Tekla Structural Designer 2021

Engineers Handbooks

April 2022

Contents

1 Engineers Handbooks

The **Engineer's Handbooks** provide wider guidance on specific areas of the program, for example, the workflows necessary to achieve specific design objectives.

We recommend you familiarize yourself with the Engineer's Handbook topics from the list below that are relevant to the types of structure you work with:

1.1 Wind modeling handbook

Wind loads can be applied to Tekla Structural Designer models in a variety of ways. The most appropriate method to use will generally depend on the type and complexity of the structure.

- The wind model method (page 12) is suitable for regular closed structures that can be readily clothed in wall panels and roof panels.
- The open structure wind method (page 89) is suitable for open industrial structures that cannot be clothed in wall panels and roof panels.

- The simple wind method (page 97) provides a quick means to apply wind loads to a closed structure without having to create a wind model.
- For tall buildings you may require the assistance of wind specialists to perform wind tunnel testing (page 108). The information they supply can then be imported to Tekla Structural Designer model in the form of diaphragm loads.

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

Click the links below to find out more:

- Use of a wind model to create wind loads (page 12)
- The open structure wind method (page 89)
- Manually applied wind loads and simple wind loads (page 97)
- Wind tunnel testing and diaphragm loads (page 108)

Use of a wind model to create wind loads

This method is typically used for enclosed structures and requires at least one wall panel, or roof panel to exist before it becomes available.

In this method, you 'clothe' the structure in wind and roof panels, and then create a wind model by running the Wind Wizard.

The wizard creates wind zone loads that are subsequently decomposed to the structure during analysis.

Wind loadcases are then created for the required directions.

- Overview of the wind model method (page 12)
- ASCE 7 Wind wizard (page 15)
- EC1991 1-4 Wind wizard (page 30)
- BS6399-2 Wind wizard (page 54)
- IS 875 (Part 3) Wind Wizard (page 71)
- Wind model loadcases (page 75)
- Wind model load decomposition (page 80)

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

Overview of the wind model method

This guide provides an outline of the basic steps required to use the wind model method.

The basic steps required to undertake the wind model method are as follows:

Clothe the structure in wind wall and roof panels

The Wind Model calculations depend on the geometry and inter-connectivity of the wall panels and roof panels that envelope the building. You must therefore define the model together with its wall and roof panels before you run the **Wind Wizard...**.

NOTE You can, should you wish, use Tekla Structural Designer purely for wind assessment - by setting up a model consisting only of wall panels and roof panels (no members). Tekla Structural Designercan then determine the wind loading on the building envelope.

In order to get the best results you should ensure that you define the largest possible sizes for the wind wall and roof panels. You may compromise the results if you define many small panels rather than one large one. (The calculation of the reference height in particular can be unconservative.)

Applying Wall Panels

A single wall panel is determined to be a single planar surface. The outward face is vitally important for determining the wind direction relative to the wall, that is windward or leeward.

It is recommended that you check the outward faces are as you intend by ensuring they are all shaded in the same color (the one assigned to 'Wind Wall - Front' in Settings > Scene). The inward faces will all be shaded in a different color. To correct any mistakes, choose the **Edit** --> **Reverse** command and then click once on a wall panel to switch its direction. Note that connected wall panels are checked to ensure that the normal directions are consistent whenever automatic zoning is carried out, for example at the end of the **Wind Wizard...**. If there is a problem it is indicated on the **Project Workspace** --> **Wind** tab, with affected panels being marked.

Once a wall panel has been placed the following additional panel properties can be specified:

- **Rotation angle** - defines the span direction, 0° is horizontal and 90° is vertical.
- **Is a parapet wall** - you can indicate whether the wall panel is a parapet or not.

  NOTE If a building face comprises a parapet above a wall, you should not attempt to model this as a single wall panel. It should be input as an upper and lower panel, with the upper panel being set as a parapet.

- **Gap** - (Head Codes: EC and BS only) where the gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funnelling. A zero gap value explicitly means ignore funnelling, for example where this building and the adjacent one are sheltered by upwind buildings
- **Solidity** - (Head Codes: EC and BS only) If you set the wall panel as a parapet, then you also need to indicate the Solidity of the parapet. (Wall panels that are not parapets automatically adopt a solidarity of 1.0).
- **Decompose to** - for wall panels that are not parapets, you can indicate how the wall load is decomposed on to supporting members. See Wind model load decomposition (page 80)

To set this information as you require, select the wall panels and then use the Properties Window to make changes.

Applying Roof Panels

A single roof panel is determined to be a single planar surface. The orientation of a roof panel is automatically determined when placed based upon the slope vector - the line of maximum roof slope.

Initially the roof type is set to 'Default'. This is interpreted as Flat if the roof slope < 5 degrees, otherwise it is interpreted as Monopitch. You should select the roof panel and then use the Properties Window to adjust the roof type as necessary for all other situations (i.e. For Duopitch, Hip Main, Hip Gable or Mansard).

The span direction is also set in the Properties Window, this is defined as an angle, where 0° is parallel to the X axis and 90° is parallel to the Y axis.

Perform the gravity design

We recommend that you perform an analysis and design at this stage for the gravity loading only, but this is not essential.

Run the wind wizard

Once the model has been 'clothed' in wall panels and roof panels, the Wind Wizard (located on the Load toolbar) guides you through the process of intelligently 'applying' wind to the resulting building envelope.

The wizard uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location.

Having defined the wind directions in which you are interested, on completion of the wizard the appropriate wind zones on the roofs and walls of your structure are automatically calculated.

Related topics

EC1991 1-4 Wind wizard (page 30)

ASCE 7 Wind wizard (page 15)

BS6399-2 Wind wizard (page 54)

IS 875 (Part 3) Wind Wizard (page 71)

Review the wind zones

The resulting wind model is accessed from the Project Workspace Wind tab. Wind Views can also be opened as required for each wind direction.

From here, you can set the type of each roof to achieve the correct zoning, and can then tailor the zoning to account for particular features in more detail, if you so require.

Define the wind loadcases

The Wind Loadcases dialog (located on the Load toolbar) can then be used to automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases the appropriate wind pressures are calculated on each zone.

NOTE It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

Related topics

Wind model loadcases (page 75)

Review wind zone loads

Wind zones can be graphically displayed for each wind direction from the appropriate Wind View. Once the wind loadcases have been created you can also display the wind pressures and zone loads for each loadcase.

Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the Design toolbar.

ASCE 7 Wind wizard

This topic will discuss in detail the Wind wizard when using ASCE7

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

Once the wall and roof panels are in place, you use the Wind Wizard on the Load toolbar to define sufficient site information to calculate the velocity pressures for the required wind directions and heights around the building.

NOTE Unless explicitly noted otherwise, all clauses, figures and tables referred to in the topics in this section are from ASCE 7-10. References (page 89) 1.

Topics in this section

Scope (ASCE7 Wind Wizard)

The scope of ASCE7-10 Wind Wizard encompasses:

- Choice of method:
    - ASCE/SEI 7-10 - Directional Procedure Part 1 - Rigid Buildings of All Heights
    - ASCE/SEI 7-10 - Envelope Procedure Part 1 - Low-Rise Buildings
- The input of appropriate basic wind data is your responsibility.
- Having defined wall panels and roof panels (defaults are standard wall, flat or pitched roof depending on the slope), you are able to specify the type in more detail e.g. monoslope / mansard etc.
- Wherever possible the wind parameters are determined for you but conservatively, you are able to override the values should you wish to.
- Given the above, zoning is semi-automatic, with full graphical feedback.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).
- There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind loadcases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Limitations (ASCE7 Wind Wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind Wizard extensive reference has been made to the References (page 89) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

The shape of the building must lie within the shapes that are valid according to ASCE7-10 Clauses 27.1.2 and 28.1.2:

- It should be a regular shaped building or structure
- It should not have response characteristics making it subject to across-wind loading, vortex shedding, instability due to galloping or flutter; or have a site location for which channelling effects or buffeting in the wake of upwind obstructions warrant special consideration.

Although the software will generate wind loads for many situations - it is up to you to accept that the loads generated are suitable according to ASCE7-10.

Other documented limitations include:

- Buildings must be enclosed or partially enclosed.
- Open sided buildings are not considered.
- Only rigid buildings are considered, not flexible buildings.
- Zones and loadcases are not generated for components and cladding.
- You will need to establish and enter the wind data yourself.
- Roof types must be set manually.
- Barrel-vault and domed roofs are not considered.
- Parapets and free-standing canopies are not considered.
- Roof Overhangs are not explicitly handled.
- There is no special handling for Multi-Bay roofs as they are not covered explicitly in ASCE 7-10.
- There is no special handling for troughed roofs as they are not covered explicitly in ASCE 7-10.

Loaded area

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.
- Orthogonal wind directions at the definition of the user.

Design wind load

The design wind load is not explicitly checked to ensure it is greater than the minimum of 16 psf - (as per Clause 27.4.7 for the Directional method, or 28.4.4 for the Envelope procedure.)

However, the average wind pressure for each of Windward, Leeward and Side directions is provided for you to manually check this is satisfied.

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind loadcases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Choice of Method (ASCE7 Wind Wizard)

Property	Description
Rigid Buildings of All Heights: Directional Procedure Part 1	Choose either the All Heights method, or the Low-Rise Buildings method as required.
Low-Rise Buildings (<=60ft or 18m): Envelope Procedure Part 1	
Apply Open Structure Wind Load	With this box checked, additional wind forces are applied to those members, ancillaries and equipment that have the **Apply**

Property	Description
	Open Structure Wind Load property selected in their properties. **NOTE** This option is only displayed if at least one entity has been selected to have open structure wind load applied. For more information, see: The open structure wind method (page 89)
Next	Depending on whether you choose the Low-Rise or the Rigid Buildings method, clicking **Next** takes you to either the Low rise building - Geometry page, or the Rigid building of all heights - Geometry page.

Envelope Procedure Part 1 - Low-Rise Buildings (Chapter 28)

The Low-Rise Building method is explicitly limited to buildings where the mean roof height does not exceed the least horizontal dimension and is less than or equal to 60 ft, (ASCE7-10 Clause 26.2). It is implied that the method should only be used for simple rectangular box-shaped buildings, however, you are given the final responsibility for determining the applicability of this method to your model.

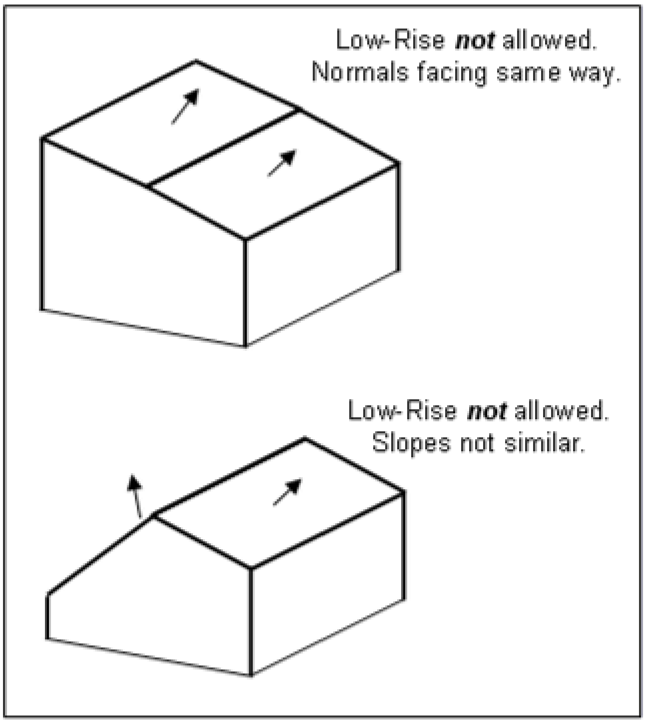
Failure to comply with any of the following conditions will explicitly prevent access to this method:

- There must be 4 walls which must be connected sequentially and form a "simple" quadrilateral in plan form, (all internal corners between 82.5° and 97.5°).
- Each wall must be almost vertical (>80°)
- Each wall must either have 4 sides, forming a "simple" quadrilateral in elevation, or 5 sides forming a convex shape, (allowing for gable ends of buildings).
- The roof system must be one of the following:-
    - Single quadrilateral roof with type "Flat"

      NOTE Where a building has a single roof with a low slope, (e.g. 2°), the default will be "Pitched", but if you consider the building suitable for the Low-Rise method, you can force the roof type to "Flat" rather than "Monoslope".

    - Two quadrilateral "Pitched" roofs which must have a single edge in common. The roofs must face in opposite directions and have similar slopes, (less than 3° difference).
    - Two quadrilateral "Hip Main" roofs which must have a single edge in common and either 1 or 2 hip gable roofs which must be triangular. The

"Hip Main" roofs must face in opposite directions and have similar slopes, (less than 3° difference).



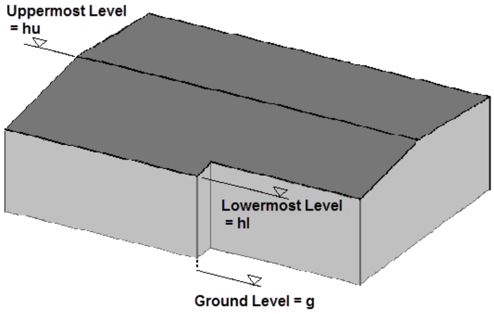
- In particular, the Low-Rise Method is not allowed with Monoslope or multi-span roofs.

Low Rise Buildings - Geometry (ASCE7 Wind Wizard)

The following geometry items are required for the Low Rise Buildings method. You can override calculated dimensions using your engineering judgement.

Property/Buttons	Description
Property	
Ground level	If for some reason, the level 0.0 feet in the Tekla Structural Designer model does not correspond to the ground level, e.g. you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. The default is zero. The allowed maximum is the minimum wall or roof height. Changing the value in this field will cause the Mean Roof Height to be

Property/Buttons	Description
	recalculated, unless you have chosen to override that dimension.
Orientation of Longitudinal Direction relative to axes (Figure 28.4-1)	This information is required to control the orientation of the Transverse and Longitudinal directions, and thus the Tekla Structural Designer Wind Directions. For simple roof shapes, *ASCE7- 10 Wind Wizard* will calculate a default value as below. • Single flat roof - orient along longest side, • 2 Duopitch roofs - orient along common edge, • 2 Hip Main roofs - orient along common edge. • Changing the value in this field will cause either or both of the Longitudinal and Transverse Dimensions to be recalculated, unless you have chosen to override them.
Roof Angle	The field is always visible, even if there are no roof panels in your model. It is calculated as follows: • No roof panels - θ = 0°. • Else - use maximum angle for all roof panels in the model. You are able to override the calculated value by checking the box; The limits are 0° to 80°.
Mean Roof Height, h (Clause 26.2)	The field is calculated as follows: • No roof panels -h is maximum reference height for walls, • Else if θ <=10° - use maximum eaves height, (h e), for all roof panels in the model, (see Figure 28.4-1) • Else - use maximum Mean Roof height, (h), for all roof panels in the model. In the example below, $h = (h_u + h_l)/2 - g$

Property/Buttons	Description
	 h is limited to a maximum of 60 ft or the Least Horizontal Dimension, whichever is lower. You are able to override the calculated value by checking the box.
Longitudinal and Transverse Dimensions (Clause 26.3 and Figure 28.4-1)	These are similar to the Overall building X dimension and overall building Y dimension (Clause 26.3) for the Rigid Buildings of All Heights Method. These dimensions are calculated from the smallest enclosing rectangle (considered over all roof and walls only), relative to the given orientation of the Longitudinal Direction. You are able to override each calculated value by checking its box. These values will then be used to derive the L and B dimensions for each wind direction. For Longitudinal Wind Directions, L = Longitudinal Dimension, B = Transverse Dimension. For Transverse directions, L = Transverse Dimension, B = Longitudinal Dimension.
Buttons	
Next	Clicking **Next** takes you to the Basic wind data page.

Rigid Buildings of All Heights - Geometry (ASCE7 Wind Wizard)

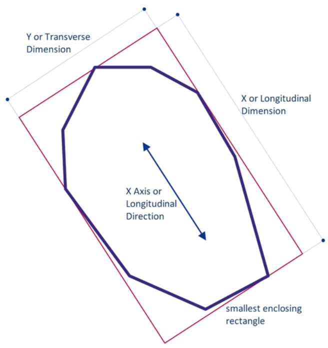
The following geometry items are required for this method. You can override calculated dimensions using your engineering judgement.

Property/Buttons	Description
Property	
Ground Level in Model (Ignore Wind Below)	If for some reason, the level 0.0 feet in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. The default is zero. The allowed maximum is the minimum wall or roof height. Changing the value in this field will cause the Mean Roof Height to be recalculated, unless you have chosen to override that dimension.
Orientation of Principal Axes	This is similar to the Orientation of Longitudinal Direction relative to axes (Figure 28.4-1) (page 20) for Low-Rise Buildings. Although there is no need to distinguish between the Longitudinal and Transverse axes for this method, the wind X axis will be aligned to this direction with the Y axis at right angles. The vortex view will be slightly different to the Low-Rise one to reflect that. Due to the potential complexity of the model, the **Wind Wizard...** will only attempt to determine the correct angle for the axes if the building is also suitable for Low Rise, otherwise zero is used as the default.
Mean Roof Height, h (Clause 26.2)	For this method, the actual reference height is only be used for the pressure on windward walls. For all other walls, and roofs, a single height is used to determine the pressure for each direction. By default, the height is calculated as for Low-Rise Buildings except that there is no single value of θ, so the

<table>
<tr><th>Property/Buttons</th><th>Description</th></tr>
<tr><td></td><td>eaves height is only used if all the roof angles are <= 10°. You are able to override the calculated value by checking the box.</td></tr>
<tr><td>Level of Highest Opening in Building, z_i (Clause 27.4.1)</td><td>For this method, q_i is evaluated at height h for all cases except for positive internal pressure in partially enclosed buildings, where it should be evaluated at the level of the highest opening. However, the clause allows h to be used even for this case, so the default is for the box to be checked and the level to be automatically updated as the Mean Roof Height changes.</td></tr>
<tr><td>Overall Building X Dimension and Overall Building Y Dimension (Clause 26.3)</td><td>These dimensions are calculated from the smallest enclosing rectangle (considered over all roof and walls only), relative to the given orientation of the Principal Axes - see figure below. You are able to override each calculated value by checking its box. These values will then be used to derive the L and B dimensions for each wind direction.

For X Axis, L = X Dimension, B = Y Dimension.

For Y Axis, L = Y Dimension, B = X Dimension:


</td></tr>
<tr><td>Design Pressure Factor (Figure 27.4.8 and Clause 27.4.6)</td><td>This defaults to 75%, but the commentary suggests that this may not cover all cases so you are allowed to change it. A single factor is used for all Torsional loadcases.</td></tr>
</table>

Property/Buttons	Description
Eccentricity (Figure 27.4.8 and Clause 27.4.6)	This defaults to 15% but again, the commentary suggests that this may not cover all cases so you are allowed to change it. A single factor is used for all Torsional loadcases.
Buttons	
Next	Clicking **Next** takes you to the Basic wind data page.

Basic Wind Data (ASCE7 Wind Wizard)

Once the geometry has been confirmed, (for either method), you are then required to enter the basic wind data. The only difference between the two methods is that the Gust Effect Factor field is only shown for the All Heights Method.

Property/ Buttons	Description
Property	
Basic Wind Speed (Clause 26.5.1)	A strictly positive value is required with 90mph being the default.
Hurricane-Prone Region (Clause 26.2)	The default is cleared, i.e. region not prone to hurricanes.
Directionality Factor, Kd (Clause 26.6, and Table 26.6-1)	The default is 0.85, range 0.85 to 1.0 inclusive. **NOTE** There is no cross-checking to ensure that the model has met the load combination criteria. If that does not apply, then it is your own responsibility to enter the correct value of 1.0.
Enclosure Classification (Clause 26.10)	Options are "Enclosed" and "Partially Enclosed" with default being "Enclosed". **NOTE** "Open" structures are not handled in the current version of the program
Gust Effect Factor (Clause 26.9.1)	This will only be shown for the All Heights Method. The default value is 0.85.
Principal Axes	There are always four directions shown for principal axes, being based on the Orientation of the Longitudinal Direction or Principal Axes depending on the method. You are not able to add, delete or modify any direction.

Property/ Buttons	Description
Exposure Category (Clause 26.7.3)	Options are "B", "C" or "D" with default being "B".
Topographic Feature (Clause 26.8 and Figure 26.8-1)	Options are as follows, with the default being "None": • "None" - no feature, i.e. K_{zt} = 1.0. • "2D Ridge" • "2D Escarp" - 2D Escarpment • "3D Hill" - 3D Axisymmetrical Hill
Crest Height, H (Figure 26.8-1)	Height of the hill or escarpment relative to the upwind terrain. The behaviour of this field depends on the Feature type as follows: • "2D Ridge" - non-zero values allowed, (negative indicates a valley). • "2D Escarp" - strictly positive values allowed • "3D Hill" - strictly positive values allowed
Crest Length, Lh (Figure 26.8-1)	Distance upwind of crest to where the difference in ground elevation is half the height of the hill or escarpment.
Distance to Crest, x (Figure 26.8-1)	Distance upwind or downwind from the crest to the building site. The value may be positive to indicate downwind, or negative to indicate upwind.
Buttons	
Next	Clicking **Next** takes you to the Results page.

Results (ASCE7 Wind Wizard)

The final page of the Wind Wizard is a summary of the velocity pressure results for the principal axes. You are able to use the **Details...** button to obtain additional information, including the values of intermediate factors used in the calculations.

Finishing the Wind Wizard

When you click **Finish**, the **Wind Wizard...** generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 16) have been encountered.

Related topics

Wind Zones - ASCE7 Low Rise Building Method (page 27)

Wind Zones - ASCE7 All Heights Method (page 28)

Wind Zones - ASCE7 Low Rise Building Method

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

Wind Directions

Eight zone directions evenly spaced at 45 degree intervals about the longitudinal direction are defined as follows:

- Long (B) 1 - (longitudinal direction + 22.5°, uses data for +X' principal axis)
- Trans (A) 1 - (longitudinal direction + 67.5°, uses data for +Y' principal axis)
- Trans (A) 1 - (longitudinal direction + 112.5°, uses data for +Y' principal axis)
- Long (B) 4 - (longitudinal direction + 157.5°, uses data for -X' principal axis)
- Long (B) 3 - (longitudinal direction + 202.5°, uses data for -X' principal axis)
- Trans (A) 3 - (longitudinal direction + 247.5°, uses data for -Y' principal axis)
- Trans (A) 2 - (longitudinal direction + 292.5°, uses data for -Y' principal axis)
- Long (B) 2 - (longitudinal direction + 337.5°, uses data for +X' principal axis)

These enable the modeling of two sets of zones per principal axis with the correct reference corner in each case.

Wall Zones

The Wind Wizard automatically generates wall zones, where possible, for each *Direction of MWFRS Being Designed* and *Reference Corner* in accordance with Figure 28.4-1.

Roof Zones

The **Wind Wizard...** automatically generates roof zones, where possible, for each *Direction of MWFRS Being Designed* and *Reference Corner* in accordance with Figure 28.4-1.

For flat roofs and for MWFRS parallel to the ridge line we assume Note 8 is not applicable.

Where Note 7 applies, we assume the dimension to the zone 2/3 boundary is measured horizontally.

Automatic Zoning

Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°.

Wind Zones - ASCE7 All Heights Method

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

Wind Directions

Eight zone directions evenly spaced at 45 degree intervals starting from the longitudinal direction are defined as follows:

- +X - (longitudinal direction, uses qz for 1st principal axis)
- +X+Y - (longitudinal direction + 45°, uses q_z for 1st and 2nd principal axes)
- +Y - (longitudinal direction + 90°, uses q_z for 2nd principal axis)
- -X+Y - (longitudinal direction + 135°, uses q_z for 2nd and 3rd principal axes)
- -X - (longitudinal direction + 180°, uses q_z for 3rd principal axis)
- -X-Y - (longitudinal direction + 225°, uses q_z for 3rd and 4th principal axes)
- -Y - (longitudinal direction + 270°, uses q_z for 4th principal axis)
- +X-Y - (longitudinal direction + 315°, uses q_z for 1st principal axis)

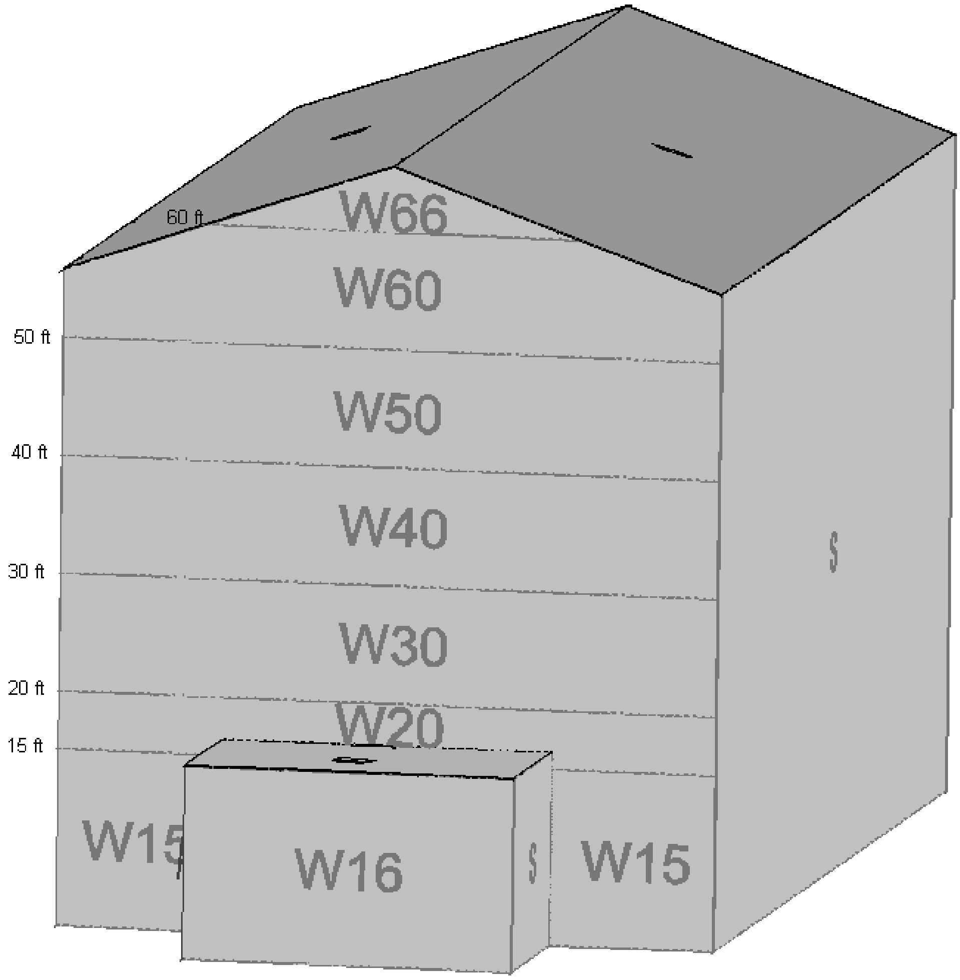
The 4 directions +X,+Y,-X,-Y enable the modeling of design wind loadcases 1 and 2 from Figure 27.4-8. The 4 intermediate directions +X+Y etc. enable the modeling of loadcases 3 and 4.

Wall Zones

The **Wind Wizard...** automatically generates wall zones, where possible, in accordance with Figure 27.4-1.

Windward Walls are split horizontally over intermediate heights. Zones are labelled with a 'W' followed by the height at top of zone rounded to nearest foot (or meter if using metric units). Each zone uses the same C_p, (0.8 from Wall C_p table in Figure 27.4-1), but a different velocity pressure, q_z, z being determined at the top of the zone.

Complex wall shapes are dealt with by splitting zones intelligently as illustrated below:



Leeward walls have a single zone, 'L', using a Wall C_p from the table in Figure 27.4-1, with interpolation for L/B. The velocity pressure, q_h is used for all such zones.

Side walls have a single zone, 'S', using the same C_p, (-0.7 from the Wall C_p table in Figure 27.4-1). A single velocity pressure, q_h is used for all such zones.

Skew Walls

There is no guidance in ASCE7-10 for walls not orthogonal to the principal axes. However, the **Wind Wizard...** will apply zones in those cases and it is your responsibility to check the wind loads adopted.

Roof Zones

The **Wind Wizard...** automatically generates Windward and Leeward roof zones ('W' and 'L' respectively, where possible, in accordance with Figure 27.4-1. This is possible for Windward and Leeward roof panels with θ >= 10°, because there is only one zone. However, for θ < 10° and Side roof panels, automatic zoning will not be carried out for all cases.

Negative and positive values of C_p are determined for each zone from the Roof C_p table in Figure 27.4-1.

For non-principal axis directions, it is assumed that roof loads are not required and so special zones are created, named "Zero". Such zones will result in zero loading on the relevant roof panels. However, if the you choose to do so you can change these to non-standard zones and enter coefficients to force loads to be applied.

When calculating the area reduction factor we use the slope area not the plan area.

Mansard roofs are not automatically detected, i.e. it is your responsibility to set the Roof Type manually. Generally, Mansard roofs are handled exactly the same as if they are:

- Flat (for slope < 0.1°)
- Pitched (for slope >= 0.1°)

There is no guidance on what to do with other multipitch roofs, so they are treated as pitched roofs, whether they have been flagged as Mansard or not.

Side roofs are split into to 4 zones depending on the size of the roof. See the Roof C_p table in Figure 27.4-1.

- Zone 1 - 0 to h/2, interpolating between 2 values for h/L if necessary.
- **NOTE** Upper value for interpolation can be reduced linearly with the sloped area of this zone, (see ** in Figure 27.4-1).
- Zone 2 - h/2 to h, interpolate between -0.9 and -0.7 for h/L if necessary
- Zone 3 - h to 2h, interpolate between -0.5 and -0.7 for h/L if necessary
- Zone 4 - > 2h, interpolate between -0.3 and -0.7 for h/L if necessary

Automatic Zoning

Automatic zoning will apply to all Windward and Leeward roof panels with θ > 10°, because there is only one zone. However, for θ < 10° and Side roof panels, automatic zoning will not be carried out for all cases.

EC1991 1-4 Wind wizard

This topic will discuss in detail the Wind wizard when using the Eurocode BS EN1991-1-4

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS EN 1991-1-4.

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the peak wind velocity and velocity pressures for the required wind directions and heights around the building, (that is the Reference Heights (z_e and z_i) for each wall panel or roof panel).

The wind velocity calculations are automated, with the data source for the calculations being either:

- Input directly for the worst case,
- Input directly for each direction,
- taken directly from the BREVe database which is based upon the Ordnance Survey data of Great Britain (only available for users working to the UK or Ireland National Annex).

Topics in this section

Design Codes and References

Unless explicitly stated all calculations in the EC1991 1-4 Wind Wizard are in accordance with the relevant sections of EC EN1991 1-4 (Ref. 3) (page 89) and the chosen National Annex. It is essential that you have a copy of the code and National Annex with you while assessing wind on any structure.

We would recommend having the following books to hand when using the software:

- Designers' Guide to EN 1991-1-4. Euro Code 1 : Actions on Structures, General Actions Part 1-4 : Wind actions. (Ref. 6) (page 89)
- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. (Ref. 7) (page 89)

In addition, you may find the following book useful:

- Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009. (Ref. 5) (page 89)
- Unless explicitly noted otherwise, all clauses, figures and tables referred to in this section of the handbook are from EC EN1991 1-4. (Ref. 3) (page 89)

Scope (Eurocode EC1991-1-4 Wind wizard)

There is no guidance in the standard for anything other than a cuboid building. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings. It is therefore the user's responsibility to ensure that the wind loading generated by the software meets the needs of any building with a shape that is beyond the scope of BS EN 1991-1-4:2005.

The scope of EN1991 1-4 Wind wizard encompasses:

- Enveloping the building with wall panels and roof panels is undertaken in Tekla Structural Designer in the normal manner. There is only limited validation of the envelope defined (for example connected wall panels must have consistent normal directions). The onus is on you to model the building shape as completely and as accurately as you determine necessary.
- Basic Wind Velocity and Peak Velocity Pressure is determined.
- Having defined wall panels and roof panels (defaults are standard wall, flat or monopitch roof depending on the slope), you are able to specify the type in more detail e.g. multi-bay, monopitch / duopitch etc.).
- The main wind parameters, are calculated for you but conservatively, (for example Crosswind Breadth, b, is determined for the enclosing rectangle of the whole building). Wherever possible other parameters are determined conservatively, but you are able to override the values should you need to.
- Given the above, zoning is semi-automatic, (not attempted for roofs with more than 4 sides which are defaulted to single conservative coefficient), with full graphical feedback.
- The software follows the UK NA (Ref. 3) (page 89) recommendation that BS6399 roof zones and coefficients are used, including Mansard, Multipitch and Multibay roofs.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).

Limitations (EC1991-1-4 Wind wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind wizard extensive reference has been made to the References (page 89) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

WARNING You should seek specialist advice for building shapes that are not covered by the Standard - BS EN 1991-1-4:2005.

- EC1 1-4 does not treat downwind re-entrant corners as special cases - see BS6399 Clause 2.4.3.1 c). So, they are ignored in the software and no warnings are given.
- EC1 1-4 does not handle stepped profiles, or inset storeys - see BS6399 clauses 2.4.4.2 and 2.5.1.7. Hence the software does not handle them automatically, but does generate warnings if such cases are detected - so you can manually edit the zones according to your engineering judgement.
- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered
- Dominant Faces are not explicitly handled - Clause 7.2.9 (5). However, you can use Table 17 to calculate the necessary Cpi value or values and manually apply to a loadcase or individual zone loads.

Loaded areas

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.
- Orthogonal wind directions at the definition of the user.

Overall loads

- Lack of correlation of pressures between the windward and leeward sides. For Overall loadcases, the software automatically reduces the windward and leeward wall pressures only. EC1 1-4 and the UK NA both suggest that the reduction "may" be applied to roofs as well.
- Division by Parts rule for "slender" buildings -Clause 7.2.2 and Figure 7.4 - not applied.

- Friction Forces - Clause 5.3 (3), equation 5.7 and Clause 7.5.

  During the "Update Zones" process, checks are performed to see if the effects can be disregarded, (Clause 5.3 (4)), and a 'Friction needed' warning is generated if not. When they cannot be disregarded you will need to manually model the friction forces as lateral loads in a separate loadcase and include them in your combinations.

Beneficial loads

- Asymmetric and Counteracting Pressures and Forces - Clause 7.1.2 and NA.2.23. Beneficial loads are not automatically removed - instead you are able to flag individual loads to be reduced to zero.

Singapore National Annex - Minimum horizontal loads

- The Foreword to the Singapore National Annex to EN 1991-1-4 Wind Actions has a minimum horizontal load requirement (1.5% characteristic dead weight). Therefore if this National Annex has been applied, it is the users responsibility to check that this requirement has been met (by ensuring that the horizontal component of the factored wind load is greater).

Finland National Annex

- We do not consider thermal inversions for buildings > 100m tall

Norway National Annex

- We do not consider the transition zones between changes in terrain category.

Wind loading on wall panels

Automatic zoning applies to all wall panels subject to the limitations described below:

- Vertical Walls on rectangular buildings -Clause 7.2.2 - the assumption for wall wind forces is that the building is rectangular or close to being rectangular.
- Wall panels that are more than 15° from the vertical are outside the scope.
- Internal Wells are not covered by EC1 1-4 and in any case are not automatically identified but you can manually edit the zones to apply the roof coefficient or otherwise as you see fit - see BS6399 Clause 2.4.3.2a.
- EC1 1-4 does not specify how to treat recesses in side walls - see BS6399 clauses 2.4.3.2 b) and 2.4.3.3 and 3.3.1.5. So, they are ignored but warnings are given.

Wind loading on roof panels

- Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, i.e. all of the internal angles < 180°
- Special care should be taken for winds blowing on duopitch with slopes that differ by more than 5°. If the wind is blowing on the steeper slope (that is that the less steep slope is downwind of ridge), the downwind slope should be set to be a flat roof with mansard at eaves for this wind direction.
- Mansard and Multipitch Roofs are not detected automatically, although certain special cases can be handled if you set the appropriate type manually - see EC1991 1-4 Wind Zones.
- BS 6399 Table 8 curved and mansard eaves - zones start from edge of horizontal roof.
- Roof Overhangs are not explicitly handled. It is suggested that you should define two separate roof panels - one forming the overhang and the other covering the inside of the building. For a small overhang, you can then manually define Cpi values to be the same as Cpe for the adjacent wall panel, (Clause 7.2.1 (3)). Reference 6 (page 89), p45, implies that larger overhangs can be manually handled by using BS6399, Clauses 2.5.9.3 and 2.6.3, i.e. standard external coefficients for the top surface and Table 18 for the internal coefficients.

NOTE The only slight issue here is that there are two sets of edge zones which will occupy a slightly larger area than strictly necessary.

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind loadcases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Using the EC1991-1-4 Wind wizard

This section runs through each page of the wizard and discusses the various options.

Data Source page

You can choose to enter one set of Worst-Case data or different values for each direction to be considered.

TIP Additional options are provided for using BREVe data when working to the UK or Ireland National Annex. For further details see: Using the EC1991 1-4 Wind Wizard with BREVe data (page 41).

The remaining choices on the Data Source page are:

Property	Description
Consider Orography	If you select this check box, then the orographic data, (manually entered), is used to determine the Orography Factor c_o as clause A.3. When calculating c_{alt}, the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, c_o is 1.0 for all heights and c_{alt} is the same for all directions, using the Site Altitude.
Consider Tall Neighbouring Structures	If the conditions in clause A.4 are met, then the wind loads need to be based on height z_n, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then z_n will be used as the reference height for all wall panels and roof panels in the model. **NOTE** If working to the Sweden NA, Tall Neighbouring Structures are not considered.
Consider Obstructions	With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, h_{dis} as (A15) in clause A.5. Otherwise the obstructions are ignored and h_{dis} is taken as zero. **NOTE** If working to the Sweden NA, Obstructions are not considered.
Apply Open Structure Wind Load	With this box checked, additional wind forces are applied to those members, ancillaries and equipment that have the **Apply Open Structure Wind Load** property selected in their properties. **NOTE** This option is only displayed if at least one entity has been selected to have open structure wind load applied. For more information, see: The open structure wind method (page 89)
Next	Clicking Next takes you to the **Basic data** page below.

Basic data page

This page is used you to define the site details.

Property	Description
Air density	You need to enter air density at the site.
Ground level	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.
Fundamental Basic Wind Velocity - Clause 4.2 and NA.2.4	Reference should be made to the National Annex being worked to when determining an appropriate basic wind speed.
Season Factor, c_{season}	Valid range 0.01 to 10.0 - default 1.0.
Probability Factor, c_{prob}	Valid range 0.01 to 10.0 - default 1.0.
Default Height for Internal Pressure (z_i)	Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, (z_i), defaulting to the height of the structure. Leaving Use Building Height checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.
Peak Factor, k_p (Sweden NA only)	The Peak Factor was introduced by the Swedish EC NA (EK11) update. Using the default value (3.0) is the equivalent of the old calculation method prior to the update.
Region (Norway NA only)	A Region is needed (Area 1, Area 2 or Area 3) representing three different height zones in the country.
Site Altitude (Norway NA only)	You need to enter the basic altitude that you want to use for the site directly. This is the altitude of your model's base.
Next	Depending on whether you chose worst case data, or data for each direction on the Data Source page, clicking **Next** either takes you to the **Roughness and Obstructions (Worst case)** page or the **Roughness and Obstructions (Data for each Direction)** page.

Roughness and Obstructions (Worst case) page

If you select the Worst Case Data Source, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself.

Property	Description
Terrain Category	Reference should be made to the National Annex being worked to when determining an appropriate Terrain Category. Depending on the terrain category selected and National Annex being worked to, you may also be required to enter some of the following data: • Average height of upwind buildings, • Upwind spacing of surrounding buildings, • Upwind distance from sea to site, • Upwind distance from edge of town to site.
Next	Depending on your selections on the Data Source page, clicking **Next** either takes you to the **Orography (Worst Case)** page, the **Tall Neighbouring Structure** page, or the **Results** page.

Roughness and Obstructions (Data for each Direction) page

If you select the **Other - Data for each Direction Data Source**, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself. However, most of the data is then dependent on the wind direction, so you must also make your choice of wind directions on this page.

Property	Description
Direction	Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update these using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined. Each row of the grid operates in a similar manner to the relevant fields of the **Roughness and Obstructions (Worst case)** page.
Next	Depending on your selections on the Data Source page, clicking Next either takes you to **Orography (Data for each Direction)** page, the **Tall Neighbouring Structure** page, or the **Results** page.

Orography (Worst Case) page

If Consider Orography was checked, then the next page of the Wizard for the Worst Case Data Source allows you to enter the data for Orography.

Property	Description
Orographic Feature (Clause A.3)	Options are: • None - no feature, i.e. c_o = 1.0. • 2D Escarp - Cliffs and Escarpments, • 3D Hill - Hills and Ridges.
Altitude of Upwind Base of Feature, A	This value is used to calculate C_{alt} instead of the Site Altitude because the Orography is significant. **NOTE** C_{alt} will be calculated at z_e for each wall and roof panel, not z_s.
Effective Crest Height, H (Figures A.2 & A.3)	Effective height of the feature.
Length of Upwind Slope, L_u (Figures A.2 & A.3)	Actual length of the upwind slope in the wind direction.
Length of Downwind Slope, L_d (Figures A.2 & A.3)	Actual length of the downwind slope in the wind direction.
Horizontal Distance to Crest, x (Figures A.2 & A.3)	Distance upwind or downwind from the crest to the building site.
Orography factor, $c_o(z)$ and Turbulence factor, k_l (Norway NA only)	When working to the Norway NA you are not required to enter the above factors; instead you enter the orography factor and turbulence factor directly for the defined 3D or 2D orographic feature.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall**

Property	Description
	Neighbouring Structure page , otherwise it takes you to the **Results** page.

Orography (Data for each Direction) page

The wind directions defined on the previous page are maintained and you are not able to update them.

Property	Description
	Each row of the grid operates in a similar manner to the relevant fields of the Orography (BREVe) page (page 46) page.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall Neighbouring Structure** page , otherwise it takes you to the **Results** page.

Tall Neighbouring Structure page

For all methods, if **Consider Tall Neighbouring Structure** was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)

Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, h_{high}, Largest Horizontal Dimension of Tall Neighbour, d_{large} and Distance to Tall Neighbour, x are all as described on Figure A.4 of the code.

Property	Description
Average Height of Neighbours , h_{ave} (Figure A.4)	The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If **Override calculated dimension** is cleared, then the value will be updated whenever the wizard is run, otherwise the user-value is used.
Height of this structure, h_{low} (Figure A.4)	The field is for information only - difference between top of highest wall / roof panel and ground level in the model.
Next	Click Next to go to the **Results** page.

Results page

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but except for the **Other - Data for Each Direction method**, you are able to update this using the **Dir.** buttons and / or changing the direction value as required. (Note : Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the **<<Details>>** button to obtain additional information, including the values of intermediate factors used in the calculations.

Property	Description
Other - Worst Case Data	The calculation of q_p is very similar to the BREVe Method, (see above), except that the worst case data has been entered by you, and this page allows you to enter your own values for C_{dir}. As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each direction.
Other - Data for each Direction	The calculation of q_p is very similar to the BREVe Method, (see above), except that the data has been entered by you for each direction only so a direct calculation can be performed instead of taking the worst case over a range of sectors. Also this page allows you to enter your own values for C_{dir}. As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each direction.
Finishing the Wind Wizard	When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions. **NOTE** Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 32) have been encountered.

Using the EC1991 1-4 Wind Wizard with BREVe data

This page steps you through the EC1991-1-4 Wind wizard when using BREVe data

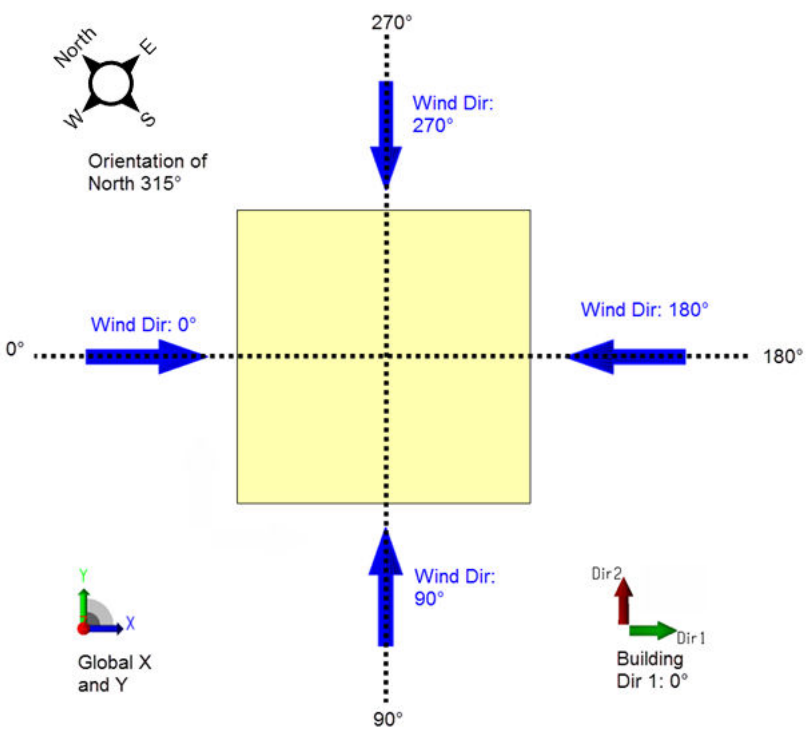
NOTE This option is only available when either the UK or Ireland National Annex has been selected.

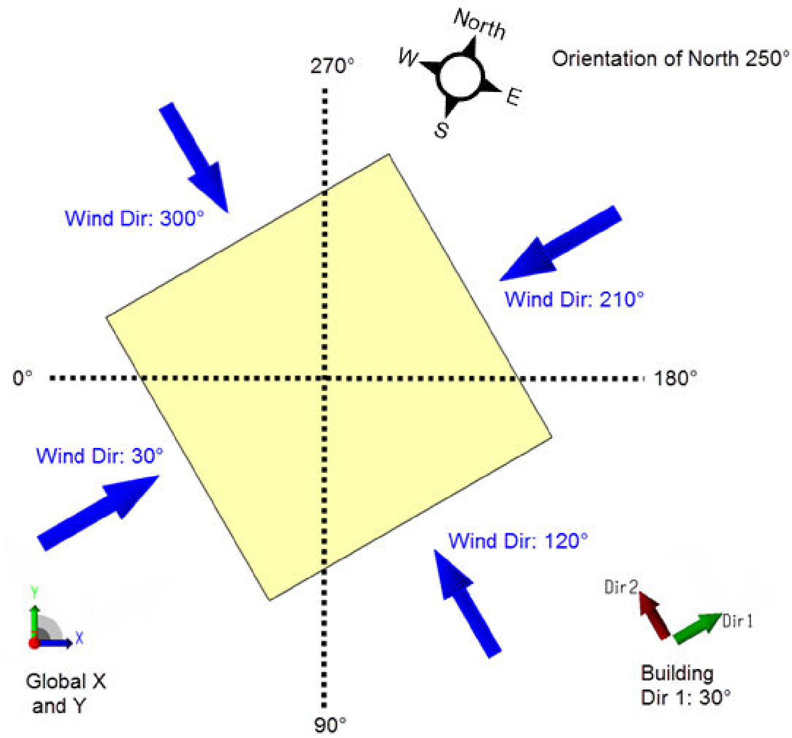
Data Source (BREVe) page

Assuming you choose to specify the site data using BREVe Grid Ref data the remaining choices on the Data Source page are:

Property/ Buttons	Description
Property	
Consider Orography	If you select this check box, then the orographic data, (either recovered by BREVe for the site or manually entered), is used to determine the Orography Factor c_o as clause A.3. When calculating c_{alt}, the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, c_o is 1.0 for all heights and c_{alt} is the same for all directions, using the Site Altitude.
Consider Tall Neighbouring Structures	If the conditions in clause A.4 are met, then the wind loads need to be based on height z_n, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then z_n will be used as the reference height for all wall panels and roof panels in the model.
Consider Obstructions	With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, h_{dis} as (A15) in clause A.5. Otherwise the obstructions are ignored and h_{dis} is taken as zero.
Apply Open Structure Wind Load	With this box checked, additional wind forces are applied to those members, ancillaries and equipment that have the **Apply Open Structure Wind Load** property selected in their properties. **NOTE** This option is only displayed if at least one entity has been selected to have open structure wind load applied. For more information, see: The open structure wind method (page 89)
Buttons	
Next	If you have chosen to use BREVe Grid Ref data, clicking **Next** takes you to the **BREVe location** page , otherwise it takes you to the **Basic Data** page.

