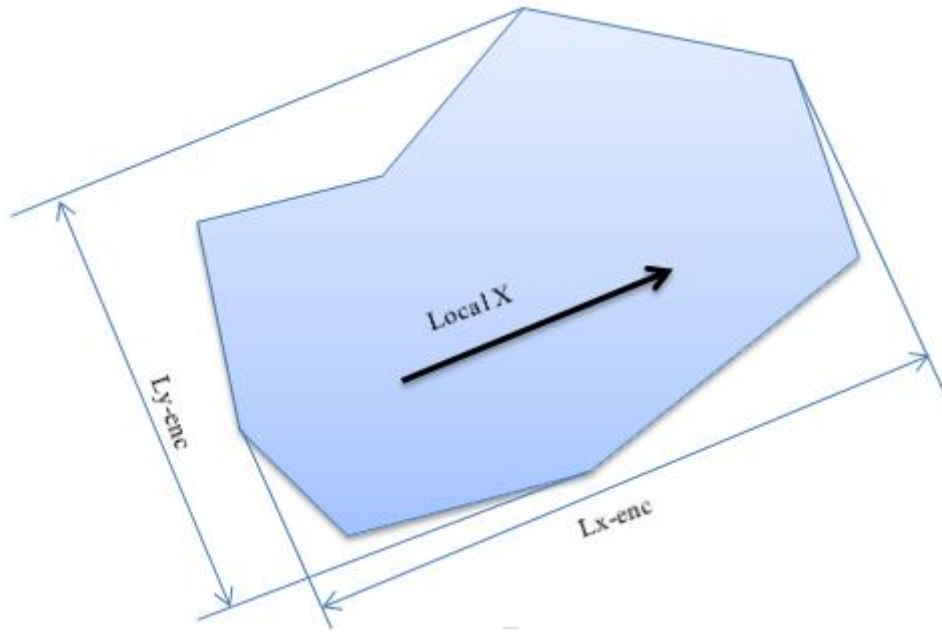
To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel rotation angle):

L_{x-enc} = maximum overall length of the panel measured parallel to local X

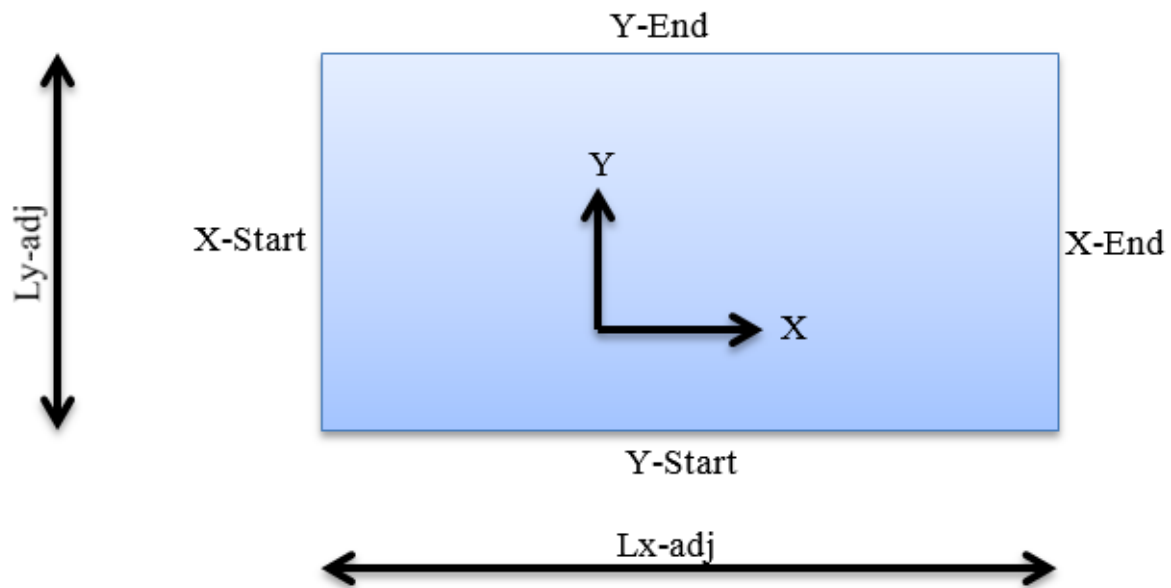
L_{y-enc} = maximum overall length of the panel measured perpendicular to local X

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)

Typical flat slab design procedure

In *Tekla Structural Designer* an interactive design approach is required for flat slab design because the patch and panel design are inter-dependant.

Overall Slab Design Workflow

1. [Split/join panels as necessary and set up Pattern Loading](#)
2. [Analyse All \(or Design All\)](#) - to establish analysis results
3. [Consider Simple \(linear\) Deflection](#)
4. [Select a Level \(or sub-model\)](#) to be designed and within that level:
 - a. [Add Patches](#)
 - b. [Design Panels](#)
 - c. [Review/Optimise Panel Design](#)
 - d. [Design Patches](#)
 - e. [Review/Optimise Patch Design](#)
 - f. [Add and Run Punching Checks](#)

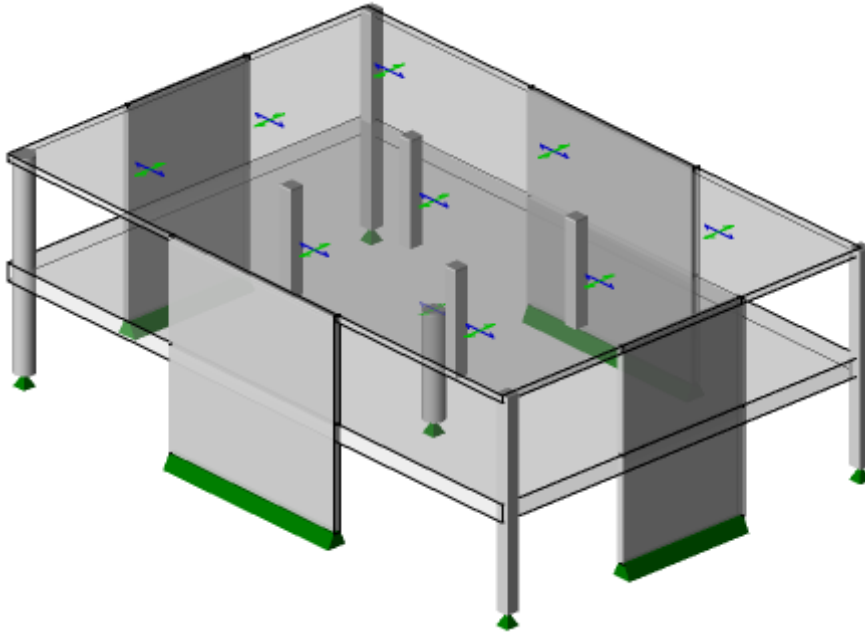
5. [Rigorous Deflection Check](#)

A rigorous deflection check could fit in at any point after step 4d above. If deflection is an issue that dictates the reinforcement provision, then optimising panel and patch reinforcement could be wasting time. On the other hand it is perhaps worth knowing the full extent of reinforcement added to satisfy deflection?

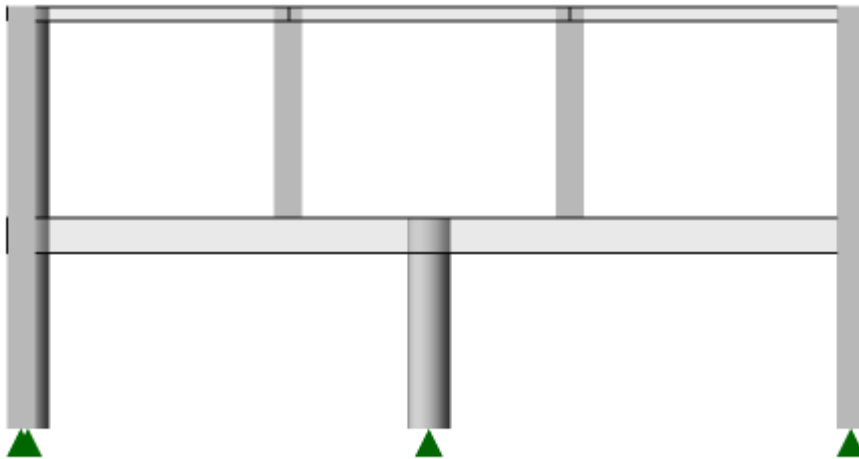
6. Move to next level or sub-model and repeat steps 4 and 5.
7. [Create Drawings and Quantity Estimations](#)
8. [Print Calculations](#)

Flat slab design example

A simple flat slab model as shown below is used in order to demonstrate the techniques involved in the slab design process.

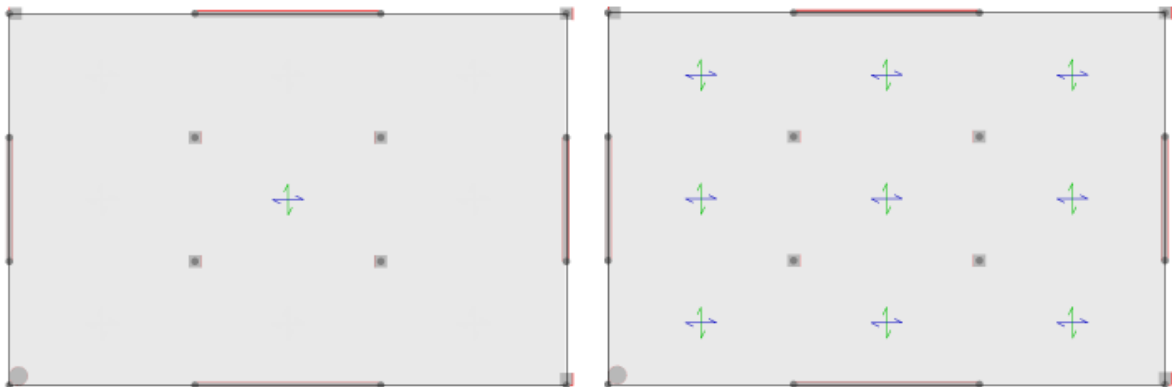


Note that there is a transfer level at the first floor:

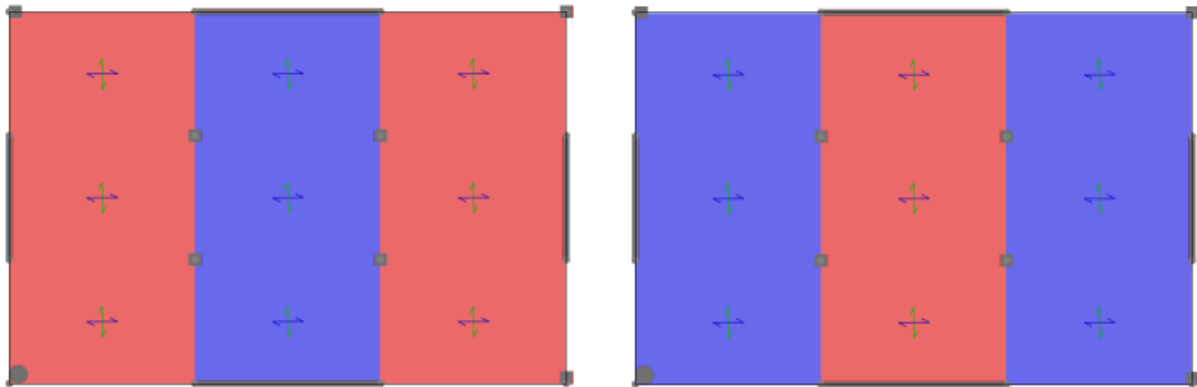


Split/join panels as necessary and set up Pattern Loading

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 meshed 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.



Analyse All (or Design All)

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analysed 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.



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).*

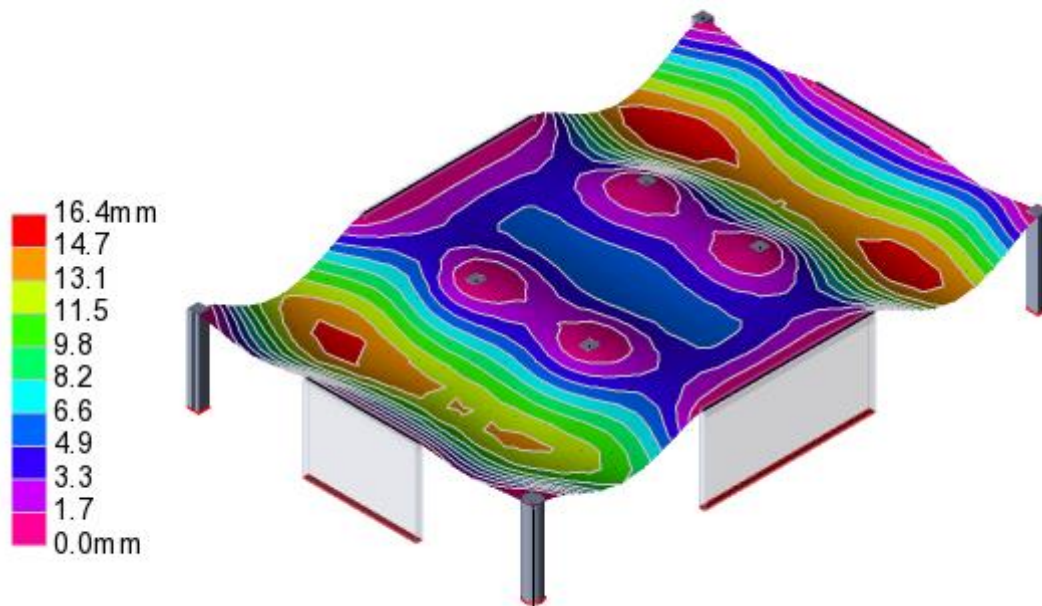
Consider Simple (linear) Deflection

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).



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/m ax=0.0/16.4mm

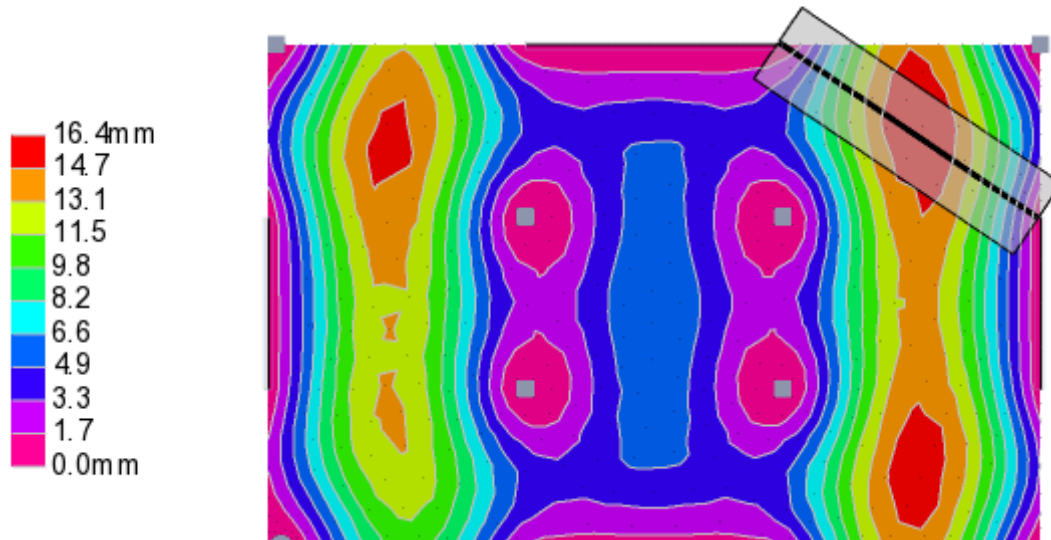


For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

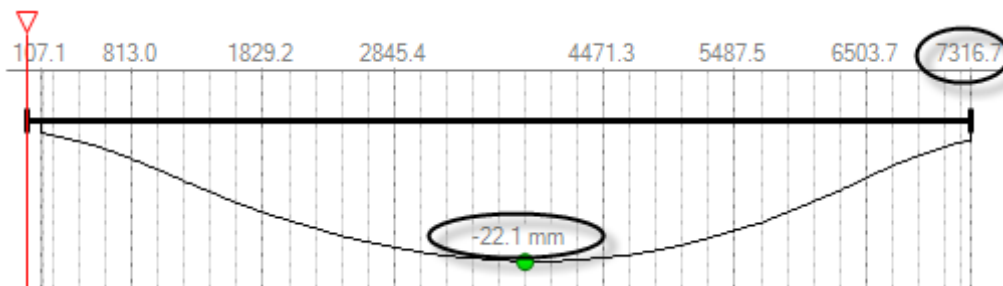
If the Head Code you are working to prevents you from running a [Rigorous Deflection Check](#) you can still use a "Deemed-to-Satisfy" method to assess deflections utilizing slab strips as outlined below:

1. With the level view displayed in 2D, click **Create Strip** and create a strip between supports.

2D Deflection XYZ min/max=0.0/16.4mm



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.