BREVe location page

Property/ Buttons	Description
Property	
Grid Ref.	This shows the grid reference of the site which you have picked through BREVe, irrespective of the method you use to define the site location.
Orientation of building known	If you know the orientation of the building with respect to North, then you can define this information by checking this box. You can then define a value which relates the building direction axes of your Tekla Structural Designer model to geographic north.
Orientation of North	The orientation of North is defined using the same convention as is applied to the orientation of the Building Direction Arrows. This can best be understood by reference to a couple of examples: In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°. The resulting relation between the building axes and North is as shown below: 

Property/ Buttons	Description
	In the second example the building direction has been input with Dir 1 = 30° and the orientation of North has been set to 250° In this case the building axes are related to North as shown below: 
Buttons	
	Using BREVe, there are 2 methods available for you to define the site location:
Site By Ref...	You can define the grid reference of the site. You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid reference from a postcode or given location, for example www.streetmap.co.uk or www.multimap.co.uk
Site By Map...	You can pick the site from a Land / Town Map, • You can pick the site from a Orography Map.

Property/ Buttons	Description
	• You can pick the site from a ground roughness Category Map, The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Basic Data (BREVe)** page.

Basic Data (BREVe) page
This page allows you to review the site details taken from the BREVe database.

Property/ Buttons	Description
Property	
Site Altitude, A	The basic site altitude of your model's base.
Air Density	Air density at the site.
Ground Level	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.
Fundamental Basic Wind Velocity ($v_{b,map}$) - Clause 4.2 and NA.2.4	The value required is defined as "the characteristic 10 minutes mean wind velocity, irrespective of wind direction and time of year, at 10m above ground level in open country terrain with low vegetation such as grass and isolated obstacles with separations of at least 20 obstacle heights", but is the value before the altitude correction is applied. (Valid range 1.0 to 1000 m/s).
Season Factor, c_{season}	Valid range 0.01 to 10.0 - default 1.0.
Probability Factor, c_{prob}	Valid range 0.01 to 10.0 - default 1.0.
Default Height for Internal Pressure (z_i)	Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, (z_i), defaulting to the height of the structure. Leaving Use Building Height checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.

Property/ Buttons	Description
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Roughness and Obstructions (BREVe)** page.

Roughness and Obstructions (BREVe) page

This page of the Wizard automatically defaults the data for ground roughness and obstructions for you.

Property/ Buttons	Description
Property	
Terrain Category	Options available are: • **Sea** - this setting is for sites where the distance to sea is between 0 and 1 km, not for offshore sites. As the worst case must be for wind blowing across the sea, there is no need to specify data for upwind buildings or distance in town. • **Country** - the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town, • **Town** - for this category you need to specify data for upwind buildings and distance to the edge of the town, so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for h_{ave} For this category, the Upwind distance from edge of town to site cannot be greater than the Upwind distance from sea to site.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Orography (BREVe)** page.

Orography (BREVe) page

NOTE When using Breve Data you should really leave it up to the system as to whether orography is significant or not. If you chose not to consider orography, the actual factor is not applied; however the site data is still displayed to allow you to check that your decision to ignore it is reasonable.

Property/ Buttons	Description
Property	
Orographic Feature (Clause A.3)	Options available are: • **None** - no feature, i.e. c_o = 1.0. • **2D Escarp** - Cliffs and Escarpments, • **3D Hill** - Hills and Ridges.
Altitude of Upwind Base of Feature, A	This value is used to calculate C_{alt} instead of the Site Altitude because the Orography is significant. **NOTE** C_{alt} will be calculated at z_e for each wall and roof panel, not z_s.
Effective Crest Height, H (Figures A.2 & A.3)	Effective height of the feature.
Length of Upwind Slope, L_u (Figures A.2 & A.3)	Actual length of the upwind slope in the wind direction.
Length of Downwind Slope, L_d (Figures A.2 & A.3)	Actual length of the downwind slope in the wind direction.
Horizontal Distance to Crest, x (Figures A.2 & A.3)	Distance upwind or downwind from the crest to the building site.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	If on the Data Source page you chose to consider tall neighbouring structures, clicking **Next** takes you to the **Tall Neighbouring Structure**, otherwise it takes you to the **Results (BREVe)** page.

Tall Neighbouring Structure page

For all methods, if Consider Tall Neighbouring Structure was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)

Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, h_{high}, Largest Horizontal Dimension of Tall Neighbour, d_{large} and Distance to Tall Neighbour, x are all as described on Figure A.4 of the code.

Property/ Buttons	Description
Property	
Average Height of Neighbours, h_{ave} (Figure A.4)	The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If Override calculated dimension is cleared, then the value will be updated whenever the wizard is run, otherwise the user-value is used.
Height of this structure, h_{low} (Figure A.4)	The field is for information only - difference between top of highest wall / roof panel and ground level in the model.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to got to the **Results (BREVe)** page.

Results (BREVe) page

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update this using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the **<<Details...>>** button to obtain additional information, including the values of intermediate factors used in the calculations.

BREVe Data

BREVe determines the parameters required to calculate $q_p(z)$ for each height in the building at 30° intervals, (0° to 330°).

For each required wind direction the worst case q_p is used for each height, based on splitting the difference to the next direction, with a maximum of ±45 degrees. Within these ranges q_p is not interpolated.

Theoretically, it is possible for a quadrant to use different 30° directions for each height, so the critical wind direction is not displayed in the summary.

The Vortex view shows the Peak Velocity Pressures calculated for each reference height for each 30° sector.

Finishing the Wind Wizard

When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 32) have been encountered.

EC1991 1-4 Wind Zones

Enter a short description of your topic here (optional).

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

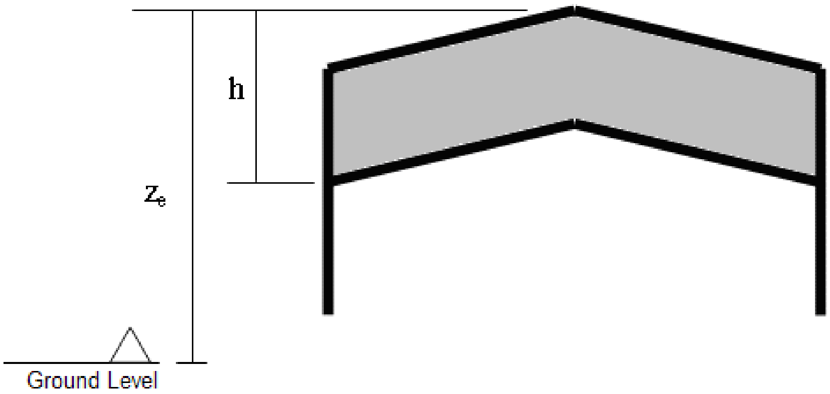
Basic Geometry

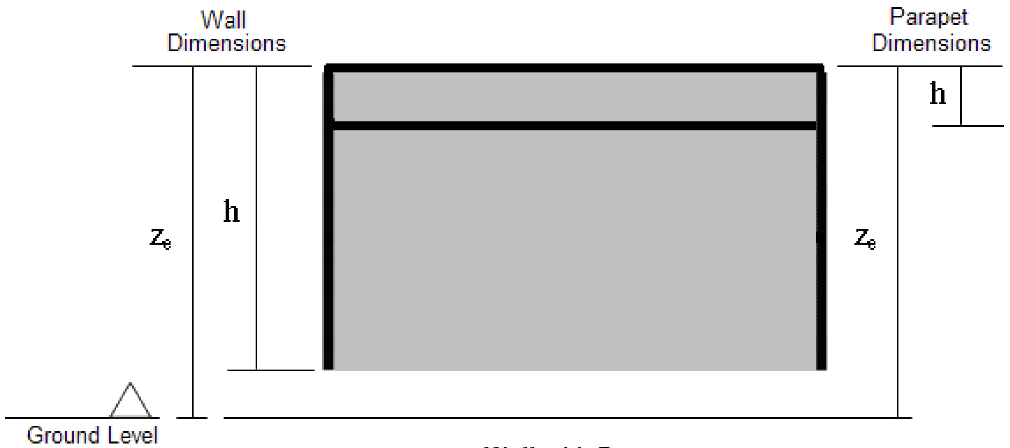
The basic building geometry is assessed as follows:

- Reference Height (z_e) - is taken as the difference between highest point on wall or roof panel and ground level.
- Wall height (h) - is taken as the difference between highest and lowest points on the wall panel.

  These definitions apply to wall panels without parapets and the actual parapets. Wall panels with parapets above them will take their highest point from the parapet. See the diagram below.

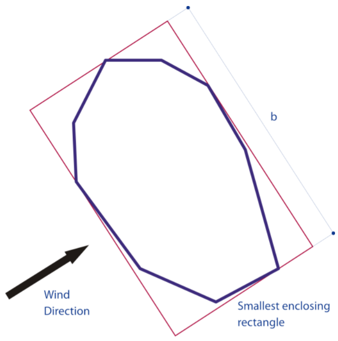
Wall above Roof





Wall with Parapet

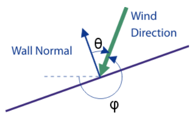
- Roof height (h) - as z_e - taken as the difference between highest point on wall or roof panel and ground level. This definition does not handle the upper roof of inset storey but is conservative.
- The Building Breadth, b is calculated from the smallest enclosing rectangle around the whole building (considered over all roof and wall panels only) for the given direction. You can override the calculated value in case the Tekla Structural Designer model does not include the whole building.



Wall Zones

Wall Type

We assess each wall panel to determine if it is a windward, leeward or side wall. We classify the type of wall dependent on q:



- θ ≤ 60 deg - Windward,
- θ ≥ 120 deg - Leeward,
- Other walls are classed as Side.

Windward walls	Windward walls have a single zone and Table 7.1 is used with interpolation for h/d.
Leeward walls	Leeward walls have a single zone and Table 7.1 is used with interpolation for h/d.
Side walls	In all cases, side walls have the relevant number of zones from Figure 7.5 and Table 7.1 is used. There is no guidance in the standard for Irregular Flushed Faces, Recesses and Downwind Re-entrant Corners that are covered in BS6399. However, it is reasonable to automatically detect Irregular Flushed Faces and process them as described in BS6399 Clause 2.4.4.1 and Figure 14. The program also detects potential Recesses but only generates a warning and no special handling occurs. Downwind re-entrant corners are conservatively ignored.

Parapets	Parapets are assessed for return corners and then classified as windward, windward oblique, leeward or leeward oblique dependent on q. Depending on the classification, parapets will either have a single zone, or up to seven zones (determined by extrapolation from Figure 7.19). Table 7.9 is used with interpolation for Solidity and l/h. **NOTE** An "r" suffix on zone name indicates return corners have been detected. Side Parapets are Special Zero zones, i.e. no nett pressure.

Roof Zones

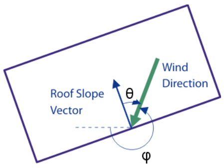
Roof zones are automatically generated where possible for each wind direction. In essence each roof panel is assessed in its own right based on its properties. The interconnectivity of touching roof panels is not generally considered.

NOTE The Advisory note on page 34 of the UK NA is followed so that zones and coefficients are generated according to BS6399.

Direction

Internally the roof slope vector (line of maximum slope) is determined from the normal vector, with its direction always giving a positive slope angle, i.e. the roof slope vector must always point up the slope.

We calculate the angle between the wind direction and projection of roof slope vector onto horizontal plane (q in range -180° to +180°).

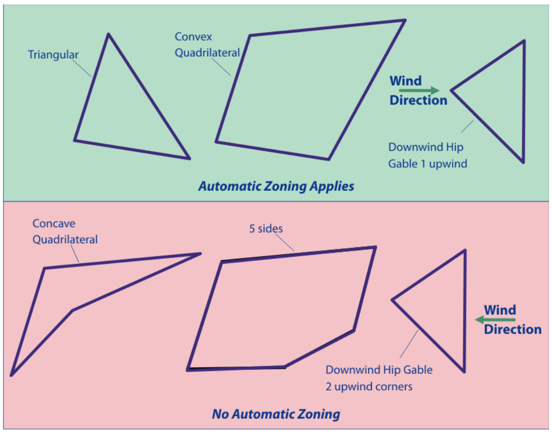


Scaling Dimension, e

The scaling dimension e = min(b, 2h)

Automatic Zoning

Automatic zoning normally only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°. However, additionally, it only applies to Hip Gable roofs if they are triangular, and Hip Main roofs if they are quadrilateral. Further, Downwind Slope Hip Gables must not have 2 upwind corners.



Dimensions	All zone dimensions are specified in plan.
Flat Roofs	See BS 6399 Clause 2.5.1, Figure 16 and Table 8.
Monopitch Roofs	See BS 6399 Clause 2.5.2.3, Figure 19 and Table 9.
Duopitch Roofs	See BS 6399 Clause 2.5.2.4, Figure 20 and Table 10.
Hip Gable	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Hip Main	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Mansard Roofs	If you manually set the connected roof types to Mansard, then the program will correctly identify the special cases in BS6399 Figures 17c, 22a and 22b, and use the correct tables and values. See BS 6399 Clauses 2.5.1.6.2 & 2.5.4
Multi-bay Roofs	We allow you to interpret BS 6399 Clause 2.5.5 and Figure 23 as you think appropriate and manually define the roof types and sub-types accordingly. You also have the ability to manually set the multi-bay position for each roof panel for each wind direction: • Not Multi-Bay - for this wind direction (conservative default), • Upwind Bay - first bay of many for this wind direction, • Second Bay - for this wind direction, • Third or more Bay - for this wind direction. Where the reduction applies, the values of all coefficients are reduced according to Table 12.

Non-Automatic Zoning

Where automatic zoning does not apply, the system creates a single zone covering the entire roof as follows:

• Hip Gable - B for upwind, B for downwind, D for side,

- Flat - B,
- Monopitch - B,
- Duopitch - B for upwind, A for downwind, B for side,
- Hip Gable - B for upwind, B for downwind, D for side,
- Hip Main - B for upwind, A for downwind, D for side.

BS6399-2 Wind wizard

This topic will discuss in detail the Wind wizard when using the British Standard BS 6399-2.

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS 6399-2.

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the effective wind speeds and dynamic pressures for the required wind directions and heights around the building, (that is the Reference Height (H_r) for each wall panel or roof panel).

The wind speed calculations are automated, the data source for the calculations is either:

- taken directly from the BREVe database, which is based upon the Ordnance Survey data of Great Britain,
- input directly.

It should be noted that BS6399-2:1997 recommends that the Standard Method requires assessment of orthogonal loadcases for wind directions normal to the faces of the building. The wizard permits you to create wind load for any wind direction and thus it is up to you to create those loads for the directions most appropriate to your structure.

Topics in this section

Design Codes and References

Unless explicitly stated all calculations in the BS 6399-2 Wind Modeller are in accordance with the relevant sections BS 6399-2:1997 incorporating Amendment 1 and corrigendum No. 1. (Ref. 4) (page 89) It is essential that you have a copy of this code with you while assessing wind on any structure.

Your attention is particularly drawn to BS6399-2:1997 - Clause 1.1. **For building shapes which are not covered by the Standard you will need to seek specialist advice.**

We would recommend having the following books to hand when using the software:

- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. (Ref. 7) (page 89)
- Wind and Loads on buildings: Guide to Evaluating Design Wind Loads to BS6399-2:1997. (Ref. 8) (page 89)

Unless explicitly noted otherwise, all clauses, figures and tables referred to in this handbook are from reference 4 (page 89).

Scope (BS6399-2 Wind wizard)

In the main, BS6399-2:1997 addresses rectilinear buildings. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings using the standard method. For more information, please refer to reference 7 (page 89) (section 2.5.3.2.4, page 82 and 2.5.4.3.3 pages 89-90).

The scope of BS 6399-2 Wind Modeller encompasses:

- Enveloping the building with wall panels and roof panels is undertaken in Tekla Structural Designer in the normal manner. There is only limited validation of the envelope defined (for example connected wall panels must have consistent normal directions). The onus is on you to model the building shape as completely and as accurately as you determine necessary.
- Choice of method:
  - BS6399-2:1997 - Standard Method - Standard effective wind speeds with standard pressure coefficients,
  - BS6399-2:1997 - Hybrid Method - Directional effective wind speeds with standard pressure coefficients.
- Basic Wind Speed and Dynamic pressure is determined.
- Having defined wall panels and roof panels (defaults are standard wall, flat or monopitch roof depending on the slope), you are able to specify the type in more detail e.g. multi-bay, monopitch / duopitch etc.).
- The main wind parameters, are calculated for you but conservatively, (for example Crosswind Breadth, B, is determined for the enclosing rectangle of

the whole building). Wherever possible other parameters are determined conservatively, but you are able to override the values should you need to.

- Given the above, zoning is semi-automatic, (not attempted for roofs with more than 4 sides which are defaulted to single conservative coefficient), with full graphical feedback.
- Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).

Limitations (BS6399-2 Wind wizard)

This page discusses the limitations to the wind wizard.

Throughout the development of the Wind Modeler extensive reference has been made to the References (page 89) and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

Geometry

DANGER You should seek specialist advice for building shapes that are not covered by the Standard - see Clause 1.1 of BS6399-2:1997.

- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Parapets and free-standing canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered.
- Dominant Openings are not explicitly handled - Clause 2.6.2. However, you can use Table 17 to calculate the necessary C_{pi} values and manually apply to a loadcase or individual zone loads.

Loaded areas

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

Wind direction

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.

- Orthogonal wind directions at the definition of the user.

Beneficial loads

- No automatic reduction is made for beneficial load. When you edit the Zone Load Data for a wind direction, having generated wind loadcases, there is an option to allow for beneficial loads.

Wind loading on wall panels

Automatic zoning applies to all wall panels subject to the limitations described below:

- Wall panels that are more than 15° from the vertical are outside the scope - Clause 2.4.1.5.
- The inset storey clause 2.4.4.2 b) is not implemented. You can edit the zones manually according to your engineering judgement to include zone E if you consider this necessary.
- Wall panels of internal wells are not automatically identified - Clause 2.4.3.2a. You can manually edit the zones to apply the roof coefficient to the wall panels.

Wind loading on roof panels

- Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, i.e. all of the internal angles < 180°
- The inset storey clauses 2.5.1.7 a) and b) are not implemented. In clause a) the software sets H_r and H equal conservatively. You are obviously able to edit the zones manually according to your engineering judgement to include the further zones indicated in Figure 18 should you consider this necessary.
- It should be noted that in Table 8 for curved and mansard eaves, the zones start from edge of horizontal roof and not from the edge of the feature.
- Special care should be taken for winds blowing on duopitch with slopes that differ by more than 5°. If the wind is blowing on the steeper slope (that is that the less steep slope is downwind of ridge), the downwind slope should be set to be a flat roof with mansard at eaves for this wind direction.
- Mansard and Multipitch Roofs are not detected automatically, However, you can manually apply the relevant roof type, apex type and bay position parameters for each appropriate wind direction to match the requirements of Figure 22 and Figure 23 - see BS6399-2 Wind Zones.
- Roof Overhangs are not explicitly handled. It is suggested that you should define two separate roof panels - one forming the overhang and the other covering the inside of the building. You can then define C_{pi} values manually

to either have the same coefficient as the adjacent wall, (Clause 2.5.8.2 Small Overhangs), or as an open sided building (Clause 2.6.3).

Additional wind loads

There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind loadcases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Using the BS6399-2 Wind Wizard with BREVe data

Using the BS6399-2 Wind Wizard with BREVe data

Method page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

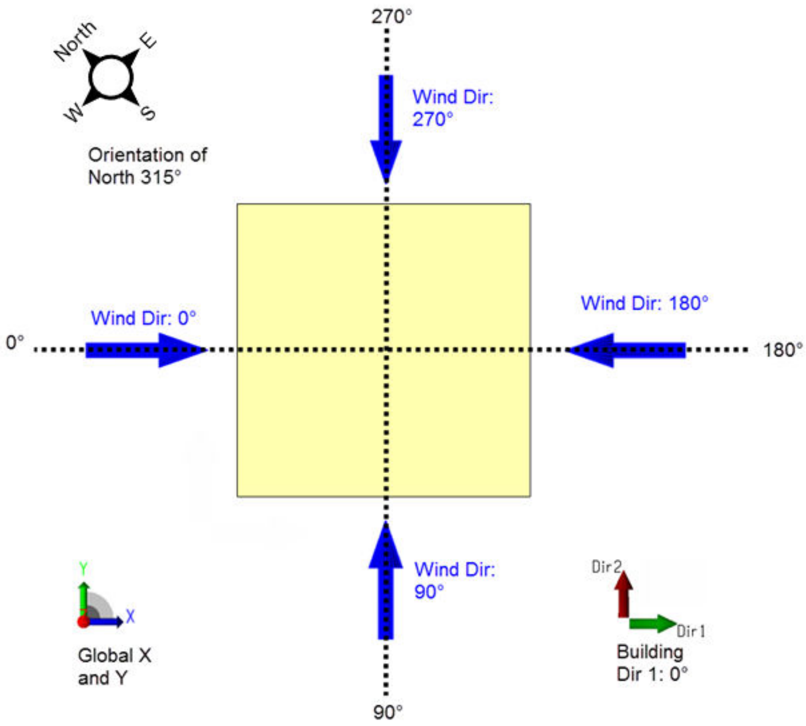
Property/ Buttons	Description
Property	
	There are two calculation methods available:
Standard	Uses standard effective wind speeds with standard pressure coefficients,
Hybrid	Uses directional effective wind speeds with standard pressure coefficients.
	Assuming you have are going to specify the site data using BREVe Grid Ref data there are two options for the source of the wind data: • BREVe - UK National Grid Ref. • BREVe - Irish Grid Ref
Buttons	
Next	If BREVe is the data source, clicking **Next** takes you to the **BREVe location page**; if Other is the data source clicking **Next** takes you to the **Other Location** page.

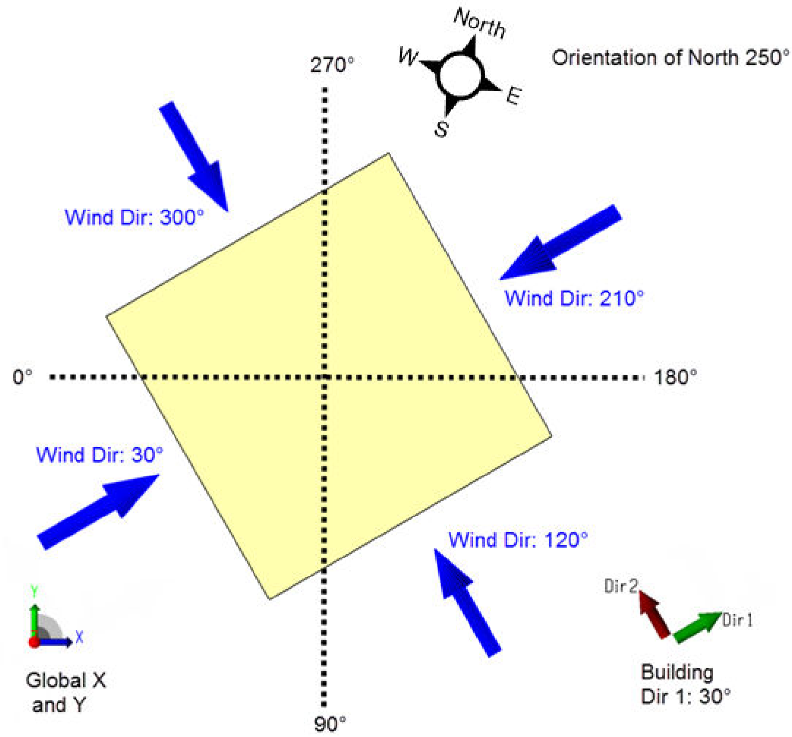
BREVe location page

This page allows you to define the location of the site using the BREVe database, and to define various options to be considered in the wind analysis.

Once you have retrieved the data for a site from the BREVe database you can edit these to take account of your local knowledge of the site.

Property/ Buttons	Description
Property	
Grid Ref.	This shows the grid reference of the site which you have picked through BREVe, irrespective of the method you use to define the site location.
Site Altitude, A	You are able to override the altitude determined by BREVe by entering a value directly here.
Air Density	You need to enter air density at the site.
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly.
Orientation of building known	If you know the orientation of the building with respect to North, then you can define this information by checking this box. You can then define a value which relates the building direction axes of your Tekla Structural Designer model to geographic north. If you want to use the Hybrid method, then you must know and define the building orientation. For the Standard method, the orientation is not essential. If you don't define the building's orientation then North is not shown in graphics views and all the S_d values are set to 1.0.
Orientation of North	The orientation of North is defined using the same convention as is applied to the orientation of the Building Direction Arrows. This can best be understood by reference to a couple of examples: In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°. The resulting relation between the building axes and North is as shown below:

Property/ Buttons	Description
	 In the second example the building direction has been input with Dir 1 = 30° and the orientation of North has been set to 250° In this case the building axes are related to North as shown below:

Property/ Buttons	Description
	
Consider Topography	If you select this check box, then BREVe uses the topographic data it recovers for the site and determines the Altitude Factor S_a in accordance with Clause 2.2.2.2.3. Otherwise the topographic data is ignored and S_a is calculated in accordance with Clause 2.2.2.2.2. **NOTE** In theory the topography could be significant for some directions and not for others.
Consider Obstructions	With this box checked, BREVe uses the obstruction data it recovers for the site and determines the Effective Height H_e as defined in Clause 1.7.3.3. Otherwise the obstructions are ignored and H_e is taken as H_r - see Clause 1.7.3.2.
Buttons	
	Using BREVe, there are 2 methods available for you to define the site location:
Site By Ref...	You can define the grid reference of the site. You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid

Property/ Buttons	Description
	reference from a postcode or given location, for example www.streetmap.co.uk or www.multimap.co.uk
Site By Map...	You can pick the site from a Land / Town Map, • You can pick the site from a Orography Map. • You can pick the site from a ground roughness Category Map, The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 65).

Using the BS6399-2 Wind Wizard with other data

This section runs through the wizard when "other data" is specified.

Method page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

Property/ Buttons	Description
Property	
	There are two calculation methods available:
Standard	Uses standard effective wind speeds with standard pressure coefficients,
Hybrid	Uses directional effective wind speeds with standard pressure coefficients.
	The remaining topics in this section assume you have chosen to enter the site data manually (Other).
Buttons	
Next	Assuming Other is the data source, clicking **Next** takes you to the **Other Location** page.

Other location page

This page allows you to define the site details when information is not available from the BREVe database, for instance if it is located outside of the UK.

Property/ Buttons	Description
Property	
Site Altitude	You are able to override the altitude determined by BREVe by entering a value directly here.
Air Density	You need to enter air density at the site.
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly.
Buttons	
Cancel	Click this button to cancel the wizard.
Previous	Click this button to go back to the previous page of the wizard.
Next	Click **Next** to move to the **Other Standard Wind data** page, or if the Hybrid method was selected to the **Other Hybrid Wind data** page.

Other Standard Wind data page

If you select the Standard Method and Other Data Source, then the next page of the Wizard allows you to enter the wind data yourself.

Property/ Buttons	Description
Property	
Basic Wind Speed	You need to enter the basic wind speed at the site.
Ground Roughness	The following settings are available: • Sea - this setting is for sites where the distance to sea is between 0 and 1 km, (see Clause 1.7.2), it is not for offshore sites., • Country - the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town, • Town - for this category you need to specify data for upwind buildings and distance to the edge of the town,

Property/ Buttons	Description
	so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for H_o For this category, the 'Upwind distance from edge of town to site' cannot be greater than the 'Upwind distance from sea to site'.
Consider Topography / Altitude Factor, S_a	When this box is checked, you need to use your own topographic data and determine the Altitude Factor S_a in accordance with Clause 2.2.2.2.3. Otherwise S_a is calculated in accordance with Clause 2.2.2.2.2 and you are not able to override it.
Season factor	You need to enter the season factor (default 1.0).
Probability factor	You need to enter the probability factor (default 1.0).
Buttons	
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 65).

Other Hydrid Wind data page

If you select the Hybrid Method and Other Data Source then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself. However, most of the data is then dependent on the wind direction, so you must also make your choice of wind directions on this page.

Property/ Buttons	Description
Property	
Direction	Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update these using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined. Each row of the grid operates in a similar manner to the relevant fields of the Other Standard Wind data page
Consider Topography / Altitude Factor, S_a	Reference 8 (section 4.10, page 26) essentially recommends using the Standard Method approach to topography even for the Hybrid Method. So, when

Property/ Buttons	Description
	calculating the Terrain and Building Factor, S_b, we ignore the effects of topography, that is we take S_h = 0. When the box is checked, you need to use your own topographic data and determine the Altitude Factors S_a as defined in Clause 2.2.2.2.3. Otherwise S_a is calculated as defined in Clause 2.2.2.2.2 and you are not able to override it. **NOTE** In theory the topography could be significant for some directions and not for others.
Season factor	You need to enter the season factor (default 1.0).
Probability factor	You need to enter the probability factor (default 1.0).
Buttons	
Next	Click **Next** to move to the Results page (BS6399-2 Wind Wizard) (page 65).

Results page (BS6399-2 Wind Wizard)

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

This is the start of your topic.

BREVe Standard Method

Initially this method creates 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North). Except for the Hybrid Method with Other Data, you can update the wind directions either by using the 'Dir.' buttons or by changing the direction value as required.

Separately, for each relevant parameter of the Standard Method, BREVe determines the worst case over all its 30° sectors. If the orientation of the building is not known, then S_d is taken as 1.0 for all directions. Otherwise we determine the worst case S_d for each direction. You cannot override the system value in either case.

The worst case S_d is based on splitting the difference to the next direction, with a minimum of ±15° and maximum of ±45°. Within these ranges S_d is interpolated.

For each reference height in the model, we then calculate the site wind speed (V_s using equation 8) and thus the effective wind speed (V_e using equation 12) and the dynamic pressure (q_s using equation 1) for each direction. When calculating actual loads on walls and roofs, we use the q_s value for the relevant reference height, but the Results page only shows the maximum values for each direction.

The Vortex view shows the effective wind speed calculated for each reference height for each 30° sector. Since a single worst case value is used for each parameter, the speeds for different sectors only differ due to S_d provided that the orientation of the building is known.

BREVe Hybrid Method

In this case, BREVe uses the directional method to determine the parameters required to calculate V_s using equation 8, for each height in the building at 30° intervals, (0° to 330°) taking the diagonal dimension `a' as the default 5.0m. (The size effect factor is applied when determining individual loads). We then use equation 27 to determine V_e and equation 16 for q_s.

For each required wind direction the worst case V_e is used for each height, based on splitting the difference to the next direction, with a maximum of ±45 degrees. Within these ranges V_e is not interpolated.

Theoretically, it is possible for a quadrant to use different 30° directions for each height, so the critical wind direction is not displayed in the summary.

The Vortex view shows the effective wind speed calculated for each reference height for each 30° sector.

Other Standard Method

The calculation of V_e and q_s are very similar to the BREVe Standard Method, (see above), except that the worst case data has been entered by you, and this page allows you to enter your own values for S_d.

As there is no data for each 30° sector, the Vortex view only shows the effective wind speed calculated for each reference height for each direction.

Other Hybrid Method

The calculation of V_e and q_s are be very similar to the BREVe Hybrid Method, (see above), except that the data has been entered by you for each direction only so a direct calculation can be performed instead of taking the worst case over a range of sectors. Also this page allows you to enter your own values for S_d.

As there is no data for each 30° sector, the Vortex view only shows the effective wind speed calculated for each reference height for each direction.

Finishing the Wind Wizard

When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no Limitations (page 56) have been encountered.

BS6399-2 Wind Zones

At the end of the **Wind Wizard...**, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.

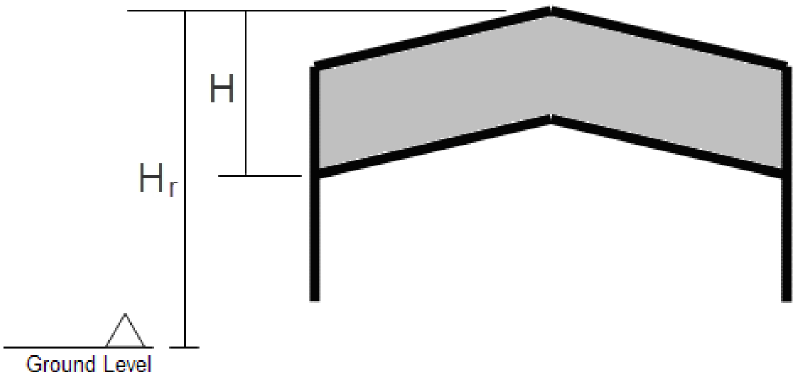
Basic Geometry

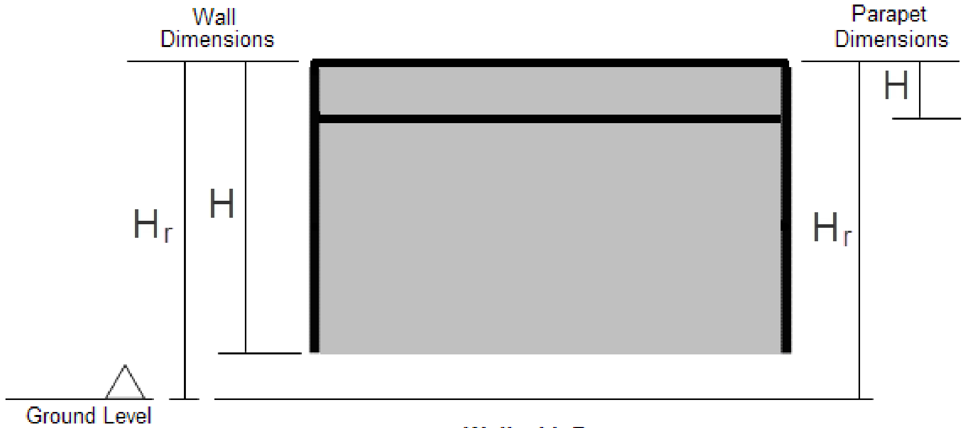
The basic building geometry is assessed as follows:

- Reference Height (H_r) - is taken as the difference between highest point on wall or roof panel and ground level.
- Wall height (H) - is taken as the difference between highest and lowest points on the wall panel.

  These definitions apply to wall panels without parapets and the actual parapets. Wall panels with parapets above them will take their highest point from the parapet. See the diagram below.

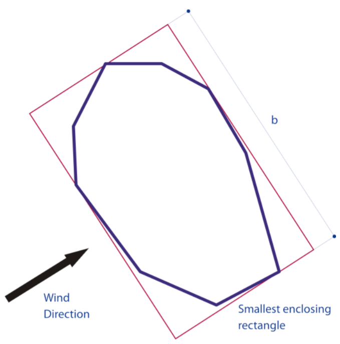
Wall above Roof





Wall with Parapet

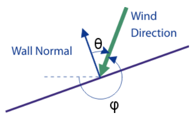
- Roof height (H) - is taken as the difference between highest point on wall or roof panel and ground level. This definition does not handle the upper roof of inset storey but is conservative and only affects the scaling dimension, b - see Clause 2.5.1.7.
- The Building Breadth, B is calculated from the smallest enclosing rectangle around the whole building (considered over all roof and wall panels only) for the given direction. You can override the calculated value in case the Tekla Structural Designer model does not include the whole building.



Wall Zones

Wall Type

We assess each wall panel to determine if it is a windward, leeward or side wall. We classify the type of wall dependent on q:



- θ ≤ 60 deg - Windward,
- θ ≥ 120 deg - Leeward,
- Other walls are classed as Side.

Windward walls	Windward walls have a single zone and Table 5 is used with interpolation for D/H.
Leeward walls	Leeward walls have a single zone and Table 5 is used.
Side walls	Side walls are assessed for recesses (narrow or wide), irregular flushed faces, downwind re-entrant corners. In all cases, side walls have the relevant number of zones. Table 5 is used.

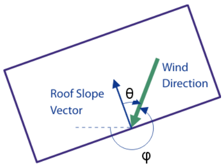
Roof Zones

Roof zones are automatically generated where possible for each wind direction. In essence each roof panel is assessed in its own right based on its properties. The interconnectivity of touching roof panels is not generally considered.

Direction

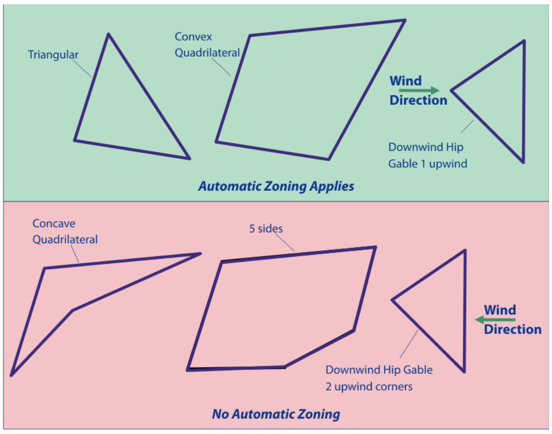
Internally the roof slope vector (line of maximum slope) is determined from the normal vector, with its direction always giving a positive slope angle, i.e. the roof slope vector must always point up the slope.

We calculate the angle between the wind direction and projection of roof slope vector onto horizontal plane (q in range -180° to +180°).



Automatic Zoning

Automatic zoning normally only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°. However, additionally, it only applies to Hip Gable roofs if they are triangular, and Hip Main roofs if they are quadrilateral. Further, Downwind Slope Hip Gables must not have 2 upwind corners.



Dimensions	All zone dimensions are specified in plan.
Flat Roofs	See BS 6399 Clause 2.5.1, Figure 16 and Table 8.
Monopitch Roofs	See BS 6399 Clause 2.5.2.3, Figure 19 and Table 9.
Duopitch Roofs	See BS 6399 Clause 2.5.2.4, Figure 20 and Table 10.
Hip Gable	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Hip Main	See BS 6399 Clause 2.5.3, Figure 21 and Table 11.
Mansard Roofs	If you manually set the connected roof types to Mansard, then the program will correctly identify the special cases in BS6399 Figures 17c, 22a and 22b, and use the correct tables and values. See BS 6399 Clauses 2.5.1.6.2 & 2.5.4
Multi-bay Roofs	We allow you to interpret BS 6399 Clause 2.5.5 and Figure 23 as you think appropriate and manually define the roof types and sub-types accordingly. You also have the ability to manually set the multi-bay position for each roof panel for each wind direction: • Not Multi-Bay - for this wind direction (conservative default), • Upwind Bay - first bay of many for this wind direction, • Second Bay - for this wind direction, • Third or more Bay - for this wind direction. Where the reduction applies, the values of all coefficients are reduced according to Table 12.

Non-Automatic Zoning

Where automatic zoning does not apply, the system creates a single zone covering the entire roof as follows:

- Flat - B,
- Monopitch - B,
- Duopitch - B for upwind, A for downwind, B for side,
- Hip Gable - B for upwind, B for downwind, D for side,
- Hip Main - B for upwind, A for downwind, D for side.

IS 875 (Part 3) Wind Wizard

This topic discusses the wind wizard when using the IS 875 (Part 3) code.

To access this configuration of the **Wind Wizard...** the Wind Loading Code has to be set to IS 875 (Part 3).

Once the wall and roof panels are in place, you use the **Wind Wizard...** to define sufficient site information to calculate the effective wind speeds and dynamic pressures for the required wind directions and heights around the building.

The wizard permits you to create wind load for any wind direction, it is up to you to create those loads for the directions most appropriate to your structure.

Unless explicitly stated all calculations in the IS 875 (Part 3) Wind Modeller are in accordance with the relevant sections of IS:875 (Part 3) - 1987 Second Revision, Reaffirmed 2003 and including Amendments 1, 2 and 3.

Wind Options page

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

Property/Buttons	Description
Property	
Data Source	The wind velocity calculations are automated, the data source for the calculations is either: • Input directly for the worst case, • Input directly for each direction.
Consider Topography	When this box is checked, you will be asked to enter your own topographic data later in the Wizard.
Consider Cyclonic Effect	Choose this option to have Cyclonic Effect considered.
Clad frame - correlated pressure and suction	Choose this option for correlated pressure and suction in clad frames.

Property/Buttons	Description
Apply Open Structure Wind Load	With this box checked, additional wind forces are applied to those members, ancillaries and equipment that have the **Apply Open Structure Wind Load** property selected in their properties. **NOTE** This option is only displayed if at least one entity has been selected to have open structure wind load applied. For more information, see: The open structure wind method (page 89)
Buttons	
Next	Click Next to go to the Basic Data page.

Basic Data page

Property/Buttons	Description
Property	
Ground Level in model	If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example you have used a site datum rather than a building datum, then this field allows you to set the appropriate value so that the reference heights for the wind can be calculated correctly. This field also gives the lowest level to which wind load is applied in the structure.
Orientation	IS 875 defines the zones assuming the longest principal axis of the building is in the Tekla Structural Designer global Y direction. Due to the potential complexity of the model, Tekla Structural Designer will not attempt to determine the correct angle for the axes, (i.e. default is zero) but you are able to enter a more appropriate value. **NOTE** The convention for Wind Directions is: wind along global +X is 0 degrees; along global +Y is 90 ; along global -X is 180 ; along global -Y is 270.
Building Height Exposed to Wind	Calculated from difference between Ground Level and max level of roof and wind wall panels in the model, with 'override calculated' option to permit user editing.
Building Width and Building Length	The smallest bounding rectangle is determined (considered over all roof and walls only), relative to the given orientation. The building length defaults to the

<table>
<tr><th>Property/Buttons</th><th>Description</th></tr>
<tr><td></td><td>maximum and the width to the minimum of the two dimensions. The 'override calculated' options permit user editing.</td></tr>
<tr><td>Building Type</td><td>The following options are available:
Temporary
Low hazard to life
All general buildings
Important buildings</td></tr>
<tr><td>Building Class</td><td>This is determined as follows:
A if max(Building Height Exposed to Wind, Building Width and Building Length) < 20m
B if max(Building Height Exposed to Wind, Building Width and Building Length) 50m
C if max(Building Height Exposed to Wind, Building Width and Building Length) > 50m</td></tr>
<tr><td>Basic Wind Speed</td><td>You need to enter the basic wind speed at the site.</td></tr>
<tr><td>Probability factor</td><td>The value of the probability factor is determined from the following table (Table 1):
<table>
<tr><td></td><td colspan="5">Basic Wind Speed</td><td></td></tr>
<tr><td>Building Type</td><td>33</td><td>39</td><td>44</td><td>47</td><td>50</td><td>55</td></tr>
<tr><td>Temporary</td><td>0.82</td><td>0.76</td><td>0.73</td><td>0.71</td><td>0.70</td><td>0.67</td></tr>
<tr><td>Low hazard</td><td>0.94</td><td>0.92</td><td>0.91</td><td>0.90</td><td>0.90</td><td>0.89</td></tr>
<tr><td>All general</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td><td>1.00</td></tr>
<tr><td>Important</td><td>1.05</td><td>1.06</td><td>1.07</td><td>1.07</td><td>1.08</td><td>1.08</td></tr>
</table></td></tr>
<tr><td>Buttons</td><td></td></tr>
<tr><td>Next</td><td>Click Next to go to the Terrain page.</td></tr>
</table>

Terrain page

The next page of the Wizard allows you to enter the terrain data.

Property/Buttons	Description
Property	
Terrain Category	• Category 1 = Exposed open terrain few obstructions (Default) • Category 2 = Open terrain scattered obstructions • Category 3 = Terrain with numerous closely spaced obstructions • Category 4 = Terrain with numerous large high closely spaced obstructions
Fetch a distance	The velocity profile for a given terrain category does not develop to full height immediately and in this release Tekla Structural Designer only deals with a single terrain profile so you are warned if the velocity profile has not developed over the fill height of the building for the defined fetch.
Buttons	
Next	Click Next to go to the Topography page.

Topography page

If Consider Topography was checked, then the next page of the Wizard allows you to enter the topography data.

Property/Buttons	Description
Property	• None - no feature (default) • 2D Escarp - Cliffs and Escarpments • 3D Hill - Hills and Ridges
Feature Height, Z	Effective height of the feature.
Site dist from crest, X	Distance upwind or downwind from the crest to the building site (-ve = upwind).
Site height above mean ground level, H	Height above mean ground level.
Upwind Slope Length, L	Actual length of the upwind slope in the wind direction.
Downwind Slope Length	Actual length of the downwind slope in the wind direction.
Buttons	

Property/Buttons	Description
Next	Click Next to go to the Results page.

Results page

The final page of the Wizard is a summary of the results.

The final page of the Wizard is a summary of the results.

$V_{z\ max} = V_b \times k_1 \times k_2 \times k_3$ (m/s)

p_z max = 0.6 x V_z max2 (N/m^2)

The above is for the highest point in the building only.

Property/Buttons	Description
Buttons	
Finish	When you click Finish, the **Wind Wizard...** generates the wind zones for the entire building for each of the specified wind directions. Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no issues have been encountered.

Wind model loadcases

Once the wind wizard has been run, the next step is the creation of appropriate wind loadcases. This is achieved using the **Wind Loadcases** dialog which is accessed by clicking **Wind Loadcases** on the **Load** toolbar.

NOTE The **Wind Loadcases** dialog is only available after the wind wizard has been run.

The **Auto** button within the **Wind Loadcases** dialog automatically generates wind loadcases.

NOTE The **Auto** button is greyed out and hence unavailable, if wind loadcases have already been defined. These would need to be deleted to use the **Auto** button again.

Alternatively, you can create loadcases manually using the **Add** button and proceeding as follows:

- specify which direction the loadcase will be created
- set default values for all the zone loads generated in the loadcase
- the loadcase is generated automatically, (but can be overwritten if required)

Try to use engineering judgment to identify the critical loadcases for design. This ensures that the number of load combinations to be considered are minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Once generated, the wind loadcases are standard loadcases which can be included in combinations in the normal manner.

Creating loadcases for the Low Rise Buildings method

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the **Load** tab).

The **Auto** button on the dialog provides various options to control the number of loadcases before they are generated (its intention being to prevent generation of very large numbers of loadcases). Alternatively you can create loadcases manually using the **Add** button.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard loadcases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases for design. This ensures that the number of load combinations to be considered are minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Adding wind loadcases

By clicking the **Add** button you are able to generate loadcases one at a time.

- The loadcase name is auto generated from the other input parameters, but it can be edited if required.
- The direction the loadcase will be created for is selected from the droplist.
- The internal pressure coefficient is entered directly.

  NOTE Depending on the wind code you have selected, the default is the negative value depending on the Enclosure Classification from:

  - AISC 7-16: Table 26.13-1, p271
  - AISC 7-10: Figure 26.11-1, p201
  - AISC 7-05: Figure 6-5, p47

- You are able to flag if it is a Torsional Loadcase or not: when flagged, reduced pressure loads are applied to parts of the standard zones (by applying a net pressure factor of 25%).

- You are able to flag if roof loads are to be generated for the loadcase or not.

Auto generating wind loadcases

By clicking the **Auto** button you are able to control the total number of loadcases generated.

Up to 4 loadcases can be generated for each direction, using positive and negative GCpi derived from the Enclosure Classification, (see Figure 26.11-1, p201). If a direction is included, but neither +ve GCpi or -ve GCpi have been checked, then that is interpreted as a loadcase with GCpi of zero. You are also given the option to include the torsional loadcases or not, (default not), but it is assumed that Roof loads are required.

By clicking **OK** the loadcases are generated, but you can then make any changes required before clicking **OK** once more to close the Wind Loadcases dialog.

Creating loadcases for the All Heights method

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the **Load** tab).

The **Auto** button on the dialog provides various options to control the number of loadcases before they are generated (its intention being to prevent generation of very large numbers of loadcases). Alternatively you can create loadcases manually using the **Add** button.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard loadcases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized.

Adding wind loadcases

By clicking the **Add** button you are able to generate loadcases one at a time.

- The loadcase name is auto generated from the other input parameters, but it can be edited if required.
- The direction the loadcase will be created for is selected from the droplist.
- The internal pressure coefficient is entered directly.

NOTE Depending on the wind code you have selected, the default is the negative value depending on the Enclosure Classification from:

- AISC 7-16: Table 26.13-1, p271

- AISC 7-10: Figure 26.11-1, p201
- AISC 7-05: Figure 6-5, p47

- You are able to flag if it is a Torsional Loadcase or not: when flagged, reduced pressure loads are applied to parts of the standard zones (by applying a net pressure factor of 25%).
- For torsional loadcases, you are able to flag if positive or negative eccentricity is applied.
- You are able to flag if loads are to be created for roof zones or not.

NOTE This option is provided to allow for the following note, (depending on the wind code you have selected):

- AISC 7-16: Figure 27.3-1, Note 7, p276
- AISC 7-10: Figure 27.4-1, Note 9, p207
- AISC 7-05: Figure 6-6, Note 9, p49

- If loads are created for roof zones, you are able to flag if positive or negative pressure coefficients are to be used for roof zones.

Auto generating wind loadcases

By clicking the **Auto** button you are able to control the total number of loadcases generated.

Up to 8 loadcases can be generated for each direction. You may choose positive and/or negative GCpi derived from the Enclosure Classification, (see Figure 26.11-1, p201). If you include a direction, but have not checked either +ve GCpi or -ve GCpi, then that is interpreted as a loadcase with GCpi of zero. Similarly you may choose positive and/or negative Cp values for roof zones - checking neither indicates Roof Loads to be ignored for this direction. Torsional loadcases will not be included by default, but can be added with positive or negative eccentricity.

By clicking **OK** the loadcases are generated, but you can then make any changes required before clicking **OK** once more to close the Wind Loadcases dialog.

Creating loadcases EC1991 1-4

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the **Wind Loadcases** dialog box (accessed from the Load toolbar).

The **Auto** button on the dialog creates a default set of loadcases in each of the directions, i.e.

- -0.3 for C_{pi} with -ve roof C_{pe}; not Overall

- -0.3 for C_{pi} with +ve roof C_{pe}; not Overall
- +0.2 for C_{pi} with -ve roof C_{pe}; not Overall
- +0.2 for C_{pi} with +ve roof C_{pe}; not Overall
- Overall with zero for C_{pi}; -ve roof Cpe

Alternatively you can create loadcases manually using the **Add** button. You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard loadcases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Wind loadcases dialog

As well as specifying which direction the loadcase will be created for, this dialog allows you to set default values for all the zone loads generated in the loadcase.

Item	Description
Fields	
Structural Factor - Automatically calculate seperate c_s and c_d factors	The UK NA states that the Structural Factor, $c_s c_d$ may be separated into a size factor c_s and a dynamic factor c_d, i.e. it is still acceptable to apply Clause 6.2 (1) a) to d) and set $c_s c_d$ = 1, or use Annex D. Where **Structural Factor - Automatically calculate separate c_s and c_d factors is checked**, c_d is calculated using Figure NA.9 and c_s using Table NA.3. **NOTE** The option to automatically calculate c_c and c_d is not available for other National Annexes.
Structural damping	The δ_s value is used to determine the dynamic factor c_d. It is only visible if the **Separate Factors** box is checked. See Table F.2.
Name	The loadcase name is auto generated from the other input parameters, but it can be edited if required.
Direction	The direction the loadcase will be created for is selected from the droplist.

Item	Description	
Overall	You are able to flag if the loadcase is to be specifically used for examining the overall behaviour of the structure by checking this box. **NOTE** It may be necessary for you to create a second copy of the loadcase with this check box cleared if the loadcase is also used for designing elements.	
b+h	When designing elements, (beams, columns, braces etc), Table NA.3 in the UK NA implies that b and h should be the width and height respectively of an element. Due to the nature of the loads in the program, it is not practical to do this automatically, and so you should specify a value to be used in the loads generated for this loadcase (default 5.0m). If separate factors are not to be used, (i.e. use combined c_sc_d), then this value is redundant. If separate factors are to be used, but **Overall** is checked on the row, then the b+h cell is marked inactive. In this case, for each wall and roof, the program calculates b+h using b & h from the zone properties for the relevant wind direction.	
Use +ve Cpe	Where 2 sets of coefficients are given in a table for roof zones, this field indicates if the negative or positive C_{pe} value is to be used.	
Cpi	Default Internal Pressure Coefficient (-0.3, 0.0, +0.2 or other value) - to be calculated by you from Clause 7.2.9.	
Buttons		
Add	Click this button to add a single wind loadcase.	
Delete	Click this button to delete a wind loadcase.	
Auto	Click this button to create a default set of loadcases in each of the directions.	

Wind model load decomposition

Click the links below to find out about decomposition options available for panels when using the wind model method.

- Wall panel load decomposition (page 81)
- Roof panel load decomposition (page 85)
- Alternative wind load decomposition method for complex models (page 87)

- Limitations of wind decomposition to diaphragms (page 101)

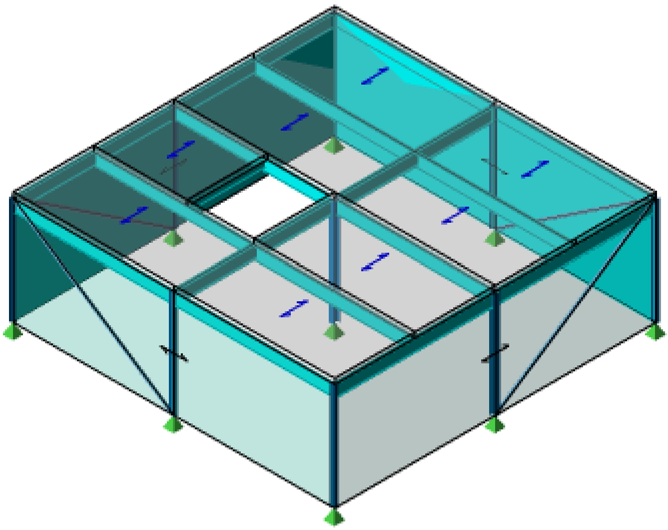
Wall panel load decomposition
This topic will discuss the decomposition that occurs when considering wind load applied to wall panels.

Decomposition options

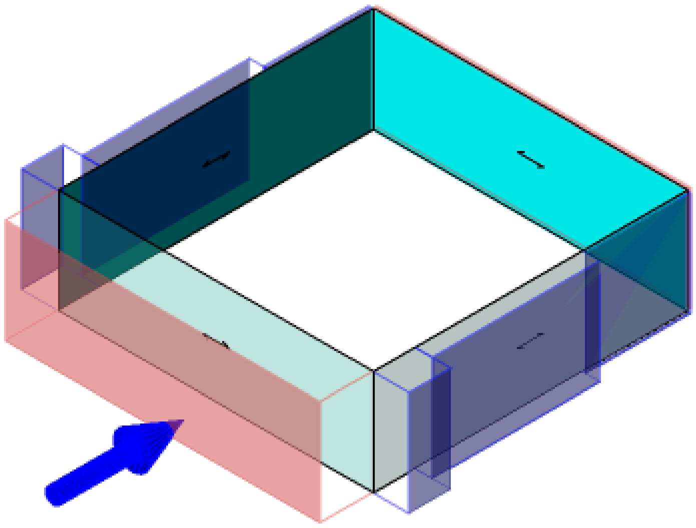
Wind zone loads are decomposed on to the structure according to the **Decompose to** wall panel property. The available options are:

- Members
- Nodes (default)
- Rigid Diaphragms

To demonstrate the effect of the different **Decompose to** values, consider a braced steel frame clad in wind wall panels as shown below:



For wind direction 0, the wind model produces Zone loads as follows:



The topics below illustrate how the zone loads shown above are decomposed using the three different **Decompose to** values.

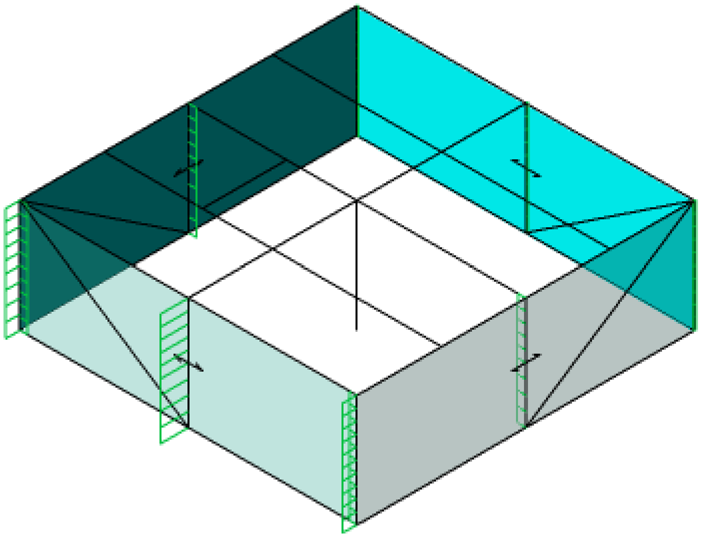
Decompose to Members

Decomposition to members is similar to the roof panel decomposition, the span direction of the wind wall panel determines the direction of the one way decomposition.

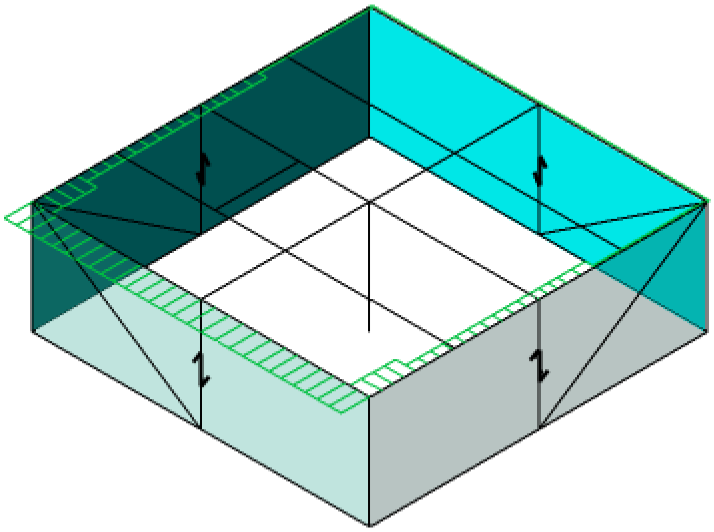
All element types within the wind wall plane are considered with the exception of bracing members.

Decomposition to members allows the generation of UDL's on portal stanchions and gable posts without the need to model side rails.

For the model shown above, choosing **Decompose to Members** produces the following load decomposition:



Decomposition to Members (rotation angle 0 degrees)



Decomposition to Members (rotation angle 90 degrees)

Decompose to Nodes

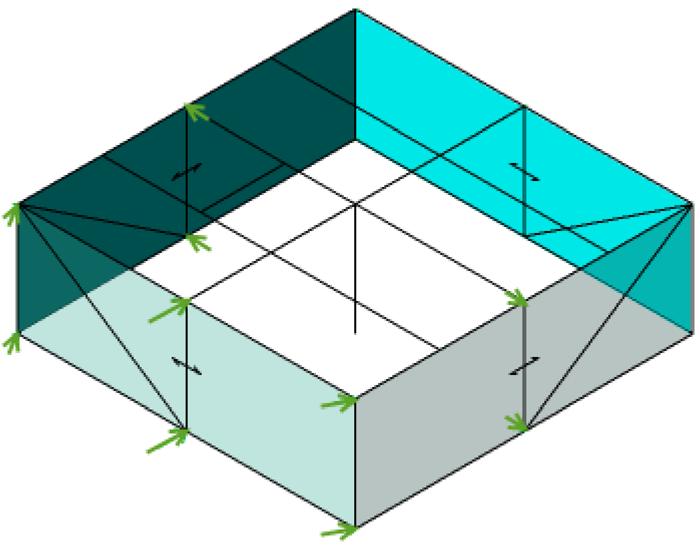
Decomposition to nodes is the **default** setting and results in nodal loads on the supporting members. This setting is typically appropriate to avoid lateral loads on simple beams.

All element types within the wind wall plane are considered with the exception of bracing members.

The initial decomposition is the same as for members, with the direction of the one way decomposition being specified by the span direction of the panel. A second decomposition stage is then undertaken to convert the member loads to nodal loads:

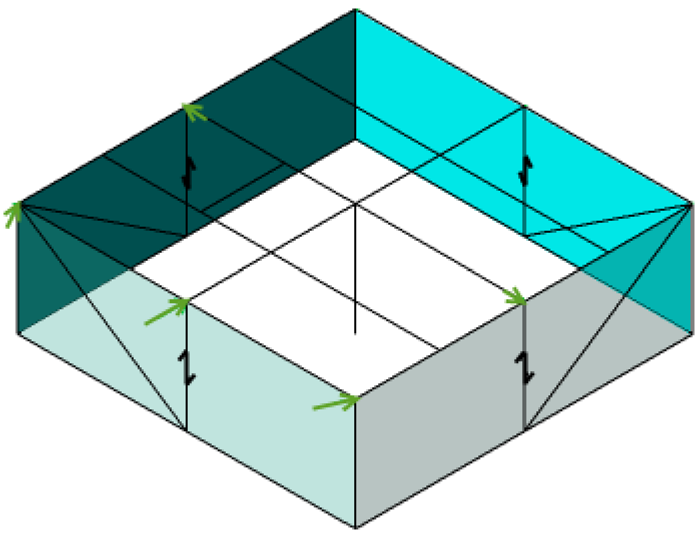
- Full/partial UDLs and VDLs on elements (lengths of beams/columns between nodes) are distributed back to nodes as if the elements were simply supported at either end.
- The final nodal load is the sum of all incoming element loads.

For the model shown above, choosing **Decompose to Nodes** produces the following load decomposition:



collapse

Decomposition to Nodes (rotation angle 0 degrees)



Decomposition to Nodes (rotation angle 90 degrees)

NOTE In the example above, when the rotation angle is 0 degrees, some of the nodal loads are applied directly to supports.

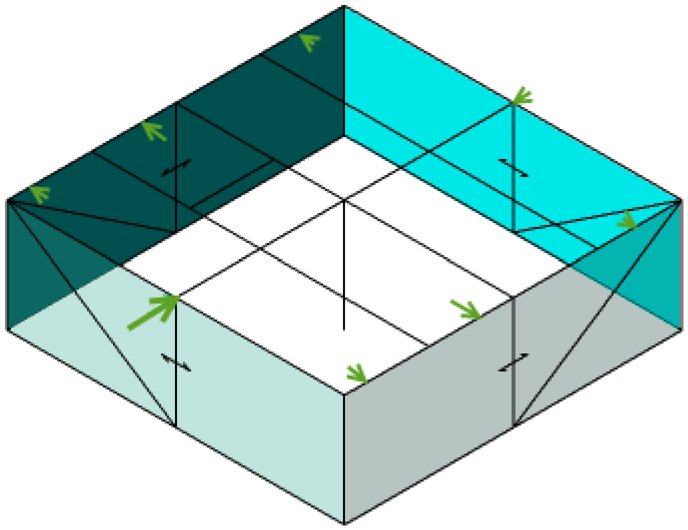
Decompose to Rigid Diaphragms

Decomposition to rigid diaphragms does not consider the span direction of the wall panel (so the rotation angle is irrelevant).

It is particularly useful for flat-slab structures, as the alternative **Decompose to Member** or **Node** decomposition methods require supporting members that may not exist in the model.

NOTE All rigid diaphragms within the wind wall height are considered for decomposition irrespective of whether they are physically connected to the wind wall.

For the model shown above, choosing **Decompose to Rigid Diaphagm** produces the following load decomposition:



Decomposition to Diaphragms - each Zone load is decomposed as a separate point load on the diaphragm

Validation of Panels set to Rigid Diaphragm Decomposition

The following validation checks are performed on wind wall panels set to **Decompose to Rigid Diaphragms**:

- Each panel must be rectangular
- The top level of every wall panel must align with a rigid diaphragm
- Each panel may be sub-divided into zones, but only by horizontal lines
- Unlike for **Decompose to Member** or **Node**, each wall panel does not need to have supporting members along its edges.

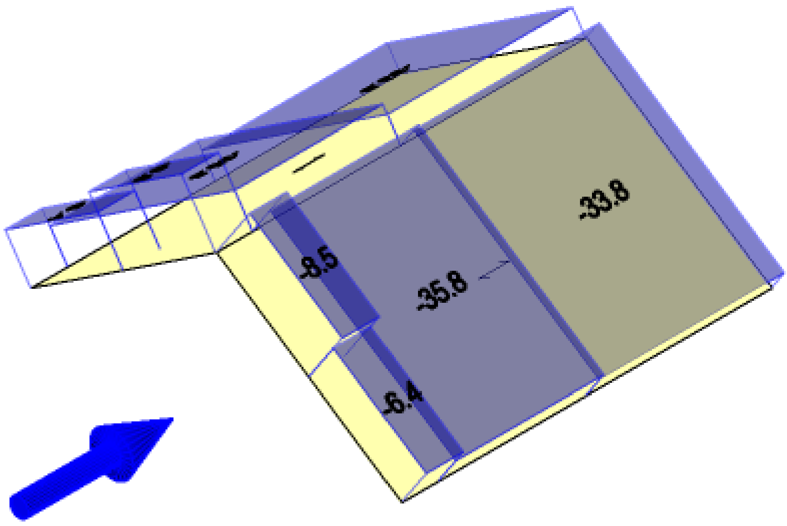
Roof panel load decomposition

This topic will discuss the decomposition that occurs when considering wind load applied to roof panels.

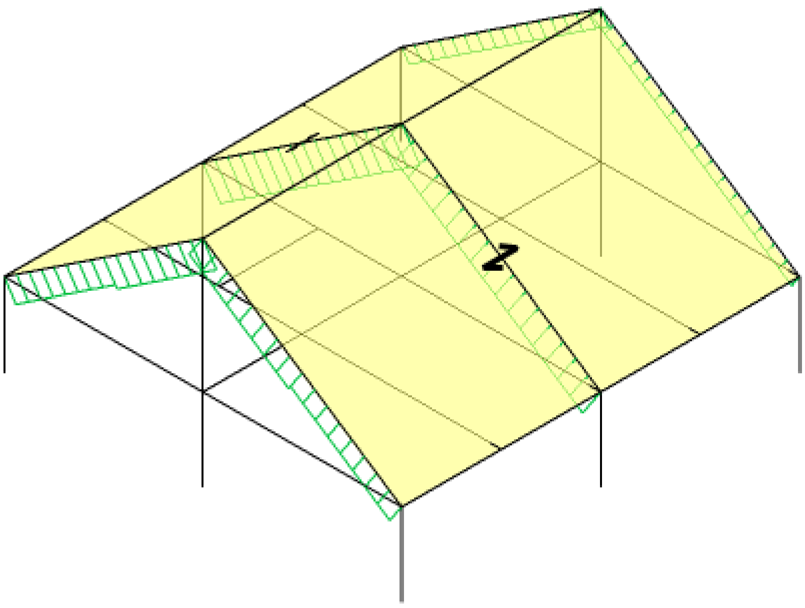
Roof panel load decomposition

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

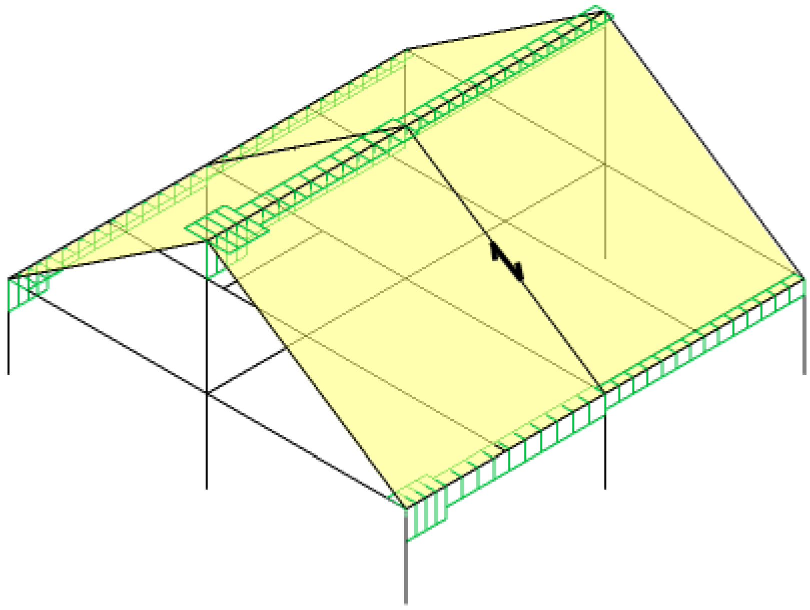
All types of elements within the roof panel plane (except bracing) are considered during the load decomposition.



Zone loads on roof.



Roof panel decomposition (rotation angle 0 degrees)



Roof panel decomposition (rotation angle 90 degrees)

Alternative wind load decomposition method for complex models

For complex models the placement of wind walls required by the Wind model method can be a time consuming operation.

In such a situation (provided the model has suitable rigid diaphragms), the following alternative offers a quicker and simpler way to apply approximate wind loads:

- Use engineering judgment to clothe the structure with an arrangement of simplified wind wall panels around its bounding box,
- set the wind walls to decompose to rigid diaphragms,
- proceed with the Wind model method

Simplified wind wall panels example

In theory, because wind walls set to **Decompose to Rigid Diaphragms** don't have to physically connect to the diaphragms, even a very complex model could be clothed with just four simplified wind walls defined along its bounding box, as illustrated below:

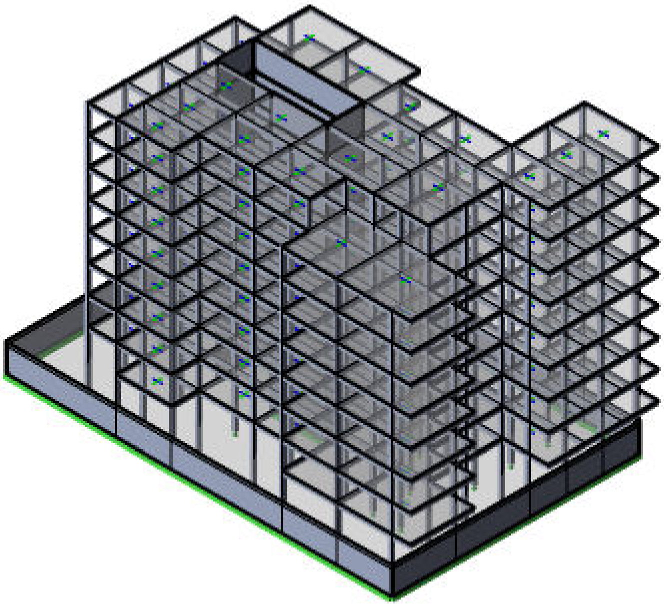


Figure1. Complex model prior to placement of wind walls

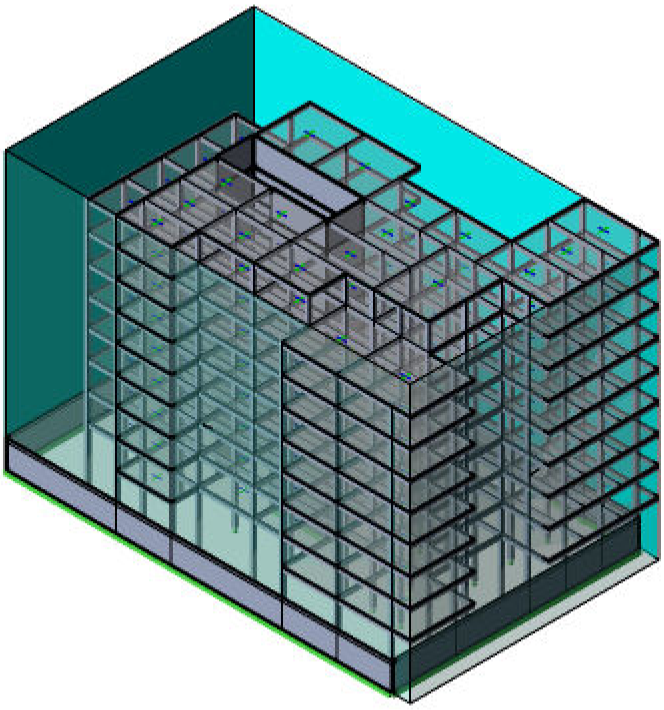
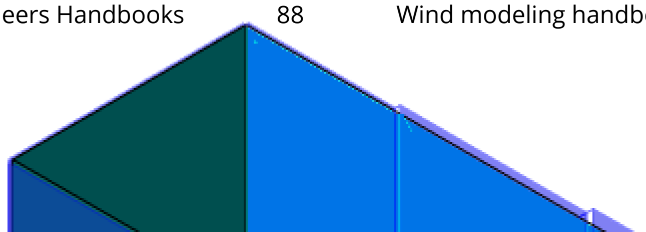


Figure2. Simplified wind walls placed around the "bounding box"



loads (based on the rectangular building envelope) which could then be refined at a later stage if necessary.

NOTE If this method is adopted you are strongly advised to review the wind zones that are formed, and the resulting decomposed loads to ensure they meet your expectations - it will not give good results for all models.

If the decomposed loads produced by the above approach are not satisfactory, you might decide to take greater control by applying the loads manually.

References

1. **ASCE/SEI 7-10**. Minimum Design Loads for Buildings and Other Structures. **ASCE, 2010. ISBN: 978-0-7844-1085-1**.
2. **Kishor C. Mehta and James M. Delahay (2004)**. Guide to the Use of the Wind Load Provisions of **ASCE 7-02. ASCE Press. ISBN: 0-7844-0703-7. British Standards Institution (25/04/05)**. Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions. BS EN 1991-1-4:2005.
3. **British Standards Institution (September 2008)**. UK National Annex to Eurocode 1: Actions on structures. NA to BS EN 1991-1-4:2005.
4. **British Standards Institution (July 2002)**. Loading for Buildings - Part 2: Code of practice for wind loads. BS6399-2:1997.
5. **British Standards Institution**. Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009.
6. **Cook, N.J**. Designers' Guide to EN 1991-1-4. Euro Code 1 : Actions on Structures, General Actions Part 1-4 : Wind actions. **Thomas Telford, London.ISBN 978-0-7277-3152-4**.
7. **Cook, N.J. (1999)**. Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. **Thomas Telford, London. ISBN: 0 7277 2755 9**.
8. **Bailey, C.G. (2003)**. Guide to Evaluating Design Wind Loads to BS6399-2:1997. **SCI Publication P286**.
9. BREVe software package version 3. **Copyright © 2009 CSC (UK) Ltd; BRE Ltd; Ordnance Survey.**

The open structure wind method

Tekla Structural Designer provides a quick means to apply wind loads to open structures in the form of automatically calculated UDLs or point loads.

In this method, you select **Apply open structure wind load** in the properties of those entities to be loaded and then determine the load values by running the Wind Wizard with the **Apply open structure wind load** option selected.

The following entities can be loaded in this way:

- Structural members (beams, columns etc)
- Line ancillaries
- Area ancillaries
- Equipment

The above approach can be used in conjunction with the method for enclosed structures if wall and roof panels have also been defined.

NOTE The open structure wind method is not currently available for the British Standard BS 6399-2, or Australian AS:1170.2 wind code variants.

Click the links below to find out more:

- Open structure wind method assumptions and limitations (page 90)
- Wind load on open structures calculations (page 91)
- Open structure wind method workflow (page 94)

Open structure wind method assumptions and limitations

Assumptions

- All wind loads are assumed to act perpendicular to the members, equipment and ancillaries
- Applicable for all 1D structural members irrespective of their characteristic except
    - analysis elements
    - DELTABEAM
    - FABSEC beams

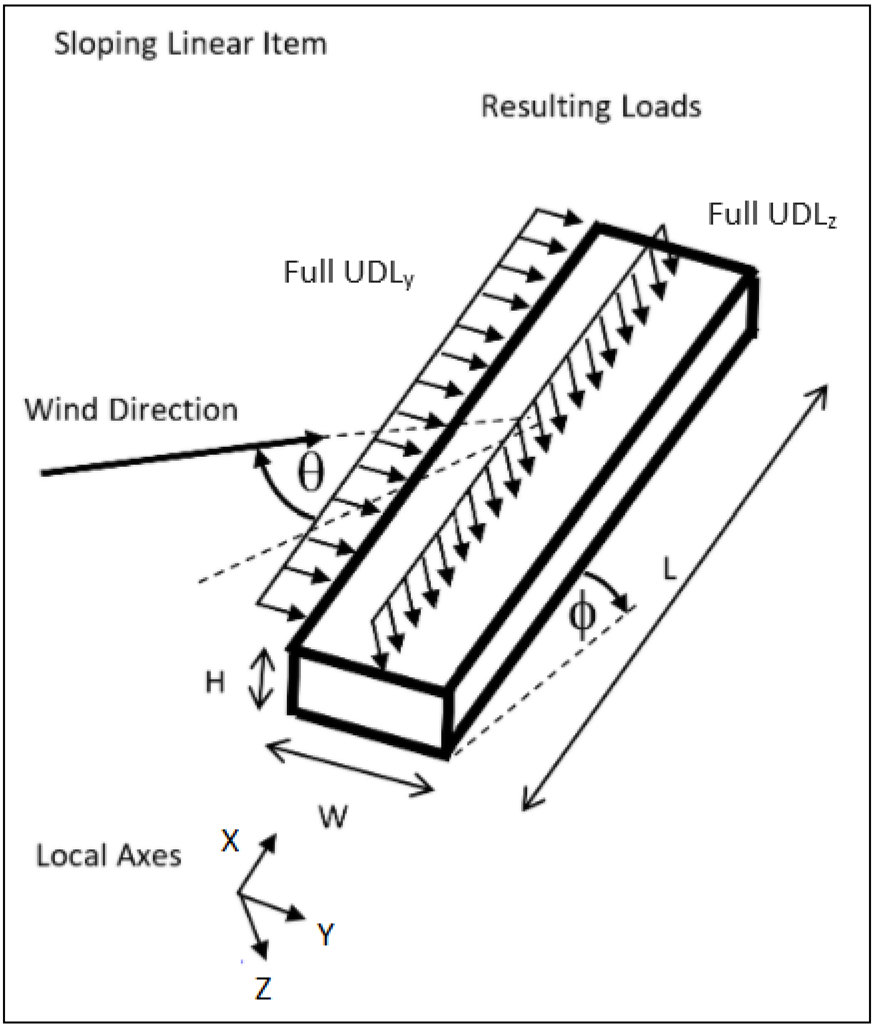
Limitations

- Only currently applicable for US, EC and IS wind codes
- Cannot be applied on structural walls
- Shielding effect is not considered
- There are no frictional wind forces accounted on any surfaces

Wind load on open structures calculations

Members and ancillaries

Wind load is determined as two full length UDLs, one acting in local y (minor) and one acting in local z (major) on the member.

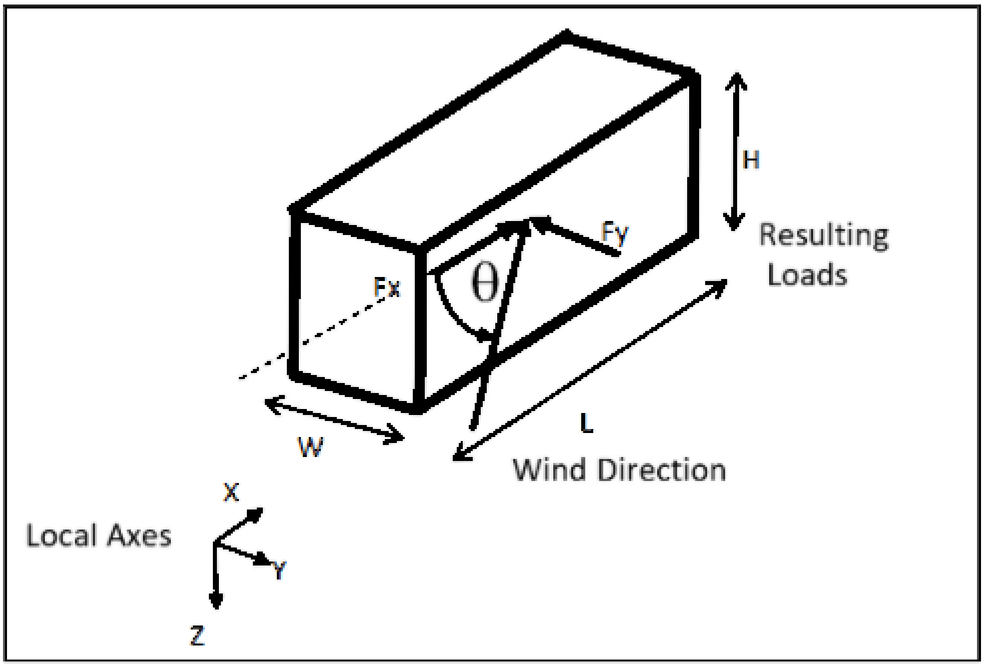


Full UDL_{Major}		z x C_f x Effective $Area_{xy}$ x Cos(θ) x Sin(φ) / L =
Full UDL_{Minor}		z x C_f x Effective $Area_{xz}$ x Sin(θ) / L =
where		

z		wind pressure calculated at the height of the member for that wind direction (for vertical member average is taken at top and bottom level)
C_f		shape factor =
Effective Area_{xy}		effective area of member in xy =
Effective Area_{xz}		effective area of member in xz =
θ		angle of the wind to the member x axis projected vertically into the horizontal plane
φ		angle of the member to the horizontal plane =
L		length of the member =

Equipment

Wind load is determined as a single point load acting on the equipment.



The point load is applied to the CoG of the equipment at mid height.

F_{Wind}	=	z x C_f x Effective $Area_{xz}$ x Sin(θ) + Effective $Area_{yz}$ x Cos(θ))
F_x	=	F_{Wind} x Cos(θ)
F_y	=	F_{Wind} x Sin(θ)
where		
z	=	wind pressure calculated at the height of the equipment for that wind direction
C_f	=	shape factor
Effective $Area_{xy}$	=	effective area of equipment in xy
Effective $Area_{xz}$	=	effective area of equipment in xz
θ	=	angle of the wind to the equipment x axis projected vertically into the horizontal plane
L	=	length of the equipment

NOTE All wind loads considered are horizontal, no vertical loading is considered.

Shape factors and effective area factors

The default shape factors (C_f) come from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE

The default shape factors and effective area factors assigned to new members/ancillaries can be edited from **Model Settings > Loading > Wind Loading**.

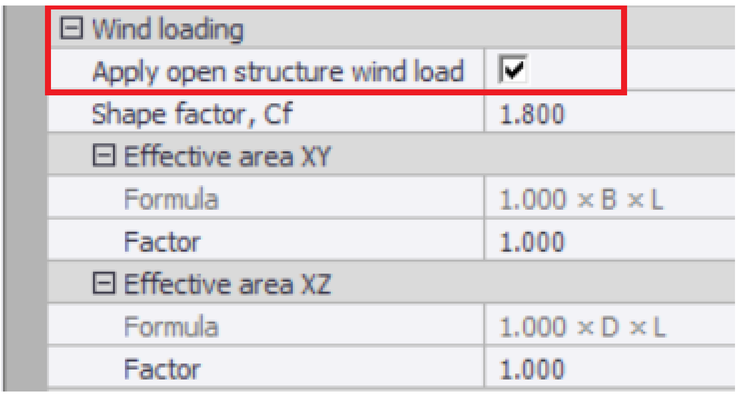
The shape factors and effective area factors for existing members and ancillaries can be edited in the properties window.

Open structure wind method workflow

The basic steps required to apply this method are as follows:

Select members to have open structure wind load applied

The first step is to confirm the **Apply open structure wind load** property is set as required for those structural members, ancillaries, and equipment to which the loads are to be applied. (By default this property is *off* for structural members, but *on* for ancillaries and equipment.)



It can be toggled *on* and *off* directly in the properties window as above, or it can be set graphically via Show/Alter State > .

For members and ancillaries, when it set *on*, additional properties become available, allowing the shape factor and effective area factors used in the wind load calculation to be customized.

For members only, each of these properties can be set for all stacks/spans, or individual stacks/spans.

Run the wind wizard

Having selected the members to have open structure wind loads applied, the next step is to run the Wind Wizard.

On the first page of the wizard you must select the **Apply Open Structure Wind Load** option.

NOTE The above option is only available if at least one entity has been selected to have open structure wind load applied.

Step-through the remaining pages of the wizard to generate wind model.

The wizard uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location.

Related topics

EC1991 1-4 Wind wizard (page 30)

ASCE 7 Wind wizard (page 15)

BS6399-2 Wind wizard (page 54)

IS 875 (Part 3) Wind Wizard (page 71)

Define the wind loadcases

The Wind Loadcases dialog can then be used to define the loadcase information for the directions you require.

NOTE If you have a totally open structure without any wall or roof panels the C_{pi} values are immaterial and can be set to 0.0.

Once the loadcases have been defined, wind loads (page 91) are automatically calculated for those entities selected to have open structure wind load.

NOTE It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

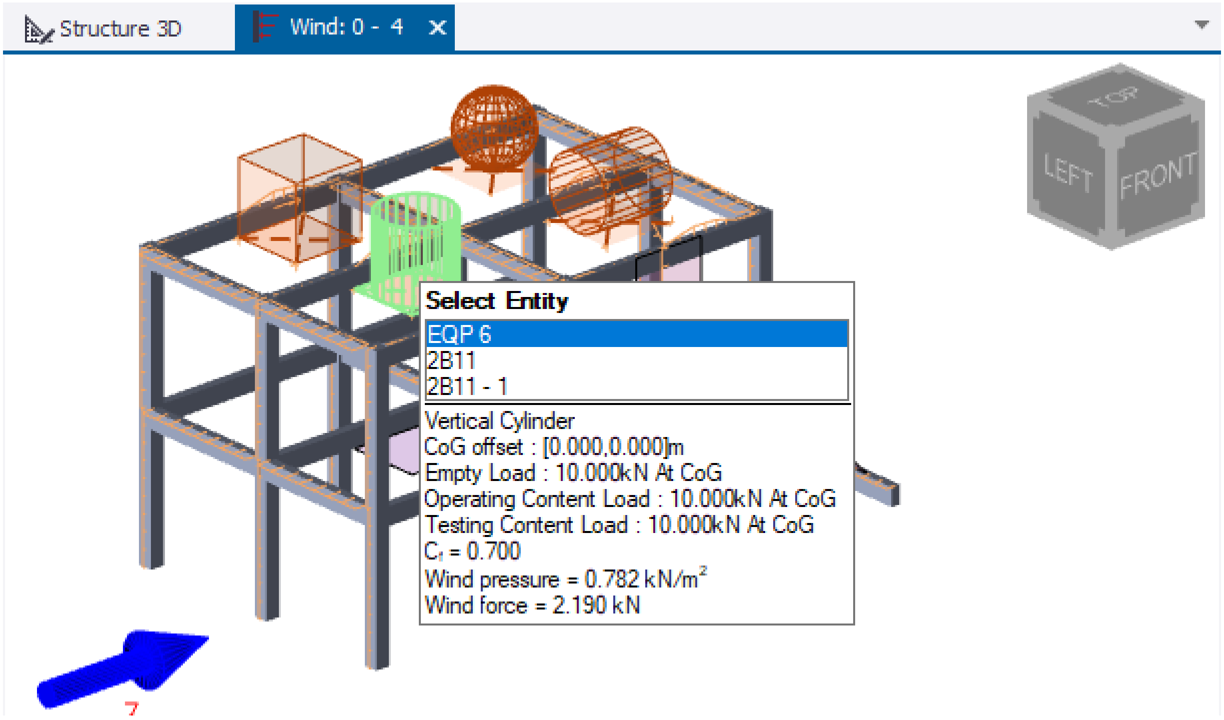
Related topics

Wind model loadcases (page 75)

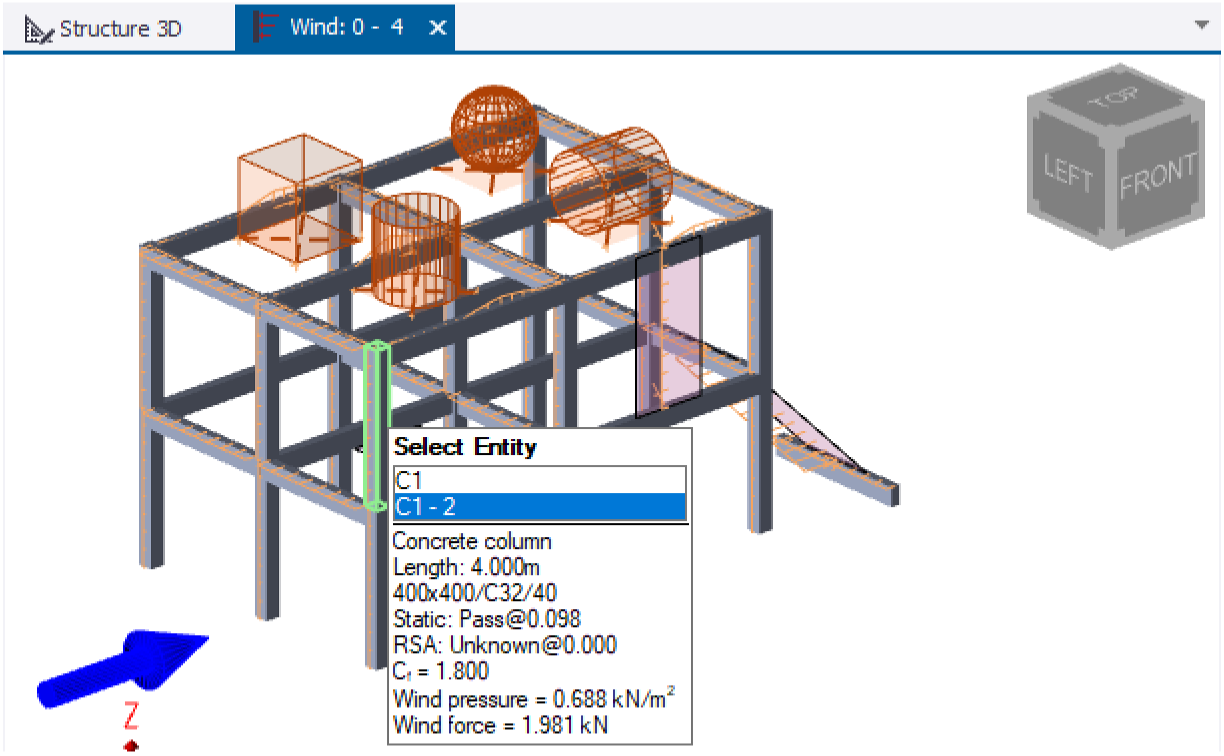
Review wind loads

The open structure wind loads on members, ancillaries and equipment can be reviewed by opening a wind view for the required direction from the Project Workspace Wind tab and selecting the required loadcase.

Hovering the cursor over an entity provides displays wind loading information in the Select Entity tooltip. The shape factor, wind pressure and wind force are all listed.



For members, the same information is displayed by hovering over the required stack/span.



Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the Design toolbar.

Create an open structure wind loads report

These loads are automatically included in the .

Manually applied wind loads and simple wind loads

This is a quick method of applying wind to open or enclosed structures without requiring you to create a wind model.

The loads are applied manually, either as panel, member, or structure loads in the usual way, or as simple wind which are then decomposed to diaphragms during the analysis.

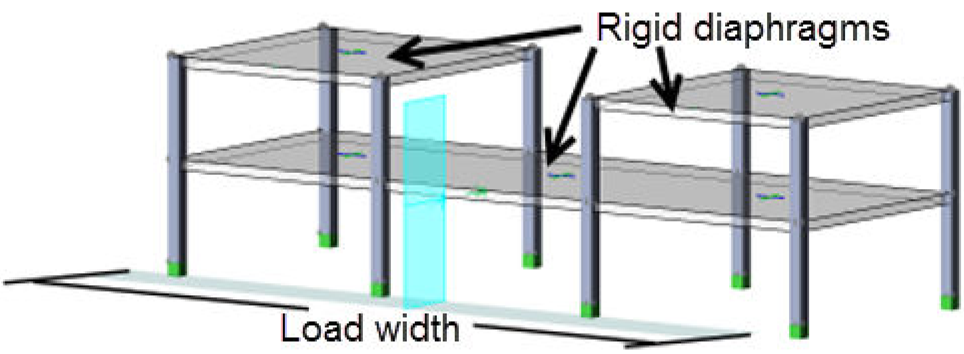
Click the links below to find out more:

-
- Limitations of wind decomposition to diaphragms (page 101)
- Simple wind example (page 97)

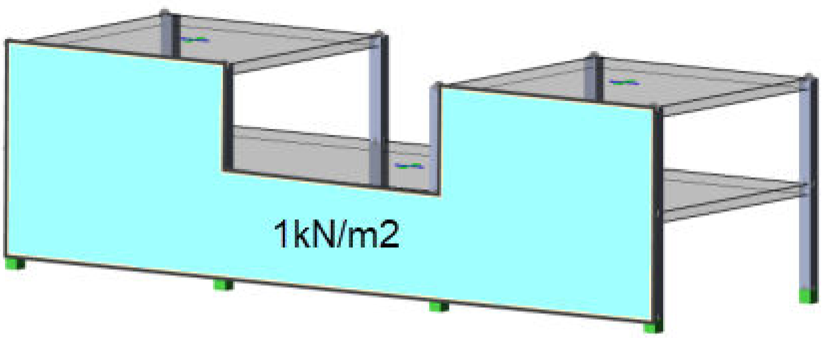
Simple wind example

Each simple wind load is defined as an area load over a defined width and height.

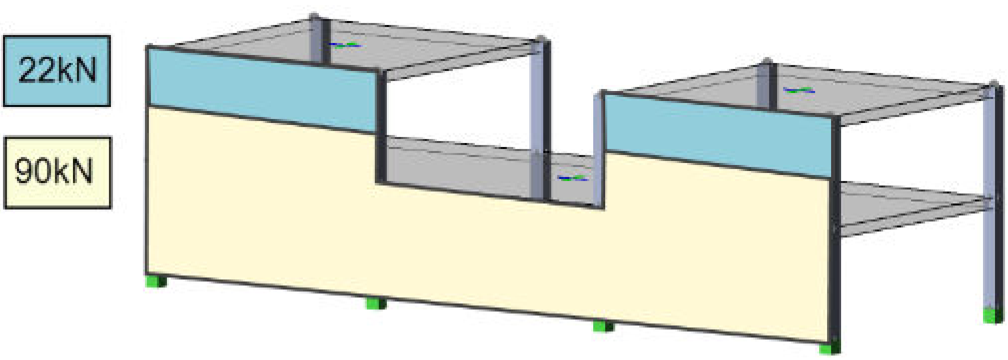
In the two story example shown below, a 1kN/m^2 load is applied over the full 22.5m width and 6m height of the building.



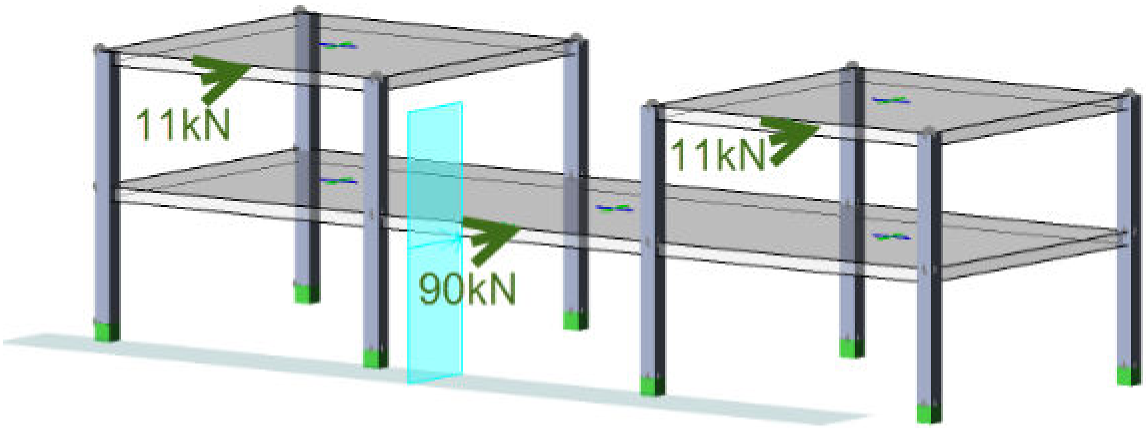
- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation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



- Load within the bounding box that hits the structure (the blue shaded area above) is decomposed to point loads on rigid diaphragms only - it does **not** get decomposed to semi-rigid diaphragms.
- All rigid diaphragms on the top or bottom level or anywhere in-between are considered, with the area load being divided between the levels before it is decomposed. In the example the area load is split between 1st and 2nd floor levels so that a total of 22kN is to be decomposed above and 90kN below.



- The load is then decomposed to the diaphragms at each level in proportion to the width of each diaphragm. Each load being applied as a nodal load in the direction of the simple wind load at the mid point of the projected load.