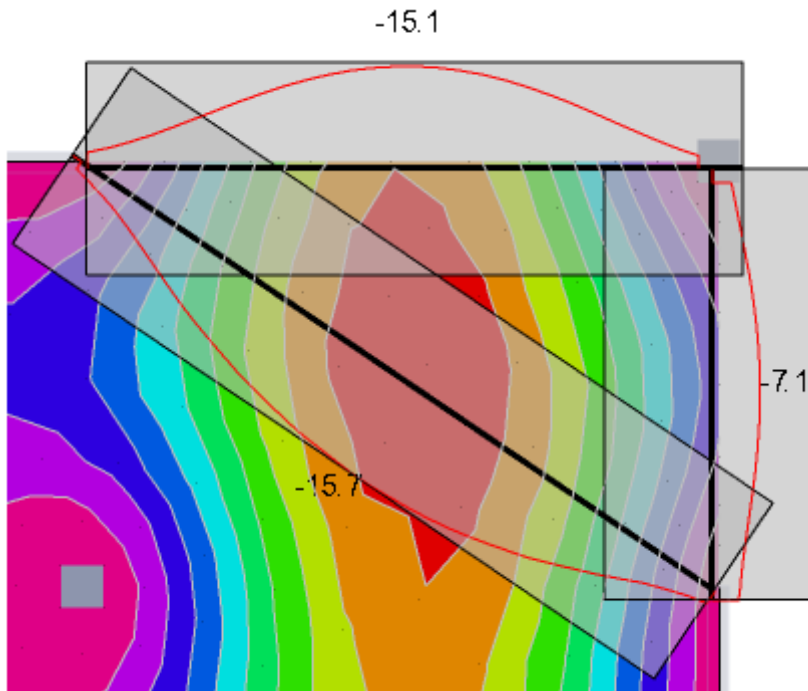
In this example there is a requirement to limit total deflection to span/250

Taking the span as the length of the slab strip: 7317mm

Deflection limit: $7317/250 = 29\text{mm}$

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.



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.

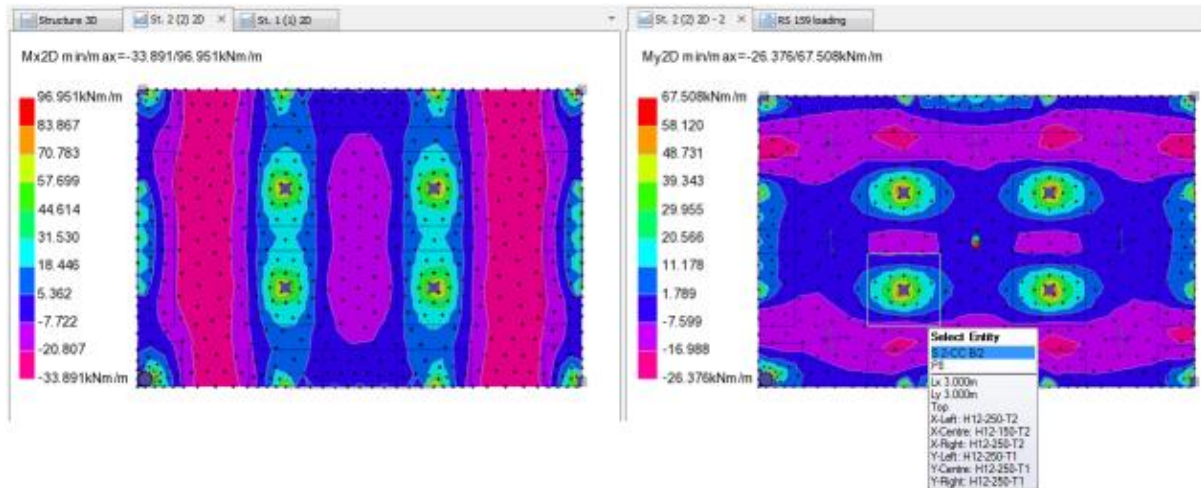
Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

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 optimisation 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



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:

1. 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

2. 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.



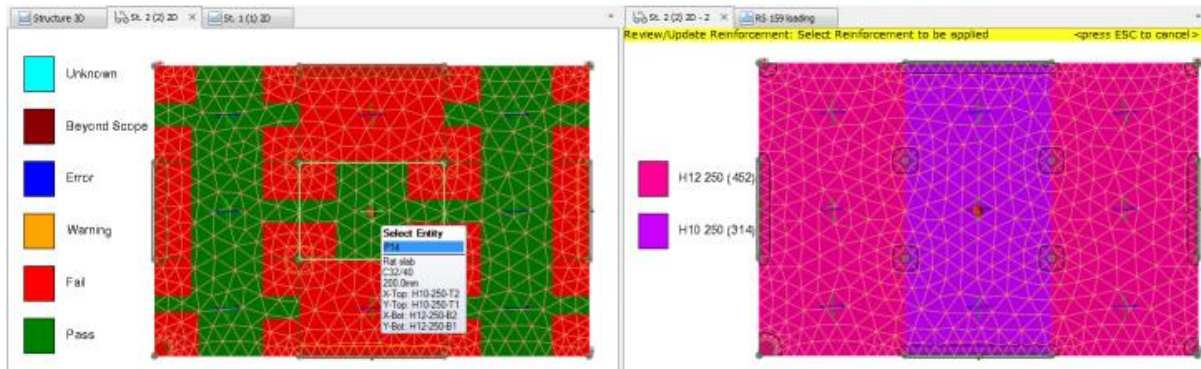
These right-click options operate on the same basis as the options for beams and columns:

- **Design Slabs** will re-design the slabs (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in the slabs regardless of the

current autodesign setting.

Review/Optimise Panel Design

Once again it is suggested that you use split **Review Views** to examine the results as indicated below.



The view on left shows **Slab Design Status**, 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 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.).



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 standardise 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 utilisations 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 rationalised 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:

1. 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

2. 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.



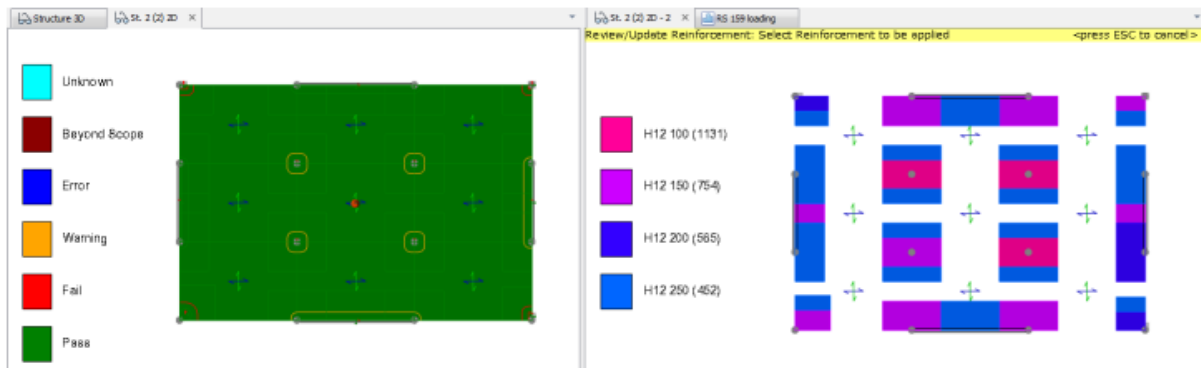
These right-click options operate on the same basis as the options for beams and columns:

- **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/Optimise Patch Design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimised)
- Wall patches - can the width be adjusted (minimised)
- 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 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



*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

Separate examples are provided to illustrate the rigorous deflection checking procedure. See:

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).

Typical slab on beams design procedure

In *Tekla Structural Designer* an interactive design approach is required for 2-way spanning slab on beam design because the patch and panel design are inter-dependant.



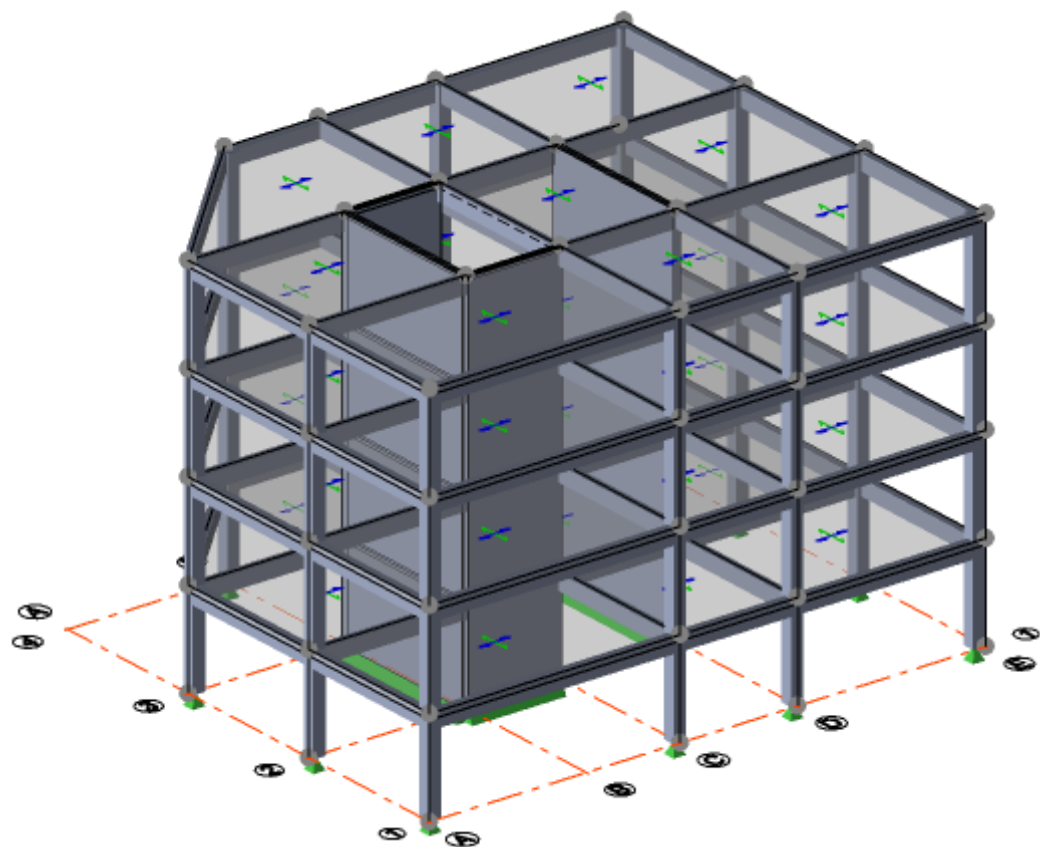
Design of slab panels that have their decomposition property specified as “one-way” is beyond scope - see: [Slab types designed by Tekla Structural Designer](#)

The below example illustrates the design approach, which typically consists of the following steps:

1. [Set up Pattern Loading](#)
2. [Design All](#) - to establish analysis results
3. [Select a Level](#) (or sub-model) to be designed and within that level:
 - a. [Add Beam and Wall Top Patches](#)
 - b. [Design Panels](#)
 - c. [Review/Optimise Panel Design](#)
 - d. [Design Beam and Wall Patches](#)
 - e. [Review/Optimise Beam and Wall Patch Design](#)
4. Move to next level or sub-model and repeat step 4.
5. [Create Drawings and Quantity Estimations](#)
6. [Print Calculations](#)

Slab on beam design example

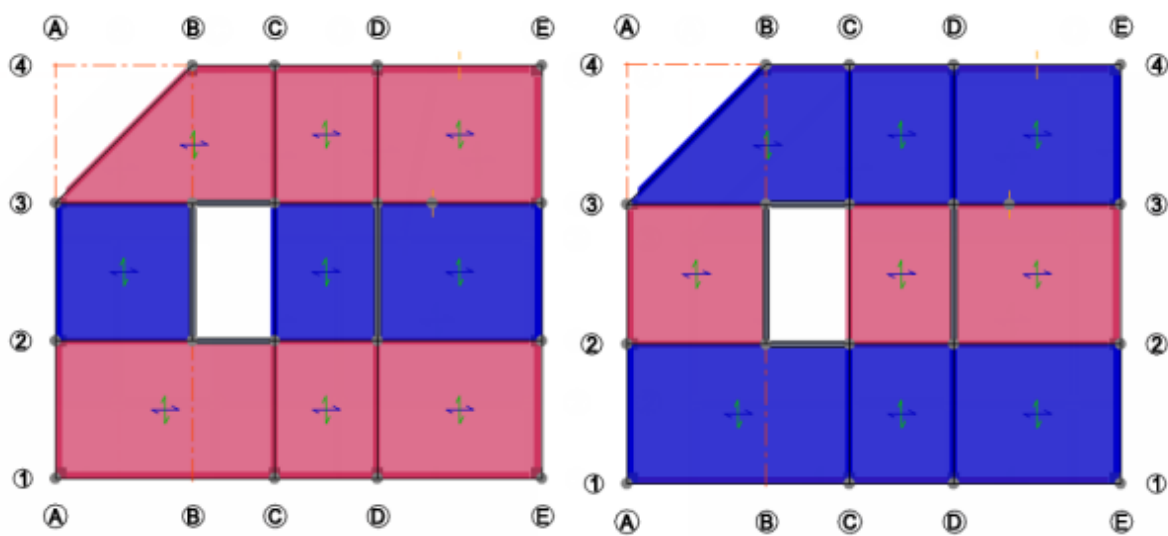
A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.



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 meshed 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.



Design All

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE Chasedown model results - as each floor is analysed 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.



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.



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.