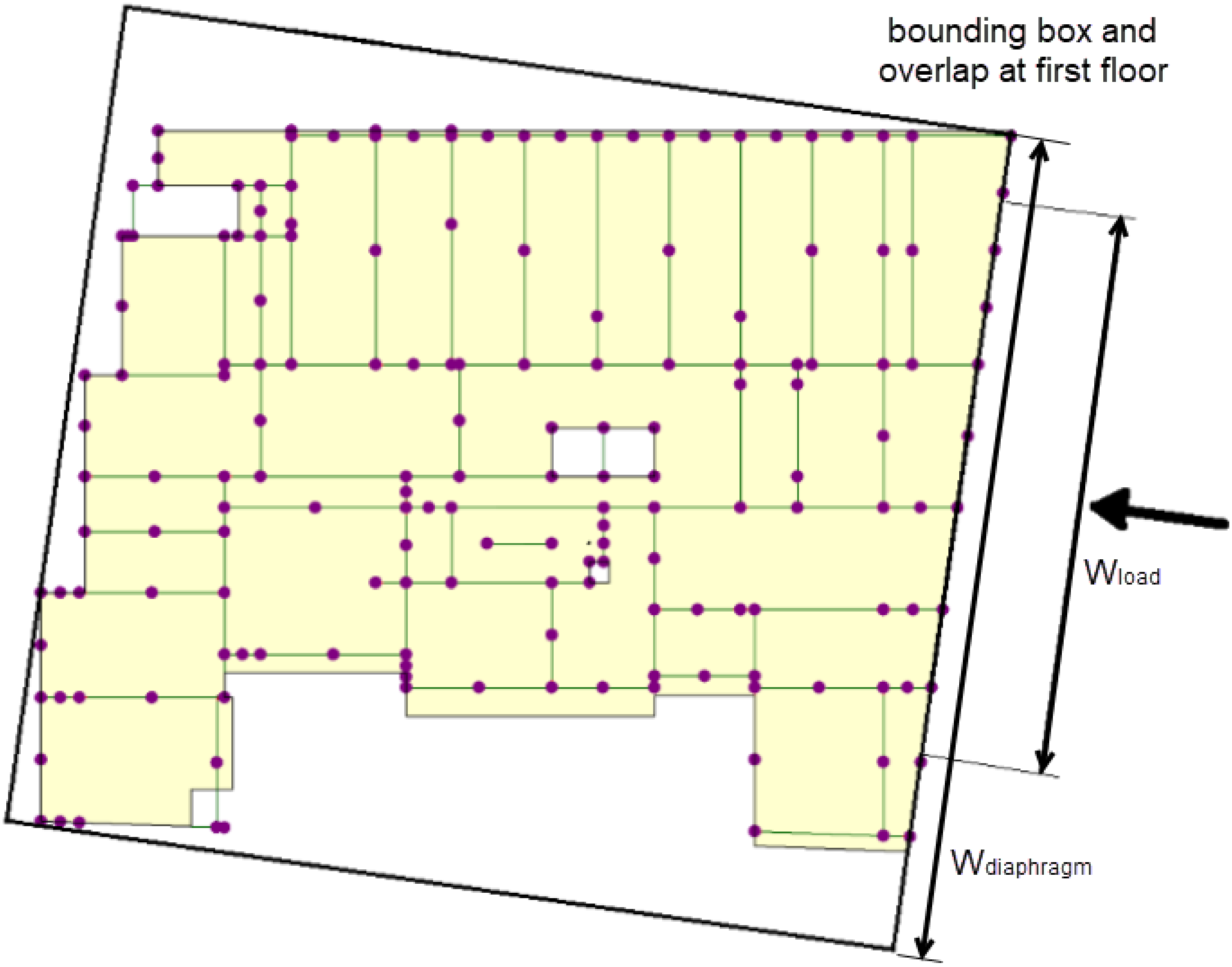
- If there are no suitable diaphragms on the top level, the load is applied at the next level down.
- Similarly if the 'Ignore diaphragms on lowest level' box is checked on the Simple Wind Loading dialog or there are no suitable diaphragms on the bottom level, the load is applied at the next level up.

NOTE If for some reason there are diaphragms at the ground level, then you may decide to check the 'Ignore diaphragms on lowest level' box in order to ensure no load is lost directly to the foundations.

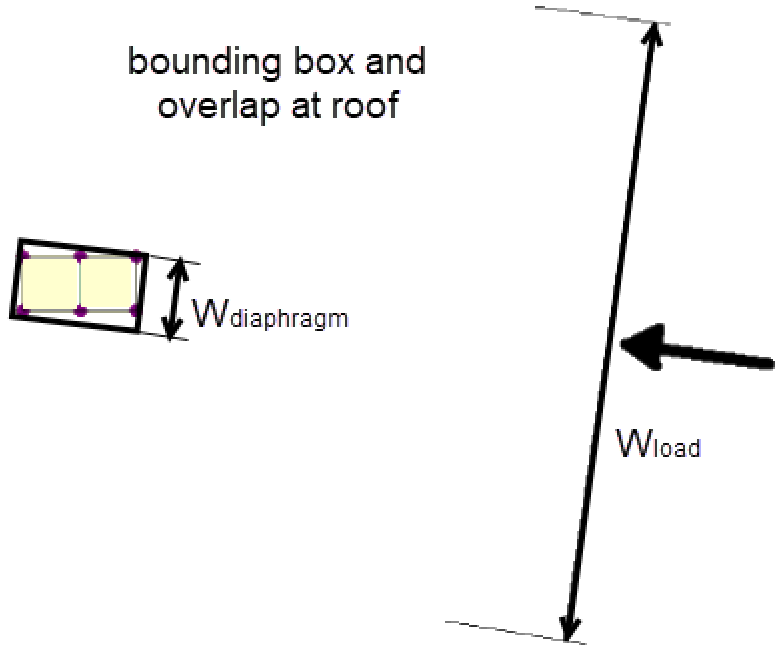
Situations where rigid diaphragms are ignored

If the projected width of a rigid diaphragm in the direction that the load is acting is less than 5% of the simple wind load width it is assumed to be ineffective.

In the below example, at first floor level the width of the diaphragm perpendicular to the load exceeds the width of the load, so the diaphragm is effective.

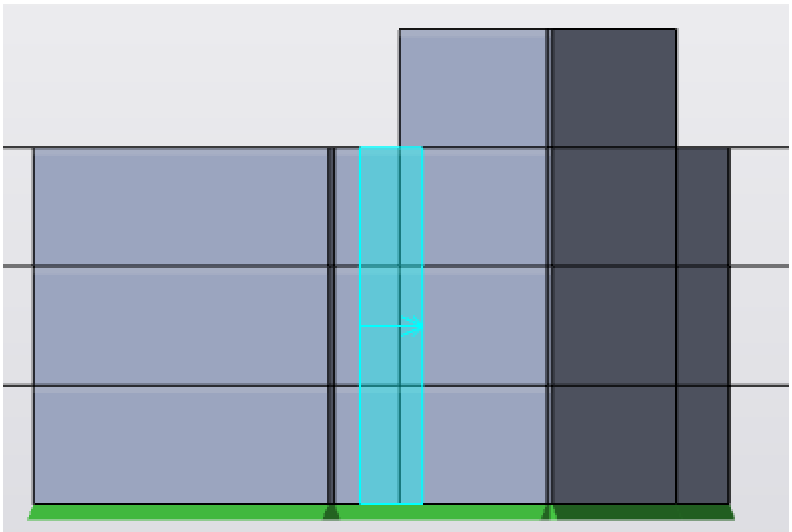


This changes at roof level, as the width of the diaphragm is now significantly smaller than the width of the load. If the diaphragm width is less than 5% of the load width, the diaphragm is ignored.



Furthermore, as this is the only diaphragm at the top level you would be prevented from applying the load, as a "Top level of Simple Wind Load must align with a rigid diaphragm" error message would be displayed.

A workaround would be to reduce the top level of the simple wind load so that it hits a diaphragm at a lower floor of sufficient width, as shown below.



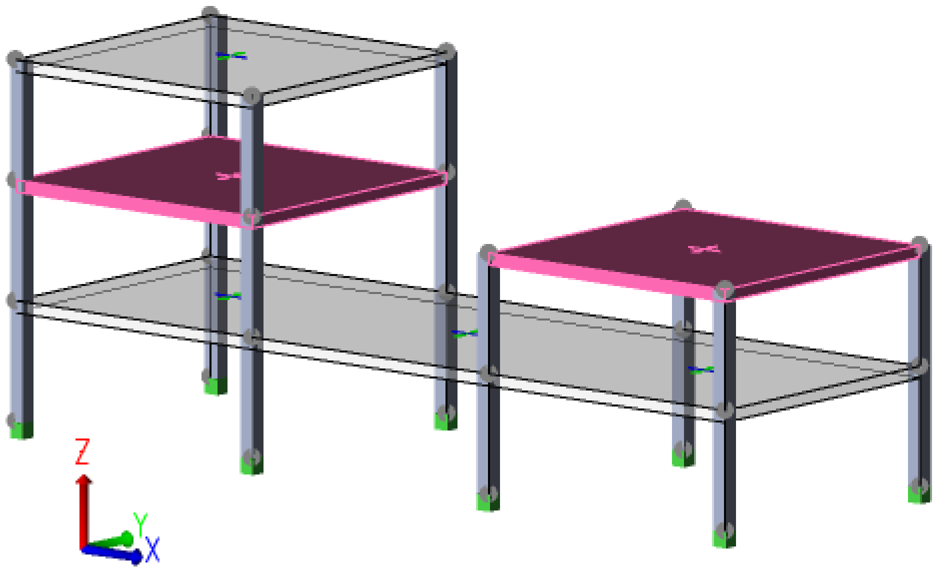
Limitations of wind decomposition to diaphragms

Certain building shapes need extra consideration if simple wind loads have been applied, or if wind loads have been applied to wall panels set to decompose to diaphragms.

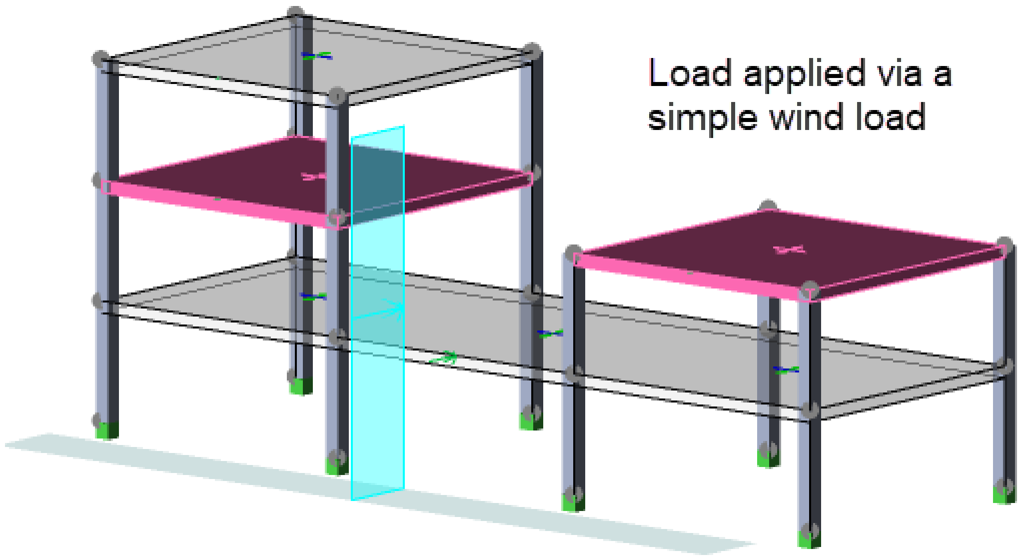
As illustrated by the following examples, buildings containing discrete towers (and thus containing disconnected rigid diaphragms) are a particular concern.

Wind load perpendicular to disconnected diaphragms

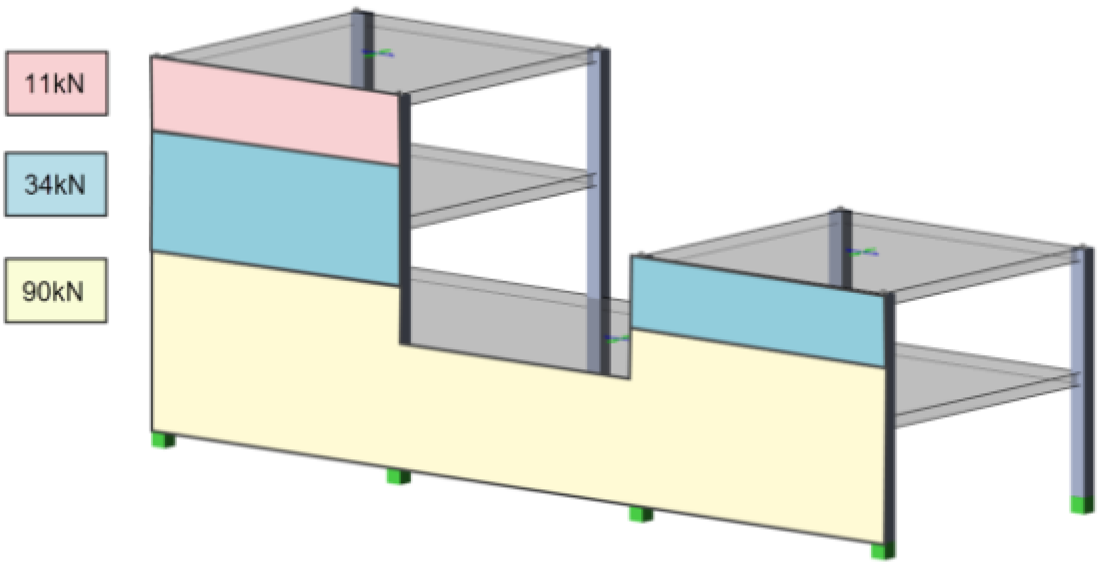
In this example wind load is to be applied in the Global Y direction, perpendicular to the disconnected diaphragms that exist in the highlighted slabs at the second floor level shown below.



An issue arises when the wind load is applied as an area load that has to be decomposed to both diaphragms. This could happen when the load is applied either via a Simple Wind load, or a wind wall panel:

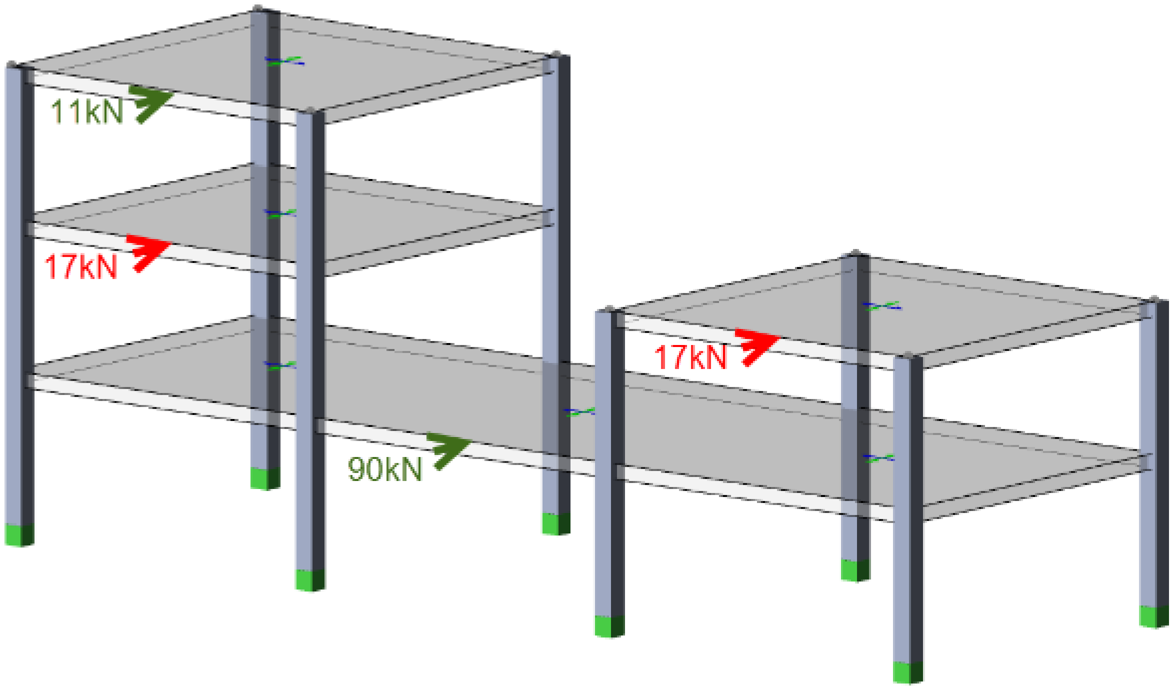


Irrespective of the method used to apply it, the area load within the building profile is shared between levels prior to decomposition.

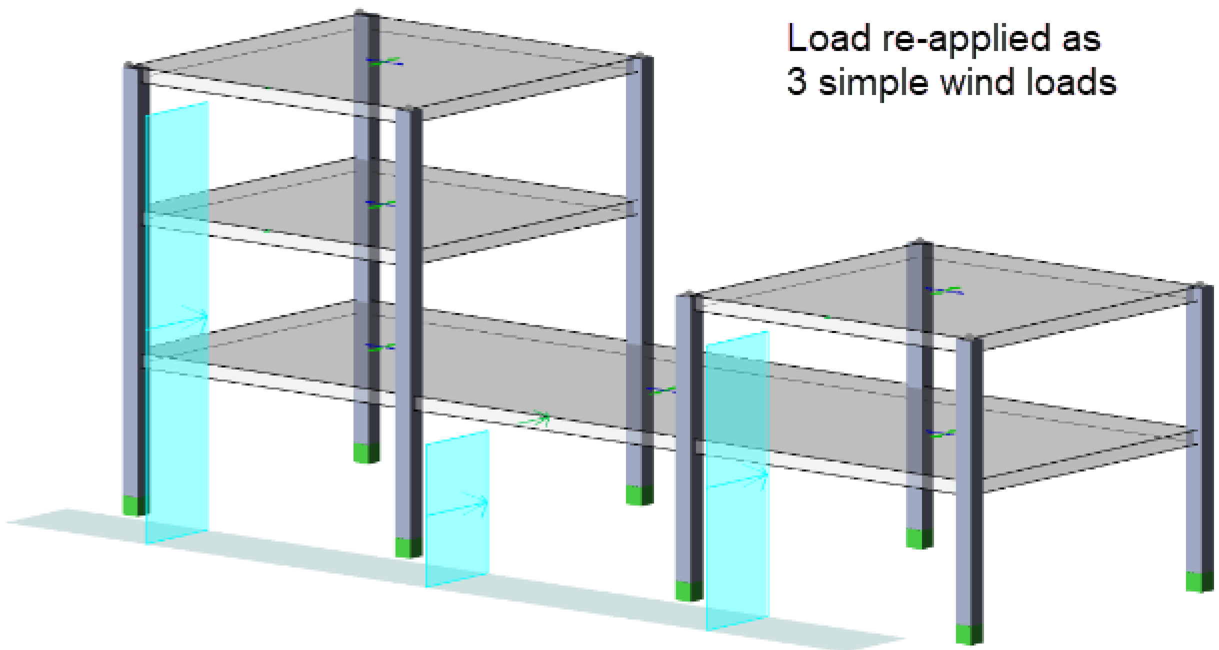


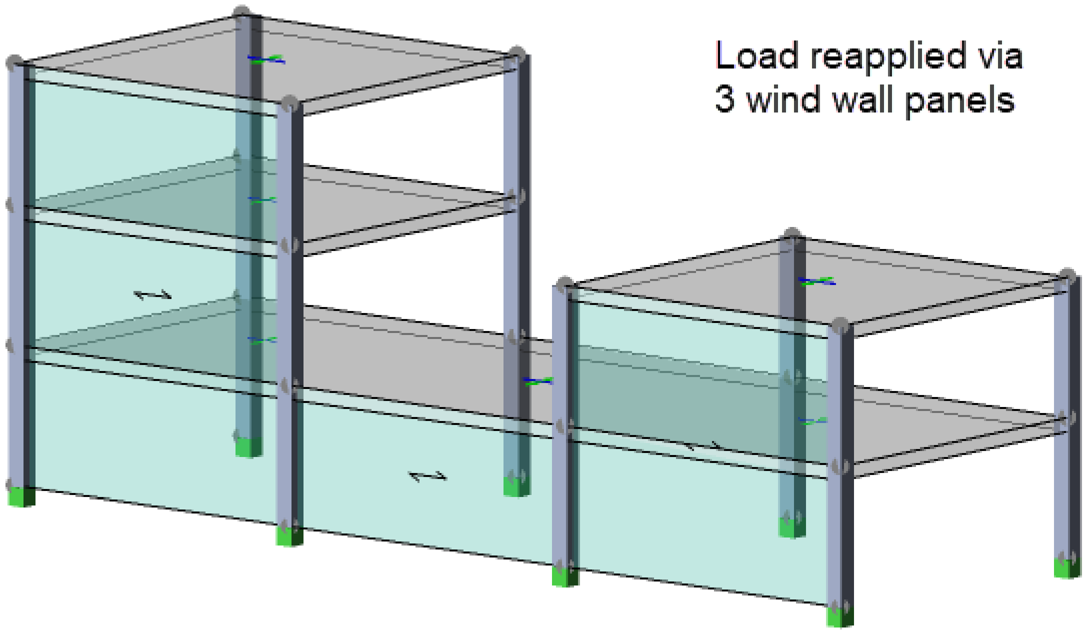
The load is then decomposed to the diaphragms at each level **in proportion to the width of each diaphragm**.

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

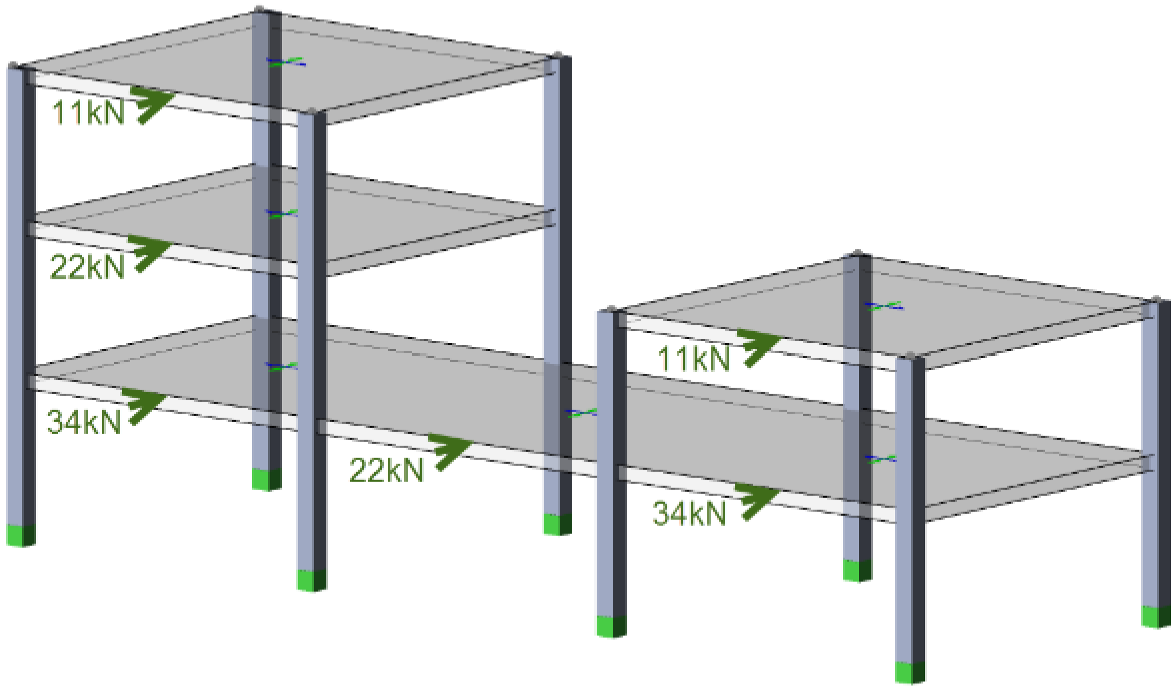


The workaround varies according to the method of loading, but basically involves replacing the original load with separate loads in each bay:



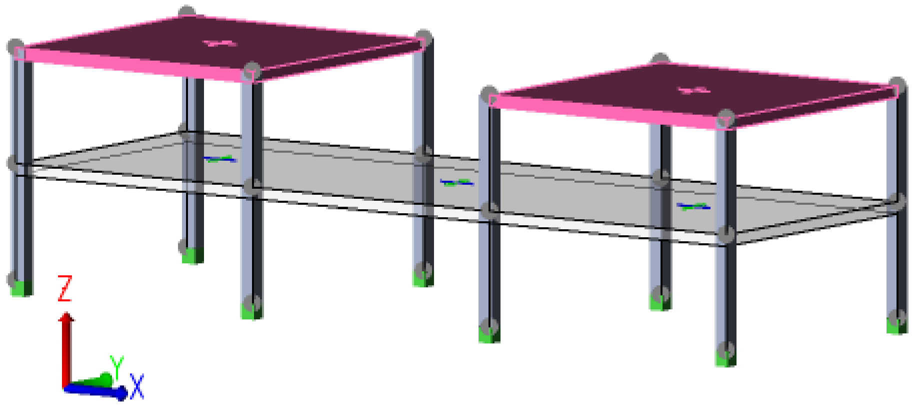


In both the above cases, the load is then decomposed as originally intended.

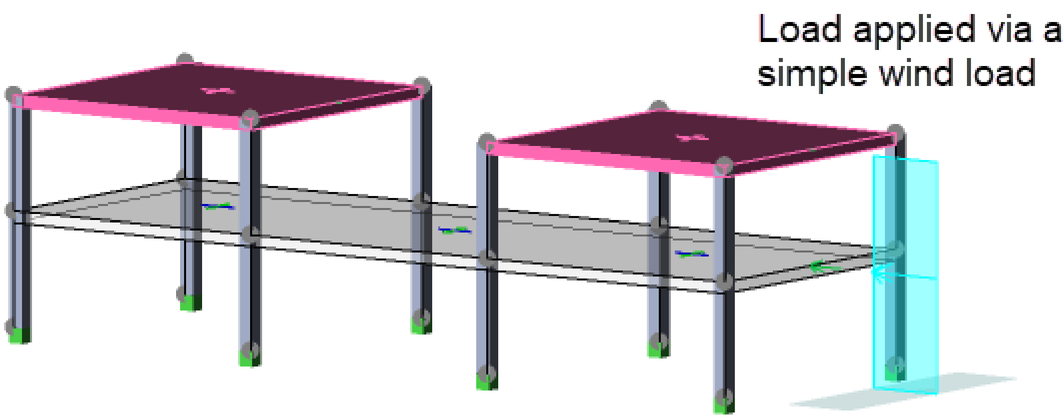


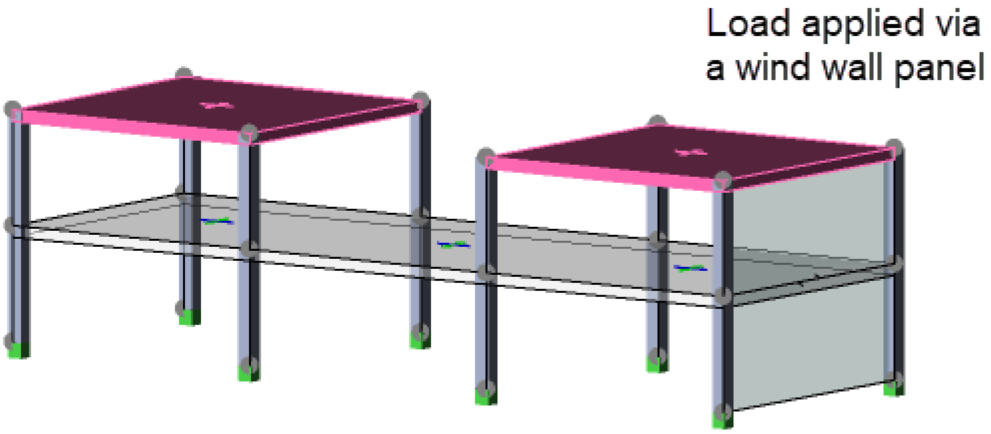
Wind load parallel to disconnected diaphragms

In this example although the two blocks are now the same height, another issue arises when the wind load is applied in the Global X direction, i.e. parallel to the disconnected diaphragms at the second floor level:

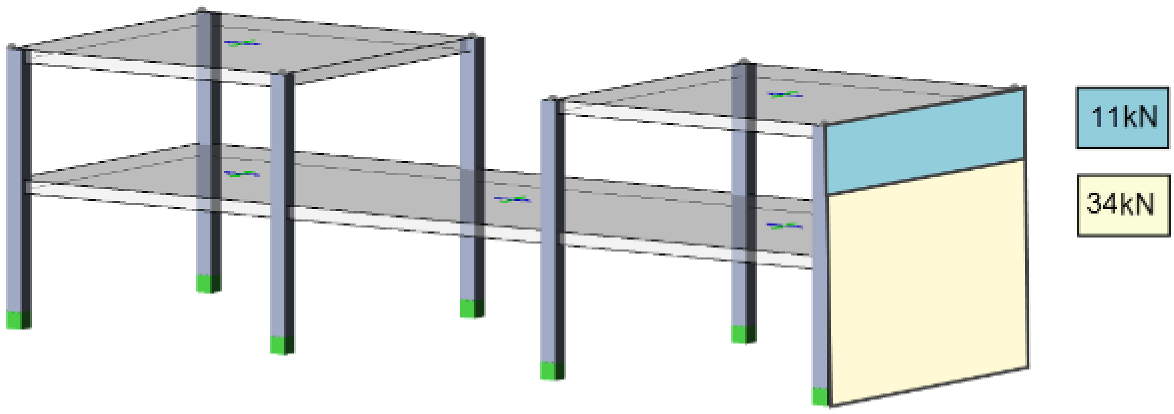


The issue arises because one diaphragm is 'hidden' from the applied load by the other diaphragm. The issue occurs irrespective of whether the load is applied via a Simple Wind load, or a wind wall panel:

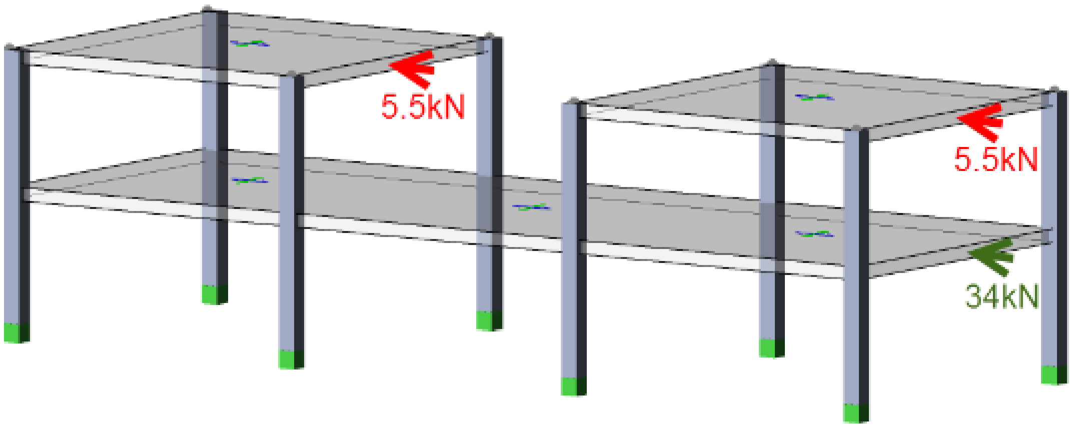




The area load within the building profile is first shared between levels prior to decomposition.



At the second floor level, instead of all the load being decomposed to the diaphragm facing the wind; because it is decomposed **in proportion to the width of each diaphragm at that level,** it ends up being shared equally to both.



To avoid the load being shared equally:

- if using wind wall panels - you would need to decompose to members or nodes instead of to diaphragms
- if using Simple Wind loads - there is no workaround, you would have to manually input the loads as diaphragm loads instead.

Wind tunnel testing and diaphragm loads

Tekla Structural Designer permits the easy flow of information required for wind tunnel testing of tall structures.

This includes:

1. The information out from Tekla Structural Designer to the wind specialists
2. The relevant results from wind specialists back into Tekla Structural Designer

Click the links below to find out more:

- Wind tunnel testing overview (page 108)
- Exporting wind tunnel data workflow (page 109)
- Using imported wind tunnel information (page 111)

Wind tunnel testing overview

There are four steps required in the process of using Tekla Structural Designer design within the process of wind tunnel testing.

1. Tekla Structural Designer creates a “Tekla Structural Designer - Wind Tunnel Report” from a vibration analysis to be used by the wind specialists.

   See: Exporting wind tunnel data workflow (page 109)

2. The “Tekla Structural Designer - Wind Tunnel Report” is sent to the wind specialists who undertake the wind tunnel tests. At the end of the tests, they create a “Wind Tunnel - Load Report” which they send back to the structural designer.

3. The designer manipulates the “Wind Tunnel - Load Report” to generate 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/level.

   See: Using imported wind tunnel information (page 111)

4. The 24 wind loadcases are combined in regular combinations to be included as part of the design process in Tekla Structural Designer

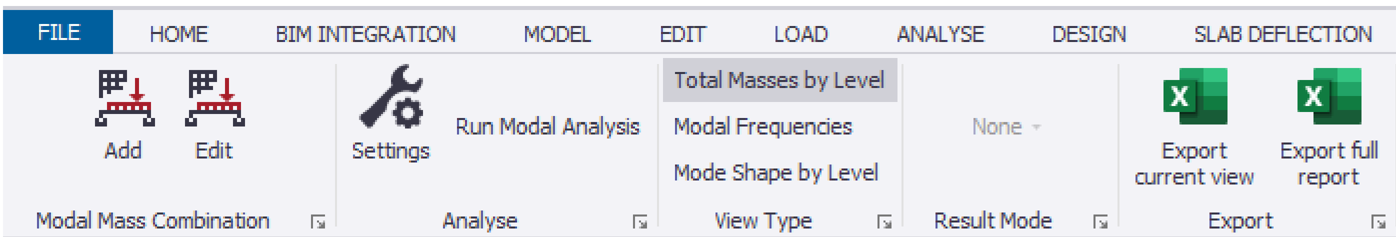
Exporting wind tunnel data workflow

Once the model has been created and had loads applied in Tekla Structural Designer, the basic workflow for generating and exporting wind tunnel data is described below.

Display the Wind Tunnel Data ribbon

The workflow is initiated by clicking **Wind Tunnel** on the Analyze ribbon.

- This enables the **Wind Tunnel Data** ribbon below:



- You can work through this from left to right...

Add modal mass combinations

1. On the **Wind Tunnel Data** tab, click **Add**.

   The opens on the **Combinations** page with a new **Modal Mass** class of combination added in the left hand pane.
2. Rename the combination as required.
3. Select the loadcases that you want to add in the combination, and click **>>**
4. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
5. Click **OK** to save the mass combination.

   Tekla Structural Designer adds the mass combination to the list of combinations in the **Loading** list.

Review the modal analysis settings

1. On the **Wind Tunnel Data** tab, click **Settings**

   The **Analysis Settings** dialog opens on the 1st Order Modal page.
2. Select the total number of modes to be calculated.
3. Review and modify the other analysis settings according to your needs.

Run a modal analysis

1. On the **Wind Tunnel Data** tab, click **Run Modal Analysis**.

   Tekla Structural Designer analyzes the model.

Review the results

After modal analysis the following results are available and can be reviewed as tabular data:

- Masses, Center of Mass, and Mass Moment of Inertia by level,
- Modal Frequencies,
- Mode shape by level

Mode shapes can also be displayed graphically if required by switching to a **Results View**.

Export the report

To export all the tabular results that can be viewed in the **Wind Tunnel Data View**, click **Export full report**.

Alternatively, you can export just the tabular results that are displayed in the current view by clicking **Export current view**.

Using imported wind tunnel information

The Tekla Structural Designer workflow for importing and using the wind tunnel information provided by wind specialists is as follows:

Create loadcases for the diaphragm loads

The wind tunnel report typically contains 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/ level.

Before the loads can be imported from the report, the engineer has to manually create wind loadcases to hold them.

Apply the diaphragm loads in each loadcase

Diaphragm loads can either be pasted into Tekla Structural Designer or entered manually.

NOTE Diaphragm loads can only be pasted at floors/levels with one or more horizontal rigid or semi rigid diaphragm.

See:

Create load combinations

Load combinations are created manually. The Wind Tunnel loadcases behave as any other "Wind" type loadcase when being used in combinations.

Analysis and design

A standard analysis and design process is followed, in which:

- the graphical analysis results for wind tunnel loadcases and combinations being shown in exactly the same way as for any other wind type loadcases and combination,
- wind tunnel loadcases are used in drift checks in the same way as other wind type loadcases,
- wind tunnel loadcases are used in design combinations in the same way as other wind type loadcases.

1.2 Stability and imperfections handbook

This handbook introduces you to the following topics:

- The four sources of stability requirements that may need to be considered, see Overview of stability requirements (page 112).

- Choosing the correct analysis type for your model and head code, see Allowing for global second-order effects (page 116).
- Catering for global imperfections, see Allowing for global imperfections (page 127).
- Catering for member imperfections, see Allowing for member imperfections (page 129).

Overview of stability requirements

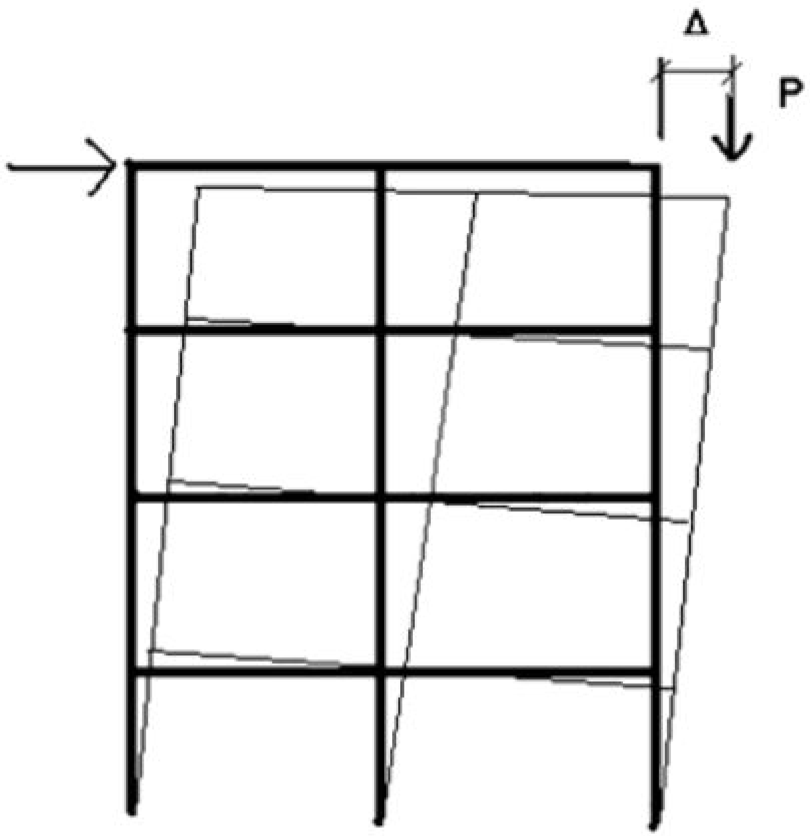
The analysis and design process has to allow for the differences between a theoretical mathematical model of a building and a more realistic representation. For example, buildings are not truly vertical when first built nor do they remain so when subject to load. These are called stability requirements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

1. **Global second-order (P-Δ) effects** to allow for deformation of the structure under load,
2. **Member second-order (P-δ) effects** to allow for deformation of the members under load,
3. **Global imperfections** due to the structure not being built plumb and square,
4. **Member imperfections** due to initial lack of straightness of the member.

There are various methods of allowing for each of these and they can be different for steel and concrete. There is also some variation based on country code.

Global second-order (P-Δ) effects

If gravity loads are applied to the deflected shape of a structure the load P applied at eccentricity Δ generates additional forces.



Provided the deflection is small:

- Structure is 'Non-sway'
- Second order effects can be ignored.

At some level these effects are no longer ignorable:

- Structure is 'Sway sensitive'
- And you have to do something to account for the second order effects.

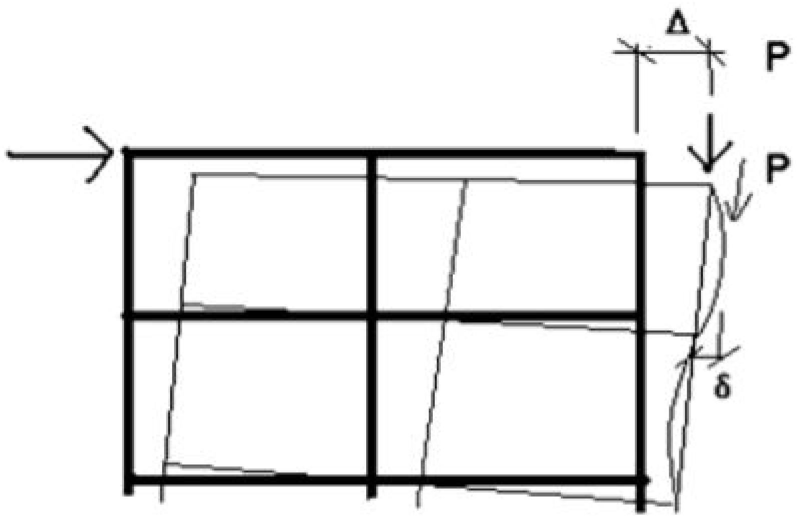
NOTE The terminology: 'Non-sway' and 'Sway sensitive' can vary between codes.

Related concept

Allowing for global second-order effects (page 116)

Member second-order (P-δ) effects

Under load members will deform between their ends:



- Member curvature introduces a displacement δ between the member ends.
- The member axial loads applied at eccentricity δ generates additional forces.

In concrete structures:

- Where deflections are small :
  - Member is 'Short' or 'Stocky' or 'Non-Slender'
  - member second order effects are considered ignorable
- At some level they are no longer ignorable:
  - Member is 'Slender'
  - effects must be catered for in the design calculations

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

NOTE The terminology: 'Short' , 'Stocky' and 'Non-slender' can vary between codes.

When must global and member second order effects be considered?

Depending on the building's overall sway classification and each member's slenderness, global and member effects must be considered as follows:

Member Effects	Global	Effects

	Non-sway	Sway
Short Member	A	C
Slender Member	B	D

A - All second order effects can be ignored

B - Global effects can be ignored - member effects must be considered

C - Global effects must be considered - member effects can be ignored

D - Global effects must be considered - member effects must be considered

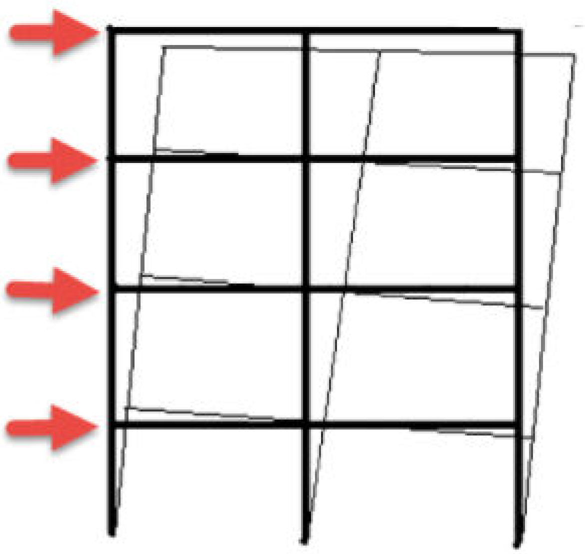
NOTE REMEMBER - every structure and every member has 2 directions!

Global imperfections

When the design code requires it you need to account for some degree of inclination (slope); typically in the range 0.2 to 0.5%

The codes allow you to cater for this in different ways:

- You could build multiple analysis models that are inclined
- You could have a single analysis model where you apply Equivalent Horizontal Forces [Eurocodes] / Notional Loads [AISC/ACI] / Notional Horizontal Forces [AUS/BS/IS] that will induce the same effect. Basically this means applying horizontal forces = 0.2 to 0.5% of the vertical forces in any combination.



In Tekla Structural Designer we use the second option.

NOTE Global Imperfections apply regardless of whether the structure is 'Non-sway' or 'Sway sensitive'.

Related concept

Allowing for global imperfections (page 127)

Member imperfections

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

Related concept

Allowing for member imperfections (page 129)

Allowing for global second-order effects

The type of analysis required to meet stability requirements and the checks performed will vary depending on the head code and material.

Choice of analysis type (ACI/AISC)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of two analysis types. These are,

- First-order analysis (not suitable for the final design of steel structures),
- Second-order analysis.

For steel structures: - Unless a number of specific criteria can be met, it is essential that your final design utilizes second-order analysis. However, second order analysis can be more sensitive to parts of your model that lack stiffness. For this reason it is recommended that you initially use first-order analysis to obtain sections and an overall building performance with which you are satisfied before switching to second-order analysis.

For concrete structures: - By choosing second-order analysis the global second-order effects are automatically catered for. However, if you determine that the structure is non-sway (see When should a concrete building be classed as non-sway? (page 116)) you may instead opt for the design to be based on the first order analysis.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 127).

When should a concrete building be classed as non-sway?

ACI 318-11 clause 10.10.5.2 states that a story can be considered as non-sway when the stability index Q is less than or equal to 0.05. (Basically, second order effects can be ignored below this value.)

Tekla Structural Designer does not report Q directly, but the Drift report does calculate an indicative stability coefficient (Δ_2/Δ_1) which is effectively the same as a moment magnifier.

From ACI Eq. (10-20):

Moment magnifier, $\delta_s = 1/(1-Q)$

Equating this to the stability coefficient gives

$1/(1-Q) = (\Delta_2/\Delta_1)$

$1-Q = 1/(\Delta_2/\Delta_1)$

$Q = 1 - 1/(\Delta_2/\Delta_1)$

For $Q < 0.05$

$1 - 1/(\Delta_2/\Delta_1) < 0.05$

$(\Delta_2/\Delta_1) < 1/0.95$

$(\Delta_2/\Delta_1) < 1.0526$

Therefore, if the Drift report determines an indicative stability coefficient (Δ_2/Δ_1) less than 1.05 it can be assumed non-sway; whereas if greater than this value a second-order analysis is required.

NOTE The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 127).

Choice of analysis type (BS)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - - see: **A practical approach to setting the analysis type** below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (regular frameworks with $\lambda_{cr} > 4$).

If the amplified forces method is selected you must also indicate which formula to use for determining the amplification factor,

If the structure is clad and if the stiffening effect of cladding is not taken into account explicitly:

$k_{amp} = \lambda_{cr}/(1.15\lambda_{cr} - 1.5)$

If the structure is unclad or clad with a direct allowance made for the stiffening effect:

$k_{amp} = \lambda_{cr}/(\lambda_{cr} - 1)$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of λ_{cr} for all stacks in the building.

The k_{amp} results are summarized for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review toolbar.
2. Select 'k_{amp}' from the View Type drop list on the Review toolbar.
3. The k_{amp} results in both directions are tabulated for each column in the building.

Second-order analysis

Full second-order analysis is not restricted to regular frameworks, but requires $\lambda_{cr} > 2$.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 127).

A practical approach to setting the analysis type

Unless λ_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to P-Δ analysis.

Note: If the rigorous second-order (P-Δ) analysis approach is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.
3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, λ_{cr}. See: **How do I assess the worst elastic critical load factor for the building?** section below.
4. If the λ_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If λ_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
   - Either, refine the design until λ_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces - see below),
   - Or, in order to use the amplified forces (k_{amp}) approach, refine the design further until λ_{cr} is greater than 4.0.
6. When a suitable λ_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate. (If the k_{amp} approach is selected you must also indicate which formula to use for determining the amplification factor, This will depend on whether the structure is clad or not and if the cladding is taken into account explicitly or not.)

7. With the analysis type still set to the full second-order, or the k_{amp} approach perform Design All (Static).

If you use the k_{amp} approach be aware that BS5950-1:2000 classes certain structures outside the scope of this method. e.g. tied portals, or structures containing tension only braces. For such structures, you would need to ensure that λ_{cr} is greater than 2.0, and use the full second-order analysis approach for the static design.

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review toolbar.
2. Select 'Sway' from the View Type drop list on the Review toolbar.
3. The elastic critical load factor in both directions ((λ_{Dir1} & (λ_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest λ value from all of the columns in either direction.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the λDir1 header until the columns are arranged in increasing order of λDir1, then repeat for λDir2.

In BS 5950-1:2000 a building can be considered as 'non-sway' when $\lambda_{cr} \geq 10$ else it is 'sway sensitive' and (global) second-order effects must be taken into account.

Note however that you are not restricted in your choice of analysis type irrespective of the value of λ_{cr} (it is your call, although we will warn you about it).

How is the elastic critical load factor calculated?

In order to determine whether a building is 'non-sway' or 'sway sensitive', Tekla Structural Designer calculates the elastic critical buckling load factor, λ_{cr}. It is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Notional Horizontal Forces (NHF) are determined. It uses 0.5% of the vertical load at the column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the NHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey

(storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of λ_{cr} for that combination as follows,

λ_{cr} =h/(200 * δ_s)

Where

h= the storey height

δ_s =the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads.

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

Choice of analysis type (Eurocode)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - see: **A practical approach to setting the analysis type** below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (basically regular frameworks with α_{cr} > 3 - see **Validity of the amplified forces method** section below). Full second-order analysis is preferred for steelwork and since it is not precluded by EC2 it can be used for concrete.

The amplified forces method is described differently in EC3 compared to EC2, whilst the presentations are different, they are both based on the amplifier, k_{amp} given as,

k_{amp} = 1/(1 - 1/α_{cr})

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of α_{cr} for all stacks in the building.

The k_{amp} results are summarized for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review toolbar.
2. Select 'k_{amp}' from the View Type drop list on the Review toolbar.
3. The k_{amp} results in both directions are tabulated for each column in the building.

NOTE The amplified forces method is not recommended for non-linear structures - a full second-order analysis should be performed instead.

Validity of the amplified forces method

EC3 Clause 5.2.2 (6)B lists limitations on the applicability of the Amp. Forces method. It is therefore your responsibility when selecting this method to ensure all of the following:

- all storeys have a similar distribution of vertical load
- all storeys have a similar distribution of horizontal load
- all storeys have a similar distribution of frame stiffness with respect to the applied storey shear forces

Also according to clause 5.2.1 (4)B limitation:

- roof slope shallow - not steeper than 1:2 (26 degs)
- axial compression in beams or rafters - N_{cr} / N_{ed} <= 11.1

Second-order analysis

Full second-order analysis is more widely applicable for steelwork structures and since it is not precluded by EC2 it can be used for concrete.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 127)

A practical approach to setting the analysis type

Unless α_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to one of the P-Δ analysis methods.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.
3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, α_{cr}. See **How do I assess the worst elastic critical load factor for the building?** section below.
4. If the α_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If α_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
    - Either, refine the design until α_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces),
    - Or, in order to use the amplified forces approach, refine the design further until α_{cr} is greater than 3.0.
6. When a suitable α_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate.
7. With the analysis type still set to the full second-order, or the amplified forces method, perform Design All (Static).

NOTE If full second-order analysis is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

NOTE If you use the 'Second-order analysis - Amp. forces method' be aware that EC3 classes certain structures outside of its scope (see above section: **Validity of the amplified forces method**). Such structures would need to be refined during gravity sizing until the elastic critical load factor is at least greater than 2.0, so that the full second-order approach can be used for the full design.

When should a building be classed as sway sensitive?

Susceptibility to second order effects is a general characteristic and is not material specific, it has just been presented differently in EC3 and EC2:

- In EC3 a building can be considered as 'non-sway' when the elastic critical load factor $\alpha_{cr} \geq 10$, else the building is 'sway sensitive' and (global) second-order effects must be taken into account.
- In EC2 the definition is slightly different - it does not use the terms 'non-sway' and 'sway sensitive'. Rather it simply defines when second-order effects are small enough to be ignored. The principle is given in Clause 5.8.2 (6) which states that they can be ignored if they are less than 10% of the corresponding first order effects. Because of the way in which the amplification factor, k_{amp} is calculated the change point is at an α_{cr} of 11 not 10. See: **Derivation of the k_{amp} formula for concrete structures** section below.

However, the intent is the same in both cases and so in Tekla Structural Designer $\alpha_{cr} \geq 10$ is taken as the change point. In any event, you are not restricted in your choice of analysis type irrespective of the value of α_{cr} (it is your call, although we will warn you about it)

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review toolbar.
2. Select 'Sway' from the View Type drop list on the Review toolbar.
3. The elastic critical load factor in both directions (α_{Dir1} & α_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest elastic critical load factor from all of the columns in either direction - this is the α_{cr} value for your building.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the αDir1 header until the columns are arranged in increasing order of αDir1, then repeat for αDir2.

Having determined an α_{cr} for your building, you then use it when deciding which is the most appropriate analysis type for design.

How is the elastic critical load factor calculated?

The elastic critical load factor, α_{cr} is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Equivalent

Horizontal Forces (EHF) are determined. These consist of 0.5% of the vertical load at each column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the EHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of α_{cr} for that combination as follows,

$\alpha_{cr} = h/(200 * \delta_{EHF})$

Where

h = the storey height

δ_{EHF} = the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

Derivation of the k_{amp} formula for concrete structures

EC2 provides two specific approaches to determine the change point below which second-order effects are small enough to be ignored:

The first specific approach is contained in Clause 5.8.3.3 which provides a pass/fail criterion to check whether the global second-order effects may be ignored. It is given as,

$F_{VEd} = k_1 * n_s/(n_s + 1.6) * S(E_{cd} * I_c)/L^2$

where

F_{VEd} = the total vertical load (on 'braced' and 'bracing' members)

k_1 = a factor that allows for cracking in the concrete of the LLRS and is a Nationally Determined Parameter (NDP)

* n_s = number of storeys

E_{cd} = the design value of the modulus of elasticity of the concrete

I_c = the second moment of area of the uncracked bracing members

L = the total height of the building

However, the above approach has a number of restrictions in its application and as a result it is not applied in Tekla Structural Designer.

The second specific approach is given in Annex H.

The method given in Annex H.1.2 is the background for the more limited method given in Clause 5.8.3.3 as described above, but it does not apply where there is significant shear deformation in the LLRS e.g. for shear walls with significant openings, hence again it is not considered in Tekla Structural Designer.

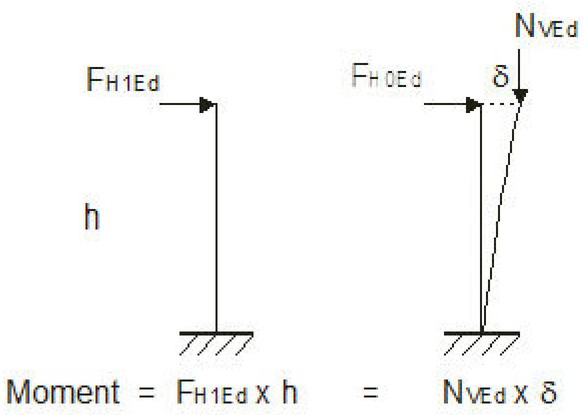
Instead, recourse is made to determining the level of second-order effect using Annex H.2. Using this approach, by rearranging Equation H.8 it is possible to provide a 'stability coefficient' $1/\alpha_{cr}$ which can be applied as the change point between non-sway and sway sensitive structures.

$F_{HEd} = F_{H0Ed} /(1-F_{H1Ed}/F_{H0Ed})$ Equation H.8

Where:

F_{H1Ed} = fictitious horizontal force, giving the same bending moments as vertical load N_{VEd} acting on the deformed structure, with deformation caused by F_{H0Ed} (first order deformation), and calculated with nominal stiffness values according to 5.8.7.2

Considering how this definition of F_{H1Ed} might apply to an imaginary cantilever of height, h, we arrive at:



1. The moment due to F_{H1Ed} is the same as that due to the vertical load N_{VEd}, so:

$F_{H1Ed} * h = N_{VEd} * \delta$

which can be rearranged to:

$F_{H1Ed} = (N_{VEd} * \delta)/h$

2. Substituting for F_{H1Ed} in Equation H.8, we have:

$F_{HEd} = F_{H0Ed} /(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

3. By defining $k_{amp} = F_{HEd}/F_{H0Ed}$ the above can be rearranged to:

$k_{amp} = 1/(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

4. Now, the EC3 Equation 5.2 for the elastic critical buckling load is:

$\alpha_{cr} = H_{Ed}/V_{Ed} * h/\delta_{HEd}$

which, when re-expressed in the terminology used in H.2 becomes:

$\alpha_{cr} = F_{H0Ed} / N_{VEd} * h/\delta_{HEd}$

and when further rearranged becomes:

$1/\alpha_{cr} = (N_{VEd} * \delta_{HEd})/(F_{H0Ed} * h)$

5. Hence $1/\alpha_{cr}$ can be substituted into the above equation for k_{amp} so that we arrive at the more well-known formula for amplification:

$k_{amp} = 1/(1 - 1/\alpha_{cr})$

NOTE Strictly, the watershed for concrete structures should be at a k_{amp} factor of 1.1 (amplification of no more than 10% due to second-order effects). Setting k_{amp} to be 1.1 and rearranging gives $\alpha_{cr} \geq 11$ i.e. a stability coefficient ≤ 0.0909 not 0.1

NOTE It is important to note that the resulting values of α_{cr} and k_{amp} are very dependent upon the analysis properties that are used and the you therefore need to carefully consider the modification factors you choose to apply via the Analysis Options.

Use of modification factors

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. Consequently design codes can require that analysis stiffness adjustment factors are applied (as the appropriate properties to use in analysis are load and time dependent).

These modification factors can be applied for each of the different materials from the Modification Factors page of the Analysis Options dialog. (which is located on the Analyse toolbar).

For non-concrete members it is also possible that you will want to apply an adjustment to material properties for various other investigations. One suggested example is the assessment of structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

Allowing for global imperfections

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

Allowing for global imperfections (ACI/AISC)

Steel code

Steelwork design to AISC 360-10 requires an allowance for global imperfections. Columns are assumed to be out of plumb by some amount and this is replicated by applying Notional Loads, (NL), in the analysis. The requirements are given in Clause C2.2b and the value of NL is given as 0.2%. However, to accommodate the requirements of Clause C2.3 an additional 0.1% has been included - so that the actual NL used is 0.3%.

Concrete code

For concrete design to ACI 318-11 there are no requirements. However, given that a building can use mixed materials, (and even for an entirely concrete building), you have the choice to include the NLs via the combinations. By default these are to the AISC requirements - i.e. 0.3%

Allowing for global imperfections (Eurocode)

The formula to calculate the global imperfections (using EHFs) is the same for both steel and concrete, see : EC2 Cl 5.3.2 (3) a) and EC3 Cl 5.2 (5)

$\varphi = \varphi_0\ \alpha_h\ \alpha_m$

Where:

φ_0 is the basic value of inclination.

α_h is the reduction factor for length or height : $\alpha_h = 2/\sqrt{h}$; $2/3 \leq \alpha_h \leq 1$

h is the length or height of the structure.

When height = 9m the maximum reduction of 2/3 will apply.

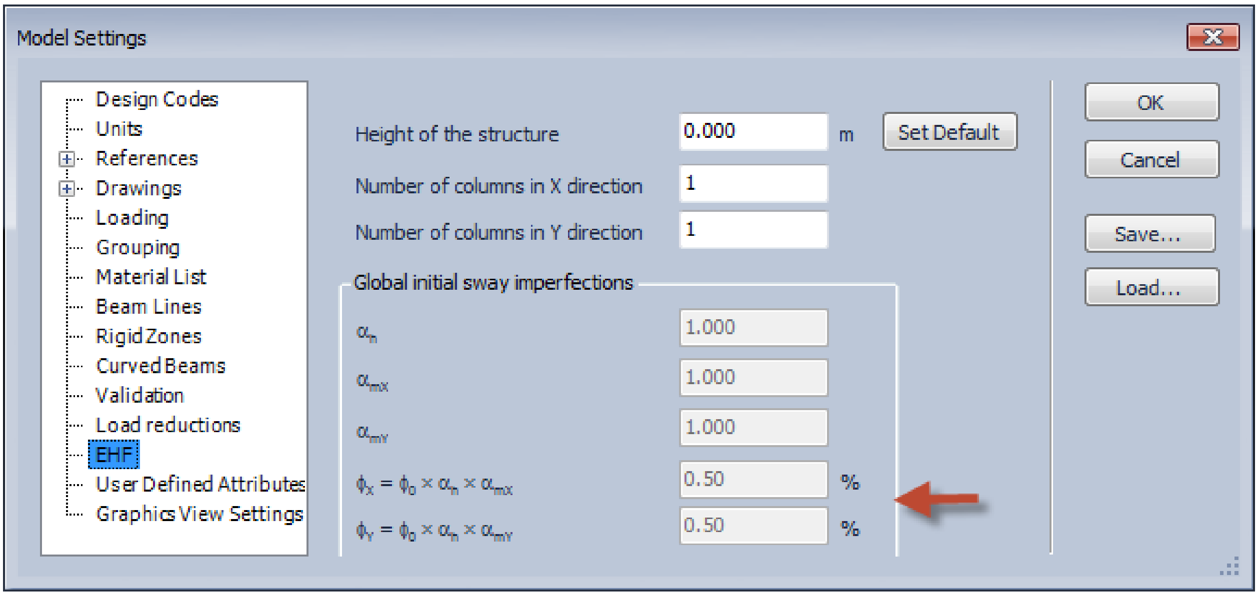
α_m is the reduction factor for number of members (EC2) or columns in a row (EC3), : $\alpha_m = \sqrt{(0.5(1+1/m))}$

m is the number of vertical members contributing to the total effect (EC2), or is the number of columns in a row including only those columns which carry a vertical load not less than 50% of the average value of the column in the plane considered (EC3).

m =	1	5	10	1000
α_m =	1	0.775	0.742	0.707

Guidance on how to count 'm' is vague and varied - however as demonstrated above, once you get above 5 or 10 it starts to make very little difference.

In Tekla Structural Designer the allowance is applied in the same manner for steel and concrete - being controlled in Model Settings as shown below:



It does require some user intervention to provide structure height and number of columns to consider. These user inputs cause adjustment of the default value of imperfection of 0.5% of the vertical load and this can be a different adjustment in the two orthogonal directions (Direction 1 and Direction 2). For example, the adjustment factor might give an EHF of 0.4% in the X-direction and 0.3% in the Y-direction.

Allowing for global imperfections (BS)

Steel code

BS5950 requires up to 0.5% of vertical loading.

Concrete code

BS8110 does not have a requirement (minimum lateral load is not the same thing).

Allowing for member imperfections

Allowing for member imperfections (ACI/AISC)

Steel code

AISC indicates that these are inherently catered for in the strut design.

Concrete code

For concrete design to ACI 318-11 there are no requirements.

Allowing for member imperfections (Eurocode)

Steel code

For steel structures designed to EC3, member imperfections are intrinsically included in the design routines for all members (beams, columns, braces). Apart from one explicit requirement, carrying out the design is all that is necessary.

The explicit requirement is from Clause 5.3.2 (6) in which member imperfection should be included as part of the analysis when the frame is sway sensitive and the axial force in members with moment connections is above a certain limit. If this situation arises Tekla Structural Designer issues a warning.

Concrete code

For concrete structures designed to EC2, explicit calculations which consider an imperfection effect are carried out as part of the design.

In Tekla Structural Designer this is achieved by adding an additional moment to the analysis results before starting design.

Allowing for member imperfections (BS)

Steel code

BS5950 inherently caters for member imperfections in the strut design curves.

Concrete code

BS8110 does not have a requirement (minimum eccentricity is not the same thing).

1.3 Static analysis and design handbook

NOTE The aim of this handbook is describe the processes that occur when running one of the combined analysis and design commands, i.e:

- Design Steel (Gravity), Design Concrete (Gravity), Design All (Gravity)
- Design Steel (Static), Design Concrete (Static), Design All (Static)

You can find the following information in this handbook:

- Overview of the combined analysis and design processes (page 131)
- 3D pre analysis processes (page 136)
- 3D analysis (page 143)
- Grillage chasedown analysis (page 144)
- FE chasedown analysis (page 144)

- Reasons for performing chasedown analyses (page 145)
- Overview of stability requirements (page 112)
- Member design stage of the combined analysis and design process... (page 152)
- Features of the three analysis types used for static design... (page 154)

Overview of the combined analysis and design processes

Overview of Design Steel (Gravity)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis. **NOTE** For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/or produce unrealistic forces. This is specified via Design Options> General.
4	Member design stage (page 152)	Steel members are designed or checked for active gravity combinations only. **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.

Overview of Design Steel (Static)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis.
4	Member design stage (page 152)	Steel members are designed or checked for all active static combinations.
5	Stability checks	Sway/Drift checks and Wind Drift checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design Concrete (Gravity)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis.

Step	Process	Description
		NOTE For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/or produce unrealistic forces. This is specified via Design Options> General.
4	FE chasedown analysis (page 144)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 144)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 152)	Concrete members and concrete walls are designed or checked for active gravity combinations only. **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.

Overview of Design Concrete (Static)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.

Step	Process	Description
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model is run (excluding pattern loading) to establish a set of design forces for the steel members. **NOTE** The analysis type (i.e. first, or second order) is specified via Design Options> Analysis. **NOTE** For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/or produce unrealistic forces. This is specified via Design Options> General.
4	FE chasedown analysis (page 144)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 144)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 152)	Concrete members and concrete walls are designed or checked for active gravity combinations only.
7	Stability checks	Sway/Drift checks and Wind Drift checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design All (Gravity)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model, with an option to mesh floors.
4	FE chasedown analysis (page 144)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 144)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 152)	All steel and concrete members and all concrete walls are designed or checked for all active gravity combinations. **NOTE** Concrete slab design is not performed **NOTE** At this point design may not fully satisfy the code requirements. To achieve this a full static design is required.
7	Stability checks	Sway/Drift checks and Wind Drift checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Overview of Design All (Static)

Step	Process	Description
1	Model validation	Checks are performed to detect specific issues before the analysis and design process begins.
2	3D preanalysis processes (page 136)	A number of pre-analysis processes are undertaken as necessary in preparation for the full analysis.
3	3D analysis (page 143)	A traditional frame analysis of the entire 3D model, with an option to mesh floors.
4	FE chasedown analysis (page 144)	Requirements: Only performed if two-way slabs exist. For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.
5	Grillage chasedown analysis (page 144)	Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.
6	Member design stage (page 152)	All steel and concrete members and all concrete walls are designed or checked for all active static combinations. **NOTE** Concrete slab design is not performed
7	Stability checks	Sway/Drift checks and Wind Drift checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

3D pre analysis processes

Pre-Analysis consists of a number of processes, such as:

- Decomposing slab and wall loads
- Preparing loadcases and combinations
- Meshing and diaphragms
- First-order gravity analysis

- Resolving vertical loads for application of global imperfections
- Generation of pattern loading

The actual pre-analysis processes performed will vary depending on the specific model that has been defined.

Overview of slab load decomposition

Decomposition of slab loads on to supporting members is automatically performed where necessary during pre-analysis.

Decomposition is not just performed for beam and slab models, the program may also need to decompose flat slab loads onto supporting columns and walls.

NOTE In Tekla Structural Designer the term "Decomposed Loads" refers to the loading on beams, columns, and walls that comes from slabs

Whether load decomposition is performed or not will depend on the analysis model, the slab properties and the **Mesh 2-way slabs in 3D Analysis** setting as follows:

<table>
<tr><th>Decomposition method specified in Slab properties</th><th>3D Analysis and Grillage chasedown models</th><th>FE chasedown model</th></tr>
<tr><td rowspan="2">Two-way</td><td>Mesh 2-way slabs in 3D Analysis option not selected (default):
• loads on two-way slabs decomposed prior to analysis</td><td rowspan="2">• it is not necessary to decompose the loads on two-way slabs prior to analysis</td></tr>
<tr><td>Mesh 2-way slabs in 3D Analysis option selected:
• it is not necessary to decompose the loads on two-way slabs prior to analysis</td></tr>
<tr><td>One-way</td><td>• loads on two-way slabs decomposed prior to analysis</td><td>• loads on two-way slabs decomposed prior to analysis</td></tr>
</table>

Potential Load Decomposition Methods

Traditionally a "tributary area" (sometimes called "yield line") loading approach would have been adopted to determine the decomposed loads, but this has limitations when dealing with complex geometry such as:

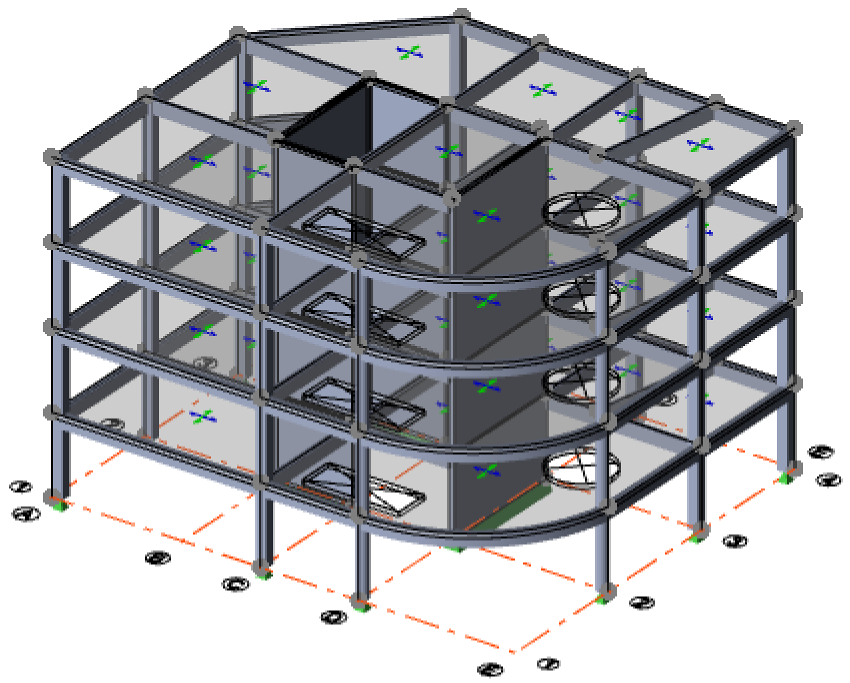
- Slabs not supported on all edges
- Complex panel shapes
- Panels with openings

Also the "tributary area" approach can only approximately handle point, line, and patch loads (by converting them to area loads).

Because of these limitations the "tributary area" method is not used in Tekla Structural Designer - instead a method referred to as FE Decomposition is applied instead. This is based on finite element analysis.

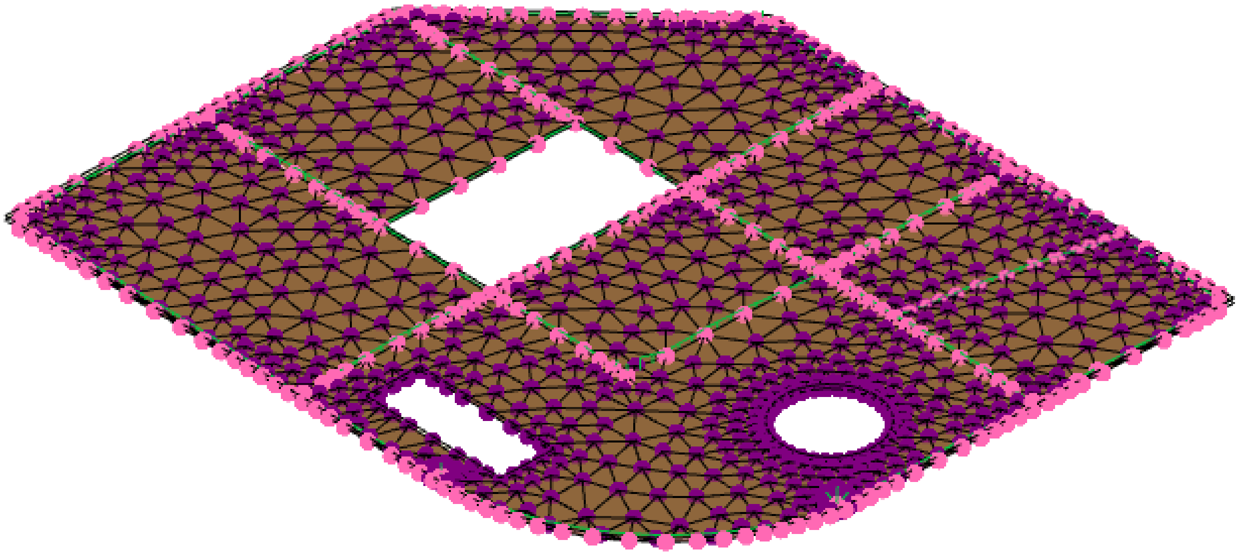
FE Decomposition Model

The FE decomposition model can be demonstrated using the following two-way slab on beams example.

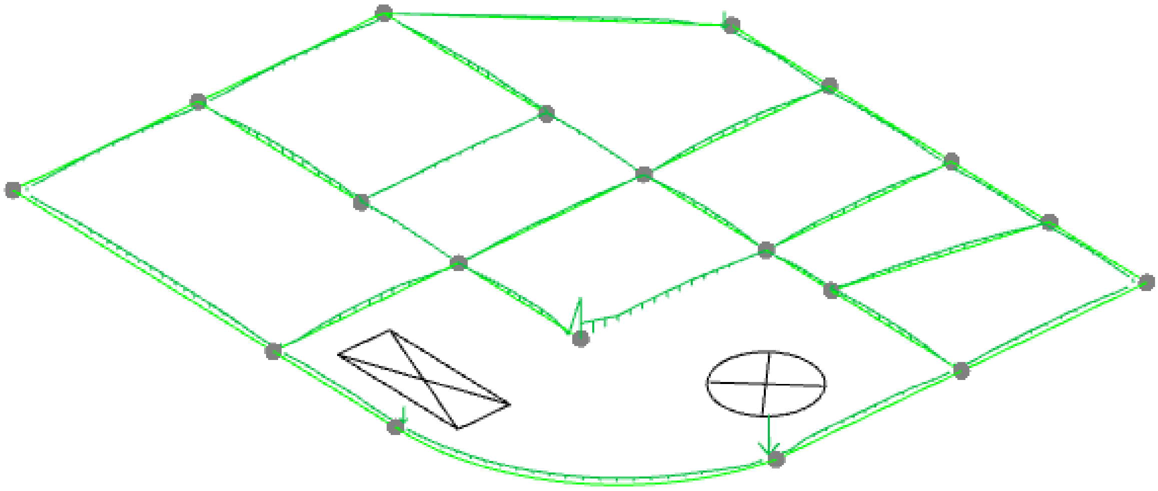


For this type of structure:

- A separate FE decomposition model is created for every floor (or sloped plane).
- Beam column and wall nodes in each FE decomposition model (shown selected in pink below) are all rigidly supported.



• Each FE decomposition model is analysed and the reactions at the rigidly supported nodes are turned back into VDLs along beams and walls, and point loads on columns.

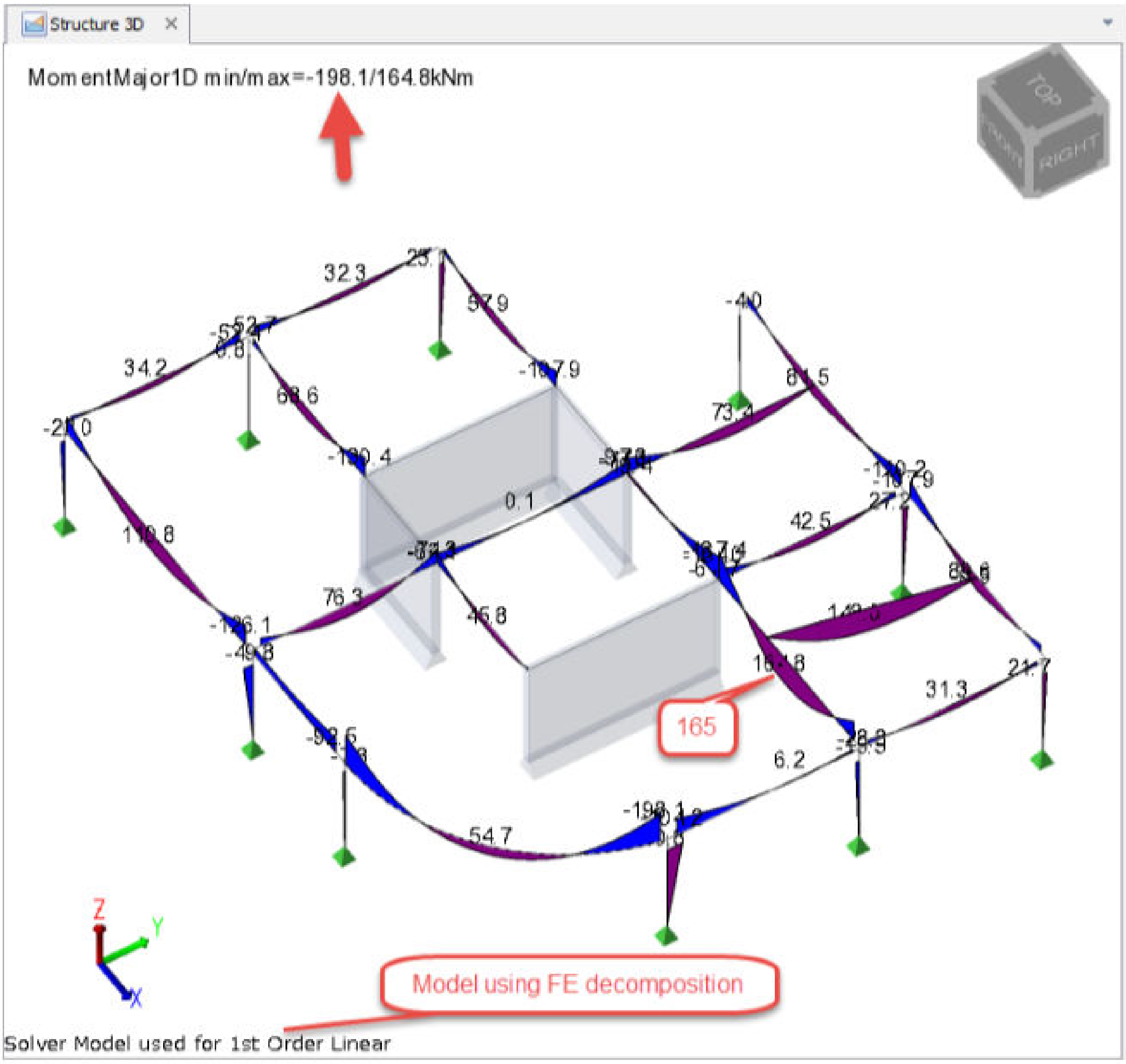


A Common Question

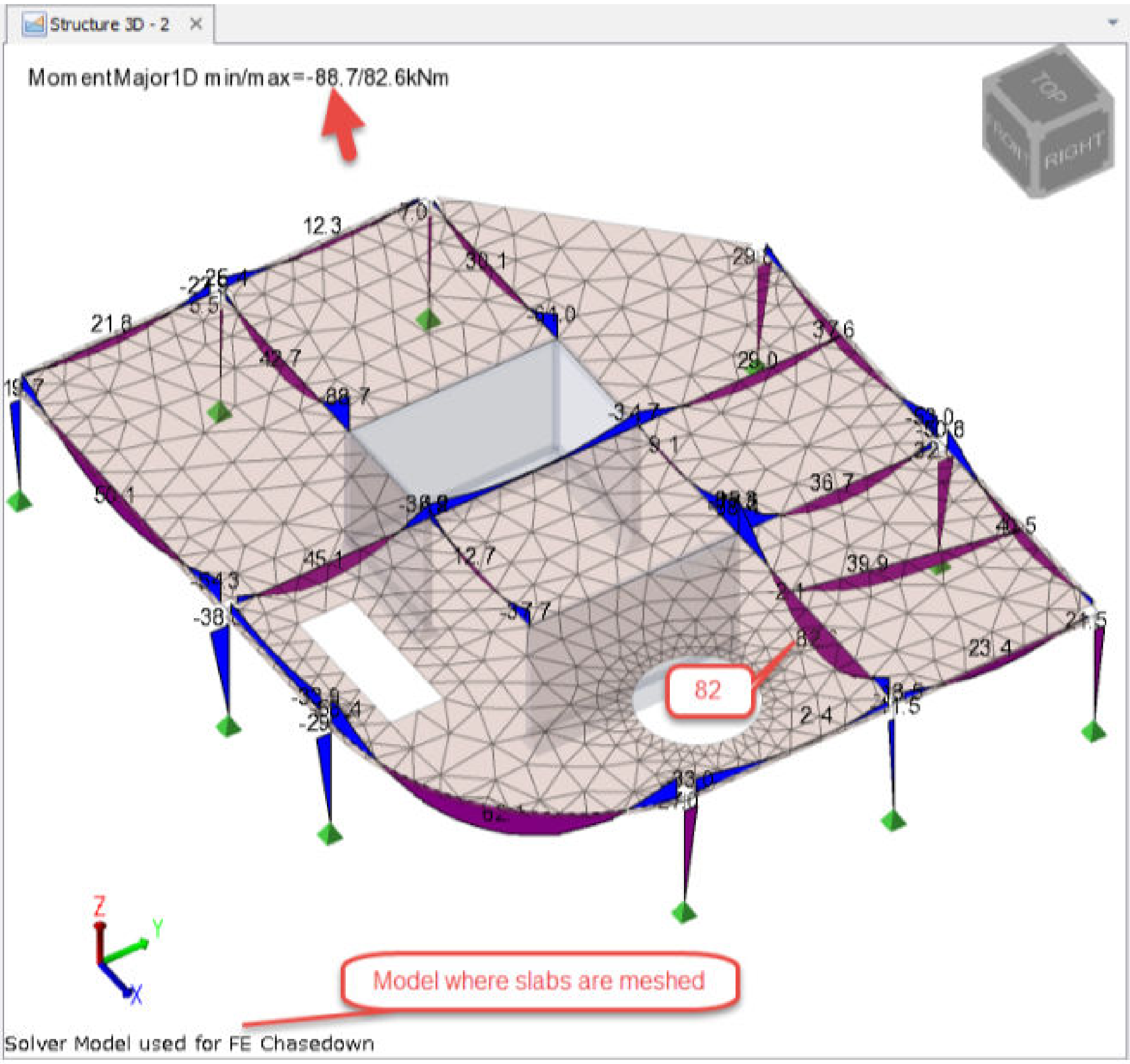
So if the slabs have to be meshed for the FE decomposition during pre-analysis, why not do away with decomposition entirely and just mesh the slabs when performing 3D analysis of the entire model instead?

Because this gives results that you don't like....

In the first run of the model below the slabs are left unmeshed, this requires the loads on the two-way slabs to be FE decomposed prior to analysis. The bending moments from the resulting 3D Analysis are as shown, (max hogging -198, max sagging 165):



The above results can be compared against a second run of the model in which the 2-way slabs are set to be meshed in the 3D analysis. FE decomposition is no longer required and the bending moments from the resulting 3D Analysis are as shown, (max hogging -89, max sagging 82)



In this second run, because the slabs are included in the 3D analysis model, some of the load is being transferred directly to the supporting columns and walls via the slabs themselves. While this is not wrong, it goes against the engineer's expectation - which would be to design the beams on the basis that they transfer all the load to the supporting members.

Overview of global imperfections

Equivalent notional horizontal loads are determined and applied during pre-analysis to cater for global imperfections (additional second order effects due to the structure not being built plumb and square). These loads are also used in seismic design to establish the base shears.

Following a first-order analysis of all gravity loadcases, the forces at the nodes at the top/bottom of each column stack/wall panel are resolved vertically. A proportion of the vertical load is determined which gives the value of the horizontal load at each point. The proportion is code dependent.

These horizontal loads are applied to the nodes in a particular direction (Direction 1 or Direction 2 or both) as specified in an individual design combination.

Overview of live/imposed load reductions

Live load reductions (US), or imposed load reductions (other head codes) 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.

Reductions are only applied to those live/imposed loadcases that have had the **Live/Imposed Load Reductions** box checked in the Loading dialog.

The reduction percentage to be applied is specified on the page in **Model Settings**. This percentage can differ depending on the number of floors being supported.

To cater for additional floors that are carried but that have not been included in the model an Assume extra floors supported value can be specified in the column and wall properties.

The methodology for live/imposed load reduction differs between national codes of practice:

Head Code: EC or BS

Levels can be designated either as "to be" or "not to be" included in the determination of the load reductions through Count floor as supported check boxes for each level in the column and wall properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular column or wall. The moments from fixed ended beams framing into a column or wall are never reduced.

For more information, see: and

Head Code: ACI/AISC

Before undertaking member design, Live and Roof Live loads are multiplied by a reduction factor R for roof live loads and other live loads independently. This reduced load is then used in combination to create design forces. The reduction factor is related to the tributary area of load carried by the particular member and also the K_{LL} factor, where K_{LL} comes from Table 4-2 in ASCE7-05/ ASCE7-10 or Table 4.7-1 in ASCE7-16 .

Essentially:

- Interior and exterior cols (no cantilever slabs) K_{LL} = 4
- Edge and interior beams (no cantilever slabs) K_{LL} = 2
- Interior beams (with cantilever slabs) K_{LL} = 2
- Cantilever beams K_{LL} = 1
- Edge cols (with cantilever slabs) K_{LL} = 3
- Corner cols (with cantilever slabs) K_{LL} = 2
- Edge beams (with cantilever slabs) K_{LL} = 1

As it is not possible to assess where cantilever slabs are and what they are attached to - you are able to change the K_{LL} factor for all column/wall stacks and beam spans as required in the Properties Window.

The default settings are:

- Columns/walls = 4
- Beams = 2
- Cantilevers = 1

For more information, see:

Head Code: AS

Before undertaking member design, imposed loads are multiplied by a reduction factor $\psi_{a.}$

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.

For more information, see:

Overview of pattern loading

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

3D analysis

A traditional frame analysis of the complete structure is always the first analysis that Tekla Structural Designerperforms in any combined analysis and design. This analysis generates a first set of results for the design of beams, columns and walls.

First Order or Second Order Analysis?

You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the Analysis page of the Design Options dialog. The actual options that are presented will vary depending on the design code being worked to.

Linear or Non Linear Analysis?

The choice of linear or non-linear analysis is made automatically:

- if the model has entirely linear properties a linear analysis is performed,
- else if any non-linear properties are detected a non-linear analysis is performed.

Grillage chasedown analysis

We know from experience that 3D Analysis on its own does not give the gravity results engineers have traditionally used or want - staged construction analysis reduces but doesn't resolve this. Therefore, Design All (Static) will also automatically undertake a grillage chasedown analysis provided concrete members exist anywhere in the model (beams, columns, or walls).

The Solver Model used for Grillage Chasedown emulates a traditional analysis and establishes an alternative second set of design forces for beams, columns and walls.

See also

Accounting for lateral loading in chasedown results (page 152)

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 two way spanning slabs on beams - the results from this analysis being required for the design of these slabs.

By default the same results are also used to generate a third set of design forces for the cast-in-place beams, columns and walls. (Whether the results are used for this purpose is controlled by a setting under **Design > Settings > Concrete > Cast-in-place > Beam, Column, Wall > General Parameters**).

A significant consideration when opting to design for the FE chasedown results is that the slabs will tend to carry a significant proportion of the load direct to the columns.

Consequently, for beam design in particular, it is unlikely that an FE chasedown could result in a more critical set of design forces than would be already catered for by the Grillage chasedown.

NOTE If duplicate levels have been specified in the Construction Levels dialog separate sub models are created and analyzed for the source and every duplicate level. This ensures that the increasing load carried by the vertical members in the lower sub models is catered for. In turn this can cause small differences in the analysis results (and consequently the design) for these sub models.

See also

Accounting for lateral loading in chasedown results (page 152)

Reasons for performing chasedown analyses

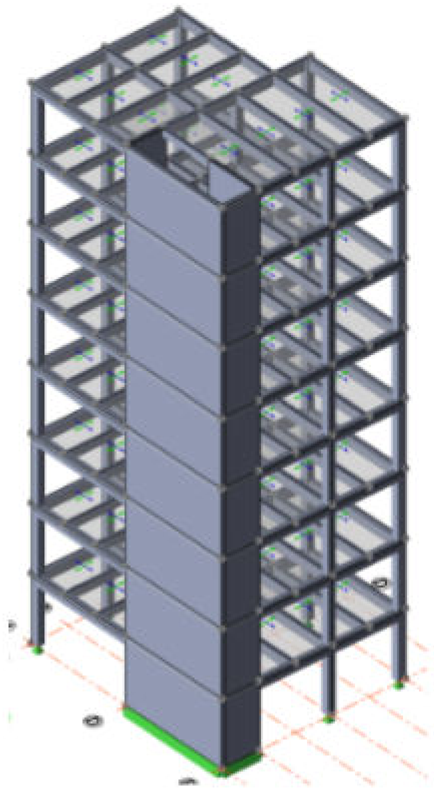
As the 3D Analysis determines a set of design forces which can then be used to design the members - why does Design (All) then carry on and do other "Chasedown" analyses?

The answer relates to expectation of results - in particular with regard to the following:

- Sway Effects under pure gravity loading
- Transfer beam designs
- Differential axial deformation

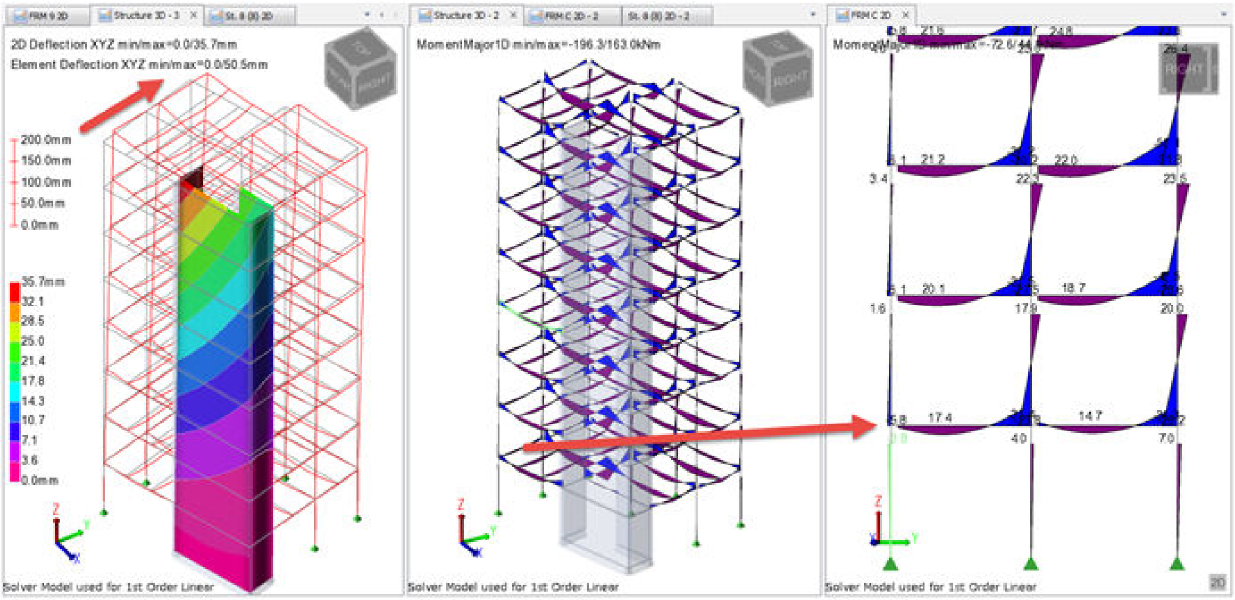
Sway Effects under pure gravity loading

Consider this 8 story model, when you click Design All, grillage and FE chasedown solver models are created in addition to the 3D Analysis model.

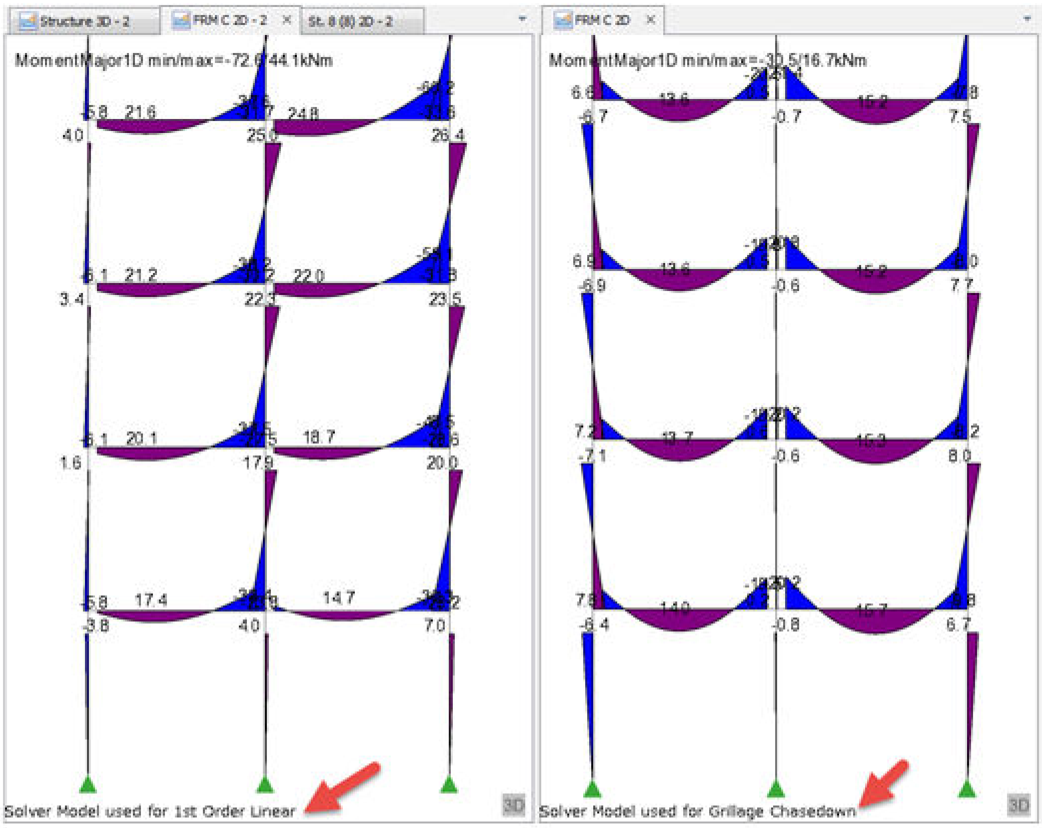


The Identified end frame below is resisting the sway - the design moments shown would be given by any analysis software.

It is an extreme example - but this result does not fit with traditional engineering expectation



The 3D Analysis results (below left) can be compared with those from Grillage Chasedown (below right).



The Grillage Chasedown results are more in line with expectation:

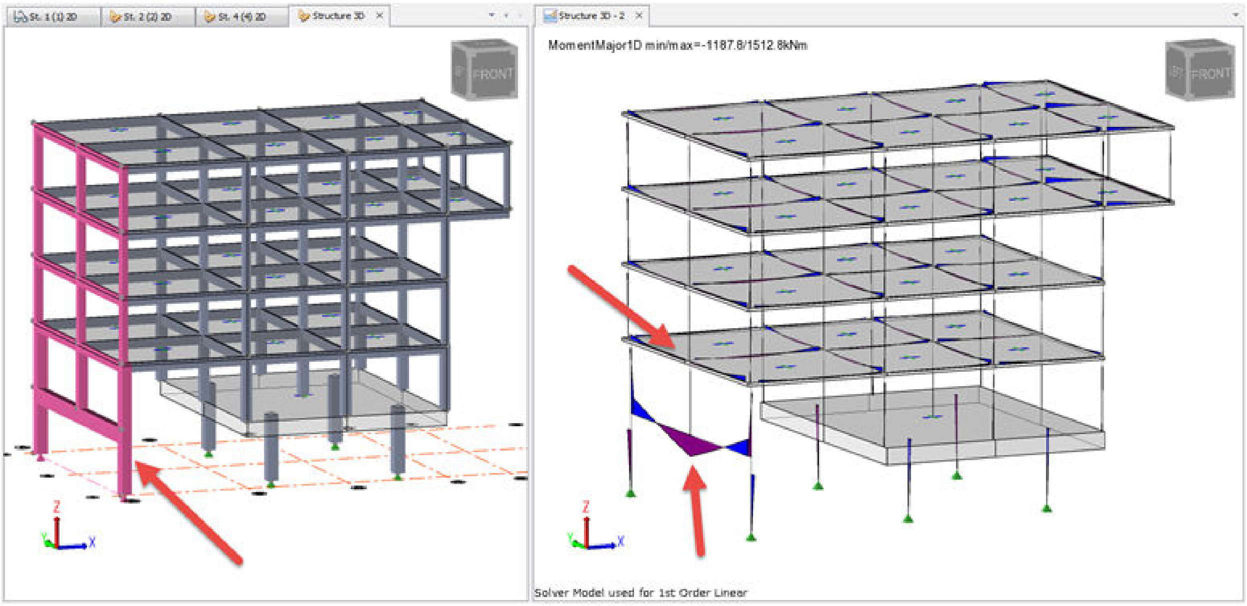
- -ve moment at LH end
- Similar moment profile at every level

Another example of why chasedown is wanted follows below....