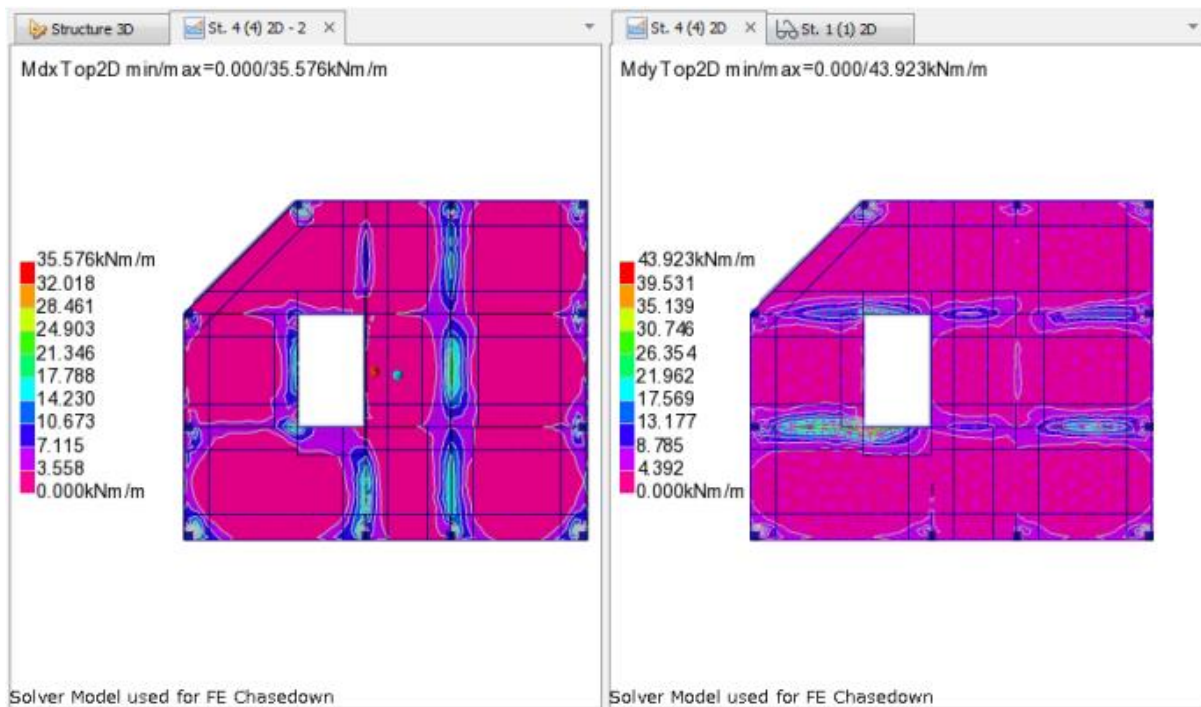
Add Beam and Wall Top Patches

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 optimisation stage.

Design Panels



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:

1. 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

2. 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.



These right-click options operate on the same basis as the options for beams and columns:

- **Design Slabs** will re-design the slabs (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in the slabs regardless of the current autodesign setting.

3. If you chose to set the top reinforcement in the slab panels to **none** but the design detects that top reinforcement is required, the affected panels will fail. In this situation you should increase the widths of the adjacent beam or wall patches before checking or designing the slab panels once again.
4. When panels are being designed (as opposed to checked), the design does not currently automatically increase reinforcement to satisfy deflection, in which case the panels might fail. In this situation you could manually increase the reinforcement until deflection is satisfied.



Adding reinforcement to resolve deflection issues can prove effective when designing to BS codes, however it is a less likely to be effective when designing to Eurocodes.

As part of the design process a span-effective depth check is performed in the appropriate span direction. If the span is the same in both directions, the more “continuously supported” direction is checked, or if both are equally supported the direction which is set to be the outer bar layer is checked.



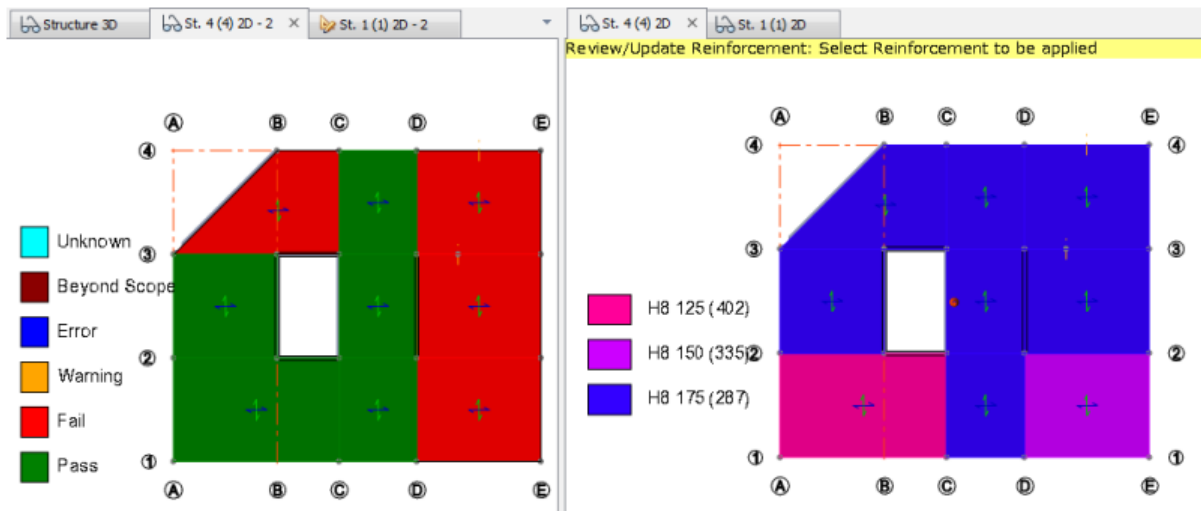
The span-effective depth check takes into account the edge category at the start and end of the span direction being checked, (the program default is to treat each edge as dis-continuous), therefore if the edges are in reality continuously supported, setting them as such will improve the result.



Non-rectangular panels are converted to idealized rectangular ones in order to perform the span-effective depth check calculations. See: [Slab on beam idealized panels](#)

Review/Optimise Panel Design

Review Views can be employed to examine the results and once again it is suggested that you use split views 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.).



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 standardise 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 utilisations 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.

Design Beam and Wall Patches

Having established and rationalised 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:

1. 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

2. 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.



These right-click options operate on the same basis as the options for beams and columns:

- **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/Optimise Beam and Wall Patch Design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimised)?
- Wall patches - can the width be adjusted (minimised)?
- 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.

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).

Performing concrete structure design

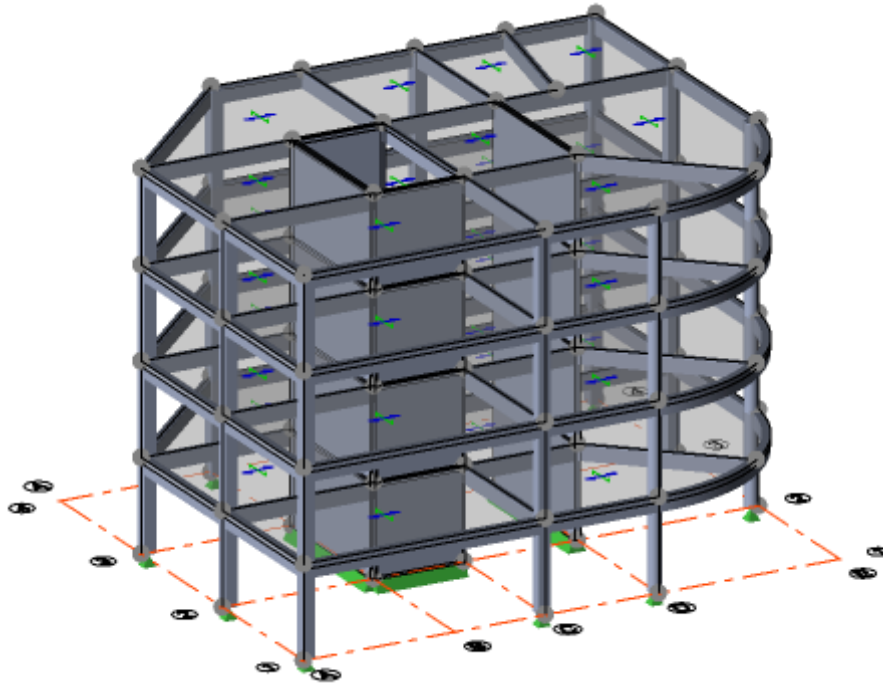
The **Design Concrete** commands on the Design ribbon can be run in order to design or check all concrete beams, columns and walls in the model.



Concrete slabs and foundations involve a more interactive process and thus have their own separate design commands.

Typical Design Concrete workflow

The following example illustrates the typical process to analyse and design all the beams, columns and walls in a concrete structure.

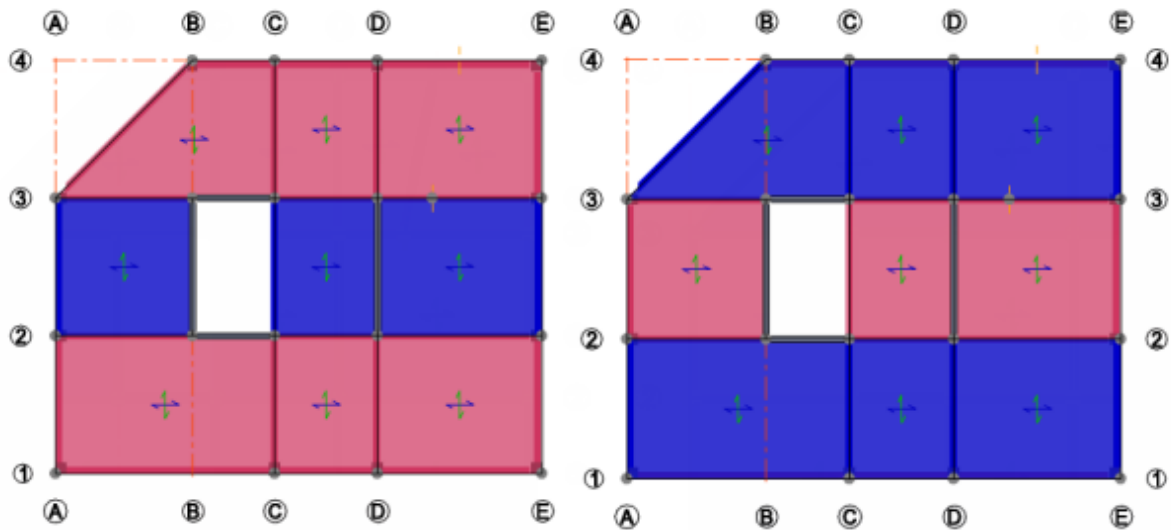


The example has been broken down into the following main steps:

1. [Set up Pattern Loading](#)
2. [Set all beams columns and walls into autodesign mode](#)
3. [Review beam and column design groups](#)
4. [Review beam, column and wall design parameters and reinforcement settings](#)
5. [Perform the concrete design](#)
6. [Review the design status and ratios](#)
7. [Create Drawings and Quantity Estimations](#)
8. [Print Calculations](#)

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.

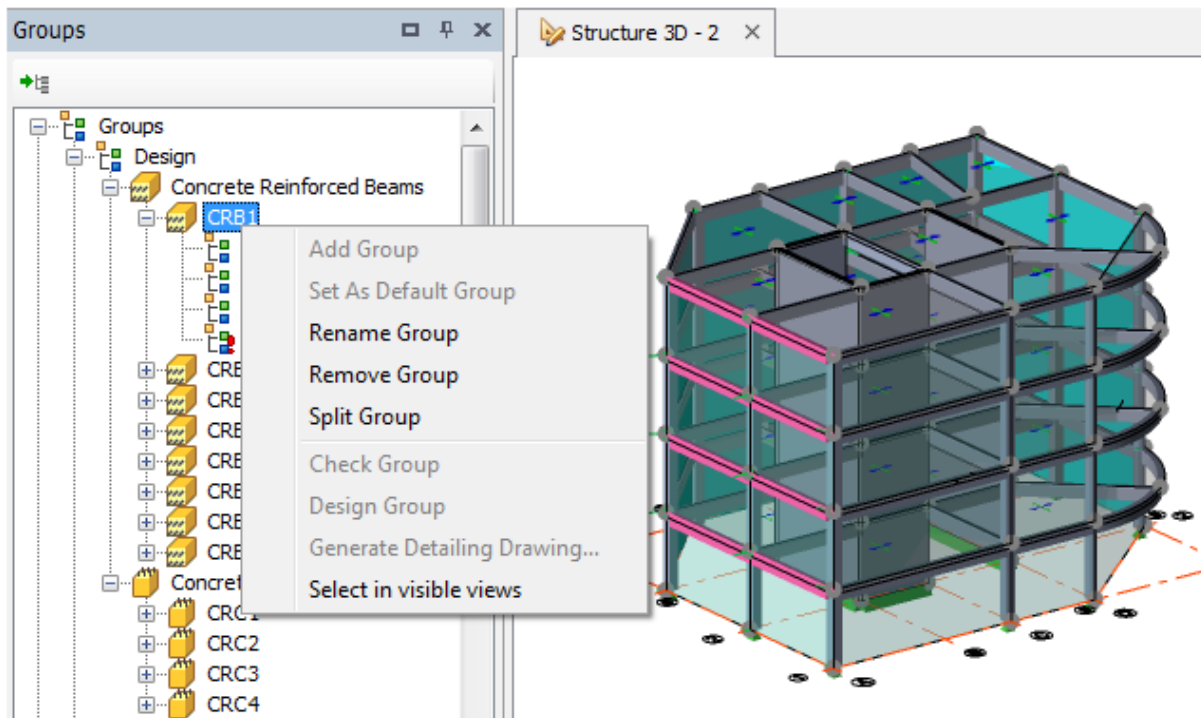


Set all beams columns and walls into autodesign mode

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**.

Review beam and column design groups

Provided that the concrete beam and column options are checked in Design Options> Design Groups, the design groups shown in the Groups tab of the Project Workspace will be applied in the beam and column autodesign processes.