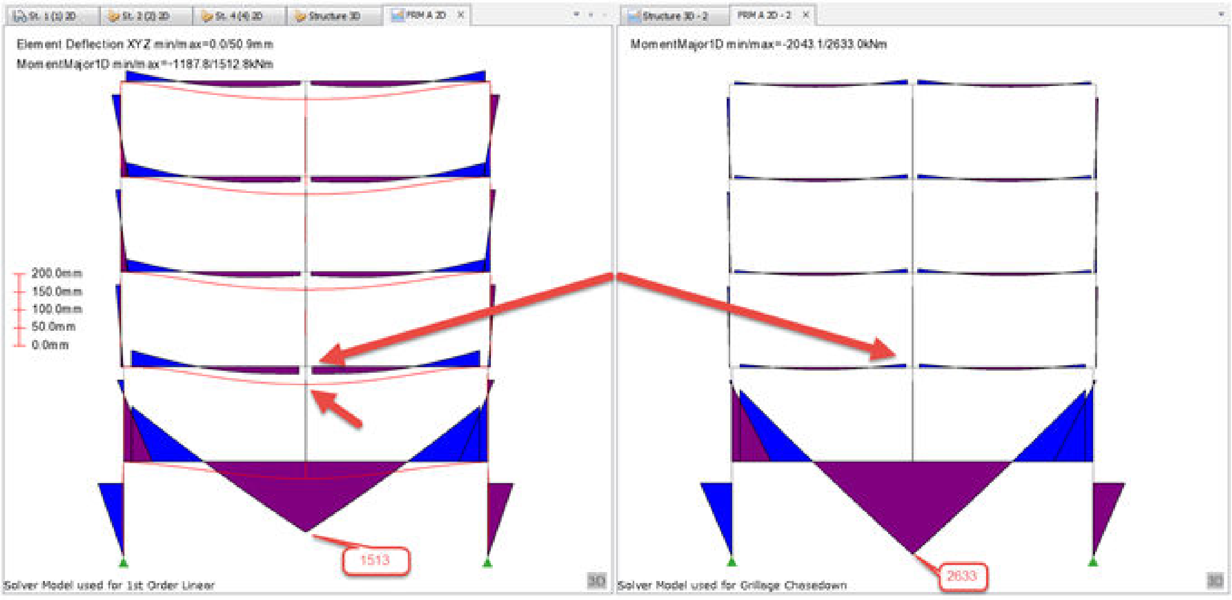
Transfer beam designs

Focus on the transfer beam in the highlighted frame.

At first glance the results look ok, but lets look more closely...



The 3D Analysis results (below left) can be compared with those from Grillage Chasedown (below right).



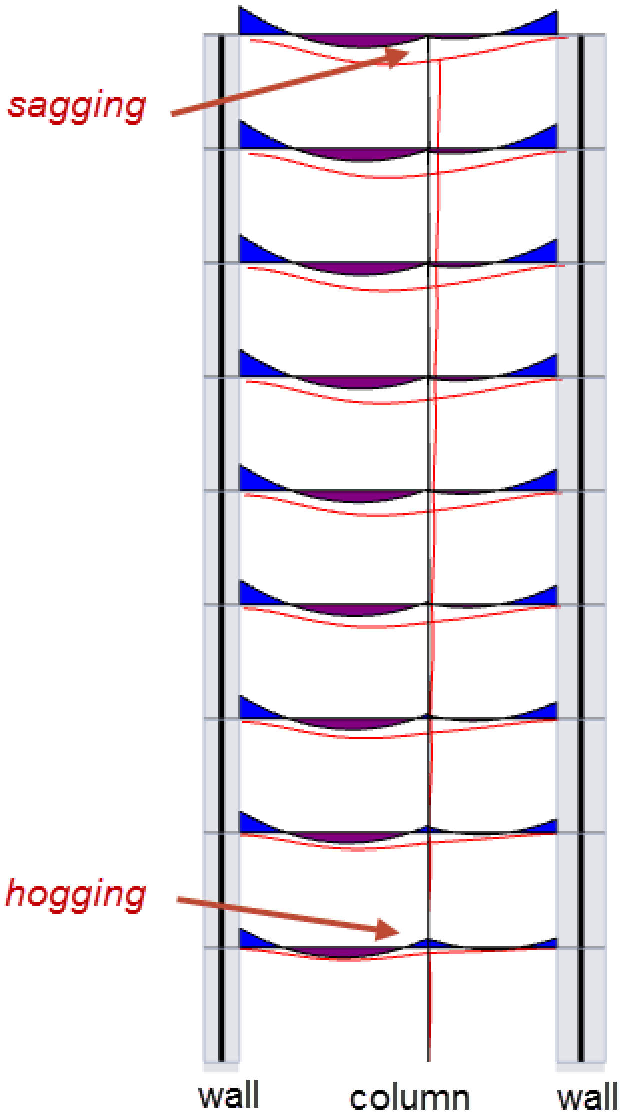
- 3D Analysis - frame deflects and loads are shared according to stiffness.
- Grillage chasedown - loads are collected floor by floor and then applied to the transfer beam - much higher moment in the transfer beam. Once again this is more in line with traditional expectation.

A third example of why chasedown is wanted follows below....

Differential axial deformation (axial shortening)

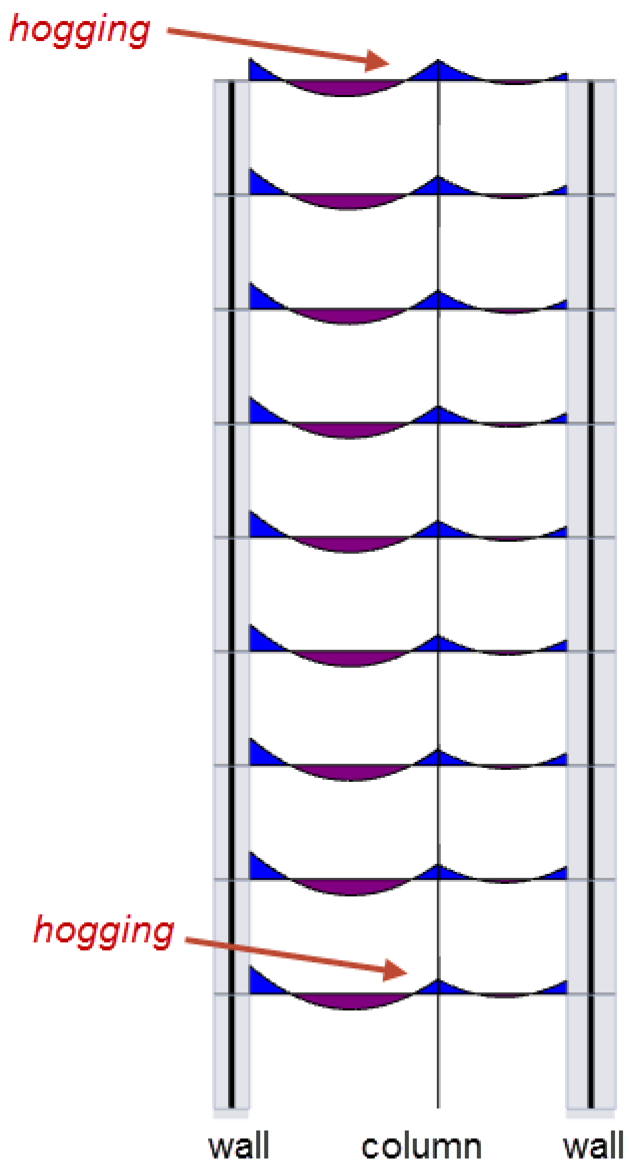
In this case a two span beam sits on walls at each end and a column in the center.

3D Analysis Results:



- Column is more highly stressed than walls and deflects downwards shedding load back to the walls.
- You end up with completely different moment profiles on beams that would traditionally be considered and designed as identical.

Grillage chasedown Results:



• the sub-models at each level are almost identical and you get very similar results over the height of the building.

Findings from the above examples

- Grillage chasedown emulates a more traditional design style where continuous beams or sub-frames are considered in isolation.
- The 3 examples show where 3D Analysis gives results that are not liked (based on traditional design expectation).
- But once you start to think about it you may conclude that actually, the 3D Analysis result should not be ignored.

- By running Design All each analysis type is performed and members are simultaneously designed for each set of results.

NOTE Deliberately extreme examples have been used to illustrate these effects and in real models the differences between the sets of results might not be as dramatic.

Accounting for lateral loading in chasedown results

It is important to note that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loadcase (wind / EHF) or from the direct analysis of any combination that includes a lateral loadcase are not valid.

Therefore in order to generate the design forces mentioned above, the chasedown analysis results are merged with the 3D building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
    a. Subtract the 1st order linear building analysis result multiplied by the relevant load factor
    b. Add the chasedown result multiplied by the relevant load factor
4. For results with Imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loadcases or combinations is not required.

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

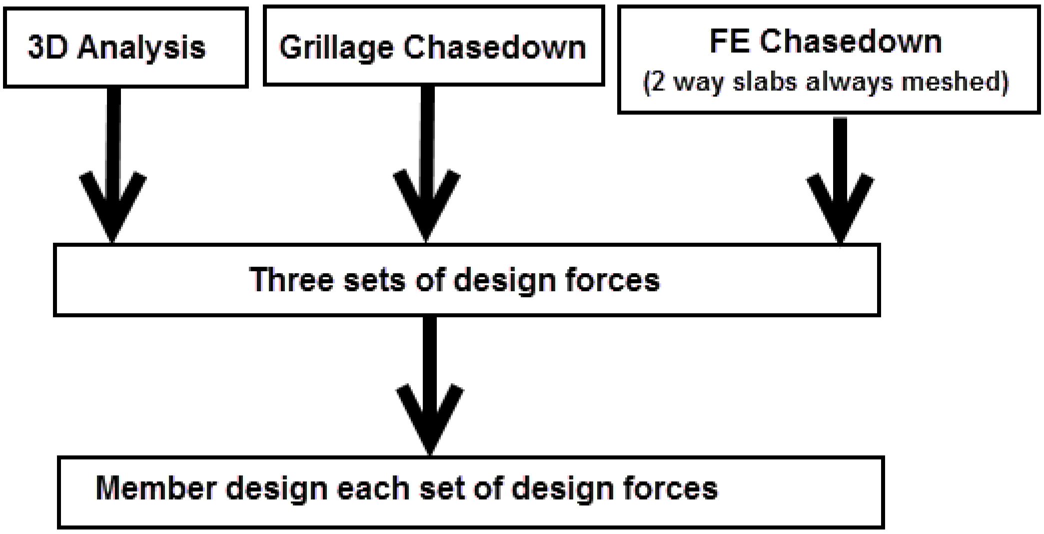
Member design stage of the combined analysis and design process

The final step in the Design All (Static) process is member design for all members for all available sets of design forces.

Steel Member Design Forces

The 3D Analysis results are the only results set used in steel member design.

Concrete Member Design Forces



Up to three sets of analysis results will exist for concrete member design as follows:

- **3D Analysis** results will always be used to design all beams, columns and walls.
- **Grillage Chasedown** results will exist for gravity loadcases if the model contains any concrete beams, in which case they will also be used to design all beams, columns and walls.
- **FE Chasedown** results for gravity loadcases will also exist if the model contains 2-way spanning slabs.

  Concrete beams can be designed for this set of results by checking the 'Design Beams for FE Chasedown analysis results' box under Design > Design Options > Concrete > Beam > General Parameters

  Columns and walls can also be designed for this set of results by checking similar boxes on their respective General Parameters pages.

Reset Autodesign

On completion of your chosen design process, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via Design Options > Autodesign.

Design Review

On completion of the Design All (Static) process the Review View and Review toolbar open automatically.

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

See also

Accounting for lateral loading in chasedown results (page 152)

Features of the three analysis types used for static design

The following table can be used to compare the features of the three analysis types used in the static design process.

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
Examples / When useful?	Gravity and Lateral analysis (Notional/Wind/ Seismic)	"Beam & Slab" buildings	Flat slab and "Beam & Slab" buildings
Special Features	• Pattern loading • Automatic EC2 sway sensitivity assessment and sway amplification • Automatically centralised analysis wires (improved rigid offsets / rigid zones) • Option to mesh slabs in the 3D analysis	• Mimics traditional design approach (sub-frame analysis) • Pattern loading	• Mimics traditional design approach (isolated floor analysis) • Slab Pattern loading
Benefits	• Member Design considers sway and differential axial deformation effects. • Caters for slabs that	Member design based on traditional sub frame is considered simultaneously with that for 3D Analysis	• Member design based on traditional sub frame is considered simultaneously with that for 3D Analysis • Irregular slab panel design

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
	contribute to the lateral load resisting system		automatically catered for
Analysis Model	3D model of entire building: • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • all 2-way slabs meshed
Analysis Method	Whole model in one pass	Each sub model sequentially from top to bottom – chasing member loads down	Each sub model sequentially from top to bottom – chasing member loads down
Analysis Type	• First order • First order - Kamp • Second order - P-D	First order	First order
Supports	External supports as defined by the user	Ends of members above/below each sub model are automatically supported	Ends of members above/below each sub model are automatically supported
Loading	Gravity and Lateral Loads	Gravity Loads only	Gravity Loads only
Forces for design			
RC Slab	Yes– All Combs	No forces	Yes – All Gravity loadcases
RC Beam	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases
RC Column	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
RC Wall	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases
Steel/Composite Members	Yes – All Combs except patterns	Not required	Not required
Foundation design	Yes – All Combs except patterns	Yes – All Gravity loadcases	Yes – All Gravity loadcases

See also

3D analysis (page 143)

Grillage chasedown analysis (page 144)

FE chasedown analysis (page 144)

Reasons for performing chasedown analyses (page 145)

1.4 Seismic analysis and design handbook

You can find the following information in this handbook:

Introduction to seismic analysis and design

Definitions

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

Various terms used in Tekla Structural Designer's seismic processes are described below:

Code Spectra

The spectra specified in a country's loading and design codes for use in ELF and RSA analysis.

Site Specific Spectra

User defined spectra for ELF and RSA which are required for locations which use another country's loading and design codes where the code spectra are not relevant.

Base Shear Combination

Also referred to as the Effective Seismic Weight Combination (ASCE7/UBC) or the Seismic Inertia Combination (Eurocode). This combination is used for modal analysis, and in the calculation of base shears, during the Seismic Analysis Process. This combination is created and modified by the Seismic Wizard only.

RSA Seismic Combinations

These combinations are created by the Combination Generator at the end of the Seismic Wizard, but can also be modified in the standard Combination dialog. They consist of 3 kinds of loadcases: Static, RSA Seismic and RSA Torsion. The Base Shear Combination is not included in this category of combination.

Static Loadcase

Standard loadcases, e.g. "Self weight - excluding slabs", "Dead", etc., and derived cases for NHF/EHF, but no patterns.

RSA Seismic Loadcase

Two loadcases, i.e. "Seismic Dir1" and "Seismic Dir2", which cannot be edited. These are created at the end of the Seismic Wizard. being derived from information supplied in the Seismic Wizard and the results of the modal analysis. No actual loads are available for graphical display.

RSA Torsion Loadcase

These cases can be generated by the Seismic Wizard and are regenerated whenever RSA Seismic Combinations are modified.

Fundamental Period (T)

Separately for Dir 1 & Dir 2, this is either defined in the Seismic Wizard, (user value or calculated), or determined in the modal analysis for the Base Shear Combination.

Level Seismic Weight

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

Effective Seismic Weight

This is the sum of the level seismic weights for all relevant levels for the Base Shear Combination.

Seismic Base Shear

The base shear is calculated separately for Dir 1 & Dir 2, for the Base Shear Combination.

NOTE In Tekla Structural Designer the base shear is displayed when you for **Cumulative Story Shear**

Square root of Summation of Square (SRSS)

The SRSS formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{k=1}^{n} (\lambda_k^2)\ )}$$

λ = Absolute value of combined "response"

λ_k = "response" value for Relevant Mode k

n = Number of Relevant Modes

Complete Quadratic Combination (CQC)

The CQC formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{i=1}^{n} \sum_{j=1}^{n} (\lambda_i\ \rho_{ij}\ \lambda_j)\ )}$$

λ = Absolute value of combined "response"

λ_i = "response" value for Relevant Mode i

λ_j = "response" value for Relevant Mode j

n = Number of Relevant Modes

ρ_{ij} = Cross modal coefficient for i & j

Cross Modal Coefficient

This co-efficient is used in the CQC method for combining modes in RSA.

$$\rho_{ij} = \frac{8\zeta^2\ (1+\beta)\beta^{1.5}}{(1-\beta^2)^2 + 4\zeta^2(\ 1+\beta\ )^2}$$

ζ = modal damping ratio

IBC/ASCE assumed = 5% (ASCE Figs 22-1 to 6)

EC8 assumed = 5% where q accounts for the damping in various materials being different to 5% (EC8 Cl 3.2.2.5)

IS codes the user can define the level of damping and this is accounted for in the above equation.

β = Frequency ratio = ω_i / ω_j

ω_i = Frequency for Relevant Mode i

ω_j = Frequency for Relevant Mode j

Overview

All seismic codes work in a similar manner from the loading view point with relatively minor differences in terminology and methodology.

It is worth noting at the start that seismic analysis determines a set of forces for which it is expected (statistically) that if those forces are designed for and other design precautions taken (additional seismic design) then in the event of an earthquake the structure may well suffer extensive damage but will not collapse and for some categories of building should actually remain serviceable.

In Tekla Structural Designer a seismic wizard gathers all the information together to setup the requirements for a seismic analysis.

From this information a number of things are determined:

- If working to ASCE7 - the seismic design category (SDC) for the building - giving amongst other things the permissible type of analysis
- The Base Shear Combination to determine the seismic base shear in the building
- The natural frequencies of the building in two horizontal directions
- The combination of the gravity and other lateral forces with the seismic load cases

Earthquakes load a building by a random cyclic acceleration and deceleration of the foundations. These are in both horizontal directions (Dir1 and Dir2) but can also be in a vertical direction too. This ground acceleration excites the building in its natural and higher frequencies.

As a result if the building is:

- In an area of low seismic acceleration, low in height and poses limited risk to life then a gross approximation can be used in analysis - assuming a % of gravity loading is applied horizontally to the building to represent the earthquake (US codes 1%, Australian codes 10%).

- In an area of moderate to low seismic acceleration, medium to low in height and does not house a significant number of people - the predominant mode excited is the 1st mode. An equivalent lateral force (ELF) approximation can be used that applies static horizontal loading distributed up the building to mimic the shape of the 1st mode of vibration in a static analysis.
- Anything else, in an area of high acceleration, tall in height and could be holding many people or be critical post-earthquake then a "more representative" analysis method of Response Spectrum Analysis (RSA) should be used. This analysis is based upon a modal analysis considering all mode shapes in the two horizontal directions in which typically 90% of the structure's mass is mobilized.

The results from the chosen method of seismic analysis are used in combination with other gravity and lateral loadcases to design both normal members and those members in seismic force resisting systems (SFRS). These latter members need additional design and detailing rules to ensure they resist the seismic forces that they subjected to during an earthquake. If working to ASCE7, the extent of these rules are determined by the SDC noted above (the higher the demand, the ‘better’ the SFRS that will be required).

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

Seismic Wizard

In Tekla Structural Designer the Seismic Wizard sets up the information required for seismic analysis - the main parameters to be input being:

- Ground acceleration - strength of the earthquake
- The Importance Level (or Importance Class) of the building - the use to which the building is being put - typically
  - I= very minor, farm and temporary buildings,
  - II= general buildings occupied by people,
  - III = buildings occupied by a large number of people
  - IV = critical buildings with a post-disaster function eg hospitals, police stations, fire stations and buildings along access route to them)
- The ground conditions upon which the building is founded (typically Hard Rock, Rock, Shallow soil, Deep Soil, Very Soft Soil)
- Building height - for low buildings the first mode is totally dominant in taller buildings other modes become significant
- Plan and vertical irregularities in the building

From this input the Seismic Wizard determines the seismic design category for ASCE7, and also the elastic design response spectrum to be used for the building.

Additionally the Wizard sets up the Base Shear Combination - the combination of loads likely to be acting on the building when the earthquake strikes.

Vertical and Horizontal Irregularities

There are typically 5 types of horizontal irregularity and 5 types of vertical irregularity - all are defined to pick up structures that have lateral framing systems and shapes in plan that will preclude the structure naturally developing a simple first mode of vibration. Since this is a basic assumption of ELF - the presence of these irregularities may preclude the use of ELF.

Torsion

When a structure's center of mass at a level does not align with the position of the center of rigidity then torsion is introduced in the structure at that level when an earthquake excites the structure. To account for this, there are three types of torsion potentially applied to levels with non-flexible diaphragms during a seismic analysis

- Inherent torsion - in a 3D analysis when the center of mass and center of rigidity at a level do not align, this is taken account of automatically
- Accidental torsion - to allow for the "miss positioning" of loads in a structure, an additional eccentricity of usually 5% of the structure's width in all relevant directions - this is accounted for with a torsion loadcase in the analysis
- Amplified accidental torsion - structures with certain SDCs and certain horizontal irregularities require an amplified accidental torsion to allow for extra effects

Modal Analysis

Using the Base Shear Combination, a modal analysis is run for two purposes:

- the natural frequencies of the building in two directions are determined to assist with the calculation of the seismic base shear that in turn is used to determine the distribution of applied loads up the building for an ELF analysis
- the frequencies and mode shapes of the building are determined that need to be included in an RSA analysis so that typically 90% of the mass in the building is mobilised during the RSA analysis

% of Gravity Load Method

The % of gravity load method is used as a means of a gross approximation of the earthquake and is only used in situations where seismic effects are considered to be low. This loadcase is combined with the relevant load factor with other gravity and lateral loadcases to determine the design forces and moments to be considered in conventional design.

NOTE This method is not applicable when working to Eurocodes.

Equivalent Lateral Force Method

The ELF method assumes that the first mode shape is the predominant response of the structure to the earthquake.

Based on the natural frequency and the Base Shear Combination, a total base shear on the structure is determined and this is then set up as a series of forces up the structure at each level (in the shape of an inverted triangle) and these deflect the structure in an approximation to the shape of the first mode.

The resulting seismic loadcases are combined with the correct combination factors with the other gravity and lateral loadcases 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 loadcases 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 loadcases in the **seismic combinations** to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

Summary of RSA Seismic Analysis Processes

RSA Seismic Analysis (1st or 2nd order) is run as a stand-alone analysis from the **Analyze** tab, or as part of the Design (RSA) process. In the latter, the use of 1st order or 2nd order is set for the static analysis is set via **Design Settings > Analysis**.

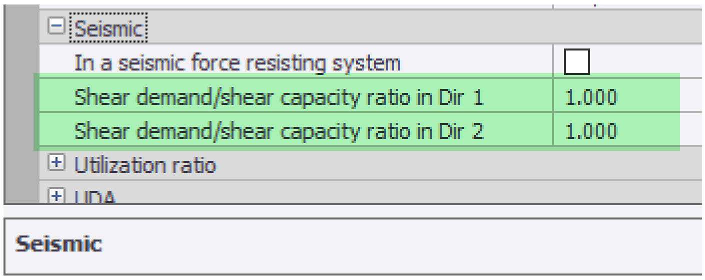
The process consists of the following steps:

Step	Process	Description
1	Model Validation	Run to detect any design issues which might exist. This is similar to standard model validation but also checks: • Base Shear Combination must exist • At least one RSA Seismic Combination must exist including at least one RSA Seismic Loadcase.
2	Modal Analysis	A 1st order modal analysis for the Seismic Inertia Combination only, which returns the standard results for that analysis type, but also the fundamental periods for directions 1 & 2.
3	Pre-Analysis for Seismic	Performs calculations for RSA Torsion Loadcases. The seismic weight and seismic torque are both calculated at this stage.
4	Static Analysis	1st Order Linear or 2nd Order Linear analysis is performed for all RSA Seismic Combinations and all their relevant loadcases, i.e. this includes Static Loadcases, but does not include RSA Seismic and RSA Torsion Loadcases.
5	RSA Analysis	A set of results is generated for a sub-set of vibration modes for each RSA Seismic Loadcase.
6	Accidental Torsion Analysis	Analysis of any RSA Torsion Loadcases that exist.

Seismic Drift

Seismic drift is assessed on a floor to floor horizontal deflection basis and there are limits for acceptability of a structure.

When working to the ASCE7 code, the engineer can directly define the shear demand / capacity ratio (beta) for columns and walls. The default value of 1.0 could be over-conservative. This option is located in the “Seismic” group of member properties, and can be set separately for 'Direction 1' and 'Direction 2' and for each stack/panel, as shown in the picture below.



- The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

The building's overall seismic drift status is displayed in the Design branch of the Status Tree in the Project Workspace.

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

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

Design Coefficients and Factors (ASCE7/UBC)

Typically three factors are determined based on the lateral force resisting systems in the structure and which account approximately for the inelastic response that occurs during the earthquake which is not accounted for directly in the analysis.

- The response modification coefficient which affects the seismic base shear.
- The overstrength factor accounts for the reinforcement steel yielding overstrength and is utilized in the concrete beam and column capacity calculations.
- The deflection amplification factor which is used in the calculation of seismic drift.

Limitations of Seismic Design

The following limitations apply:

- Where seismic design and detailing is required this is only supported in Tekla Structural Designer for the ACI/AISC and Indian Head Codes.
- It is up to the user to assess whether framing is split horizontally or vertically, whether system specific requirements need to be assessed - like mixed system moment frames, whether diaphragms are rigid or flexible - in all instances, the user will need to make the necessary adjustments for the situation in hand. The software does not handle these situations automatically.

- Linear modal analysis with non-linear element properties - currently the modal analysis is limited to a linear model so all non-linear elements are set to be linear.
- ELF can be run as 1st or 2nd order analyses, however if the Fundamental Period is determined using modal analysis the modal analysis is always run as 1st order.
- The RSA analysis itself is a 1st order linear process. For the 2nd Order RSA Seismic analysis, the peak responses are enveloped around the static results for 2nd Order Linear Analysis. Thus when the analysis is set to 2nd order in Design Options, in real terms the results are actually RSA Seismic + 2nd Order.
- Structures with linear members and supports are run using linear analyses. Structures with non-linear supports and /or members are run as non-linear in ELF but linear in modal and RSA.
- We do not consider any of the standard methods for structurally accommodating seismic actions - e.g. base isolators, damping systems
- We do not consider more accurate methods of analysis like time history analysis. As a result there are some situations with very tall buildings and very irregular buildings that Tekla Structural Designer does not cater for.
- Diaphragms - rigid and semi-rigid diaphragms (meshed floors) are available and it is the user's responsibility to ensure they are modelled suitably. Rigid diaphragms are only allowed in limited circumstances and, so called, 'flexible diaphragms' can be modelled as semi-rigid diaphragms with extremely low stiffness. Force transfer into and out of the diaphragm is not checked.
- Collector elements - no checks included.
- Non-structural elements - no checks included.

Specific limitations of steel seismic design

- Coincident V & A braces giving X type are beyond scope
- Various other requirements not checked
    - e.g. V & A braces are restrained at their intersection
    - e.g. tension braces resist between 30% and 70% of total horizontal force
    - e.g. forces in restraining members not checked
- Connections are not designed

A specific limitation of RC seismic design

- While checks can be done in both directions they are direction specific where applicable - there are no biaxial checks.

Seismic force resisting systems

Available SFRS types

The seismic design requirements for a particular member are based upon which Seismic Force Resisting System (SFRS) the member forms part. Hence, in Tekla Structural Designer you can set appropriate members as part of one of the following systems:

SFRS types included for steel members

Moment Frame Systems

- Special Moment Frames (SMF)
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Braced Frame and Shear Wall Systems

- Ordinary Concentrically Braced Frames (OCBF)
- Special Concentrically Braced Frames (SCBF)
- Special Concentrically Braced Frames (SCBF)

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types available for concrete members

Moment Frame Systems

- Special Moment Frames (SMF) - limited design.
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Walls

- Special reinforced concrete structural wall - limited design
- Intermediate precast structural wall - no seismic design performed
- Ordinary reinforced concrete structural wall

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types excluded

Everything else

- e.g. Eccentrically Braced Frames

Members allowed in the SFRS

The following member types are allowed to be part of a SFRS in Tekla Structural Designer

- Steel columns
- Steel beams
- Steel braces
- Concrete columns
- Concrete beams
- Concrete walls

The following member types are not allowed to be part of a SFRS in Tekla Structural Designer

- Any timber, cold-formed, general
- All other "Characteristics", e.g. steel joists, truss members, purlins
- Composite members
- Plated, Westok, Fabsec, concrete filled, concrete encased – selectable but no design (i.e. only rolled)

Assigning members to the SFRS

The choice of members to be part of a SFRS is entirely the engineer's responsibility.

- It is expected that all members in a frame are assigned to the SFRS.
- The assigned members should be specified to act in building Direction 1 or Direction 2

Special Moment Frames - assigning connection types at steel beam ends

For SFRS comprising of steel SMF it is necessary to ensure that the beams fail before the columns. To this end, an assessment of plastic moment capacity is made at each floor. The moment capacity is dependent upon the position of the plastic hinge, typically $(d_{col} + d_{beam})/2$. These locations can be selected appropriate to each beam end either in the beam properties.

Options are provided as follows:

- Plastic hinge position at start

  • Plastic hinge position at end

Either, $(d_{col} + d_{beam})/2$ (default)

or, $d_{col}/2 + L$

L = 0 (default)

Validation of the SFRS

There is only a small amount of validity checking for an SFRS that can be performed automatically; it remains in large part the user's responsibility to ensure that each SFRS is defined appropriately.

The following validation conditions are however detected:

1. Any A or V brace in a Seismic Force Resisting System must have the A or V as vertically released. A warning is provided in validation if this is not the case.
2. X type bracing is defined as more than one V or inverted V (A) type brace pair on the beam. When more than one A or V brace pair is detected, the additional checks required by AISC 341-05 and AISC 341-10 given in Section 8.3 are out of scope. This situation is not detected during validation, but it is identified in the seismic design, so that the beam is given a "beyond scope" status.
3. The use of K braces is not allowed in AISC 341. An error is provided in validation.
4. Tension only braces were permitted to the 05 version but had no additional requirements. In the 10 version they are only are allowed for OCBF. Thus, an error is provided in validation when a tension only brace is set as part of a SCBF and the code is the 10 version. (The same validation is also applied to compression only braces.)
5. If seismic loadcases are included in combinations and there is not at least one member assigned to each of Direction 1 and Direction 2 then a warning is issued.

Auto design of SFRS members

All SFRS members can be auto-designed to the conventional design requirements both for seismic and non-seismic combinations;

During the automatic design procedure, besides the conventional auto-design, and for seismic combination results only:

- Steel members in the SFRS are checked for seismic provisions;
- Normal weight reinforced concrete members in the SFRS are auto-designed for seismic provisions

Seismic design methods

To design members and walls for the results of a seismic analysis:

- For geographic regions categorised as "low seismicity" it is acceptable to assume "ductility class low" applies. Under these conditions the seismic analysis results can be fed into "conventional" design, see Seismic analysis and conventional design (page 169)

- Certain conditions (e.g. "high seismicity") necessitate that a "seismic" design is performed - additional design and detailing requirements have to be applied in this situation, see Seismic analysis and seismic design (page 170)

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

Seismic analysis and conventional design

ELF seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the ELF method is summarised as follows:

1. Modelling
   - No additional seismic modelling requirements
   - There is no need to assign members to a SFRS
2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required

   b. Set the site class (ASCE7), or ground type (Eurocode)

   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method

   d. Set up the relevant seismic combinations
3. Static Design

   Run the Design (Static):

   - the results of the ELF seismic combinations are fed into the design and considered in the same way as other combinations.
4. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the RSA method is summarised as follows:

1. Modelling
    - No additional seismic modelling requirements
    - There is no need to assign members to a SFRS
2. Loading and Analysis

    Run the Seismic Wizard to:

    a. Determine building height to the highest level and adjust it if required.

    b. Set the site class (ASCE7), or ground type (Eurocode)

    c. Select RSA method of analysis

    d. Set up the relevant seismic combinations
3. Static Design

    Run the Design (Static):

    - Results of the static combinations are fed into conventional design.
4. RSA Seismic Design

    Run the Design (RSA):

    - Results of the RSA seismic combinations are fed into conventional design and considered in the same way as the static combinations.
    - No additional seismic design is required
5. Calculation Output
    - A Seismic Design Report is available
    - Drift limitations are checked

See also

Seismic analysis and design handbook (page 156)

Seismic analysis and seismic design

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

These requirements vary depending upon the 'sophistication' of the SFRS. For example OMF have less stringent requirements than SMF.

ELF seismic analysis and seismic design

The overall modelling, analysis and seismic design process using the ELF method is summarised as follows:

1. Modelling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type.

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE7), or ground type (Eurocode)

   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method.

   d. Set up a modal mass combination

   e. Set up the relevant seismic combinations

3. Static Design

   Run Design (Static) to:

   a. Conventionally design all members for all non-seismic (gravity and lateral) combinations

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

   c. Perform additional seismic design for the seismic combinations for those members assigned to a SFRS

4. Calculation Output

   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and seismic design

The overall modelling, analysis and seismic design process using the RSA method is summarised as follows:

1. Modelling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type. These members will be designed and detailed according to the seismic provisions.

2. Loading and Analysis

   Run the Seismic Wizard to:

a. Determine building height to the highest level and adjust it if required.

b. Set the site class (ASCE7), or ground type (Eurocode)

c. Select RSA method of analysis

d. Set up a modal mass combination

e. Set up the relevant seismic RSA combinations

3. Static Design

Run the Design (Static) to:

- Conventionally design all members for all non-seismic (gravity and lateral) combinations .

4. Modal Analysis

At this point it is recommended that you run a 1st order modal analysis in order to confirm the model converges on a solution, (until it is able to do so, it is pointless proceeding with a full RSA Seismic Design).

5. RSA Seismic Design

Run the Design (RSA) to:

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

b. Perform additional seismic design for the RSA seismic combinations for those members assigned to a SFRS

6. Calculation Output

- A Seismic Design Report is available
- Drift limitations are checked

See also

Seismic analysis and design handbook (page 156)

1.5 Steel design handbook

To get started with designing steel structures in Tekla Structural Designer see:

- Combined analysis and design choices for steel structures (page 173)
- Steel member autodesign (page 174)
- Steel member design groups (page 176)
- Steel beam design (page 178)
- Composite beam design (page 193)
- Steel column design (page 218)

Combined analysis and design choices for steel structures

Gravity design

For many steel structures a separate gravity design is not necessary and you can simply proceed directly to a **Static design**.

In some circumstances however, you may find it preferable to adopt a two-stage workflow in which gravity members are sized prior to the full design.

The benefits of this are:

- For larger models it can significantly speed up the analysis and design time
- If lateral resisting systems are not modeled, then mechanisms can result and the analysis may not find a solution. Since these lateral resisting systems are perhaps unknown early in the project (or insufficient systems have been provided), running a gravity design has the added benefit of automatically fixing column nodes (not attached to a diaphragm) horizontally, which stabilizes the model.

The gravity design stage enables you to design all members (and hence size those members with the Autodesign property checked), for only the active gravity combinations (this will include the Construction Stage combination). As an example, beams within a rigid floor diaphragm are unaffected by lateral load, and hence they can be sized using only the gravity combinations.

Gravity design is initiated by clicking **Design Steel (Gravity)**.

After the gravity design has been completed, by default all steel members are reset to check design mode.

You may then decide to reset certain members to auto design e.g. columns and beams in 'moment frames', columns and braces in vertical or horizontal braced bays. In such cases, when the full static design is performed member 'pre-sizing' will take place and for members resisting lateral loads the new section size will be used if it is larger than that which resulted from the gravity design.

Static design

Full static design is initiated by clicking **Design Steel (Static)** or **Design All (Static)** on the **Design** toolbar.

All beams, columns and braces that do not have the **Gravity only** property checked, are designed or checked for all active combinations; Gravity only design members are designed or checked for the active gravity combinations only.

As part of the full design process a 3D Analysis is performed, for which you must select (via Design Options) the analysis type. The choice of analysis type will depend on the code being designed to.

Designing individual members for gravity only

By default, when a full static design is run, all members are designed for both gravity and lateral combinations.

You can however tell Tekla Structural Designer to only consider gravity combinations for the design of specific members. This is achieved by checking **Gravity only** in an individual member's properties, i.e.

- **Gravity only** checked - designed for gravity combinations only
- **Gravity only** unchecked - designed for all combinations types: gravity, lateral and seismic

Setting columns that do not help resist lateral loads to be designed for gravity loads only, will reduce the overall design time.

Engineering judgement will be required when identifying members as being 'gravity only'.

For example:

- if an inclined braced member connects to a beam, axial force in the brace (from both gravity and lateral loads) puts the beam into bending and therefore the beam should be designed for both gravity and lateral loads.
- potentially, beams in a sloping roof would also need to be designed for both gravity and lateral load

NOTE If a composite beam is identified as being designed for both gravity and lateral combinations, they are only designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered. Hence a warning is provided, if the ignored loads exceed a preset limit.

Steel member autodesign

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

- When **Autodesign** is not selected (i.e. check mode), you assign your desired section size to the member and Tekla Structural Designer determines if the section is sufficient.

- When **Autodesign** is selected the section type to be used is specified from a Design Section Order and Tekla Structural Designer attempts to automatically determine a suitable section.

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

  - Size constraints (page 175)
  - Design section orders

NOTE If a member type has been set to be designed using Design groups, then if at least one member of the group is set to autodesign, the whole group will be automatically designed.

Size constraints

When undertaking an Autodesign, the Size Constraints property allows you to ensure that the sections proposed by Tekla Structural Designer match any particular size constraints you may have.

For instance:

- Minimum width - you may want to ensure a minimum flange width to satisfy bearing requirements for any supported beams. To achieve this, you would enter a value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than the specified minimum width for the design of the beam.
- Maximum depth - you may want to ensure that the maximum beam depth is specified to ensure that it fits within a floor zone. To achieve this, you would enter a value as the Maximum depth and Tekla Structural Designer would not consider sections with depths greater than the specified value for the design of the beam.
- Max span/depth ratio (beams only) - you may want to set a max span/depth ratio limit above which autodesign will reject sections that are of insufficient depth.
  - The depth is taken as the depth from the section properties - for single and double angles this is the vertical leg length.
  - Limit is not applied to any member which has rotation.

NOTE Size constraints only work when undertaking an **Autodesign**

Applying a size constraint to an element

You can control which sections in a design order list are considered, by following the procedure below.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window
2. Expand All spans/stacks or the individual span/stack as required to reveal **Size Constraints**
3. Expand **Size Constraints** and set the **Max depth**, **Min depth**, **Max width**, **Min width**, or **Apply max/span/depth ratio** as required.

When an **Autodesign** is performed, the section selected from the section order list will fall within any size constraint limits set.

Steel member design groups

Why use steel design groups?

Steel members are not automatically placed into design groups. They are however, automatically put into groups, primarily for editing purposes. In this way, individual groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

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

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

NOTE Grouped design is optional and can be activated if required, via Design Options> Design Groups

NOTE A fixed set of rules are initially used to determine the **automatic member grouping**: for example beams are designated as either direction 1 or direction 2, are of a similar span, material grade and construction type, columns have the same number of stacks etc. **You can manually edit groups.**

What happens in the group design process?

When the option to design a specific steel member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the section size so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the section size is adequate for each and if this proves not to be the case, it is

increased as necessary and the revised section size is copied to all members in the group.

- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

NOTE When performing the design of a group of composite beams the changes in section size may mean that for some beams the existing stud layout will no longer remain adequate. To ensure the safe and rational design of the studs, the stud design routine is re-run at the end of the group design for all beams with the stud auto-layout active in the beam properties.

Steel design group requirements

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

Member type	Design group rules
Steel beam	• A beam element may only be in **one** design group • All beam elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled, • All beam elements in the group must have the same material properties Automatic grouping uses the following rules to automatically generate the groups, however, when manually editing the groups these are not enforced: • All beam elements in the group are of a similar span. • All beam elements have a similar direction i.e. beams are grouped as Direction 1 or 2 if they lie +/- 45 degrees to the directional axis
Steel column	• A column element may only be in **one** design group • All column elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled, • All column elements in the group must have the same material properties • All column elements in the group must have the same number of stacks

Group management

Automatic Grouping

Steel members are grouped automatically.

In Model Settings the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for automatic grouping purposes.

Manual/Interactive Grouping

It is recommended that you manually edit the groups for steel members, since members can be placed into any group without the need for similar span lengths or directional requirements. For example, edge beams, primary beams, secondary beams etc. could have different span length and directions, and be placed into the same design group using manual editing.

NOTE A member can only be placed into a single design group.

To manually reassign a member to a different group, locate it in the **Project Workspace**,**Groups tab**and drag it to a different group

NOTE A cross will appear if the element does not meet the Group requirements.

Regroup ALL Model Members

If you have made changes in Design Options that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Steel beam design

Click the following links to find out more:

- Steel beam overview (page 179)
- Steel beam fabrication (page 179)
- Steel beam restraints (page 184)
- Deflection limits (page 185)
- Camber (page 186)
- Instability factor (page 187)
- Beam web openings (page 187)
- Steel beam torsion (page 192)
- Fire check (Eurocode only) (page 193)

Steel beam overview

Tekla Structural Designer will design steel beams for an international range of doubly symmetric I-sections, C-sections, rectangular and square hollow sections, single angles, double angles and tees for many different countries and also for many specific manufacturers. Plated beam design is also available for some head codes.

Steel beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

In addition to major axis bending, minor axis bending and axial loads are also considered.

In its simplest form a steel non-composite beam can be a cantilever, or a single member between supports to which it is pinned.

It can also be a continuous beam consisting of multiple members that do not, with the exception of the remote ends, transfer moment to the rest of the structure. Each span of a continuous beam can be of different section size, type and grade.

At the remote ends of the beam there are a number of options for the end fixity depending upon to what the end of the beam is connected. These are:

- Free end
- Moment connection
- Pin connection
- Fully fixed
- Partially fixed (% of 4EI/L)
- Partially fixed (linear spring)

The beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Steel beam fabrication

Fabrication types summary - all head codes

The fabrication types that can be designed as non-composite steel beams in Tekla Structural Designer are dependent on the design code.

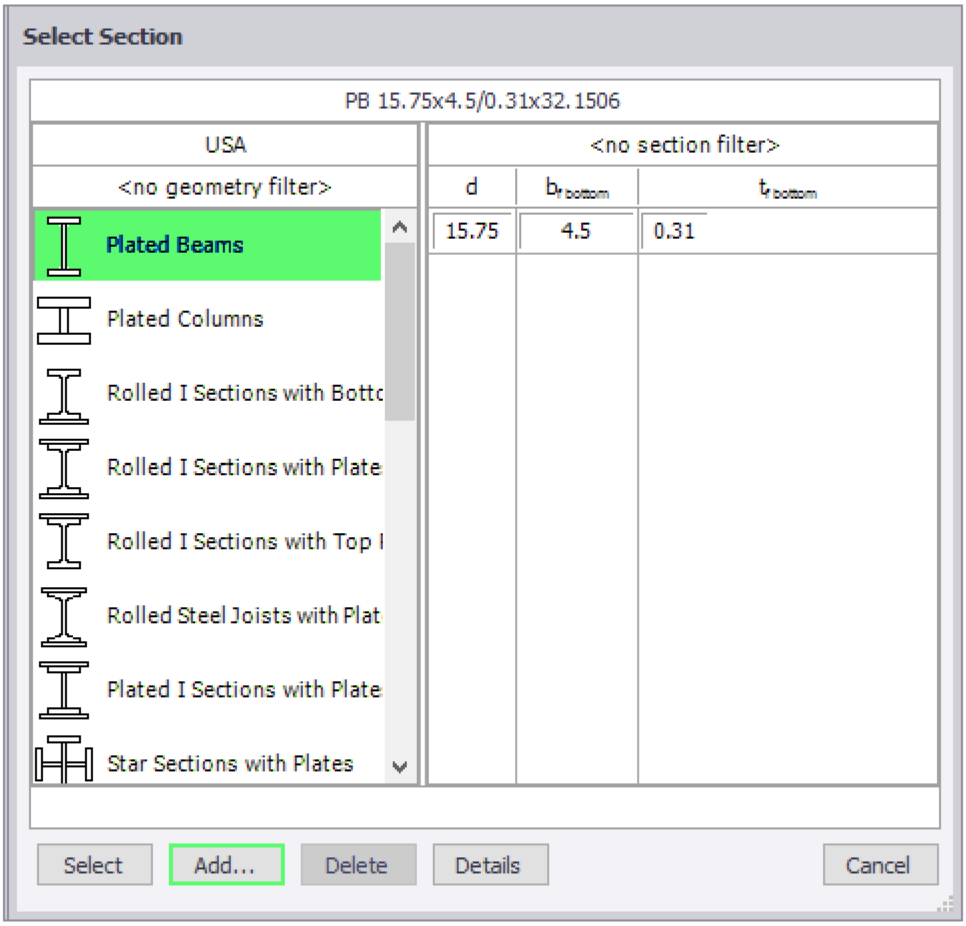
Refer to the following table for details.

	Rolled	Plated	Westok cellular	Westok plated	FABSEC	DELTABEAM
AISC	Yes	Yes	No	No	No	No
Eurocode	Yes	Yes	Yes	Yes	No	No

BS	Yes	Yes	Yes	Yes	No	No
IS	Yes	Yes	No	No	No	No
AS	Yes	Yes	No	Yes	No	No

Plated beams - AISC head code

When the Fabrication type is set to Plated, only **Plated Beams** can be designed to the AISC 360-05, 360-10 and 360-16. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.



NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for plated sections once these have been added to the sections database.

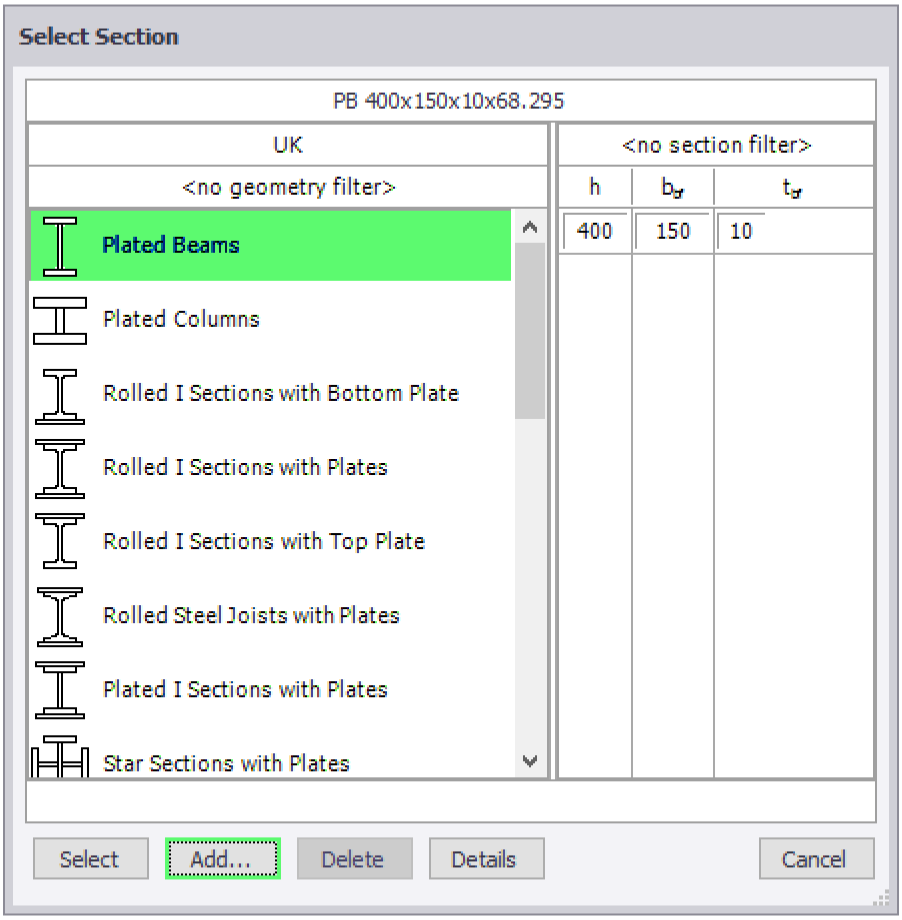
In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.

- Only parallel flange sections can be used.