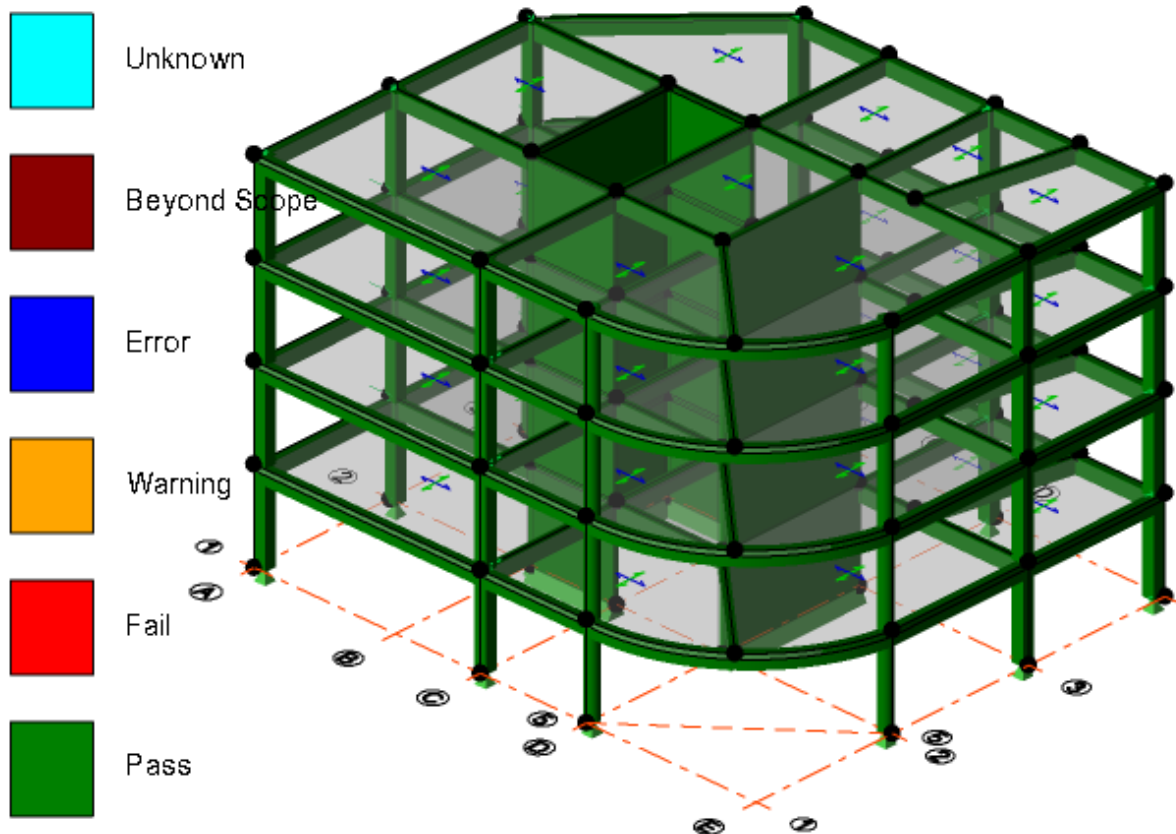
Groups will initially have been established for members sharing the same geometry, but you should consider reviewing and amending them if required.

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

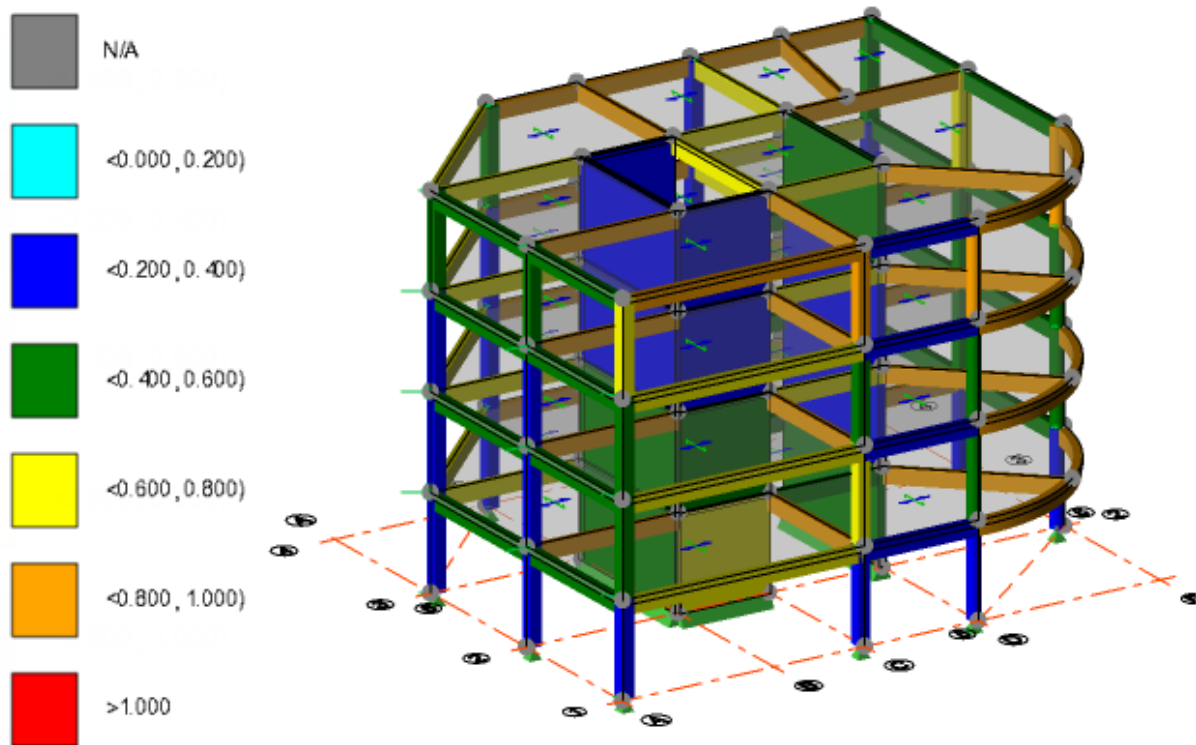
Concrete beams, columns and walls are designed by running **Design Concrete (Static)** from the Design ribbon.



Reinforcement is designed, but member sizes are not changed during this process.

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).

Reviewing Design Concrete and refining the design of individual members

A number of tools are available to assist the post-design review:

1. **Review View** - use the various tools on the [Review toolbar](#) to get an overall picture of the design results.
2. **Check Member** - to view detailed results for individual concrete members.
See: [Check Member](#)
3. **Design Member** - to quickly reselect reinforcement for an individual member, (without having to re-perform the entire structure design).
See: [Design Member](#)



Design Member is intended for individual member design, other members in the same design group are NOT updated with the revised reinforcement. a member is in a group, then if at least one member of the group is set to autodesign the whole group will be auto-designed.

4. **Interactive Design** - if required, use to actively control the reinforcement selected for an individual member.
See: [Interactive concrete member design](#)

Interactive concrete member design

Interactive concrete beam design

The completely automatic design processes, [Design Concrete \(Static\)](#), [Design All \(Static\)](#) etc. are complemented by the program's interactive beam design facility. This allows you to interact with the beam design to override the design results arising from the auto-design process.

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 Beam Design Dialog** displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for 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 cancelling or accepting the changes.

How do I open the Interactive Beam Design Dialog?

1. Right-click the member you want to design interactively and select **Interactive Design...** (Static or RSA as required) from the context menu that is displayed.

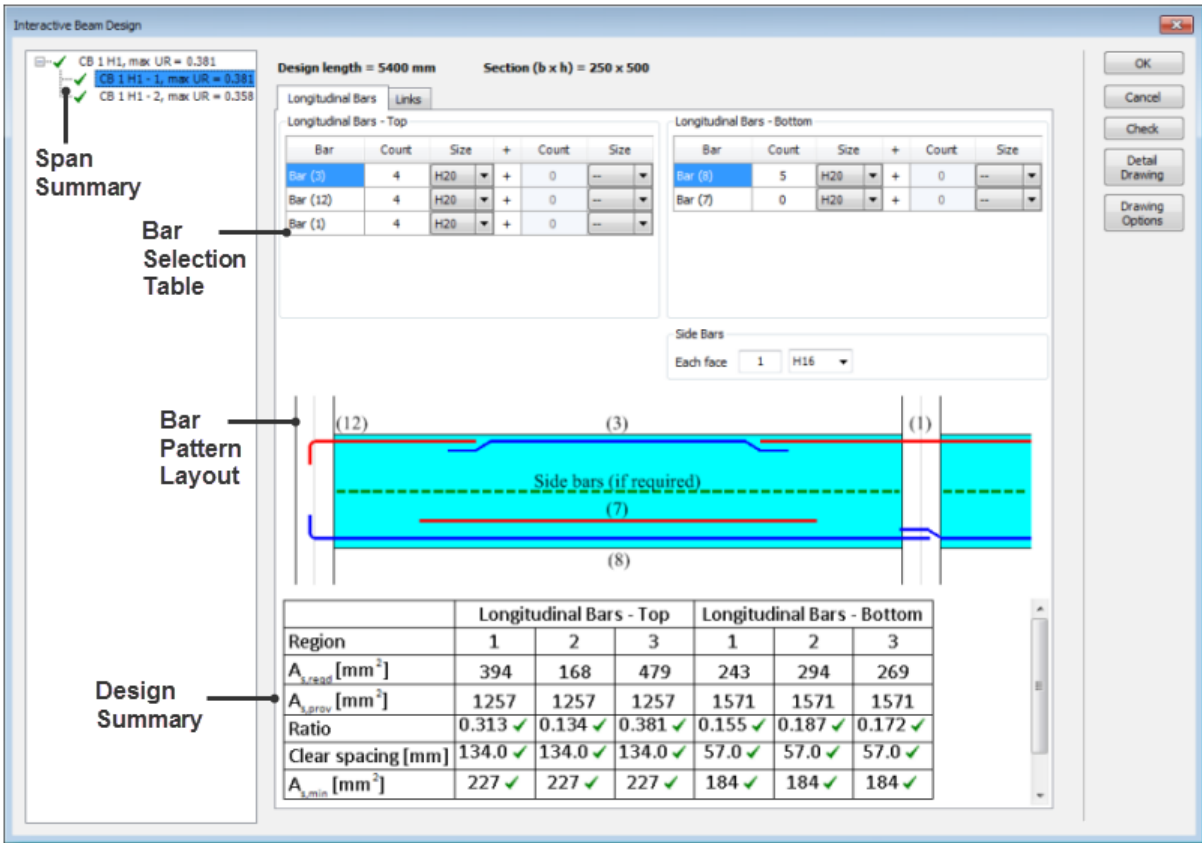
The Interactive Beam Design Dialog opens, displaying results for the existing reinforcement.

Overview of the Interactive Beam Design Dialog

When the dialog is opened, the current reinforcement and check results are shown for each beam in the 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 every single change you make to the reinforcement.

The Interactive Beam Design Dialog consists of the following areas:



Span Summary

A tree view displays the design status of each span and the associated utilisation ratio.

Click a particular span in the summary to display or edit its design in the tabbed pages.

Longitudinal Bars tab

Bar Selection Table

Used for editing the longitudinal bars into the beam.

- Each row in the table is labelled 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.)

Bar Pattern Layout

This is a schematic diagram representing the top and bottom patterns assigned to the beam.

Design Summary Table

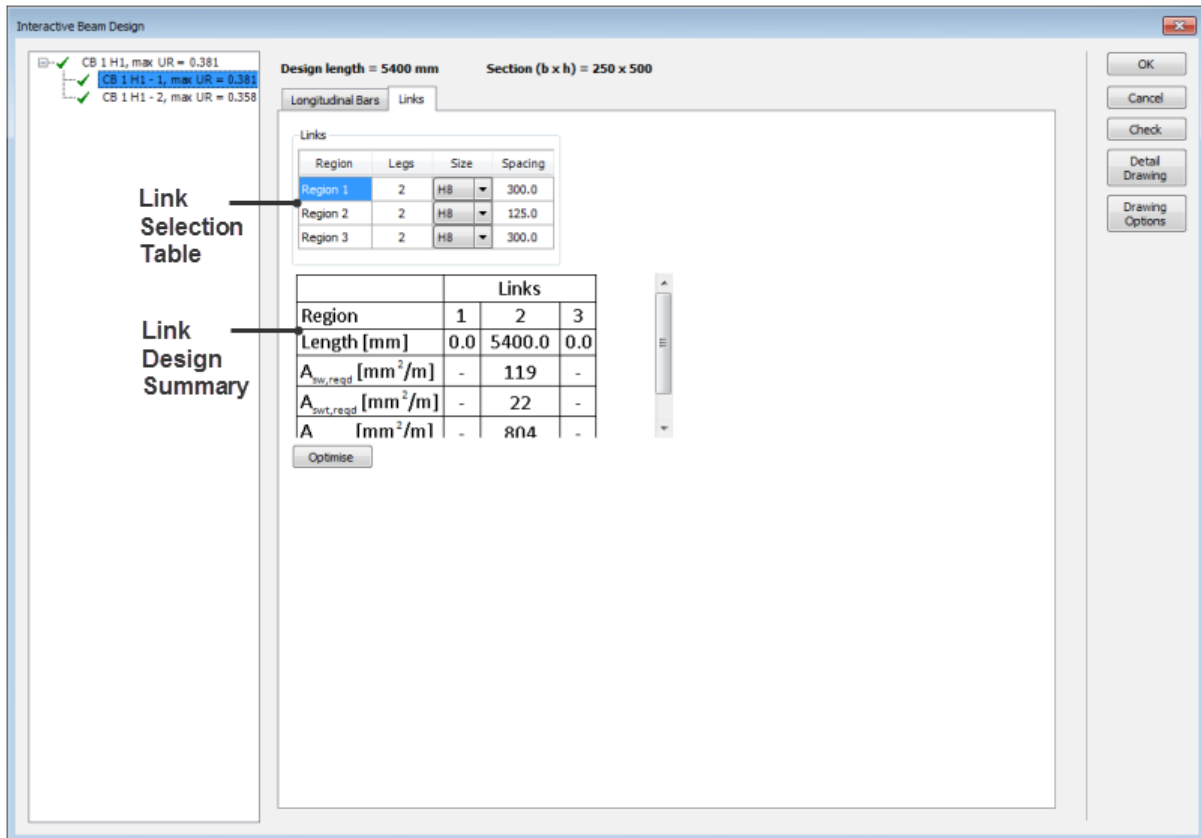
The table displays critical results for each of the design regions from all combinations:

- Area of reinforcement required, $A_{s,reqd}$
- Area of reinforcement provided, $A_{s,prov}$
- Reinforcement area utilisation ratio
- Smallest clear spacing between bars
- Minimum required reinforcement area, $A_{s,min}$

Deflection check

This checks the actual span: effective depth ratio against the limiting span : effective depth ratio.

tab



Selection Table

Specifies the number of legs, size and spacing in each of the regions.

Optimise Button

This calculates the optimum length of the central region given the reinforcement that you have selected. The button is not visible when the beam is in a design group with other beams, and is also not visible when the span is a cantilever.

Design Summary Table

The table displays the most critical result from all combinations:

- Region length
- area over spacing required for shear, $A_{sw, reqd/s}$
- area over spacing required for torsion, $A_{swt, reqd/s}$
- area provided, $A_{sw, prov}$
- utilisation ratio

Buttons

OK

Closes the dialog and saves the current design

Cancel

Closes the dialog without saving changes

Check

Opens the Results dialog displaying 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

How do I change the bar pattern?

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 concrete column design

The completely automatic design processes [Design Concrete \(Static\)](#), [Design All \(Static\)](#) etc. are complemented by the program's interactive column design facility. This allows you to interact with the column design to override the design results arising from the auto-design process.

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 Column Design Dialog** displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for ties only) of the selected bars. Interaction diagrams are also displayed for the current design.

After making changes, you are able to see the effect on the displayed results – you then have the option of cancelling or accepting the changes.

Interactive design also allows columns to be designed for user-defined forces, (for example to 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.

How do I open the Interactive Column Design Dialog?

1. Right-click the member you want to design interactively and select **Interactive Design...** (Static or RSA as required) from the context menu that is displayed.

The Interactive Column Design Dialog opens, displaying results for the existing reinforcement.

Overview of the Interactive Column Design Dialog

When the dialog is opened, the current reinforcement and check results are shown for each stack.

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 every single change you make to the reinforcement.

The **Interactive Column Design Dialog** consists of the following areas:

Stack Summary

- CC A/1, max UR = 0.885
- CC A/1 - 4, max UR = 0.885
- CC A/1 - 3, max UR = 0.598
- CC A/1 - 2, max UR = 0.598
- CC A/1 - 1, max UR = 0.395

Column Summary

Longitudinal Bars

Principal bar size: H12

Intermediate bar size: H12

Int. length	Count	Ctr spacing [mm]	Int. length	Count	Ctr spacing [mm]
1-2	1	116.6	6-7	1	101.0
2-3	0	89.5	4-5	1	103.0
3-4	1	101.0	8-1	1	116.6
5-6	1	103.0	7-8	0	89.5

Design Summary

Position	Longitudinal Bars		
	Top	Mid-fifth	Bottom
M_{ed} [kNm]	72.2	43.5	102.9
M_{res} [kNm]	155.4	185.0	188.4
Ratio	0.465	0.235	0.546
N_{ed} [kN]	550.8	558.7	564.0
N_{res} [kN]	3597.1		
Ratio	0.153	0.155	0.157
Smallest clear spacing [mm]	77.5		
$A_{\text{st,min}}$ [mm ²]	752		

Section and Concrete Grade

Section: Elbow 400x400x300

Concrete: C32/40

Stack length = 3000 mm

Containment status: Pass

Cross Section

600 mm

406.4 mm

Minor

Major

Column and Stack Summaries

The left hand pane of the dialog consists of a tree view containing a Column Summary and Stack Summaries for the individual stacks.

- The Column Summary can be used to edit the section size and grade for all stacks simultaneously. It also displays the overall design status and utilization ratio for the entire column.
- The Stack Summaries are used to review and edit the individual stack designs.

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.



In the current release only one layer of reinforcement against any shear tie edge is permitted.

Principal bar size

Used to change the size of all principal bars (all must have the same size).

Intermediate bar size (not displayed for circular columns)

Used to change the size of all intermediate bars (all must have the same size).

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 centreline 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).

Design Summary Table

The 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

Section Droplist

Used for changing the section size for the current stack.



If the droplist is used to change the section shape an autodesign is performed.

Concrete Droplist

Used for changing the concrete grade for the current stack.

Confinement status

This status is determined based on the requirements for bars being tied.

Cross-section

The drawing displays:

- Exact bar positions (drawn to scale)
- Tie locations
- Section dimensions
- Principal bar labels

tab

Use support region

Select to design support regions for the .

spacing

specifies the spacing (if support regions are applied two different spacings can be specified)

size

Used to change the size of bars (all must have the same size).

Design Summary Table

The table displays the most critical result from all combinations:

- area over spacing required, major
- area over spacing required, minor
- area over spacing provided
- utilisation ratio

Cross-section

The drawing displays:

- Exact bar positions (drawn to scale)
- locations
- Section dimensions
- Principal bar labels

Interaction diagram tab

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

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.

You can view the diagram for any analysis method, combination and position as required and the "Set Critical" button can be used to return to the critical parameters.

Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.

Design cases...

Click to open a dialog in which to add the 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).

Design cases table

Each design case added in the Design Cases dialog 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

Buttons

OK

Closes the dialog and saves the current design

Cancel

Closes the dialog without saving changes

Check

Opens the Results dialog displaying the detailed results for the current design

Detail Drawing

Creates a detail drawing for the selected member

Autodesign Stack

Runs a full reinforcement autodesign on the stack (starting from minimum reinforcement)

Drawing Options

Opens the DXF Export Preferences dialog

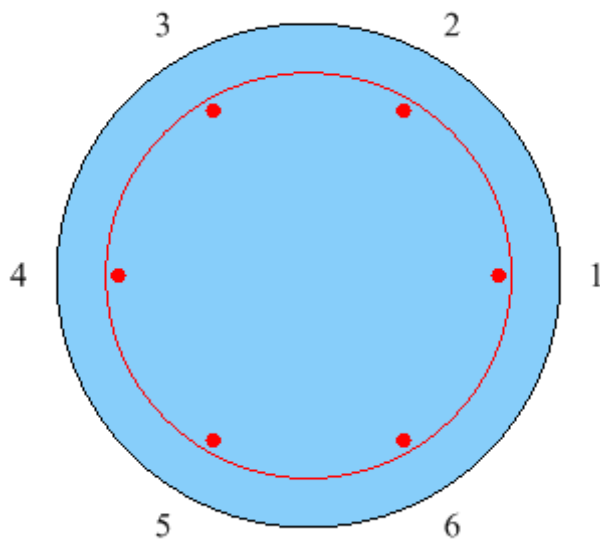
Copy...

Can be used to copy the section, grade, or reinforcement from the current stack to others.

- Reinforcement can only be copied if the section is identical
- An option is provided to autodesign stacks where the section has changed, instead of copying reinforcement

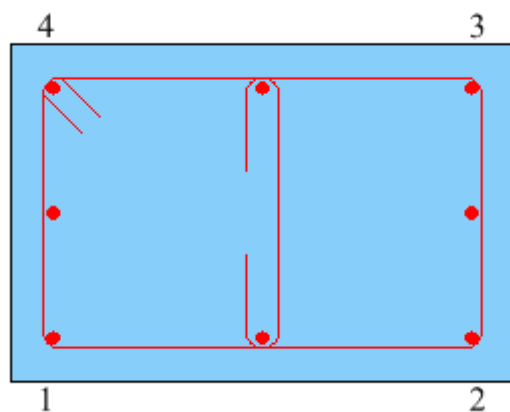
How do I arrange bars in the Interactive Column Design Dialog?

Circular Columns



Steel bars are arranged by modifying the bar size and count fields.

Rectangular and Polyline Columns



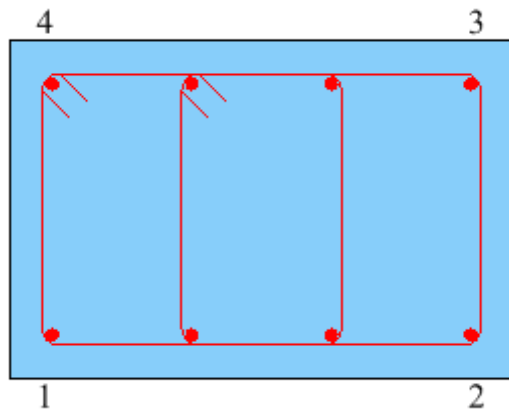
Principal bars exist at fixed locations; they are labelled with numbers in the cross-section, as shown above. You can only change the principal bar sizes, not their locations.

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	✓

arrangements in rectangular and parallelogram sections have the following basic options:

- Single ,
- Double ,
- Triple ,
- Cross .

Tie bars are used with these arrangements. arrangements in other section shapes use standard positions with additional tie bars where required.

How do I define additional column design cases?

1. In the Interactive Column Dialog, select **Additional Design Cases** 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 concrete wall design

The completely automatic design processes, [Design Concrete \(Static\)](#), [Design All \(Static\)](#) etc. are complemented by the program's interactive wall design facility. This allows you to interact with the wall design to override the design results arising from the auto-design process.

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 Wall Design Dialog** displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for only) of the selected bars. Interaction diagrams are also displayed for the current design.

After making changes, you are able to see the effect on the displayed results – you then have the option of cancelling or accepting the changes.

Interactive design also allows walls to be designed for user-defined forces, (for example to 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.

How do I open the Interactive Wall Design Dialog?

1. Right-click the member you want to design interactively and select **Interactive Design...** (Static or RSA as required) from the context menu that is displayed.

The Interactive Wall Design Dialog opens, displaying results for the existing reinforcement.

Overview of the Interactive Wall Design Dialog

When the dialog is opened, the current reinforcement and check results are shown for each stack.

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 every single change you make to the reinforcement.

The **Interactive Wall Design Dialog** consists of the following areas:

Wall Summary

Longitudinal Lateral Interaction Diagrams Additional Design Cases

☐ Use end-zones

Panel

Number of layers: 2

Reinforcement type: Loose bars

Number of rows: 25 Centre spacing = 125.0 mm ✓

Vertical bar size: H8

Additional end row bars: 0 Centre spacing = 126.0 mm ✓

Bar Location Table

Position	Vertical Bars		
	Top	Mid-fifth	Bottom
M_{Ed} [kNm]	124.5	410.7	470.7
M_{Ed} [kNm]	585.9	1959.7	2119.0
Ratio	0.212 ✓	0.210 ✓	0.222 ✓
N_{Ed} [kN]	1460.3	1517.7	1555.7
N_{Ed} [kN]	10630.5		
Ratio	0.137 ✓	0.143 ✓	0.146 ✓
Smallest clear spacing [mm]	113.9 ✓		
$A_{s,min}$ [mm ²]	2400 (0.40%)		
$A_{s,max}$ [mm ²]	12000 (2.00%)		
A_s [mm ²]	2513 (0.42%) ✓		
Other checks	✓ Pass		

Cover: provided / limiting = 25.0 / 20.0 ✓

Thickness and Concrete Grade

Thickness: 200.0 mm

Concrete: C32/40

Panel Summary

- W1, max UR = 0.476
- W1 - 4, max UR = 0.476
- W1 - 3, max UR = 0.257
- W1 - 2, max UR = 0.236
- W1 - 1, max UR = 0.222

Design Summary

Cross Section

3000 mm

200 mm

Wall and Panel Summaries

The left hand pane of the dialog consists of a tree view containing a Wall Summary and Panel Summaries for the individual panels.

- The Wall Summary can be used to edit the thickness and grade for all panels simultaneously. It also displays the overall design status and utilization ratio for the entire wall.
- The Panel Summaries are used to review and edit the individual panel designs.

Longitudinal tab (Reinforcement definition when end-zones not used)

Use end-zones

fields are displayed as below when end-zones not used

Panel Bar Location Table

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)

Longitudinal tab (Reinforcement definition when end-zones used)

Use end-zones

fields are displayed as below when end-zones used

End-zone Bar Location Table

Used for adding bars in the end-zones.

- Length - length of each end-zone
- Number of rows - the number of loose bars in each layer 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)

Mid-zone Bar Location Table

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

Longitudinal tab (Other fields)

Design Summary Table

The 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

Thickness

Used for changing the thickness of the current panel.

Concrete Droplist

Used for changing the concrete grade for the current panel.

Cross-section

The drawing displays:

- Exact bar positions (drawn to scale)
- locations
- Section dimensions

Lateral tab

Use

Select to specify .

Use support region

Select to design support regions for the .

spacing

specifies the spacing (if support regions are applied two different spacings can be specified)

size

Used to specify the size of bars (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.

Design Summary Table

The table displays the most critical result from all combinations:

- area over spacing required, major
- area over spacing provided, major
- utilisation ratio, major
- area over spacing required, minor
- area over spacing required, minor

- utilisation ratio, minor

Horizontal Reinforcement Summary Table

The table displays the horizontal bar spacing and reinforcement ratios.

Cross-section

The drawing displays:

- Exact bar positions (drawn to scale)
- locations
- Section dimensions

Interaction diagram tab

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

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.

You can view the diagram for any analysis method, combination and position as required and the "Set Critical" button can be used to return to the critical parameters.

Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.

Design cases...

Click to open a dialog in which to add the 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).

Design cases table

Each design case added in the Design Cases dialog 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

Buttons

OK

Closes the dialog and saves the current design

Cancel

Closes the dialog without saving changes

Check

Opens the Results dialog displaying the detailed results for the current design

Autodesign Panel

Runs a full reinforcement autodesign on the panel (starting from minimum reinforcement)

Detail Drawing

Creates a detail drawing for the selected member

Drawing Options

Opens the DXF Export Preferences dialog

Copy...

Can be used to copy the thickness, grade, or reinforcement from the current panel to others.

- Reinforcement can only be copied if the panel thickness is identical
- An option is provided to autodesign panels where the thickness has changed, instead of copying reinforcement

How do I define additional wall design cases?

1. In the Interactive Wall Dialog, select **Additional Design Cases** 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 panel.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all wall panels where appropriate.

The additional loading cases are now always checked whenever the regular combinations are checked.

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 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 vibration 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.

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.



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!)

Foundation Design Handbook

This handbook provides a general overview of *Tekla Structural Designer* in the context of its application to the foundation design.

The following isolated foundations can be designed:

- **Pad base** - an isolated foundation that supports a single column
- **Strip base** - an isolated foundation that supports a single wall

In addition, the following mat foundations can be designed:

- **Mat foundation** - a foundation supporting multiple columns and walls on ground springs

Isolated foundation design

•

Overview of the isolated foundation analysis model

Association with member supports

Columns and walls typically have supports at their bases (apart from transfer columns/walls where the lower end resides on a beam, slab etc.). These supports can be set to be pinned/fixed/sprung or non-linearly sprung (compression/tension only) as required.

At any time during the modelling process, you can define isolated foundations (pad bases, strip bases and pile caps) which are associated with the above mentioned supports.



Typically pad bases and pile caps can only support and be loaded by a single column, and strip bases can only support and be loaded by a single wall; however, if a ground beam is attached to the same support, loading from the beam will also be considered in the isolated foundation design.

Analysis types

Isolated foundations are designed using the results of up to three analysis types:

- 3D Analysis
- FE Chasedown Analysis

- Grillage Chasedown Analysis

Design forces and checks

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



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}



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.

Design Checks

The foundation is treated as being a rigid base resting on the soil working up to a maximum bearing pressure. Uplift is allowed.

Top and Bottom reinforcement (both directions) is designed in accordance with the selected design code.



*In the current release, top reinforcement is **not** designed for the Indian and Australian head codes.*

The checks performed as part of the design are as follows:

- bearing pressure check
- design for bending
- design for shear
- design for punching shear
- design for sliding
- design for uplift



*In the current release, the uplift check is **not** performed for the Indian and Australian head codes.*

When bases are placed at different levels and close to each other there is a potential risk that the lower base will be affected by the base pressure of the other one. A specific check is made for this and if they are too close a validation warning message is issued.

Bearing pressure calculations

To determine bearing pressures, the number of corners in bearing have to be determined. Depending on the vertical load eccentricity the potential cases are:

- Uniform pressure
- No uplift, four corners in bearing, biaxial overturning
- Uplift, two corners in bearing, uniaxial overturning
- Uplift, three corners in bearing, biaxial overturning
- Uplift, two corners in bearing, biaxial overturning
- Uplift, single corner in bearing, biaxial overturning

Once the cases are identified the corner pressures can then be calculated - in *Tekla Structural Designer* the equations for this are taken from on those published in an article by Kenneth E. Wilson published in the Journal of Bridge Engineering in 1997.