Plated beams - Eurocode head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to Eurocodes. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

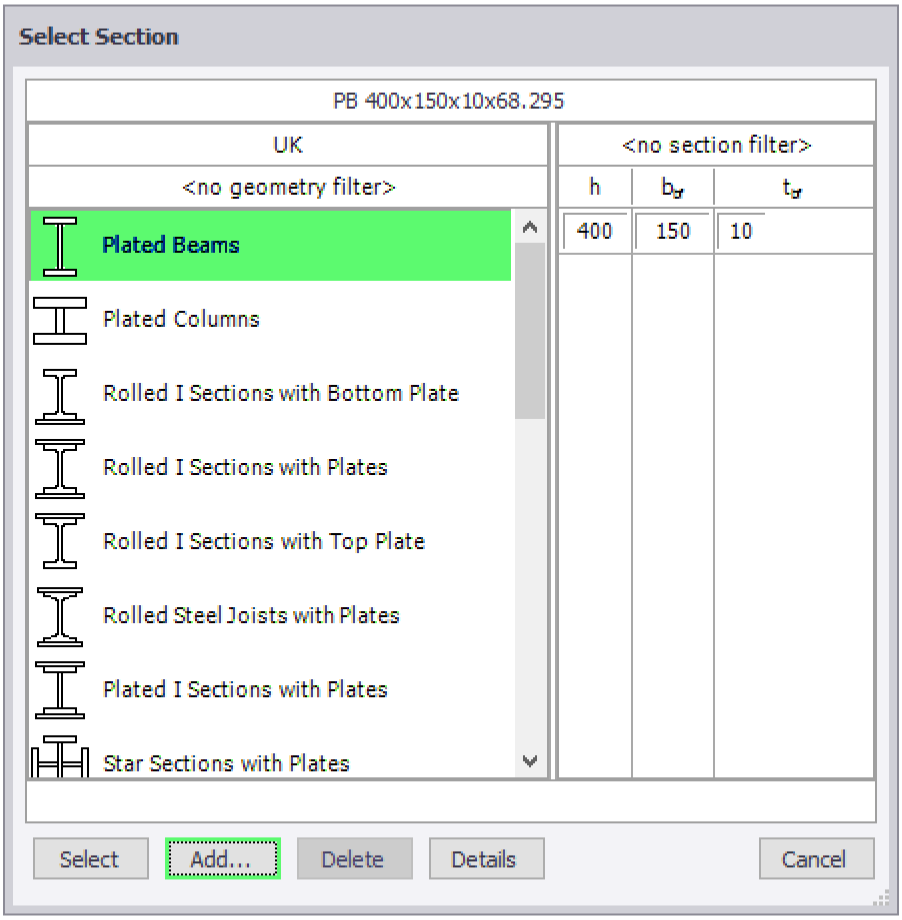


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - BS head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to BS 5950. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

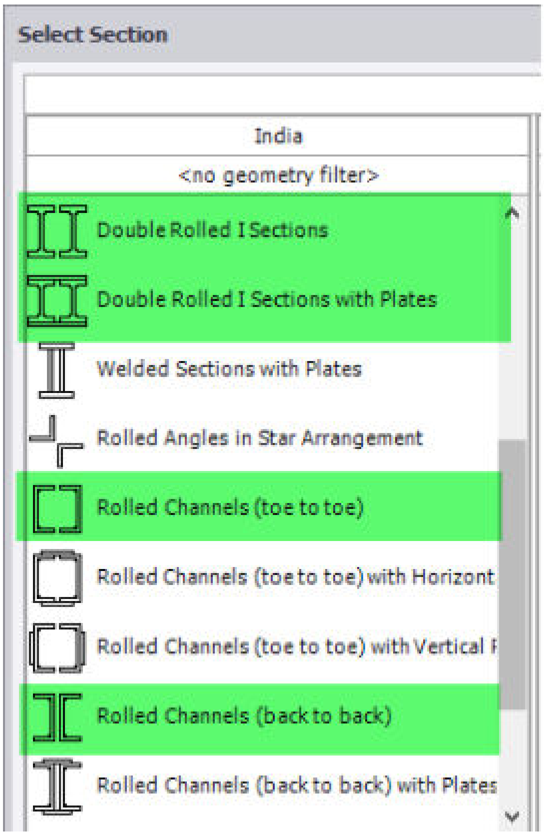


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - Indian head code

When the Fabrication type is set to Plated, a range of steel compound sections can be designed to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections
- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- High Shear case with minor axis moment is beyond scope.
- Design of the lacing or battening system is beyond scope.

- Only parallel flange sections can be used.

Plated beams - Australian head code

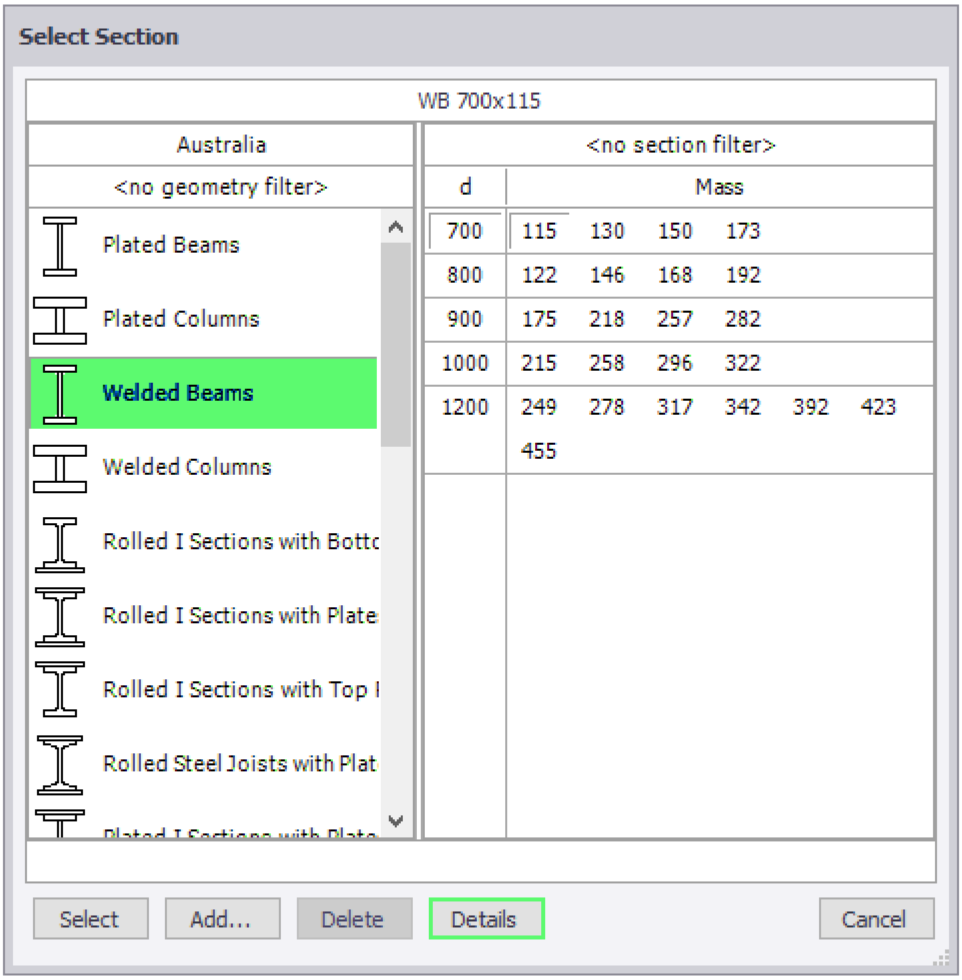
When the Fabrication type is set to Plated, only welded doubly symmetric I-section beams can be designed to the Australian code.

A pre-defined range of these is available from the **Welded Beams** page of the Select Section dialog - these can be both checked and autodesigned.

NOTE Autodesign requires a **Design section order** to be specified, the Welded Beams design section order is provided for this purpose.

While you can add to the range of welded beams the Select Section dialog by clicking the **Add...** button, any such user-defined beams can only be analyzed, but are not designed.

Plated beams available on other pages of the Select Section dialog can be analyzed, but are not designed.



Steel beam restraints

Each beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Conditions of restraint can be defined in- and out-of-plane for compression (strut) buckling and top and bottom flange for lateral torsional buckling (LTB). The effective lengths used in the buckling checks depend on the type of restraint, particularly at supports.

In all cases, the program sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. Any compression or LTB effective length can take the type “Continuous” to indicate that it is continuously restrained over that length.

LTB and Compression restraints

LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model.

The default assumption is that an incoming member connecting to either side of the beam provides lateral restraint to the top flange only, and compression restraint about the minor axis. An incoming member connecting from above or below doesn't provide lateral restraint but provides compression restraint about the major axis.

By right-clicking a member to edit its properties in the Property dialog, you are then able to edit the restraints. You can indicate also continuously restrained sub-beams and edit length factors.

Note that the same level of restraint editing is not provided in the Properties Window (although it does allow you to independently set both the top and bottom flanges as continuously restrained for the entire member length via the Top/Bottom flange cont. rest. properties).

TIP As an alternative to using the Steel Beam Property dialog, restraint settings for steel beams can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

---

Torsion restraints

Torsion restraints are currently displayed in the Property dialog for information only. In design the beam is treated as unrestrained against warping and restrained against torsion.

Continuously Restrained Flanges

In the Properties Window you can independently set both the top and bottom flanges as continuously restrained. By setting ‘Top flange cont. rest.’ and/or ‘Bottom flange cont. rest.’ to ‘Yes’ the relevant buckling checks are not performed during the design process.

Deflection limits

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

Deflection Limits allow you to control the amount of deflection in both composite beams and steel beams by applying either a relative or absolute limit to the deflection under different loading conditions.

A typical application of these settings might be:

- not to apply any deflection limit to the slab loads, as this deflection can be handled through camber,
- to apply the relative span/over limit for live load deflection, to meet code requirements,
- possibly, to apply an absolute limit to the post composite deflection to ensure the overall deflection is not too large.

Camber

Camber is primarily used to counteract the effects of dead load on the deflection of a beam. This is particularly useful in long span composite construction where the self-weight of the concrete is cambered out. It also ensures little, if any, concrete over pour occurs when placing the concrete.

The amount of camber can be specified either:

- As a value
- As a proportion of span
- As a proportion of deflection
  - If this option is selected, the engineer should identify the combination to be used for the calculations by selecting Camber adjacent to the appropriate gravity combination on the Loading combination dialog page.
  - If no combination is selected then the first gravity combination in the combination list is used.

In the latter case, if 100% of the dead load deflection is cambered out, it is also possible to include a proportion of the live load deflection if required.

Lower limits can be set for span length and calculated camber below which camber is not applied. This ensures that unwanted or impractical levels of camber are not specified.

Limits to the maximum allowable camber as well as to the minimum section web thickness to avoid web crippling can also be defined. These will be accounted for in the automatic design process affecting the section size selection.

Related information

See also

Support article: Composite Beam deflections including camber

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because, for example, there are many smaller members or sheeting, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier can be applied to steel beams, composite beams and steel columns. It is defined in the properties window by selecting Prevent out of plane instability and then entering a suitable value in the Instability factor field.

NOTE This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, there is no amplification of the minor axis inertia in the design of the member.

Beam web openings

You cannot currently automatically design sections with web openings, you must perform the design first to get a section size, and then add and check the openings. This gives you complete control of the design process, since you can add appropriate and cost effective levels of stiffening if required, or can choose a different beam with a stronger web in order to reduce or remove any stiffening requirement.

When openings are added they can be defined as rectangular or circular and can be stiffened on one, or on both sides.

Openings cannot be defined from the Properties Window, they can only be defined from the Properties Dialog, (by right-clicking on the member and selecting Edit...)

For guidance in relation to a specific head code, see:

- Beam web openings to AISC (page 188)

- Beam web openings to SCI P355 (page 188)
- Beam web openings to SCI P068 (page 191)

Beam web openings to AISC

Web Opening design limitations

- Simple beams only (single span, pin ended). Although web openings can be defined on multi-span beams design is beyond scope.
  - Cantilevers are excluded.
  - Curved beams are excluded.
  - Haunched and tapered members are excluded.
- Section type: rolled symmetrical compact I sections.
  - Plated sections are excluded.
- Steel yield strength is limited to 65 ksi
- Web opening placement and sizes: Opening depth is limited to 70 percent of beam depth. There are also two control parameters which dictate the dimensions of the opening, one being the aspect ratio, ao/ho, of the opening and the other being the opening parameter po. Both should meet the required limits.
- Multiple openings. Checks are performed to ensure that openings are spaced far enough apart so that design expressions for individual openings may be used.
- Openings cannot be placed closer than the section height to a support.
- Openings can be defined as rectangular or circular; circular openings are designed as equivalent rectangular openings.
- Openings can be reinforced on one side or both sides but always top or bottom, but no checks are performed on the reinforcement and its welds.
- No concentrated loads should be placed above an opening. If there is a point load above the opening then that combination gets a Beyond Scope status, (but only if the concentrated load is greater than 25% of the shear strength of the two tee sections).
- The nearest concentrated load should be placed at least d/2 from the edge of the opening.
- A check is performed to determine whether bearing stiffeners are necessary, if so these are not designed but a warning is shown.
- ASD design i.e. LRFD only.

NOTE For details of the web opening design checks performed see

Beam web openings to SCI P355

NOTE When the Head Code has been set to Eurocode, Tekla Structural Designer adopts the following approach to web openings which is specific to the UK National Annex.

As each web opening is added it is checked against certain geometric and proximity recommendations taken from Table 2.1 of SCI Publication P355 (see below).

Guidance on size and positioning of openings

The following general guidance on size and positioning of openings is taken from Table 2.1 Section 2.6 of the SCI Publication P355

NOTE These geometric limits should normally be observed when providing openings in the webs of beams. It should be noted that these limits relate specifically to composite beams and caution should be used in applying these limits to non-composite beams.

Parameter	Limit	
	Circular Opening	**Rectangular Opening**
Max. depth of opening:	<= 0.8h	<= 0.7h
Min. depth of Tee,	>= t_f +30 mm	>= 0.1h
Min. depth of Top Tee:	As above	As above and >= 0.1 l_o if unstiffened
Max. ratio of depth of Tees: h_b/h_t h_b/h_t	<= 3 > = 0.5	<=2 >= 1
Max. unstiffened opening length, l_o Max. stiffened opening length, l_o	- - - -	<= 1.5 h_o high shear* <= 2.5 h_o low shear <= 2.5 h_o high shear* <= 4 h_o low shear
Min. width of web post: - Low shear regions -High shear regions	>= 0.3h o >= 0.4h o	>= 0.5 l_o >= l_o
Corner radius of rectangular openings:	-	r_o >= 2 t_w but r_o >= 15 mm
Min. width of end post, s_e:	>= 0.5 h_o	>= l_o and >= h

Parameter	Limit	
	Circular Opening	**Rectangular Opening**
Min. horizontal distance to point load: - no stiffeners - with stiffeners	>= 0.5 h >= 0.25 h_o	>= h >= 0.5 h_o

* A high shear region is where the design shear force is greater than half the maximum value of design shear force acting on the beam.

Symbols used in the above table:

h = overall depth of steel section

h_o = depth of opening [diameter for circular openings]

h_t = overall depth of upper Tee [including flange]

h_b = overall depth of lower Tee [including flange]

l_o = (clear) length of opening [diameter for circular openings]

s_e = width of end post [minimum clear distance between opening and support]

t_f = thickness of flange

t_w = thickness of web

r_o = corner radius of opening

In addition, the following fundamental geometric requirements must be satisfied.

d_o <= 0.8*h for circular openings

d_o <= 0.7*h for rectangular openings

$d_o < 2 * (d_{oc} - t_t - r_t)$

$d_o < 2 * (h - d_{oc} - t_b - r_b)$

$d_2 < d_{oc} - d_o/2 - t_t - t_s/2$

$d_2 < h - t_b - d_{oc} - d_o - t_s/2$

$l_o < 2 * L_c$

$l_o < 2 * (L - L_c)$

$L_s < 2 * L_c$

$L_s < 2 * (L - L_c)$

where

d_t= the depth of the web of the upper tee section measured from the underside of the top flange

d_{oc} = the distance to the centre line of the opening from the top of the steel section

d_2 = the distance from the edge of the opening to the centre line of the stiffener

t_s = thickness of stiffener [constrained to be the same top and bottom]

t_t = the thickness of the top flange of the steel section

t_b = the thickness of the bottom flange of the steel section

r_t = root radius at the top of the steel section

r_b = root radius at the bottom of the steel section

L_c = the distance to the centre line of the opening from the left hand support

L = the span of the beam

NOTE Dimensional checks - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udls and in a low shear zone for point loads). This is because for anything other than simple loading the best position becomes a question of engineering judgment or is pre-defined by the service runs.

NOTE Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Beam web openings to SCI P068

As each web opening is added it is checked against certain geometric and proximity recommendations taken from SCI Publication P068.

Guidance on size and positioning of openings

We advise you to comply with the following positional recommendations for web openings:

- Web openings are designed using the bending moment and vertical shear values at the side of the opening where the moment is lower,
- Openings should preferably be positioned at the mid-height of the section. If not, the depth of the upper and lower sections of web should differ by not more than a factor of two,
- Openings should not be located closer to the support than two times the beam depth or 10% of the span whichever is the greater,
- The best location for any opening is between 1/5 and 1/3 of the span from a support in uniformly loaded beams, or in lower shear zone of beams subject to point loads,
- Openings should be not less than the beam depth, D, apart,

- Unstiffened openings should not generally be deeper than 0.6D or longer than 1.5D,
- Stiffened openings should not generally be deeper than 0.7D or longer than 2D,
- Point loads should not be applied at less than D from the side of the adjacent opening.

NOTE Dimensional checks - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udls and in a low shear zone for point loads). This is because for anything other than simple loading the best position becomes a question of engineering judgment or is pre-defined by the service runs.

NOTE Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Steel beam torsion

Torsion can only be checked to the AISC or Eurocode head code and only under the following conditions:

- Section geometry
  - doubly symmetric rolled I-sections
  - closed RHS/SHS/CHS sections (steel and cold formed)
  - closed HSS sections
- pinned
  - single span beams
  - applied torsion moment
  - no web openings

If any of the above conditions are contravened the check is beyond scope.

Only a check design is performed, (no auto-design for torsion).

The procedure and scope are different for Open sections (I’s) vs closed sections (HSS’s)

- Open Sections:
  - Torsion design and angle rotation check will be carried out for applied torsion forces only. “Inherent” torsion is not checked
- Closed Sections
  - “Inherent” torsion and/or physically applied torsion loads checked
  - Angle of rotation check carried out for applied forces only

Fire check (Eurocode only)

The mechanical resistance of a steel beam in case of fire can be checked in accordance with EN 1993 & national annex for the UK, Ireland, Singapore, Malaysia, Sweden, Norway, Finland or the recommended Eurocode values.

The fire check is activated via the beam properties, after which the user is required to specify:

- Load reduction factor for fire
- Required time of fire exposure
- Exposure
- Protected, or unprotected
    - If the protected option is selected, the engineer is then required to specify the fire protection material details.

The time interval for critical temperature iteration is specified in Design Settings, for both 'unprotected' and 'protected' situations.

The check compares the design shear force against the design resistance at the required time of fire exposure.

For the scope of the check and the limitations that apply, see:

Composite beam design

Click the following links to find out more:

- Composite beam overview (page 194)
- Composite beam loading (page 194)
- Composite beam fabrication (page 196)
- Composite floor construction (page 198)
- Precast concrete planks (Eurocode only) (page 205)
- Concrete slab (page 208)
- Metal deck (page 208)
- Stud strength (page 208)
- Connector layout (page 209)
- Composite beam restraints (page 216)
- Composite beam natural frequency (page 216)
- Composite beam transverse reinforcement (page 217)
- Allow non-composite design (page 218)
- Deflection limits (page 185)

Composite beam overview

Composite beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

The beams must be simply supported, single span unpropped structural steel beams.

The following are beyond scope:

- continuous or fixed ended composite beams,
- composite sections formed from hollow rolled sections,
- composite sections where the concrete slab bears on the bottom flange,
- the use of fibre reinforcement.

Beams are designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered.

NOTE If either minor axis bending or axial loads exist which exceed a limit below which they can be ignored, a warning is given in the beam design summary.

Profiled steel sheeting can be perpendicular, parallel and at any angle in between relative to the supporting beam web.

Tekla Structural Designer will determine the size of beam which:

- acting alone is able to carry the forces and moments resulting from the Construction Stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to carry the forces and moments at Composite Stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to provide acceptable deflections, service stresses and natural frequency results.

Alternatively you may give the size of a beam and Tekla Structural Designer will then determine whether it is able to carry the previously mentioned forces and moments and satisfy the Serviceability requirements.

An auto-layout feature can be used for stud placement which caters for both uniform and non-uniform layouts.

Composite beam loading

All loads must be positive since the beam is considered as simply supported and no negative moment effects are accommodated.

Construction stage loading

You define these loads into one or more loadcases as required.

The loadcase defined for construction stage slab wet concrete has a Slab wet loadcase type specifically reserved it. Clicking the Calc Automatically check box enables this to be automatically calculated based on the wet density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, this loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the wet concrete. If you do not do this then you effectively would be designing the beam on the assumption that it is propped at construction stage.

It is usual to define a loadcase for Live construction loads in order to account for heaping of the wet concrete etc.

Having created the loadcases to be used at construction stage, you then include them, together with the appropriate factors in the dedicated Construction stage design combination. You can include or exclude the self-weight of the beam from this combination and you can define the load factors that apply to the self weight and to each loadcase in the combination.

NOTE You should include the construction stage slab wet concrete loadcase in the Construction stage combination, it cannot be placed in any other combination since it's loads relate to the slab in its wet state. Conversely, you cannot include the Slab self weight loadcase in the Construction stage combination, since it's loads relate to the slab in its dry state. The loads in the Construction stage combination should relate to the slab in its wet state and any other loads that may be imposed during construction.

NOTE TIP: If you give any additional construction stage loadcases a suitable title you will be able to identify them easily when you are creating the Construction stage combination.

Composite stage loading

You define the composite stage loads into one or more loadcases which you then include, together with the appropriate factors in the design combinations you create. You can include or exclude the self-weight of the steel beam from any combination and you can define the load factors that apply to the beam self weight and to each loadcase in the combination.

The Slab self weight loadcase is reserved for the self weight of the dry concrete in the slab. Clicking the Automatic Loading check box enables this to be automatically calculated based on the dry density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, the Slab self weight loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the dry concrete. For each other loadcase you create you specify the type of loads it contains – Dead, Live or Wind.

For each load that you add to an Live loadcase you can specify the percentage of the load which is to be considered as acting long-term (and by inference that which acts only on a short-term basis).

All loads in Dead loadcases are considered to be entirely long-term while those in Wind loadcases are considered entirely short-term.

Composite beam fabrication

Fabrication types summary - all head codes

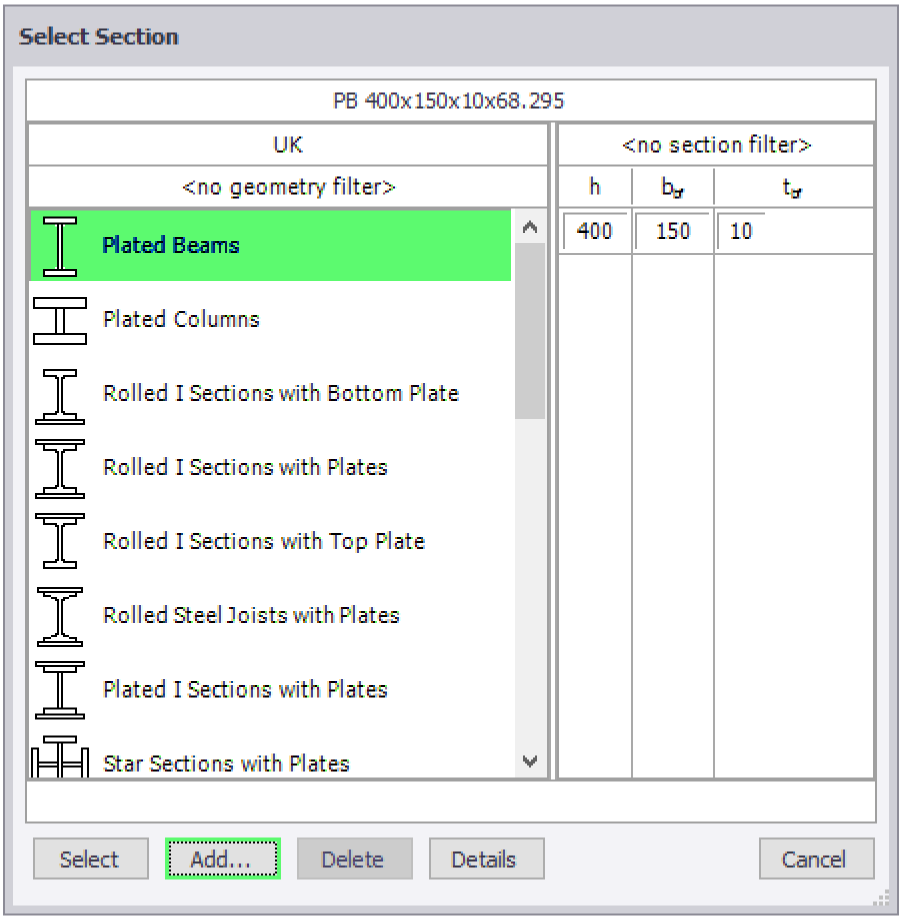
The fabrication types that can be designed as composite steel beams in Tekla Structural Designer are dependent on the design code.

Refer to the following table for details.

	Rolled	Plated	Westok cellular	Westok plated	FABSEC	DELTABEAM
AISC	Yes	No	No	No	No	No
Eurocode	Yes	Yes	Yes	Yes	No	No
BS	Yes	Yes	Yes	Yes	No	No
IS	N/A - Design of composite beams to IS codes is currently beyond scope					
AS	N/A - Design of composite beams to AS codes is currently beyond scope					

Plated beams - Eurocode head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to Eurocodes. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.

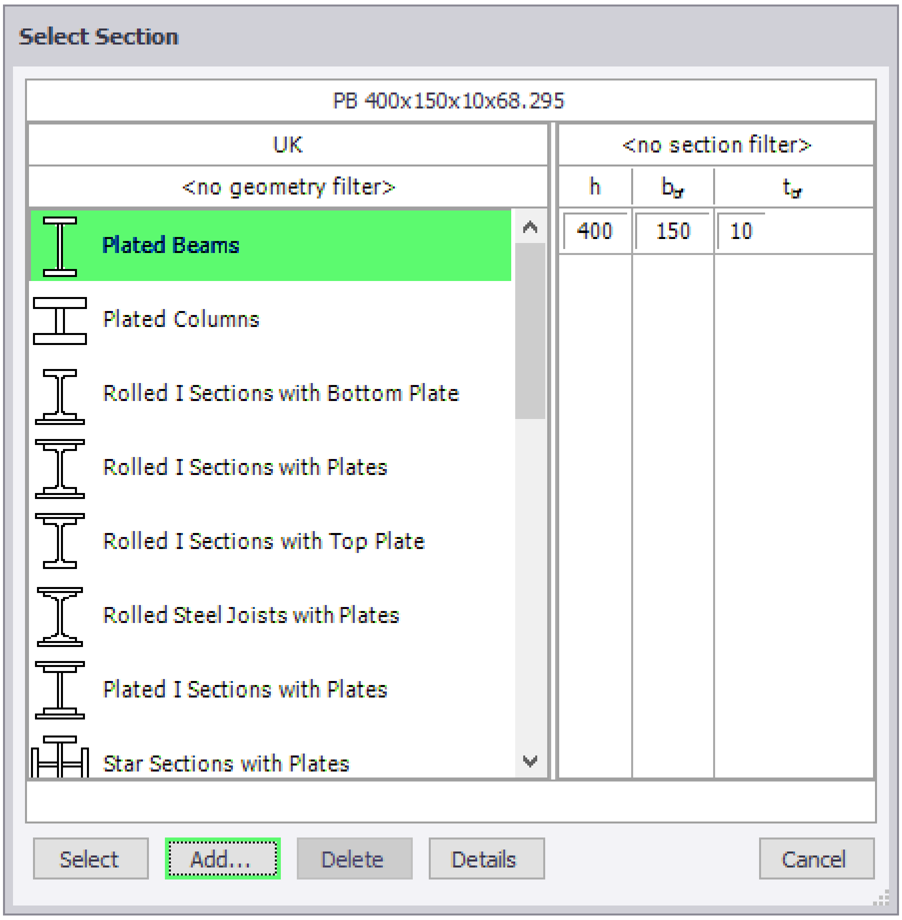


NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - BS head code

When the Fabrication type is set to Plated, only **Plated Beams** can be autodesigned or checked to BS 5950. While one plated section has been provided as a default, you can define (and save) your own plated sections by clicking the **Add...** button shown below.



NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for plated sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Composite floor construction

Deck Type, Angle and Condition

The deck type and angle used in the beam design are determined from the properties of an adjacent slab item. If there are multiple adjacent slab items with different properties, it is the users responsibility to indicate which one governs.

- When specifying the slab item properties you will find that a wide range of profiled metal decks have been included for manufacturers from many countries. PC Planks are also available, but only for the EC Head Code.

- The slab item's rotation angle relative to the global X axis is used to set the profiled metal deck as spanning at any angle between 0° (parallel) and 90° (perpendicular) to the direction of span of the steel beam.

The beam's "condition" is:

- restricted to internal if it has composite slabs attached along its full length on both sides,
- restricted to edge if it has no composite slabs on one side,
- defaulted to edge (but editable) if it has composite slabs on both sides but not along the full length.

Shear connector type

The shear connection between the concrete slab and the steel beam is achieved by using shear studs.

NOTE The use of channel connectors or Hilti™ connectors is currently beyond the program scope.

Head Code: Eurocode, BS

19mm diameter studs with 100 and 125 nominal height (95 and 120 as welded height) are offered. 22mm diameter studs are also offered but only for precast plank decks. Studs do not have a given capacity as their resistance is derived.

Effective width used in the design - Head Code: ACI/AISC

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

   NOTE If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.

   The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

Tekla Structural Designer will calculate the effective width of the compression flange, be for each composite beam as per Section I3.1a (360-05/-10).

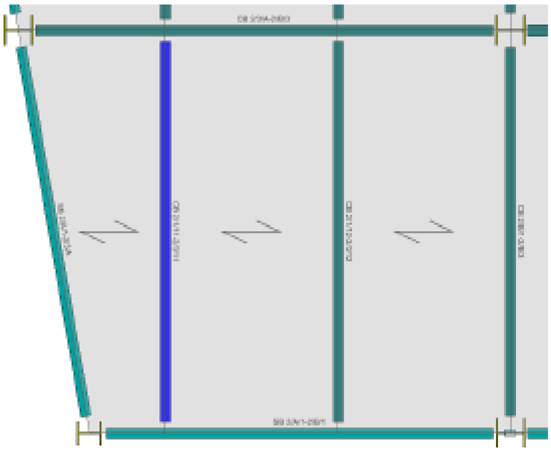
For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the center to center of supports

- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

Although the program calculates b_e, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgment may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, b_e(right) = beam span/8
- to the left of the beam, b_e(left) = one half of the shortest distance to the center line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure **Update effective width prior to design check** is checked on in **Design > Settings > Steel > Composite beam**
2. The floor construction properties will then display an Override effective width option.

a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.

b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Effective width used in the design - Head Code: Eurocode

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

NOTE If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.

The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

Tekla Structural Designer will calculate the effective width of the compression flange, b_{eff}, for each composite beam as per Clause 5.4.1.2 of EC4.

Unless hollowcore units are used, it is taken as the smaller of:

- Secondary beams: the spacing of the beams, or beam span/4
- Primary beams (conservatively): 80% of the spacing of beams, or beam span/4
- Edge beams: half of above values, as appropriate, plus any projection of the slab beyond the centerline of the beam.

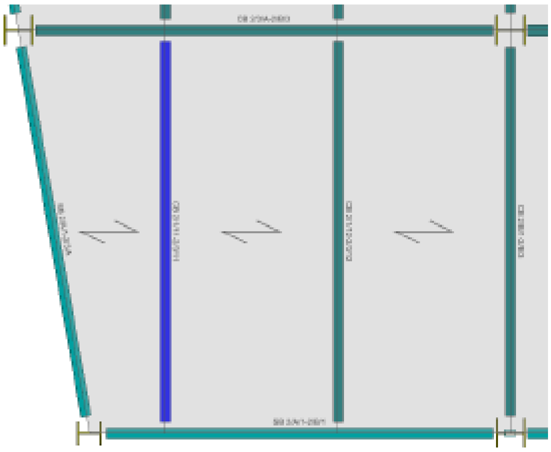
For hollowcore precast plank units only, we calculate the effective width for each side of the beam as the minimum of:

- Assumed concrete fill (500 mm) + recommended gap
- beam span/8 – span taken as the center to center of supports
- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

These effective breadths are used in both strength and serviceability calculations.

Although the program calculates b_{eff}, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgment may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, $b_{eff(right)}$ = beam span/8
- to the left of the beam, $b_{eff(left)}$ = one half of the shortest distance to the center line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure **Update effective width prior to design check** is checked on in **Design > Settings > Steel > Composite beam**
2. The floor construction properties will then display an Override effective width option.
    a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.

b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Effective width used in the design - Head Code: BS

The effective width can either be entered manually, or be calculated automatically either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the Floor construction > Effective width value you wish to use.

NOTE If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.

The Calculate button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select Calculate effective width in the properties.
2. Then click the [...] box that appears next to Calculate.
3. The calculated value is reported in the effective width property.

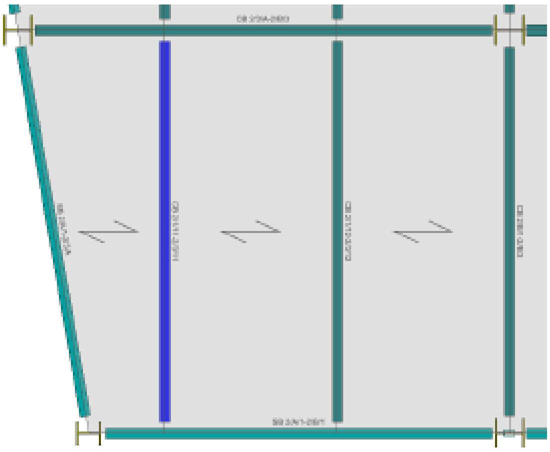
Tekla Structural Designer will calculate the effective width of the compression flange, b_e, for each composite beam as per section 4.6 of BS 5950 : Part 3 : Section 3.1 : 1990.

For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the centre to centre of supports
- one half of the distance to the centre line of the adjacent beam (for slabs spanning perpendicular)
- 40% of the distance to the centre line of the adjacent beam (for slabs spanning parallel)
- the distance to the edge of the slab

Although the program calculates b_e, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgment may sometimes be required.

For example consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, $b_{e(right)}$ = beam span/8
- to the left of the beam, $b_{e(left)}$ = one half of the shortest distance to the centre line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is possible to automatically perform the Calculate process discussed above automatically to all beams prior to a Design command.

1. Ensure **Update effective width prior to design check** is checked on in **Design > Settings > Steel > Composite beam**
2. The floor construction properties will then display an Override effective width option.
    a. With this unchecked, the effective width will be determined when a Design command is performed and the value is reported to you in the Effective width property which is not editable.
    b. With this checked, you can override the effective width value and the value you input into the effective width property is used for design. The Calculate button can still be used to advise the code based values.

Precast concrete planks (Eurocode only)

NOTE The design of composite beams with precast concrete planks is only available for Eurocodes. It is not currently supported for other Head Codes.

General limitations and assumptions

The following limitations and assumptions apply to the use of precast concrete planks:

Cross-section classification is restricted to Classes 1 & 2.

As per normal composite beams there is no requirement to check for transverse force as it is assumed there are no loads or support conditions that would necessitate this.

A balanced condition is assumed during the construction stage and the top flange of the beam is treated as laterally restrained in construction. This condition should be evaluated against the particular application, if it is not suitable then it should be cleared.

Both hollow core units and solid planks are assumed to act compositely only with perpendicular secondary beams and not primary beams parallel to the span of the PC units. Beams neither parallel nor perpendicular to the PC slab are termed angled and are also designed non-compositely.

Precast unit

Design of the precast units themselves is not carried out. It is assumed that the application and loading conditions of the particular precast unit is justified before design of the composite beam is carried out.

The ability to model slab openings is not restricted. The effect an opening has on the behaviour of the precast plank is however, not taken into account and the engineer should verify this to be safe.

P401 restricts the design of composite beams with precast concrete units to the following:

- Hollow core units with circular or circular elongated openings along their length (150mm – 260mm deep). It is assumed all hollow core units modelled will have circular or circular elongated cores. Cores with other cross-sectional geometries may need additional design and verification, this is beyond the scope of SCI P401
- Solid precast planks (75mm – 100mm deep)
- Downstand beams

Deeper units can be chosen than the sizes stated above, however design will not be carried out.

It is assumed that the concrete infill does not contribute to the overall weight of the slab.

If a solid slab is chosen, the contribution of the precast slab is ignored in resistance and stiffness calculations.

If a hollow core unit is chosen, the contribution of the concrete topping is ignored in resistance and stiffness calculations.

Steel Beam

The following applies to steel beam sections:

- Minimum flange width
  - Internal beam - 220mm for shop-welded and 235 for site welded shear connectors.
  - Edge beam – 2 * (6 * stud diameter)

  These recommendations can be reduced by decreasing the bearing - special provisions must be made after consultation with both the precast manufacturer and the steelwork providers*
- Web openings are ignored in design*
- No significant point loads are applied to the composite beam*
- Beam must not behave as a cantilever
- For solid precast units only the topping is to be included as the joints between the units may not be in good contact
- For hollow core units only the precast plank will be taken into account during design
- A warning is issued in this case and subsequent design is carried out assuming that the engineer has justified the particular condition as safe.

Bearing

A default bearing of 75mm minimum is used. This can be reduced but the engineer must consult the precast manufacturer and steelwork provider. The minimum flange width is therefore also reduced.

Concrete properties

Overall properties of the slab should be specified. It is up to the engineer to decide those that govern the overall slab (PC plank + topping). It is these properties that are used to carry out design calculations and slab self-weight.

Loading

Slab self-weight

The concrete infill in the hollow cores is not taken into account in the calculation of the overall weight of the slab.

Where either no topping or structural topping is used, both dry and wet overall self-weight is calculated from the self-weight of the precast unit plus any topping. Where a non-structural topping is used, the engineer is expected to input the overall self-weight of the slab themselves.

Significant Point Loads

Significant point loads are beyond the scope of design in SCI P401. A warning is issued if a significant point load is present on the composite beam. Subsequent design calculations are carried out assuming the point load has no effect on the composite behaviour of the beam. The engineer must carry out additional hand calculations to justify this assumption is safe.

Shear Connectors

19mm and 22mm diameter shear studs are allowed in composite design with precast planks.

Should the engineer choose to place shear connectors in pairs, no dimensional check is carried out. It is assumed that the engineer has justified their use in pairs.

Longitudinal Shear

It is assumed the shear force is divided equally between the two sides of the beam flange.

The factors that influence the longitudinal shear capacity of your composite beam are:

- Concrete strength, slab depth and slab width – you cannot change these independently for the longitudinal shear check, since they apply equally to the entire composite beam design,
- The areas of Transverse and Other reinforcement which you provide in your beam

Transverse

Transverse reinforcement is designed to ensure VEd ≤ VRd. Additional reinforcement to that detailed in design may be required for other purposes.

Refer to SCI P401 for recommended minimum bar sizes and spacing of transverse reinforcement. In the case of a solid slab, the additional mesh is ignored in transverse reinforcement calculations as only either mesh or loose bars can be chosen. Additional mesh, however, can be applied to the slab reinforcement – see “Other”.

It is possible to increase the maximum spacing of transverse reinforcement from that shown in SCI P401 Table 3.1, however it must be noted that this is being done under the engineer’s own judgement.

EN 1992-1-1, 6.2.4 is used to determine the design resistance VRd to the longitudinal shear at the potential failure surface a-a (shown in Figure 4.7 in SCI P401). Failure surface b-b however is not checked in design.

Refer to SCI P401 for recommendations on the detailing of transverse reinforcement and minimum bar length.

Other

Any “other” slab reinforcement in the topping applied to a hollow core unit is ignored in design.

Composite Moment of Inertia

When determining the moment of inertia of a composite section with a hollow core unit, the section is taken as a solid slab (i.e. hollow cores aren’t taken into account).

For stiffness calculations the concrete below the neutral axis is considered as it will contribute some stiffness. However when carrying out resistance calculations, this concrete is ignored.

Concrete slab

NOTE While you can define concrete slabs in both normal and lightweight concrete, design using lightweight slabs is only available for the Eurocode.

Eurocode

Warnings are issued in the design if you do not comply with the following constraints:

- Normal weight concrete range C20/25 - C60/75 - See EN 1994-1-1:2004 Clause 3.1(2),
- Lightweight concrete range LC20/22 - LC60/66 - See EN 1994-1-1:2004 Clause 3.1(2),
- Minimum density for lightweight concrete 1750 kg/m3 - see EN 1994-1-1:2004 Clause 6.6.3.1(1).

Metal deck

Minimum lap distance

The position and attachment of the decking is taken into account in the longitudinal shear resistance calculations.

The applied longitudinal shear force is calculated at the center-line of the beam, and at the position of the lap (if known). If the position of the lap is not known, then the default value of 0mm should be used (that is the lap is at the center-line of the beam) as this is the worst case scenario.

Stud strength

The stud properties you can choose from are appropriate to the stud source.

All types of stud may be positioned in a range of patterns.

Stud groups - Head Code ACI/AISC

You can allow group sizes of 1, 2, 3 or 4 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 3 or 4 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 2 studs in a group.

For each group that you allow you must enter the 'Set distance emid to' - as either >= 2in or < 2in and you also specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

Stud groups - Head Code Eurocode or BS

You can allow group sizes of 1 or 2 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 2 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 1 stud in a group.

For groups with 2 studs you must specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

NOTE It is up to you to check that a particular pattern fits within the confines of the rib and beam flange since Tekla Structural Designer will draw it (and use it in design) anyway.

Optimize shear interaction

If you choose the option to optimize the shear interaction, then Tekla Structural Designer will progressively reduce the number of studs either until the minimum number of studs to resist the applied moment is found, until the minimum allowable interaction ratio is reached or until the minimum spacing requirements are reached. This results in partial shear connection.

Connector layout

When running in Auto-design mode you may not want to specify the stud layout at the start of the design process. To work in this way check Auto-layout to have the program automatically control how the stud design will proceed. When the beam is subsequently designed Auto-layout invokes an automatic calculation of the required number of studs, which is optimized to provide an efficient design.

NOTE 'Auto layout' can actually be checked regardless of whether you are auto designing the beam size or checking it. The combination of 'Check' design with 'Auto layout' of studs can be used to assist you to rationalize your designs e.g.

to force a beam to be the same size as others in the building but have Tekla Structural Designer determine the most efficient layout of studs.

You may choose to perform the initial design with Auto-layout checked and then refine the spacing with Auto-layout cleared if the spacing is not exactly as you require. This may arise if for instance the theoretical design needs to be marginally adjusted for practical reasons on site.

Auto-layout for perpendicular decks

For perpendicular decks, the Auto-layout dialog provides two options for laying out the studs:

- Uniform
- Non-uniform

Uniform

The Uniform option forces placement in ribs at the same uniform spacing along the whole length of the beam.

Whether the stud groups are placed in every rib (as shown above), alternate ribs, or every third rib etc. can be controlled by adjusting the limits you set for Minimum group spacing ( ) x rib and Maximum group spacing ( ) x rib.

The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

NOTE Example:

If you set Minimum group spacing 2 x rib and Maximum group spacing 3 x rib, then the program will only attempt to achieve a solution with studs placed in alternate ribs, or studs placed in every third rib. It will not consider a solution in which studs are placed in every rib.

Additionally, if on the Studs - Strength page, you have allowed groups of 1 stud and 2 studs; then if 1 stud per group proves to be insufficient the program will then consider 2 studs per group.

Non-uniform

If optimization has been checked (see Optimize shear interaction (page 208)) studs are placed at suitable rib intervals (every rib, alternate ribs, every third rib etc.), in order to achieve sufficient interaction without falling below the minimum allowed by the code.

NOTE The optimum and minimum amounts of shear connection are defaulted to 50% and 25% respectively. These can be adjusted if required.

If optimization has not been checked, studs are placed at suitable rib intervals in order to achieve 100% interaction.

Knowing the number of studs necessary to achieve the required level of interaction, it is possible that placement at a given rib interval could result in a shortfall; the program will attempt to accommodate this by working in from the ends, (as shown in the example below). If every rib is occupied and there is still a shortfall, the remainder are 'doubled-up', by working in from the ends once more.

In this example the point of maximum moment occurs one third of the way along the span, this results in an asymmetric layout. If you prefer to avoid such arrangements you can select Adjust layout to ensure symmetrical about centerline. A redesign would then result in the symmetric layout shown below.

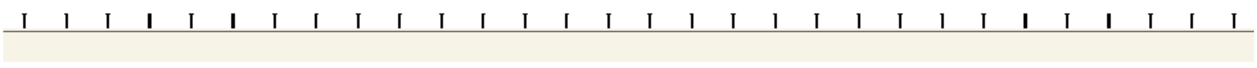
For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

Auto-layout for parallel decks

For parallel decks, the Auto-layout again provides Uniform and Non-uniform layout options, but the way these work is slightly different.

Uniform

The Uniform option forces placement at a uniform spacing along the whole length of the beam. The spacing adopted will be within the limits you set for Minimum group spacing distance and Maximum group spacing distance. If the point of maximum moment does not occur at mid span, the resulting layout will still be symmetric.



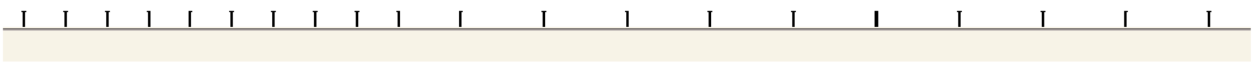
The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

Non-uniform

If optimization has been checked (see Optimize shear interaction (page 208)) studs are placed at a suitable spacing in order to achieve sufficient interaction without falling below the minimum allowed by the code.

If optimization has not been checked, studs are placed at a suitable spacing in order to achieve 100% interaction.

If the point of maximum moment does not occur at mid span, the resulting non-uniform layout can be asymmetric as shown below.



For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

Manual Stud Layout

You may prefer to manually define/adjust the group spacing along the beam. This can be achieved by unchecking Auto layout.

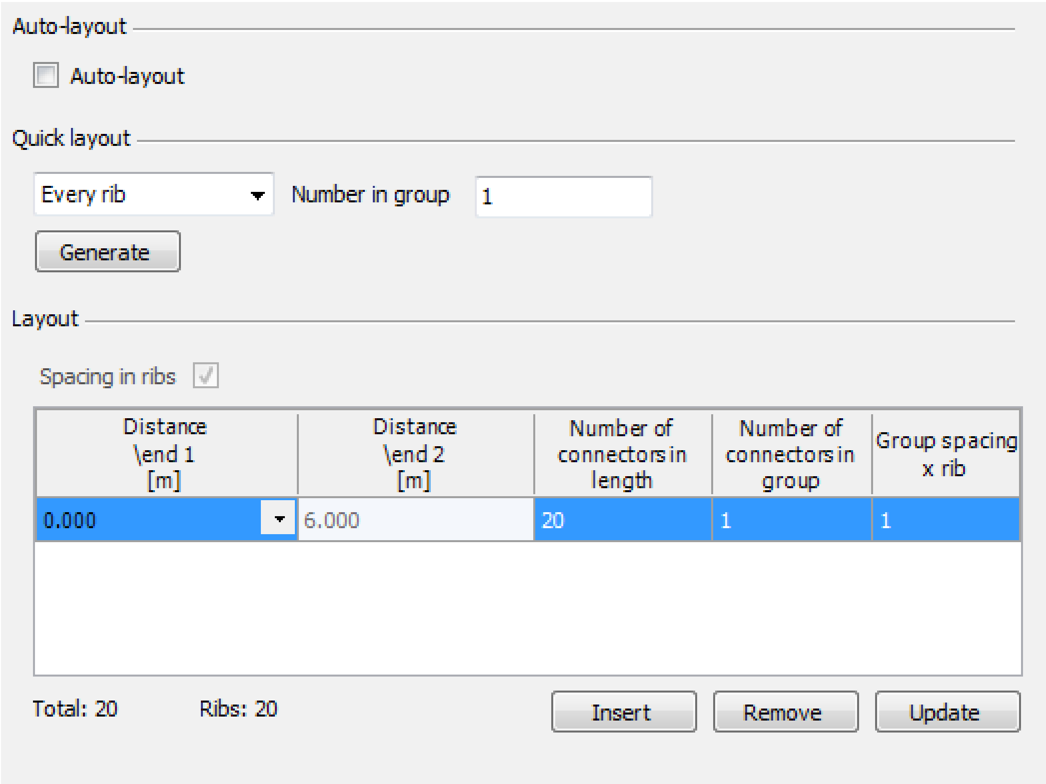
NOTE If you specify the stud spacing manually, then it is most important to note:

- the resulting design may not be the optimal design possible for the beam, or
- composite design may not be possible for the stud spacing which you have specified.

To generate groups of studs at regular intervals along the whole beam use the Quick layout facility. Alternatively, if you require to explicitly define the stud layout to be adopted for discrete lengths along the beam use the Layout table.