Pad base and strip base design procedures

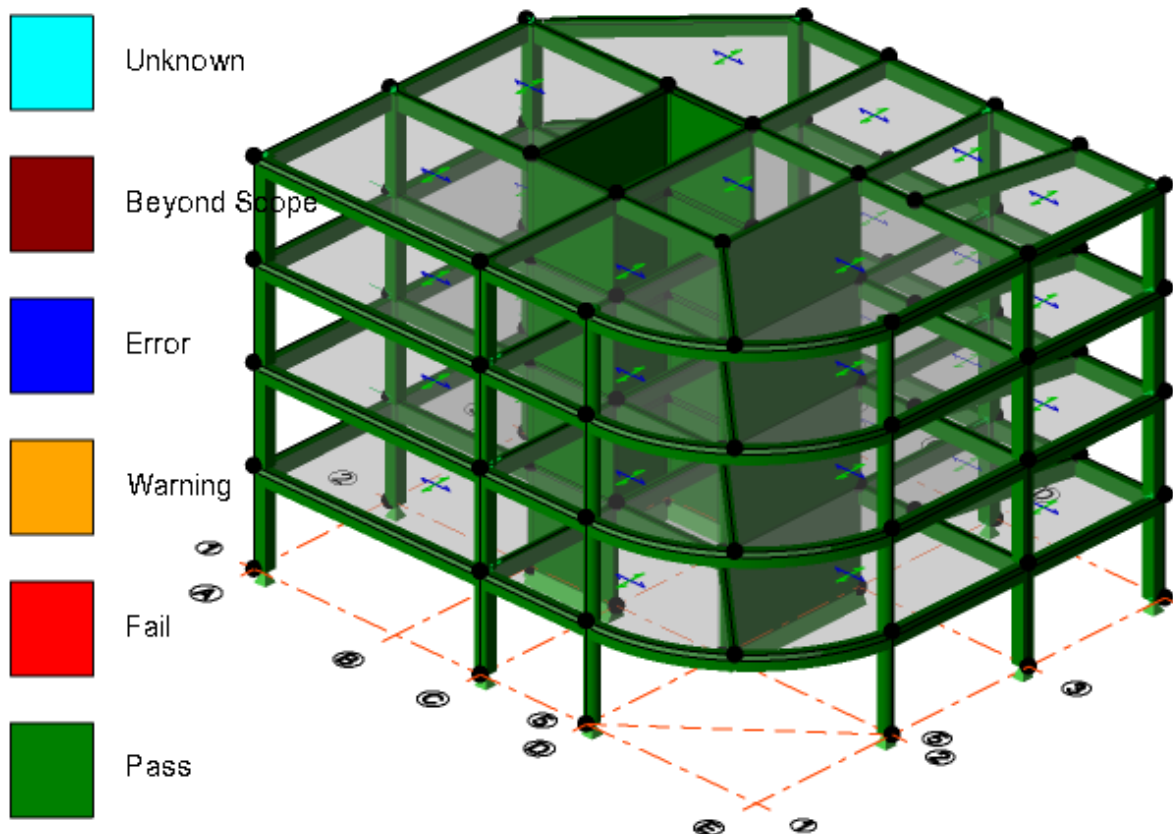
The overall procedure is demonstrated in the following [Pad base design example](#) (but is basically the same for strip bases also).

The typical steps required are as follows:

1. [Apply bases under supported columns](#)
2. [Auto-size bases individually for loads carried](#)
3. [Apply grouping to rationalize pad base sizes](#)
4. [Review/Optimise Base Design](#)
5. [Create Drawings and Quantity Estimations](#)
6. [Print Calculations](#)

Pad base design example

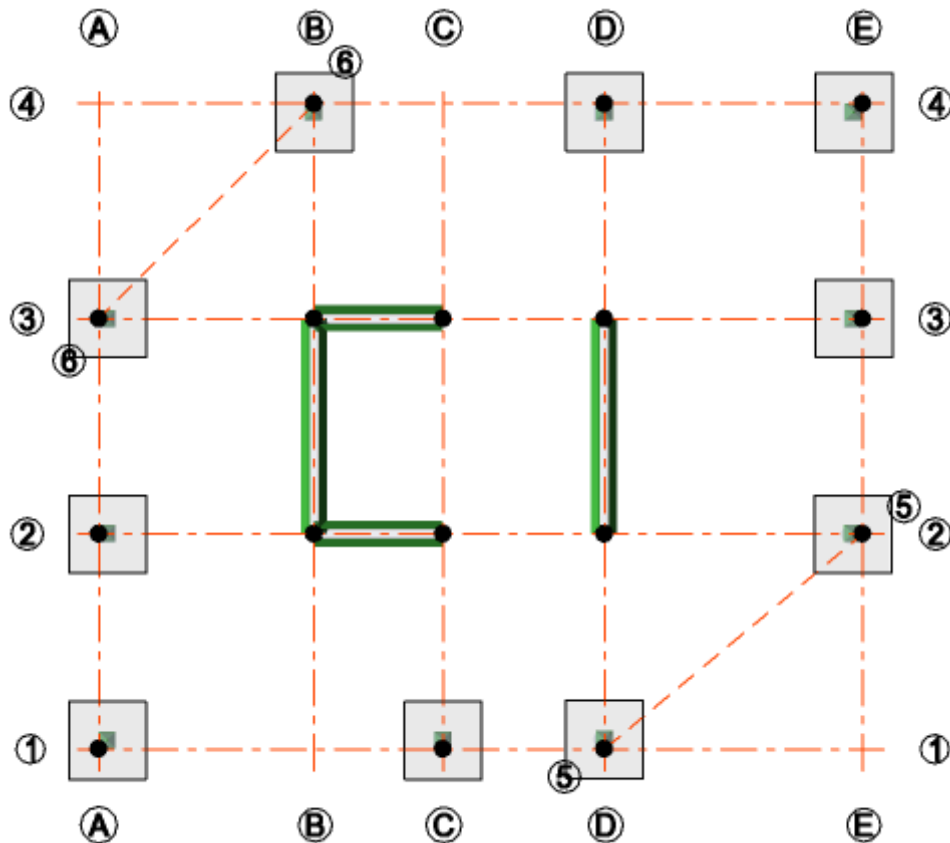
The small concrete building model shown below will be used to demonstrate the base design process.



The model has already been designed prior to placing the bases.

Apply bases under supported columns

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 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 Options 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.



The overall auto-design procedure is summarised 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 starting point, and go back to step 1.*

3. Shear Design - *increase depth, set reinforcement back to starting point, and go back to step 1*

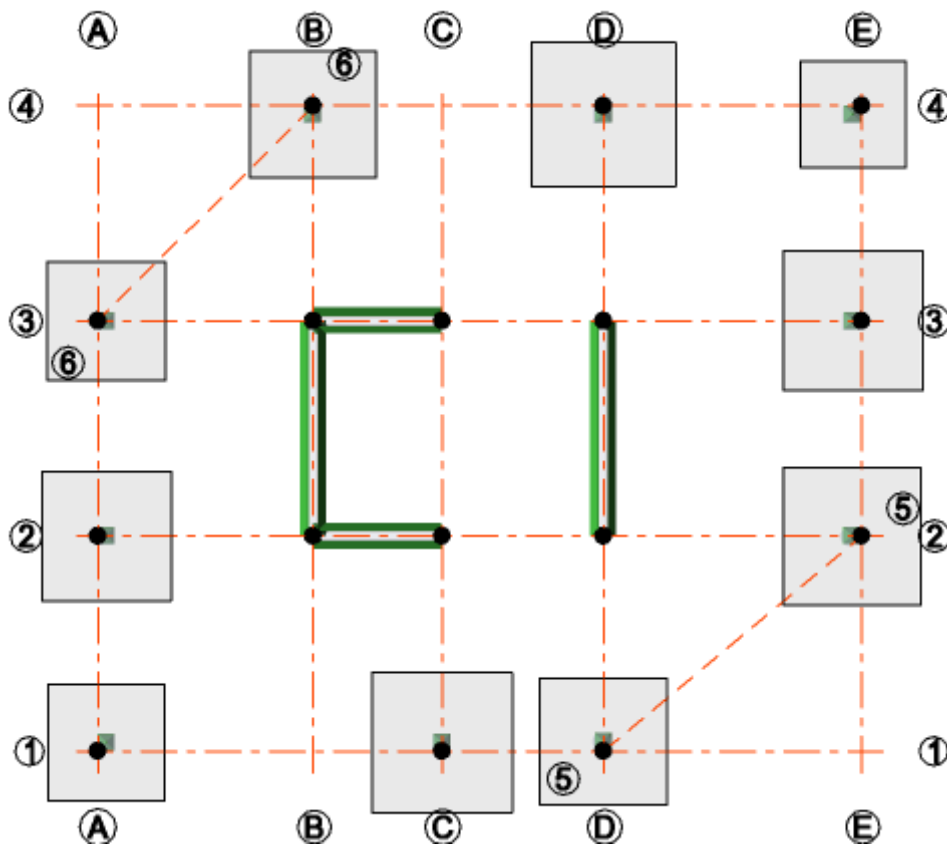
4. Punching Shear Design - *increase depth, set reinforcement back to starting point, and go back to step 1*

5. Sliding Checks - *increase depth, set reinforcement back to starting point, and go back to step 1*

At every stage, if the max allowable depth is reached the design fails.

3. From the Foundations ribbon 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.

Apply grouping to rationalize pad base sizes



Grouping can only be applied to pad bases - not to strip bases.

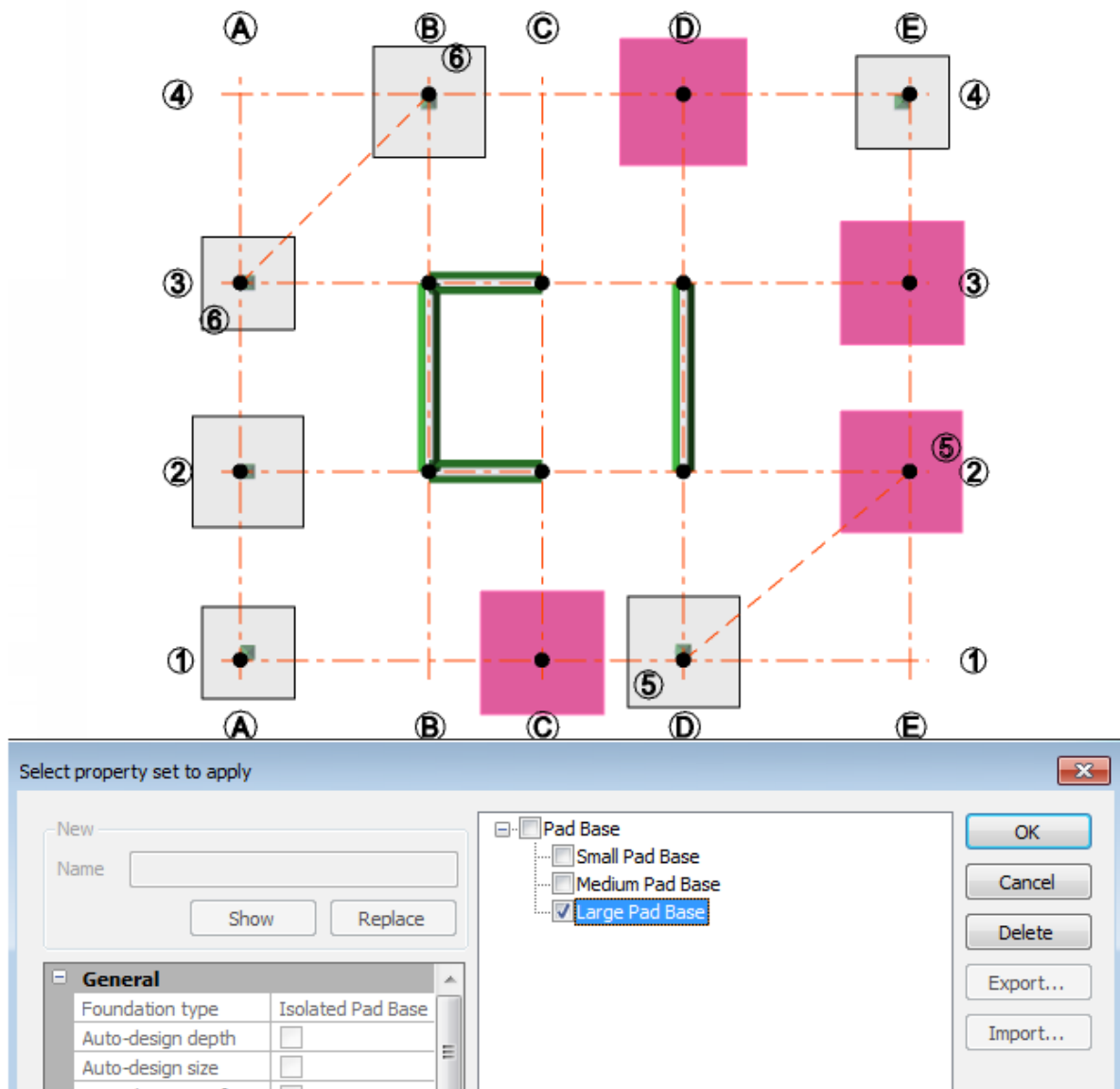
Once pad bases have been sized individually, the designs can be rationalised by activating grouping, in order to obtain one design per group sufficient for all bases within the group.

This is done as follows:

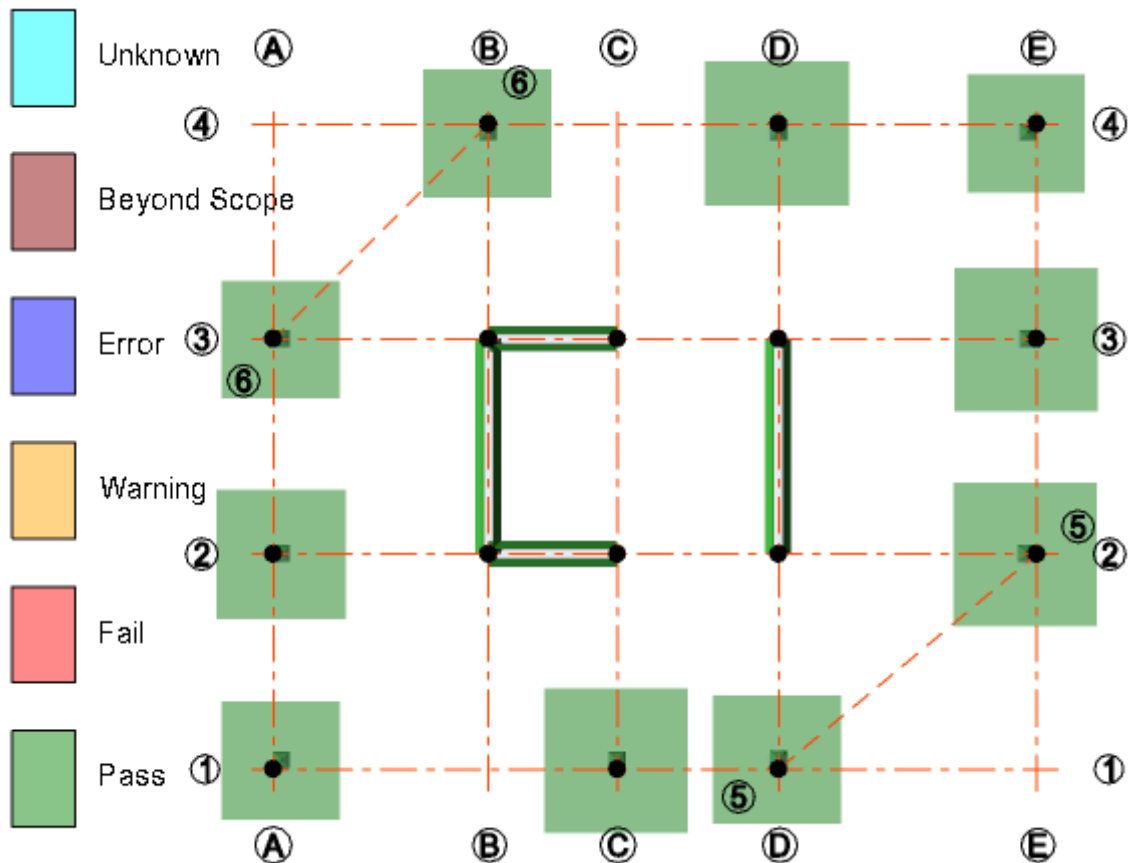
1. Select a base that you want to be in a particular group.
2. In the Properties Window, ensure it is set to auto-design.
3. Right-click the same base and from the context menu choose **Create Property Set...**
4. Select all the other bases that you want to be in the same group.
5. In the **Properties Window**, click **Apply...** to apply the property set you have just created to the selected bases.



*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 the Design Options dialog, 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.



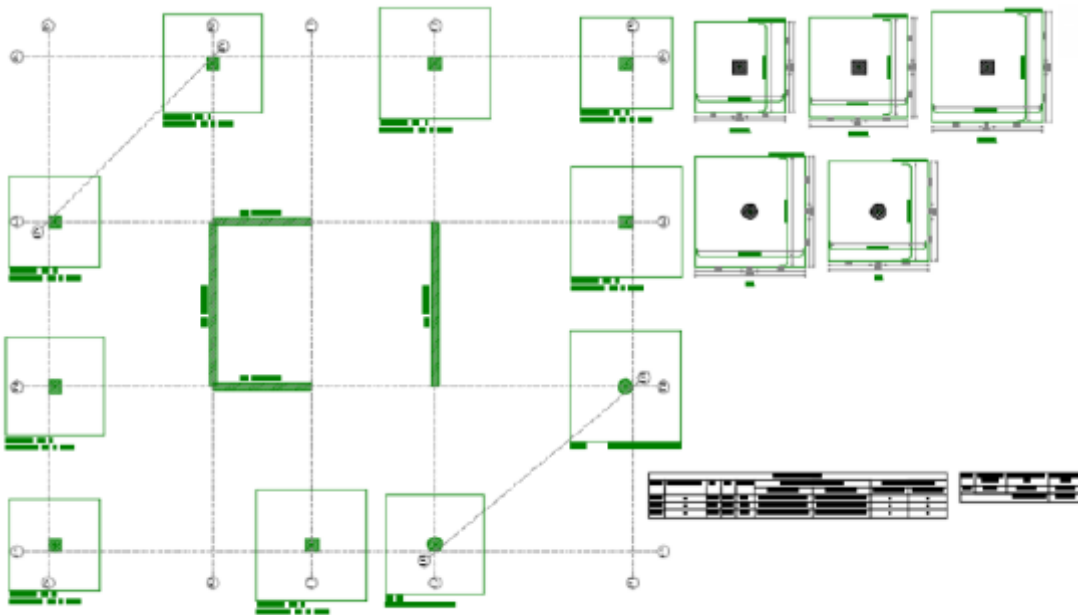
Review/Optimise Base Design

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).

Mat foundation design

Features of the mat foundation analysis model

Analysis Types

Foundation mats are designed for the results of up to three analysis types:

- **3D Analysis**
- **FE Chasedown Analysis**
- **Grillage Chasedown Analysis**

In each of the above analyses, mats are modelled as meshed 2-way slabs, either supported on ground bearing springs, or on discrete piled supports, (or a combination of both).

In both the FE and grillage chase-down models the mat and first level above the mat are always combined in a single sub-model.

Soil Structure Interaction

When **not** supported by a mat, columns and walls typically have supports at their bases.

When a mat or piled mat is introduced these supports must be removed - as the mat now supports the whole building (either on ground bearing springs, or pile springs). Consequently adding a mat means re-analysis and hence re-design of the whole building.

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.

Soil Parameters

Ground Bearing Springs

Unless you have defined discreet piled supports, the mat will need to be supported on ground bearing springs.

Allowable Bearing Pressure

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:

- i. Allowable bearing pressures are checked
- ii. Uplift (tension) is checked
- iii. If no problems then linear springs are sufficient

When non-linear springs are applied:

- i. You can have compression only
- ii. And also capped compression
- iii. Either way analysis takes longer

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 below¹ 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<800kPa)	24,000	48,000
Clayey Soil (qa>800kPa)	48,000	200,000

Horizontal Support

The degree of horizontal support provided by the ground springs can be set as:

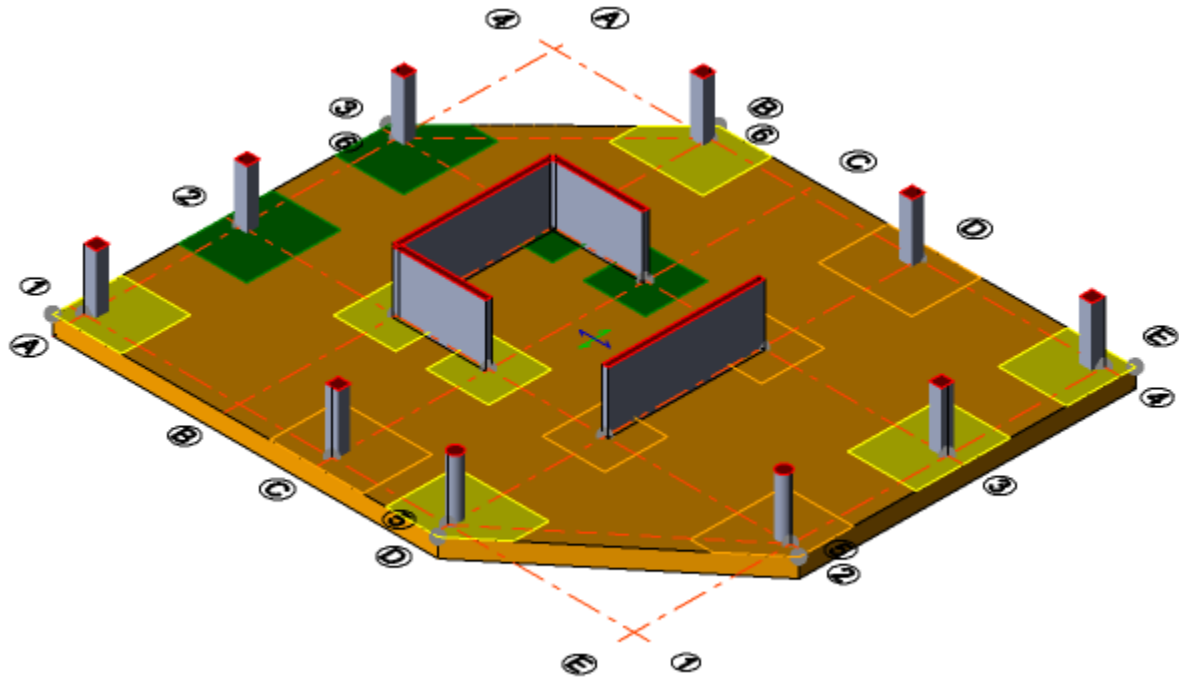
- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to "Free" a mechanism will result unless you provide additional discreet supports.

[1.](#)Reference: "Foundation Analysis and Design", Joseph E. Bowles, 1995 (Table 9-1)

Typical mat foundation design procedure

The following example illustrates the typical process to model and design a mat foundation.

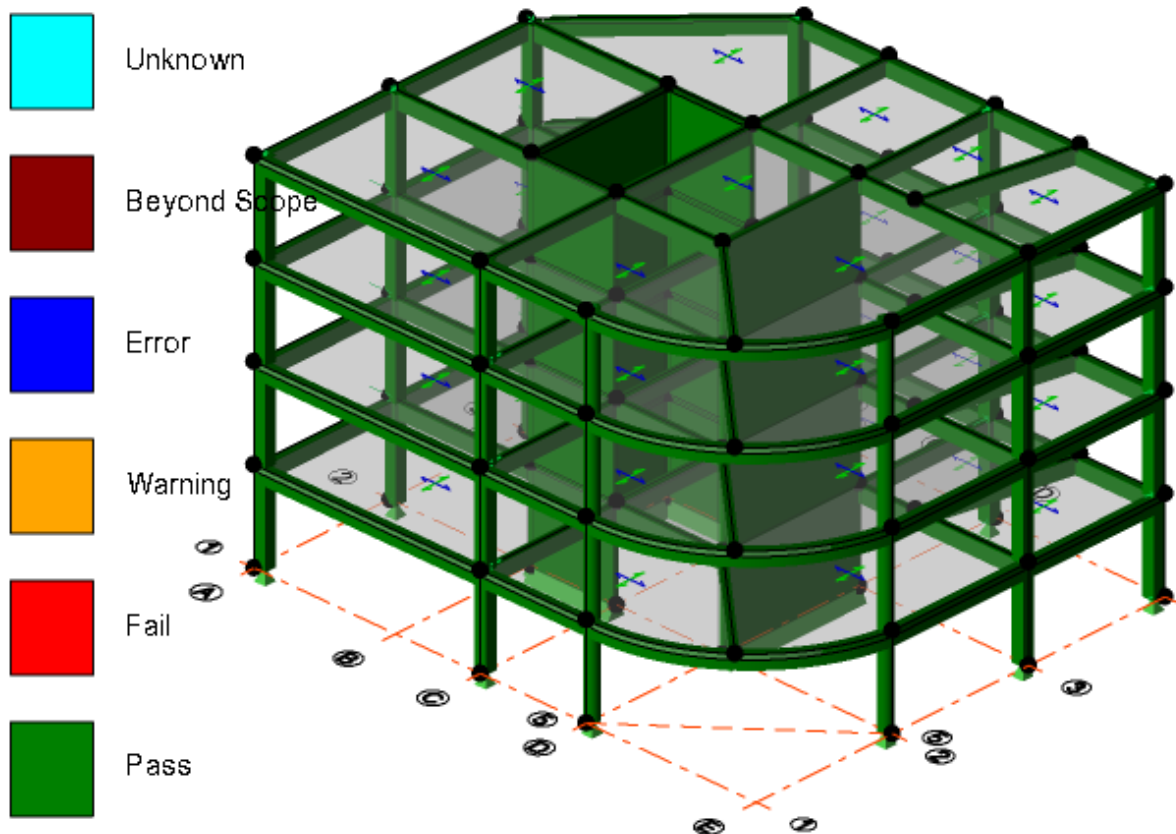


For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

The example has been broken down into the following main steps:

1. [Design the structure before supporting it on the mat](#)
2. [Create the mat, \(either with ground springs, or discreet supports\)](#)
3. [Model validation](#)
4. [Perform the model analysis](#)
5. [Check foundation Bearing Pressure and Deformations](#)
6. [Re-perform member design](#)
7. [Open an appropriate view in which to design the mat](#) and:
 - a. [Add Patches](#)
 - b. [Design Mats](#)
 - c. [Review/Optimise Mat Design](#)
 - d. [Design Patches](#)
 - e. [Review/Optimise Patch Design](#)
 - f. [Add and Run Punching Checks](#)
8. [Create Drawings and Quantity Estimations](#)
9. [Print Calculations](#)

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, (either with ground springs, or discrete supports)

Unless you have defined discrete piled supports, the mat will need to be supported on ground bearing springs. These are defined by specifying appropriate [Soil Parameters](#) in the mat properties.

You are required to manually specify the **Reduce** imposed **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.

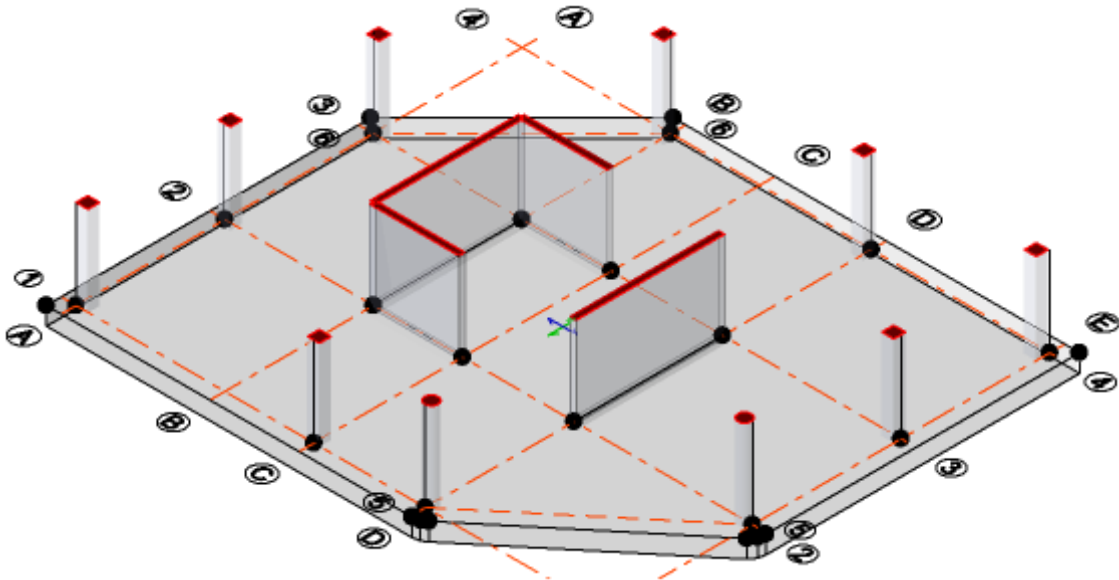
The remaining properties are the same as those used to define a typical concrete flat slab.



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:

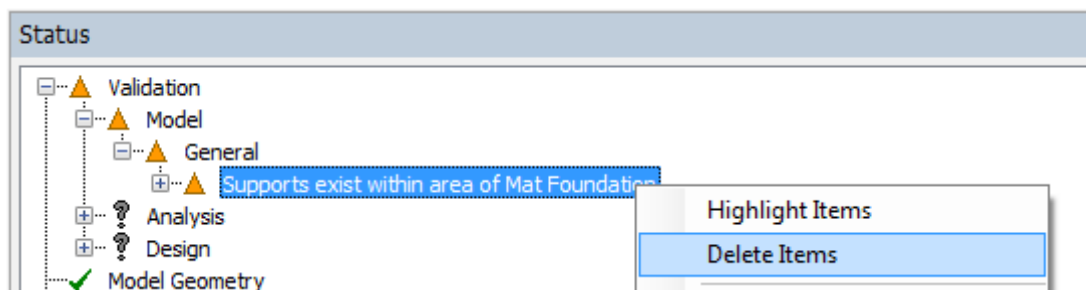
- Imposed loads reduced by 30%
- Default overhang
- mat thickness 600mm
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties



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 analysed by running **Design All (Static)**, and any seismic RSA combinations by running **Design All (RSA)**.



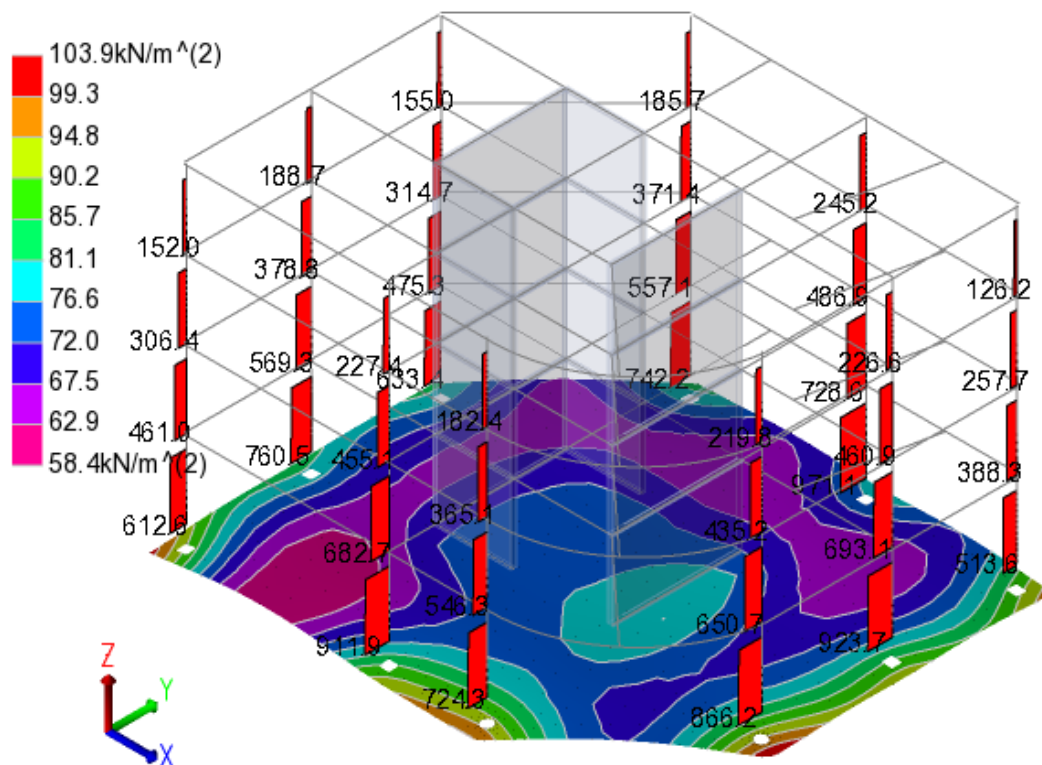
The above processes will also recheck all the member designs taking account of the effects of soil structure interaction.



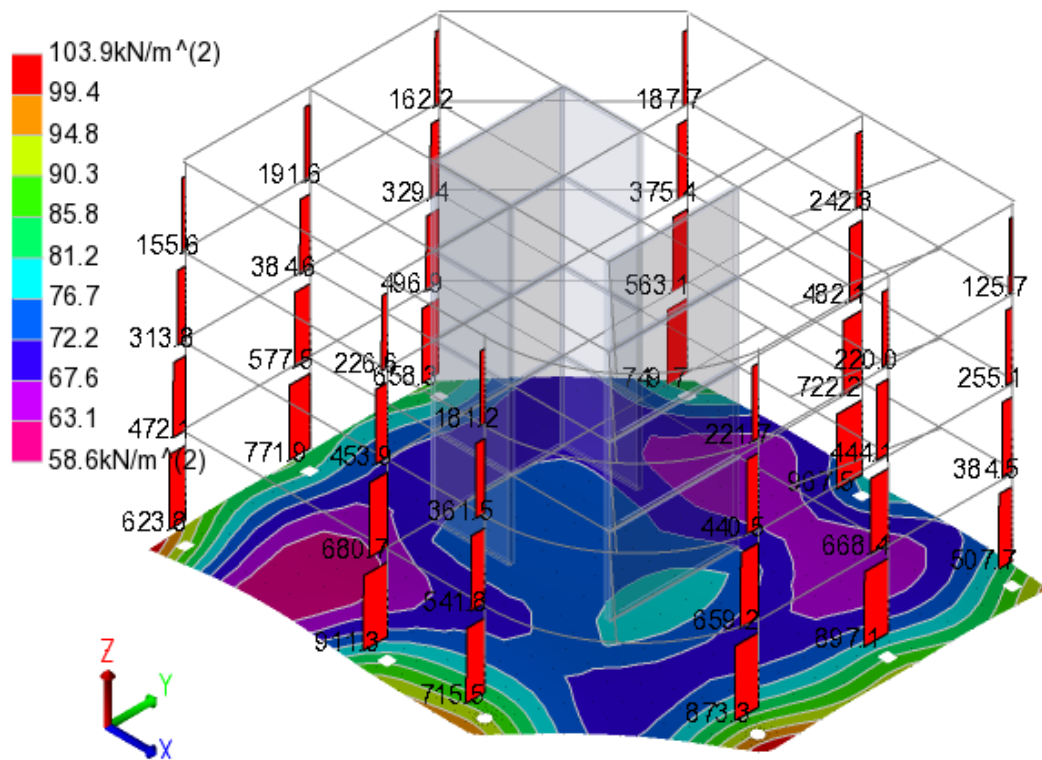
In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run the analyses that are required from the Analyse ribbon then re-run the member design at a later stage.

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.

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.



*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.

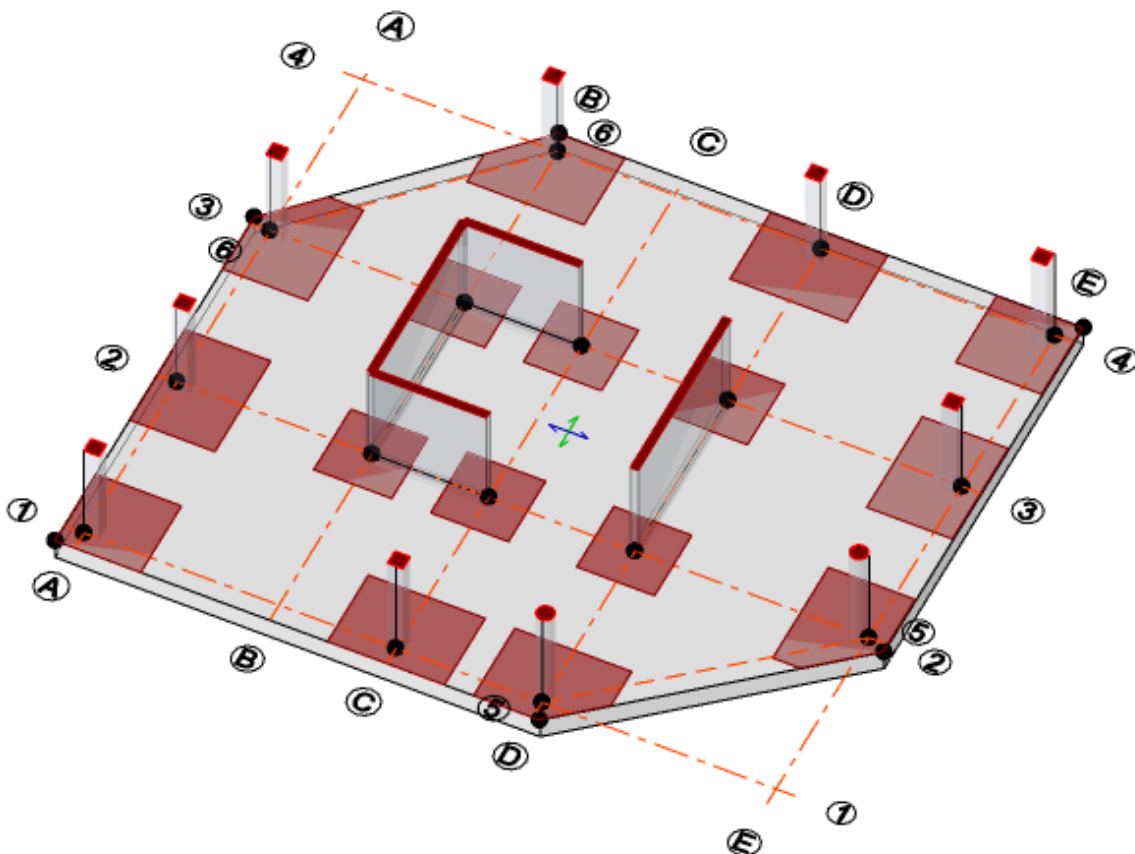


When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered.

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 optimisation stage.



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



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:

1. 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,
2. 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**.



These right-click options operate on the same basis as the options for beams and columns:

- **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/Optimise 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.)



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 standardise 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 utilisations 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 rationalised 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:

1. 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,
2. 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.



These right-click options operate on the same basis as the options for beams and columns:

- **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/Optimise Patch Design

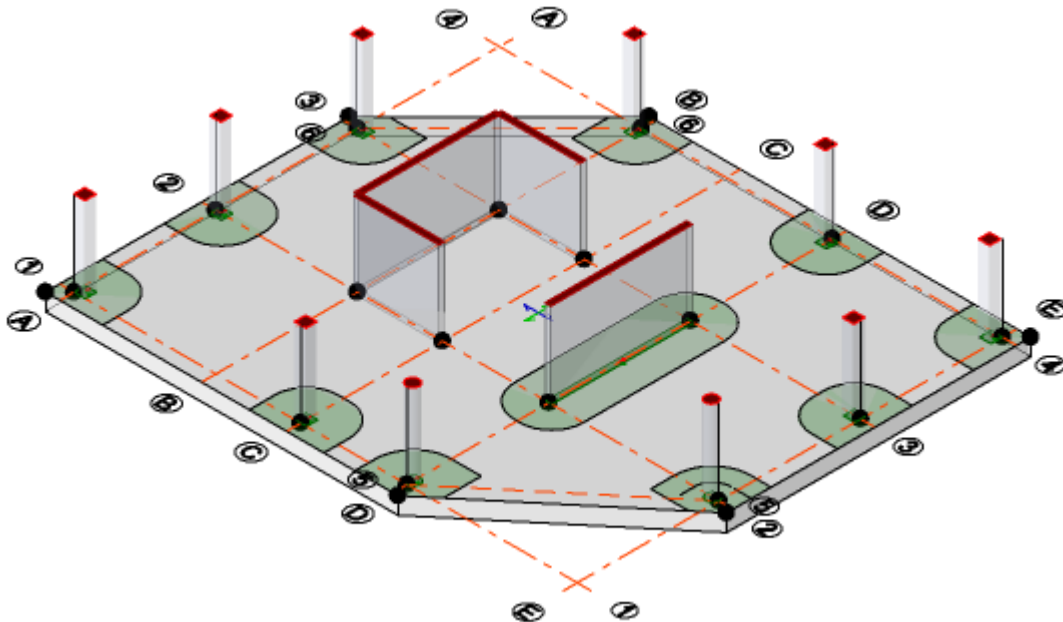
At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimised)?
- 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).

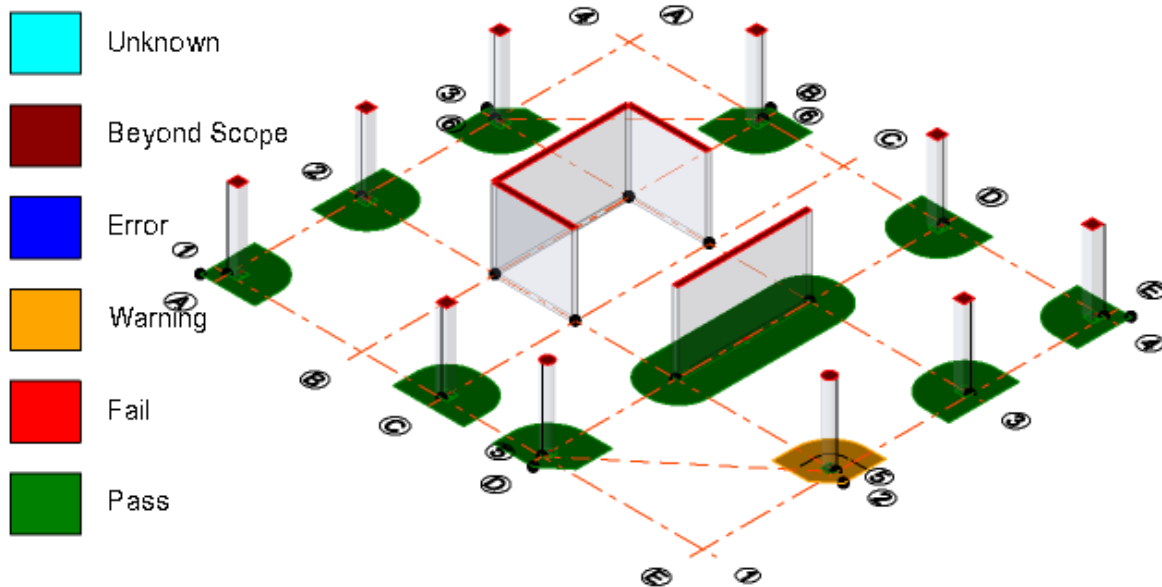
Add and Run Punching Checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added over the entire level, or structure by windowing it. You can then select any check and review the properties assigned to it.



Once added click **Design Punching Shear**

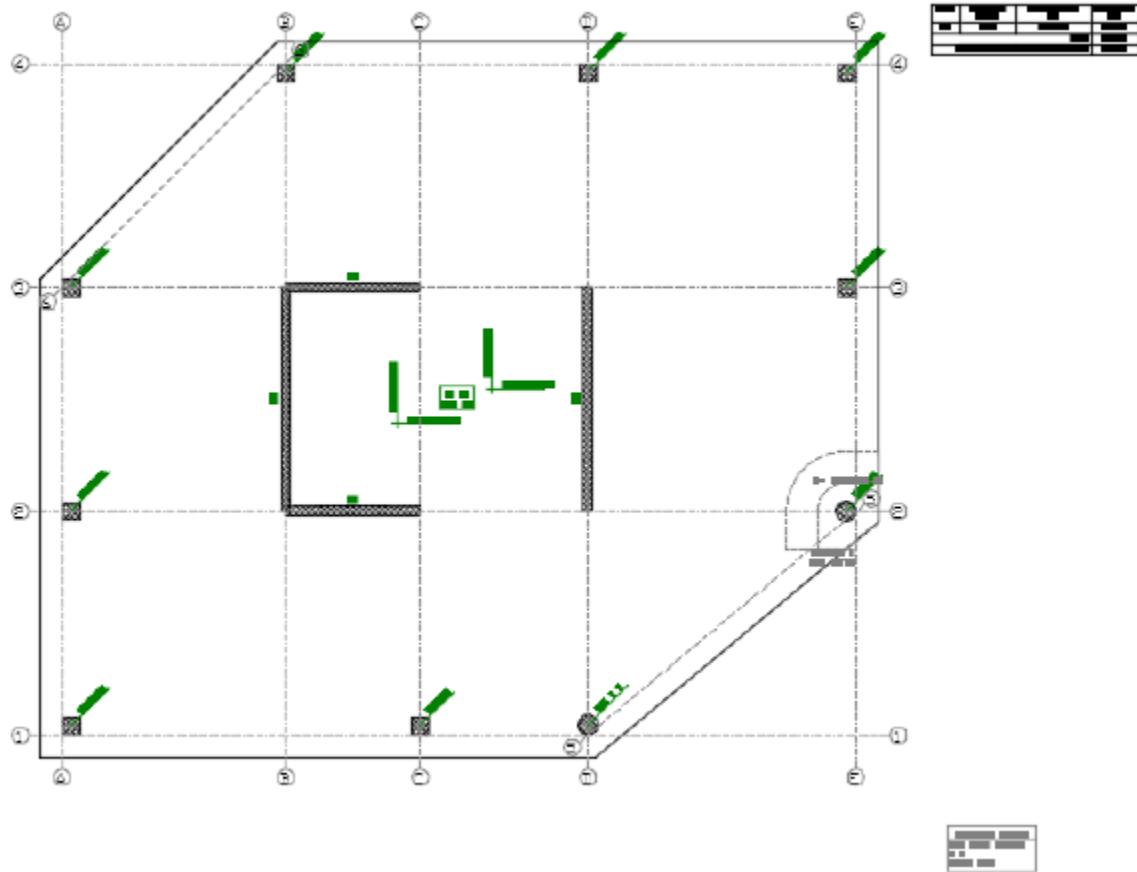


The checks are done and status is shown as:

- 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).