Manual layout for Perpendicular decks

For perpendicular decks, the dialog for manual layouts is as shown:

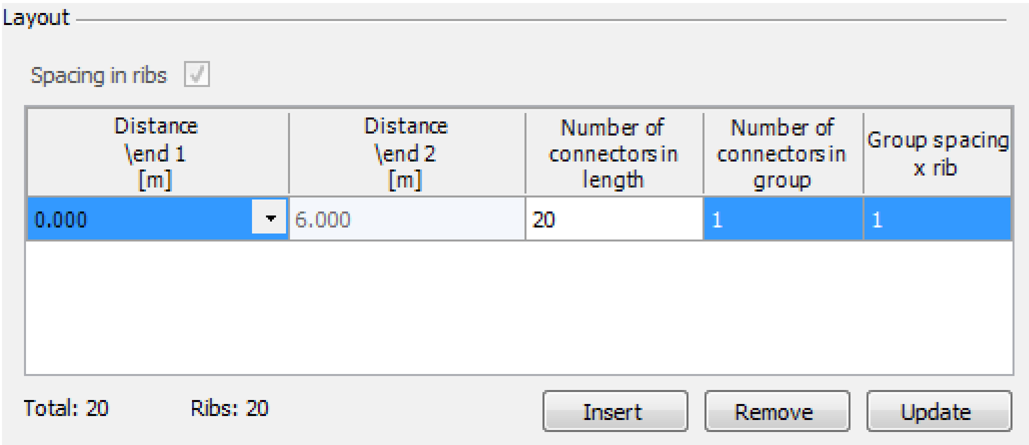


To use Quick layout, proceed in one of two ways:

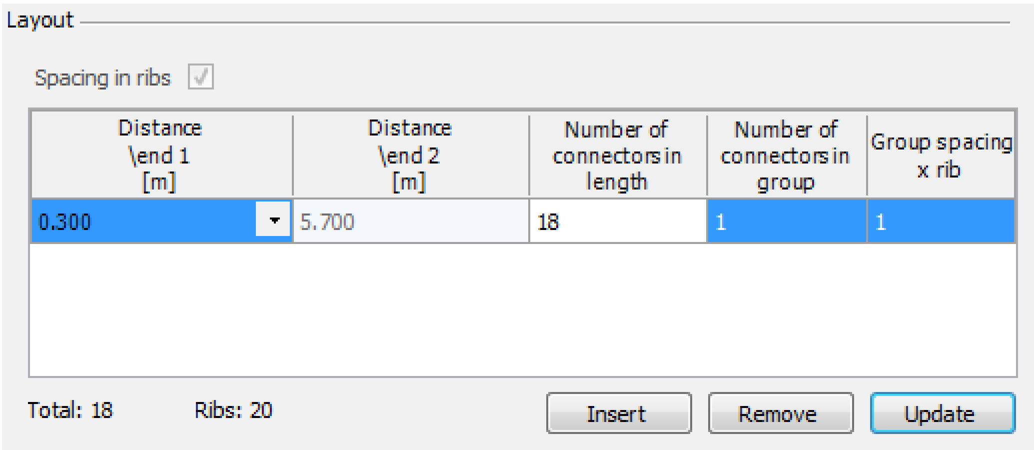
- Choose to position groups in either every rib, or alternate ribs, then specify the number of studs required in the group and click Generate.
- Alternatively: specify the total number of studs, then when you generate, if the number specified is greater than the number of ribs, one will be placed in every rib and the remainder will be 'doubled-up' in the ribs at each end starting from the supports. Similarly if the number specified is less than the number of ribs, but greater than the number of alternate ribs, one will be placed in every alternate rib and the remainder will be placed in the empty ribs. Limits of 600mm or 4 x overall slab depth, (whichever is less), are considered.

To use the Layout table:

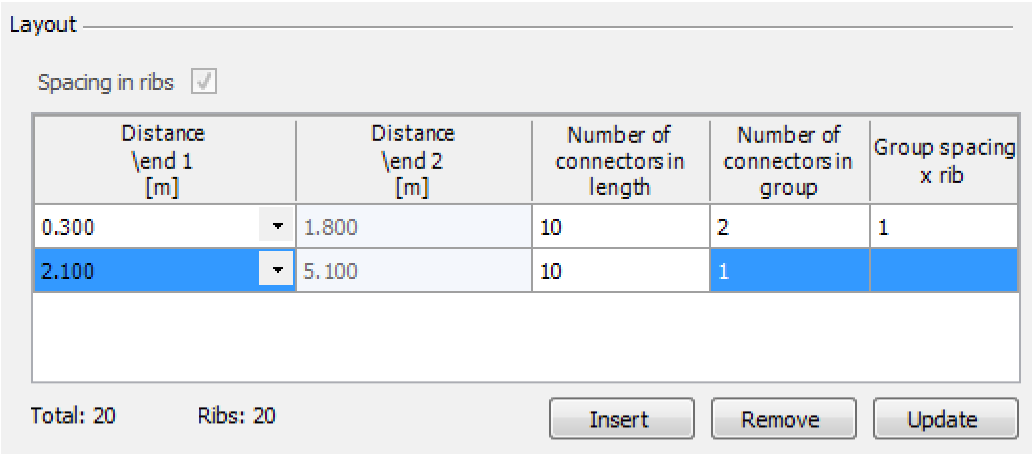
- For each segment you should define the following parameters: No. of connectors in length and No. of connectors in group; Group spacing x rib.



Your input for these parameters is used to automatically determine Distance end 2 - this latter parameter cannot be adjusted directly, hence it is dimmed.

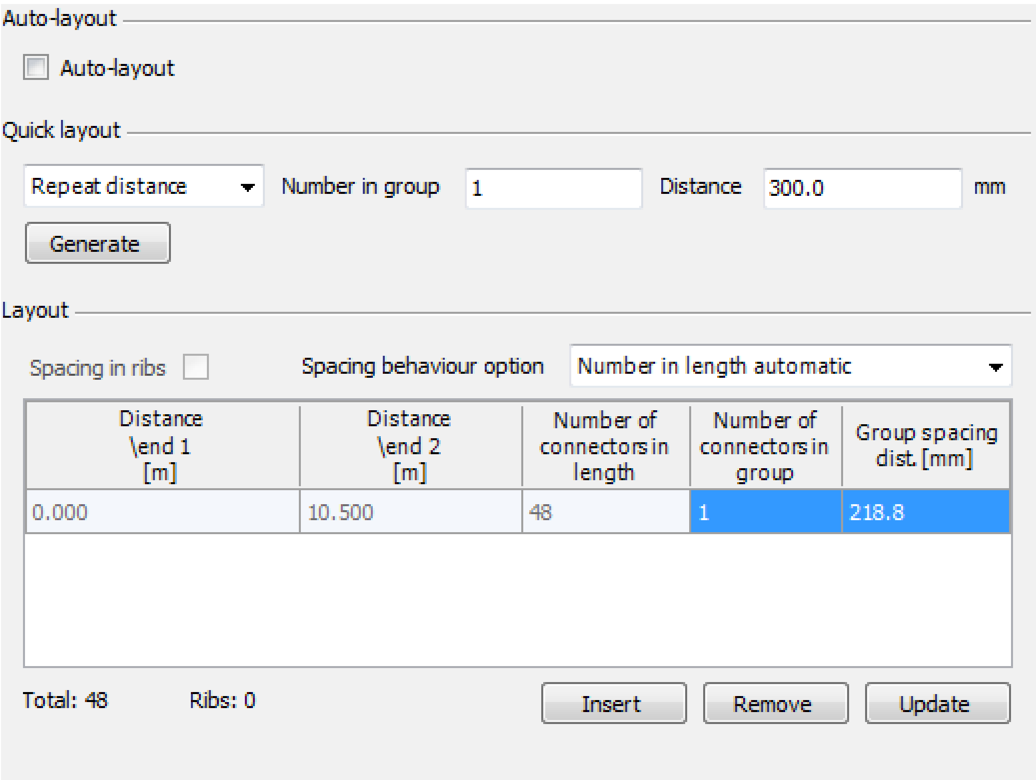


- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify a different stud layout for each segment.
- We would advise that having entered No. of studs in length, group and spacing and ignoring Distance ends 1 and 2 you click Update, this will automatically fill in the missing fields.



Manual layout for Parallel decks

For parallel decks, the dialog for manual layouts is as shown:



To use Quick layout, proceed in one of two ways:

- Choose to position groups at a set repeat distance, then specify the number of studs required in the group and click Generate.

- Alternatively: specify the total number of studs, then click Generate - the program calculates the repeat distance automatically, subject to the code limits.

To use the Layout table:

- The preferred method is to choose the option Spacing distance automatic, in which case you can adjust the No. of connectors in length and No. of connectors in group. Alternatively you could choose the option Number in length automatic and then adjust No. of connectors in group and Group spacing dist.
- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify Distance end 1 for each new segment and it's own stud layout.

Composite beam restraints

You can independently set both the top and bottom flanges of a composite beam as continuously restrained in the **Properties** window.

When the beam is initially created the decking direction is unknown until the beam is actually placed and the floor slab and direction are also created. Hence defaults are provided for each eventuality.

The defaults are:

- for perpendicular decks the deck restrains the beam top flange
- for parallel decks the deck does NOT restrain the beam top flange
- for precast decks the deck restrains the beam top flange
- for all decks the deck does NOT restrain the beam bottom flange

By setting the top flange as continuously restrained and/or the bottom flange as continuously restrained the relevant buckling checks are not performed during the design process.

When not continuously restrained, LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model. The buckling checks are based on these.

By right-clicking a member to edit its properties in the **Property dialog**, you are then able to edit the restraints. You can indicate continuously restrained sub-beams and also edit length factors.

For composite beams the buckling checks are only performed at construction stage as at composite stage they are always assumed to be fully restrained.

TIP Restraint settings for composite beams can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

---

Composite beam natural frequency

A natural frequency check can optionally be requested. When activated a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behaviour i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer has the option to perform a 1st order modal analysis.

Composite beam transverse reinforcement

This reinforcement is be provided specifically to resist longitudinal shear.

Since the profile metal decking can be perpendicular, parallel or at any other angle to the supporting beam the following assumptions have been made:

- if you use single bars they are always assumed to be at 90° to the span of the beam,
- if you use mesh then it is assumed to be laid so that the main bars are at 90° to the span of the beam.

The reinforcement you specify is assumed to be placed at a position in the depth of the slab where it is able to contribute to the longitudinal shear resistance

Automatic transverse shear reinforcement design

It is possible to automatically design the amount of transverse shear reinforcement for each beam. This is achieved in Tekla Structural Designer by checking the Auto-select option on the Transverse reinforcement tab of the Composite Beam Properties.

NOTE The Auto-select option for designing transverse shear reinforcement is only available when the beam is in auto-design mode.

If you are checking a beam, then you must specify the transverse shear reinforcement that you will provide, and then check out this arrangement.

The auto-selected bars can be tied into the stud group spacing by checking the Bar spacing as a multiple of stud spacing option. Alternatively, the spacing can be controlled directly by the user.

Bar spacing as a multiple of stud spacing

When the option Bar spacing as a multiple of stud spacing is checked, the Transverse Reinforcement tab provides the user with controls on the bar size and the multiples of stud spacing.

These can be used to achieve a selection of say, 12mm diameter bars at 2 times the stud spacing, with a slightly greater area than a less preferable 16mm diameter bars at 4 times the stud spacing.

Controlling the bar spacing directly

When the option Bar spacing as a multiple of stud spacing is not checked, the Transverse Reinforcement tab provides the user with direct control on the bar size and the bar spacing.

Allow non-composite design

Typically, at the outset you will know which beams are to be non-composite and which are to be composite and you will have specified the construction type accordingly. However, circumstances can arise in which a beam initially intended to be composite proves to be ineffective. Examples might be:

- very small beams,
- beams with a significant point load close to a support,
- beams where the deck is at a shallow angle to the beam, hence the stud spacing is impractical,
- beams where, for a variety of reasons, it is not possible to provide an adequate number of studs, and
- edge beams, where the advantages of composite design (e.g. reduced depth) are not so clear

Where Tekla Structural Designer is unable to find a section size which works compositely, you can ask for a non-composite design for the same loading. You will find that this facility is particularly useful when you right-click a key beam in the model in order to perform an individual member design.

To invoke non-composite design

1. Select the composite beam(s) as required.
2. In the Properties Window select Allow non-composite design

Steel column design

Click the following links to find out more:

- Steel column overview (page 219)
- Simple columns (page 220)

- Steel column fabrication (page 220)
- Steel column restraints (page 224)
- Steel column connection eccentricity moments (page 225)
- Splice and splice offset (page 230)
- Steel column web openings (page 231)
- Instability factor (page 187)

Steel column overview

Tekla Structural Designer allows you to analyse and design a structural steel column which can have moment or simple connections with incoming members, and which can have fixity applied at the base. The column can have incoming beams which may also be capable of providing restraint, and may have splices along its length at which the section size may vary. You are responsible for designing the splices appropriately.

In its simplest form a steel column can be a single pin ended member between construction levels that are designated as floors.

More typically it will be continuous past one or more floor levels, the whole forming one single entity typically from base to roof.

Steel columns that share moments with steel beams form part of a rigid moment resisting frame.

In all cases you are responsible for setting the effective lengths to be used appropriate to the provided restraint conditions. All defaults are set to 1.0L.

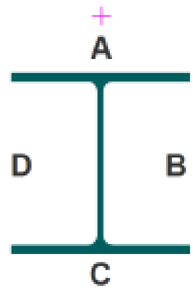
Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not considered.)

If working to either AISC or Eurocodes, the design will take into consideration additional moments resulting from the eccentricity of pinned beam end connections. If working to other head codes, while the eccentricity moments are calculated, they are not used in the design.

Steel member orientation

Tekla Structural Designer considers member orientation when displaying analysis results. Therefore, to apply the sign convention correctly, you need to know which is the end 1 and which is the end 2 of the member. For columns you also need to be able to identify the four faces: A, B, C & D.

If you select the **Direction** option for a member in **Scene Content**, Tekla Structural Designer displays a direction arrow the member which points points from end 1 to end 2 of the member. For columns the direction is always from bottom to the top and the arrow is always drawn adjacent to **Face A**. Looking down from the top of a column, Face B, C, and D then follow in the clockwise direction.



Simple columns

A steel column can be designated as a `simple column' - in which case specific design rules are required.

A simple column should not have any applied loading in its length.

Simple columns are subject to axial forces and moments due to eccentricity of beam reactions.

In order to prevent end fixity moments you would have to manually pin the ends of the column.

NOTE The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as “beyond scope” when designed.

Steel column fabrication

Fabrication types summary - all head codes

The steel column fabrication types that can currently be designed in Tekla Structural Designer are dependent on the design code, and also the construction type specified in the column properties - refer to the following table for details.

Construction: Non-composite column

	Rolled	Plated	Concrete filled	Concrete encased
AISC	Yes	No	No	No
Eurocode	Yes	No	No	No
BS	Yes	No	No	No
IS	Yes	Yes	No	No

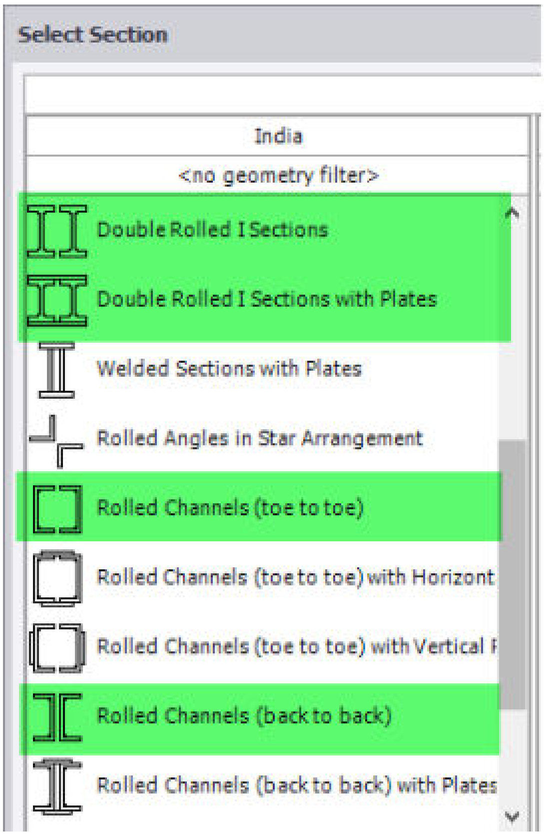
AS	Yes	Yes	No	No

Construction: Composite column

	Rolled	Plated
AISC	No	No
Eurocode	No	No
BS	No	No
IS	No	No
AS	No	No

Plated columns - Indian head code

When the Fabrication type is set to Plated, a range of steel compound sections can be designed to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections

- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- High Shear case with minor axis moment is beyond scope.
- Design of the lacing or battening system is beyond scope.
- Only parallel flange sections can be used.

Plated columns - Australian head code

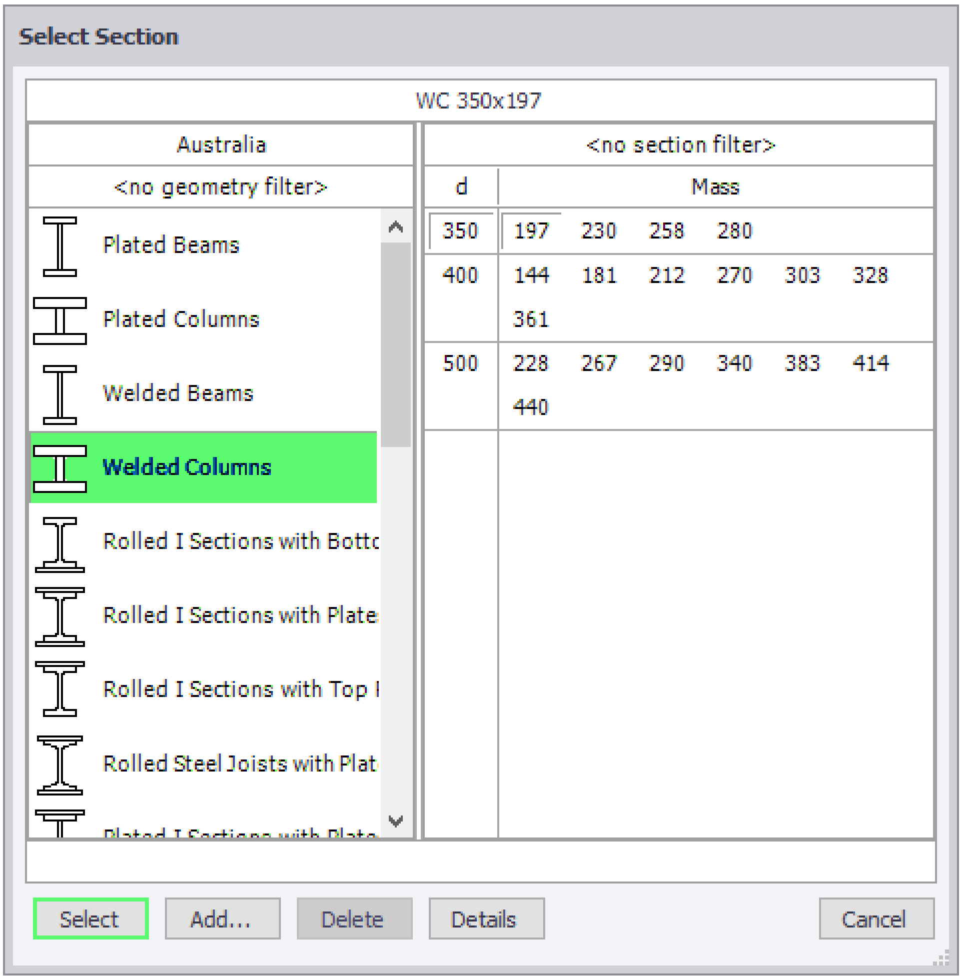
When the Fabrication type is set to Plated, only welded doubly symmetric I-section columns can be designed to the Australian code.

A pre-defined range of these is available from the **Welded Columns** page of the Select Section dialog - these can be both checked and autodesigned.

NOTE Autodesign requires a **Design section order** to be specified, the Welded Columns design section order is provided for this purpose.

While you can add to the range of welded columns in the Select Section dialog by clicking the **Add...** button, any such user-defined columns can only be analyzed, but are not designed.

Plated columns available on other pages of the Select Section dialog can be analyzed, but are not designed.



Composite, concrete filled, and concrete encased steel columns

Whilst composite columns, concrete filled hollow sections and concrete encased sections can be specified in Tekla Structural Designer they are not designed.

For composite columns and concrete filled hollow sections, the analysis uses the bare steel inertia and not the 'effective composite inertia'. This is conservative as the lower stiffness of the bare section will promote more second-order effects. On the other hand for concrete encased columns Tekla Structural Designer takes into account the encasement based on the size specified by the user.

When working to the US head code: AISC 360-16, sub-section I1.5 deals with the calculation of stiffness for concrete filled hollow sections and concrete encased sections used in the Direct Analysis Method (DAM). In accordance with C2.3, Tekla Structural Designer takes account of the standard stiffness adjustment factor of 0.8 with τb set to 1.0 given the additional notional load of 0.1% is also applied. I1.5 requires these types of member to take τb as 0.8

when considering the flexural stiffness. The user can achieve this for concrete encased columns by adjusting the Modification Factors in the . (For composite columns and concrete filled hollow sections which both use the bare steel stiffness, no adjustment is required).

Steel column restraints

Restraints to flexural and torsional buckling are determined from the incoming members that connect to the column. The buckling checks are based on these restraints.

Restraints are considered effective on a particular plane providing they are within ±45° to the local coordinate axis system.

In all cases Tekla Structural Designer sets the default unrestrained length factor between restraints to 1.0.

You have the control to set any unrestrained length to be continuously restrained over that length - when set in this way the relevant buckling check is not performed during the design process.

NOTE The Steel Column Properties window only allows you to set entire stacks as either continuously restrained or unrestrained. In order to specify restraints between the incoming members within each stack it is necessary to open the Steel Column Property dialog instead.

TIP As an alternative to using the Steel Column Property dialog, restraint settings for steel columns can be easily reviewed and adjusted graphically via Review View > Show/ Alter State. For more details see: Review and modify restraints

Lateral torsional buckling (LTB):

- Members framing into either Face B or D are by default assumed to provide full LTB restraint. You therefore need to consider whether or not your particular configuration of incoming members is capable of providing this level of LTB restraint. If necessary you can edit the default restraint provision by selecting the Face A override and/or Face C override checkboxes as appropriate.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 219)

Compression/Strut buckling:

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

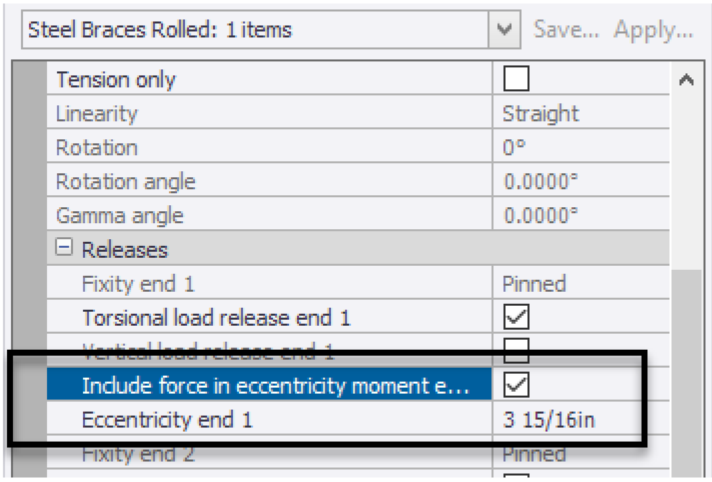
Steel column connection eccentricity moments

Overview

Nominal eccentricity moments that arise from beam end reactions are considered in Tekla Structural Designer's steel column design.

These moments do not come directly from the global analysis but instead are calculated at the ‘load analysis’ post-processing stage as follows:

- At each level the eccentricity of each connection is taken as half the depth of the supporting column, plus an additional user defined offset from the column face.
- At each level the pinned beam end reactions connecting to the column at each face are determined.
- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

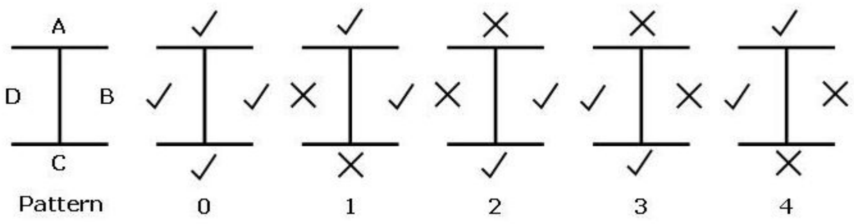
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are 'true' patterns switching live load 'on' and 'off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design

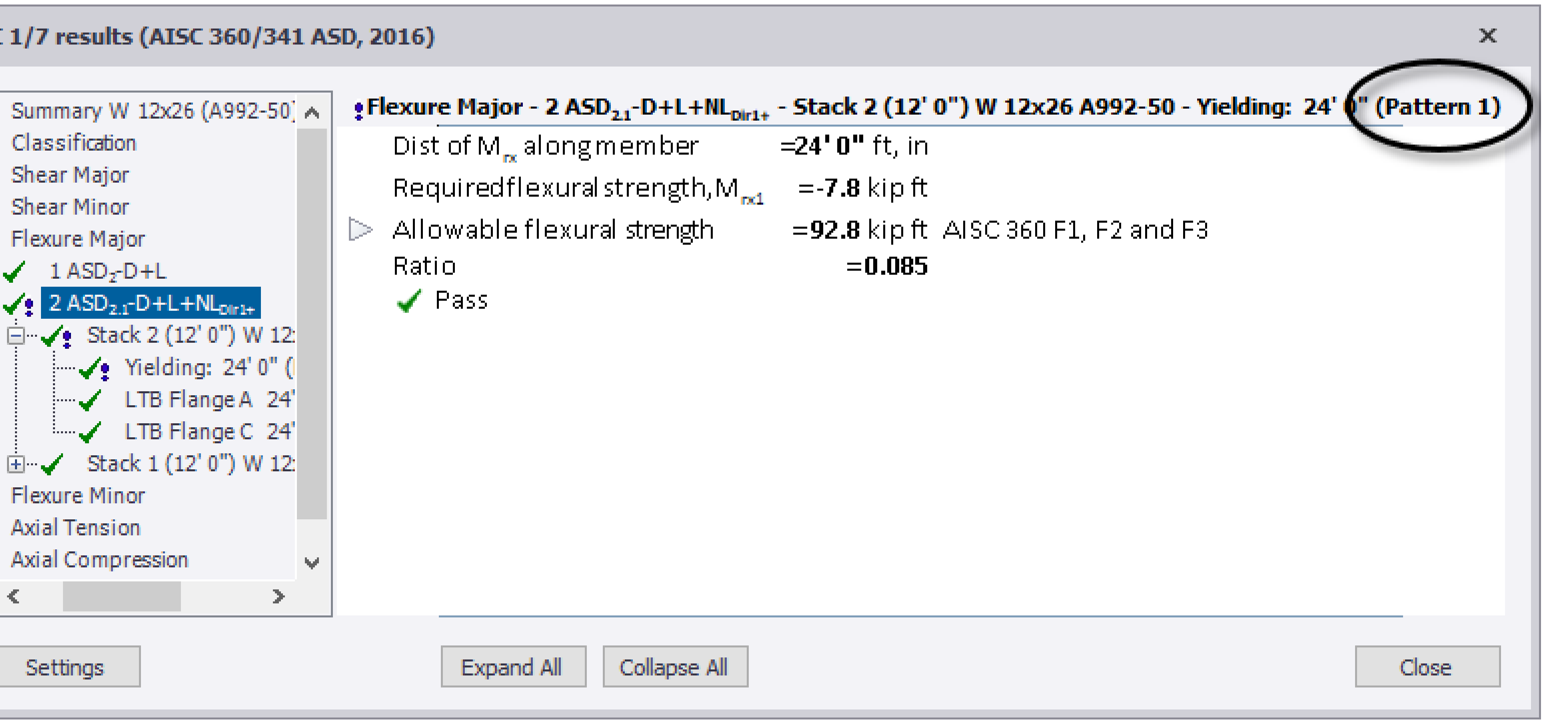
NOTE Patterned eccentricity moments are only considered in the design for the **AISC** or **Eurocode** head codes. If working to other head codes, while the patterned eccentricity moments are calculated, only the fully loaded pattern (P0) is used in the design.

In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

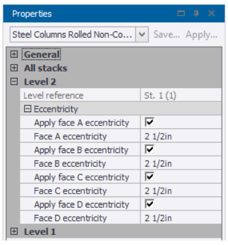
- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.
- Since it is not known whether a reduced axial force with more eccentricity moment is a worse case than full axial and a lower (or even zero, in the balanced case) eccentricity moment, the design loops through all patterns in order to consider each eventuality.
- The patterned eccentricity moments are considered in all design checks apart from 'Shear', (which is unaffected).

- To keep the design details to a manageable level, results for every pattern are not listed; the pattern which produces the governing design forces is listed in the check combination and location tree and details heading, (as shown below).



Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



As long as the option to apply eccentricity at a face is checked, the total eccentricity at that face is taken as half the dimension of the supporting column, plus the additional eccentricity from the face as specified in the **Properties** window shown above.

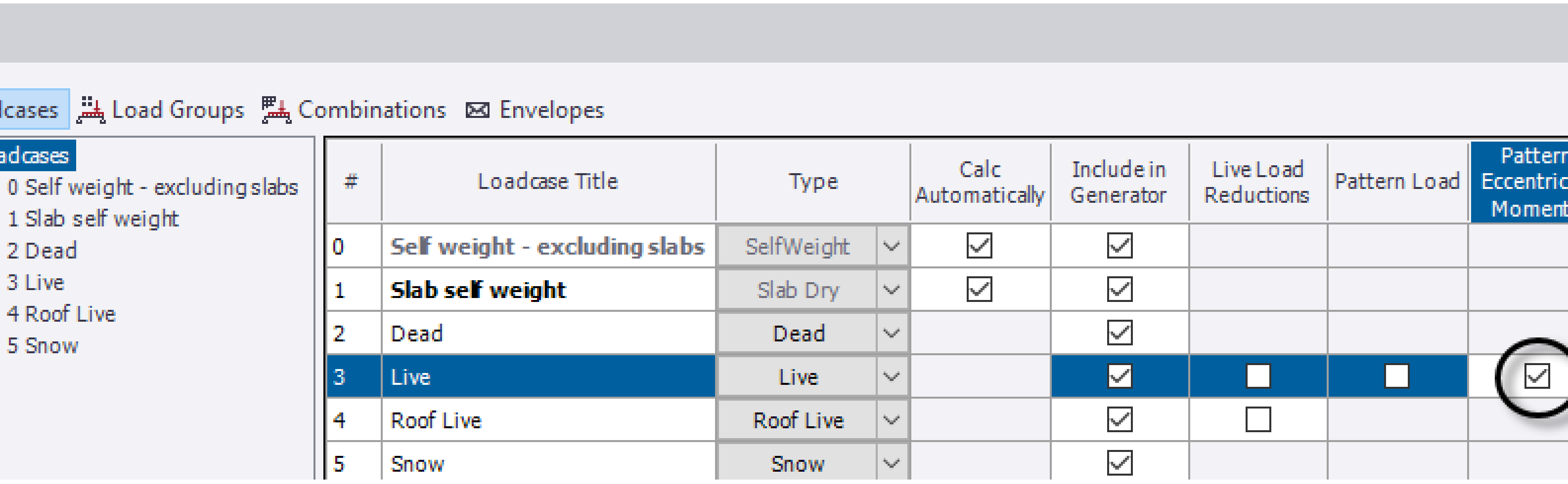
If you uncheck the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE	Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 219)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
    a. Click **Model Settings > Loading > General**
    b. Select **Use patterning of eccentricity moments for steel columns**
    c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
    a. Select a live loadcase that you want to be patterned
    b. Select **Pattern Eccentricity Moments for Steel Columns**
    c. When patterning has been selected for each of the required loadcases, click **OK**

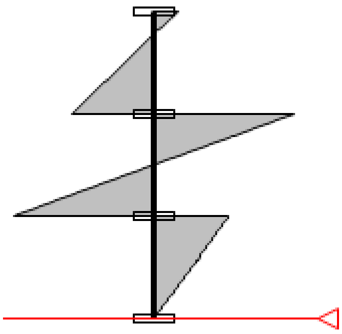


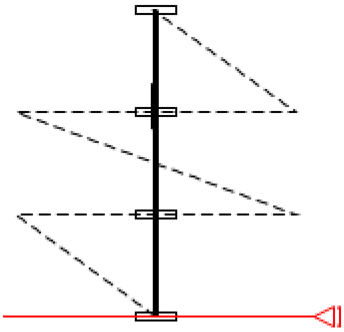
Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View.

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:



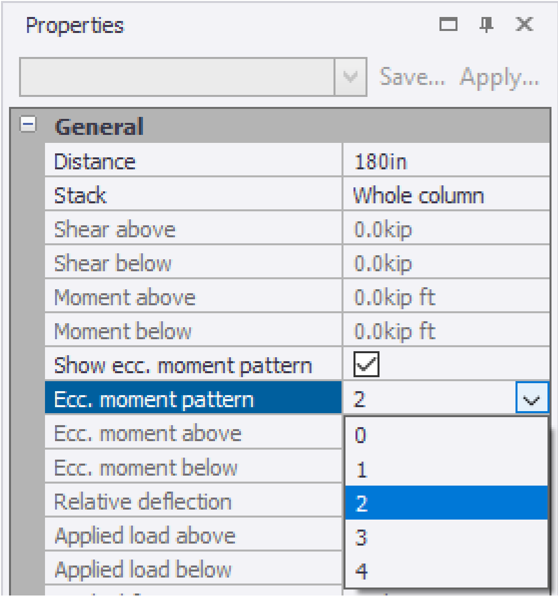


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.



Splice and splice offset

Splices are allowed at floor levels only and must be placed at changes of angle between two adjacent stacks and at changes of section size or type. A validation error will result if this is not the case. The splice can be given an offset from the floor level - the default of 500mm is considered not to be structurally significant.

You must detail the splice to resist the applied forces and moments. The detail should provide continuity of stiffness and strength. Splices given considerable offset should be designed to take account of the P-d moment (also known as moment induced by strut action) at the position as well as the forces from the analysis.

NOTE To remind you of the above requirement the following warning is issued when splice loads are exported to BIM as part of the analysis results: *"The loads given do not include any additional moment due to member buckling."* A similar warning is also issued when splice loads are included in reports. This is an explicit requirement in the British Standards (Clause 6.1.8.2 of BS 5950-1), although as the same engineering principle applies the world over, we issue it for all head codes.

Each lift (length between splices) of a general column can be of different section size and grade. Different section types within the same column are not allowed due to the particularly complex design routines that general columns require. You are responsible for guaranteeing that the splice detail ensures that the assumptions in the analysis model are achieved and that any difference in the size of section between lifts can be accommodated practically.

Steel column web openings

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

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

Web openings can be added to a column by a 'Quick-layout' process or manually.

The 'Quick-layout' process, which is activated using the check box on the Web openings dialog page, adds web openings which meet certain geometric and proximity recommendations (taken from Table 2.1 of SCI Publication P355 if the Head Code is set to Eurocode, or taken from SCI Publication P068 if the Head Code is set to BS). The openings so created are the maximum depth spaced at the minimum centers recommended for the section size.

Web openings can also be defined manually. With Quick-layout cleared, the `Add' button adds a new line to the web openings grid to allow the geometric properties of the web opening to be defined.

Column base plate design

The definition and check of column base plates is an intrinsic part of Tekla Structural Designer - all data associated with a particular column base plate is held within the model. Only simple column base design checks under Eurocode and US head code are supported in the current release.

Click the following links to find out more:

- Column base plate design workflow (page 231)

Related video

Simple column base plate workflow and design

Column base plate design workflow

NOTE To find out more about the theory, assumptions and checks performed for the head code being worked to, see: or .

Related video

Simple column base plate workflow and design

Data sources

Whilst all column base plate data is held in the model, the source of such data is several fold. This includes:

- Default data - when the column base plates within the model are set up by Tekla Structural Designer, intelligent defaults are used that can establish a part or full solution to the configuration.
- Derived data - the model already holds such items as the section size and grade of the column and concrete foundation, plus the design forces.
- Property Sets - certain data can be set to be used to modify existing properties of the column base plates e.g. base plate steel grade, bolt size etc. For more information, see:
- Added data - any individual column base plate can be edited to improve or add to the configuration e.g. additional bolts.

Generate base plates

When a steel column is created or copied a base plate is automatically generated at the lower end of the column if a support is required.

Base plates can also be created by clicking the (Add) Base Plate button on the Design ribbon and then selecting a steel column in the model. Note that moving a steel column that already has a base plate attached will automatically move the attached base plate with it.

Design columns

Base plates have the capability of *automatically sizing* when the column size changes.

This **Autosize** option, which is on as default, will update the plate size and the bolt layout when the column size changes to avoid an invalid base plate configuration. This is not to be confused with automatic design of the base plate which is beyond the scope of Tekla Structural Designer.

Edit base plate properties

Once the associated columns are sized it's possible to turn off **Autosize** and start to finalize the base plate configuration.

This is easily done based on the column size by navigating to **Structure Tree > Members > Base Plates** and selecting the column size.

You can then adjust the column base plate properties in the **Properties** window.

Alternatively, and also for more complex bolt/rod layouts, you can right click an individual base plate and select Edit from the context menu, which opens an Edit dialog for all of the base plate properties.

You can see whether your base plate configuration looks sensible by display in the Structure 3D view.

To copy, move or delete base plates

- Base plates can be moved/copied/deleted independently of the column they are associated to as long as the target location is the bottom of a valid column.
- Base plates can be moved/copied/deleted with the associated column.
    - By simply selecting the column to move/copy/delete the base plate, as default, will be selected too.
    - You can chose to only move/copy/delete the column by de-selecting the base plate from the usual dialog.

Check base plates

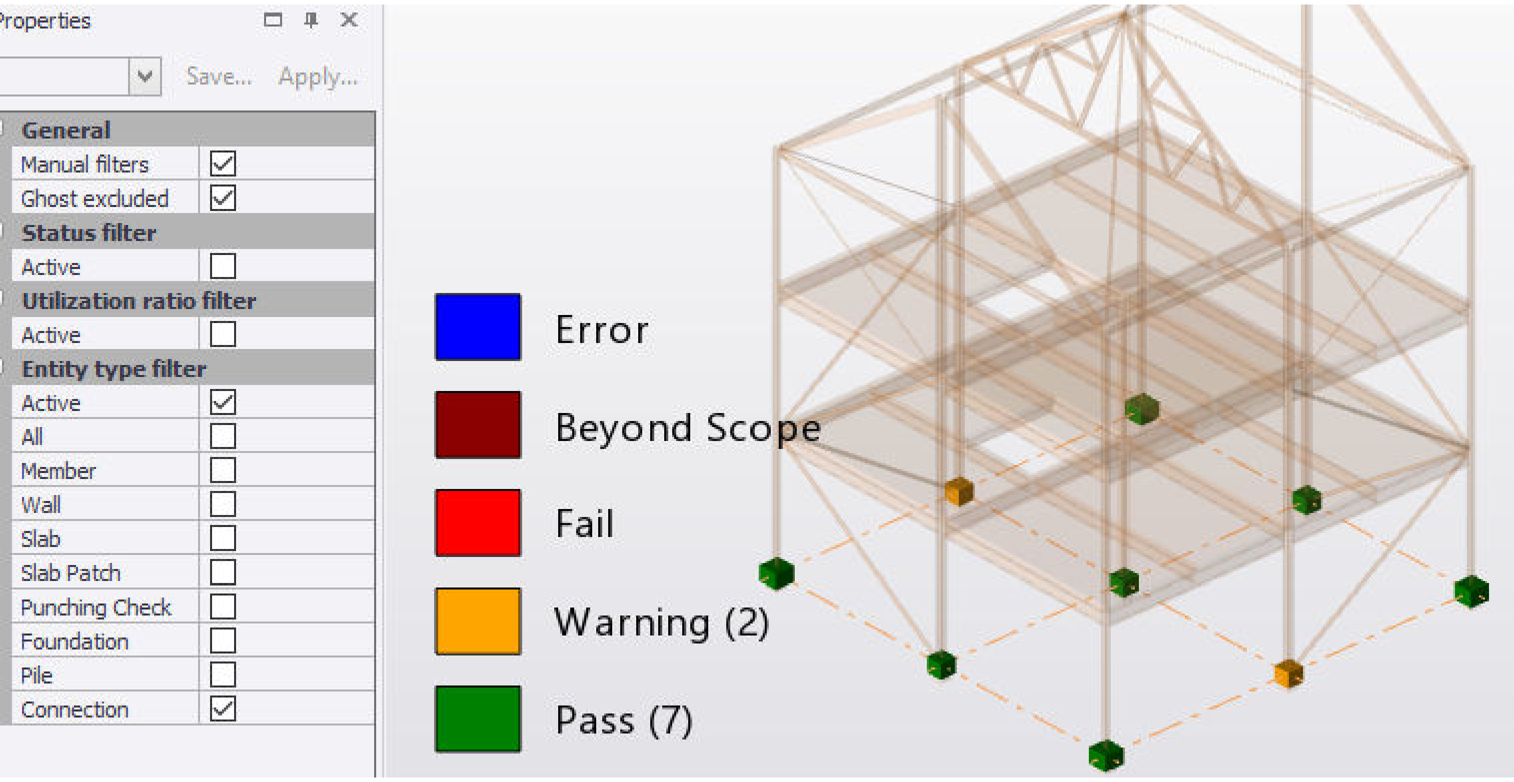
Having established and rationalized the base plate configurations it is now logical to carry out a design check.

You can check an individual base plate, or a selection of base plates and adjust them as you proceed or, once you are content with the layout, you can check them all in a single operation:

- To check an individual base plate - select it in the model (or in the Structure Tree), right-click to display the context menu and select **Check Base Plate**
- To check a selection of base plates - navigate to the **Structure Tree > Members > Base Plates**, select the column size to check, right-click and select **Check Base Plates**
- To check all base plates in the model - navigate to the **Design** toolbar and select **Check Base Plates**

Review check status and utilization

You can then use Review View to visually check which column base plates have passed and which have not, and their associated utilization ratios.



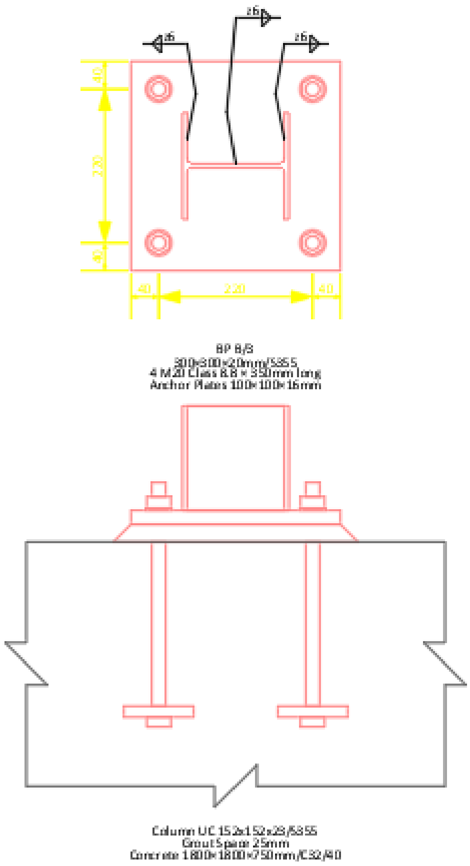
View results and print calculations

Results of your column base plate checks can be viewed on the screen by right clicking on a base plate and selecting **Check Base Plate** from the context menu.

When you are ready to create a report:

- if you require the input, diagrams, and design results you should incorporate the **Base Plates** chapter,
- if you only require a summary of base plate design check calculations you should include the **Base Plate Design Summary** chapter. (The default includes this along with design calculations for other member types in the model).
- if you require a material list containing base plate components, for example rods (US), or bolts (Eurocode), and anchor plates, these are listed in the Material Listing report, as well as the plates themselves, but not the welds.

Create drawings



Base plate detail 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.

Steel brace design

Steel brace overview

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

Steel braces can be specified as rolled I-sections, C-sections, T-sections, rectangular, square and circular hollow sections, angles, double angles, and flat sections.

NOTE If working to the US head code, a brace specified as a flat section can only be analyzed, but design is beyond scope.

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

Applied loading

The following points should be noted:

- Loads for the brace are derived from the building model.
- Element loads cannot be applied directly to the brace itself.
- Live and Roof Live load reductions are not applied.
- Moments due to self weight loading are ignored.

Design Forces

The design forces for strength checks are obtained from an analysis of the entire structure. Braces can be subject to axial compression or tension, but will not be subject to major and minor axis bending.

Input method for A and V Braces

A and V Braces should be modeled using special tools which can be found on the 'Steel Brace' drop list in the 'Steel' section on the 'Model' tab.

Although it is also possible to model the exact same brace arrangement using individual elements created using the simple 'Steel Brace' command, it is important to note that whilst the Notional Loads \ EHFs (Equivalent Horizontal Forces) calculated for models built using the A or V Brace tools are correct, this is not the case when the A or V braces are built up out of individual brace members. In this latter case, elements of the vertical loads that are supported by the bracing system are 'lost' and are not included in the Notional Load \ EHF calculations with the result that the calculated Notional Loads \ EHFs are not correct.

Steel brace in compression

Effective length factors are defined for each axis of buckling.

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

NOTE If designing to British Standards, see Steel brace in compression - BS 5950-1:2000 (page 237)

Steel brace in tension

The net area of the section is required for tension checks. This can be specified either as:

- Percentage value
- Effective net area

NOTE If designing to British Standards, see Steel brace in tension - BS 5950-1:2000 (page 237)

Steel brace in compression - BS 5950-1:2000

If either an Angle (single or double) or Channel or Tee section has been selected as the brace member then a 'Connection at each end' property box appears which allows selection of an appropriate Clause from Table 25 of the BS code to describe the connection type.

The default Clauses from Table 25 are as follows:

Single Angle	4.7.10.2a
Double Angle	4.7.10.3d
Channel	4.7.10.4b
Tee	4.7.10.5b

Notes:

1. Clauses 4.7.10.2b, 4.7.10.2c, 4.7.10.3b, 4.7.10.3c, 4.7.10.3d and 4.7.10.3e only apply to Bolted connections so in these cases 'Bolted' should be selected in the 'Connection' property box. ('Bolted' is the default connection.)
2. For Angle (single and double), Channel and Tee sections the Effective Length Factors will be auto-completed according to the connection Clause selected but these Factors can be changed if required. For Angle (single and double) sections the length Lv-v is always assumed to be L/3 and the Effective Length Factor v-v will act on this Lv-v.
3. For single Angle sections the longer leg is assumed to be the connected element unless 'Short attached' is checked on the Size page of the dialog.
4. For double unequal Angle sections, whichever leg is not the back-to-back leg is assumed to be the connected element when connection Clauses 4.7.10.3a and 4.7.10.3b are selected, and vice versa with Clauses 4.7.10.3c, 4.7.10.3d and 4.7.10.3e

Steel brace in tension - BS 5950-1:2000

Brace tension capacity is calculated according to section type as follows:

A. Hollow sections (CHS, RHS, SHS):	Gross area capacity
B. Angle, Channel, Tee sections:	Reduced effective net area capacity (Clauses 4.6.3.1 and 4.6.3.2)
C. I/H and any other sections:	Effective net area capacity (Clause 4.6.1)

Notes:

1. For section types B and C listed above, an Effective net area (Ae) should be entered either as a percentage of the gross area (Ag) or as an absolute value. The default is to use 100% of Ag, and an absolute value cannot be used if autodesign is also selected.
2. The 'Geometry' property needs to be selected manually if autodesign is also selected, otherwise the 'Geometry' property does not appear visible.

Steel joist design

Steel joist design overview

Steel joists, (or bar joists), are a specific type of members used in the United States. They are simply supported secondary members that do not support any other members - they only support loaded areas, i.e. slab and roof loads. Steel joists are constrained to standard types specified by the US Steel Joist Institute, and standardized in terms of span, depth and load carrying capacity.

In Tekla Structural Designer steel joist design is performed in accordance with the 44th Edition of the Steel Joist Institute (SJI) specification, which uses a similar approach to that embodied in AISC 360-05/10/16 LRFD and ASD.

Standard types

Steel joists are constrained to standard types specified by the Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity. There are four standard types of steel joist available in Tekla Structural Designer.

- K series joists - parallel chord steel joists (2 variations; rod or angle webs) - depths 8" to 30" with spans up to 60ft.
    - Including 2.5 K series joist substitutes - a depth of 2.5in, intended to be used for spans up to 10ft.
- KCS series joists - K series adapted and specially designed for constant moment/shear along length (position of point loads become irrelevant).

- LH series joists - long span joists - depths 18" to 48" for clear spans up to 96ft.
- DLH series joists - deep long span joists - depths 52" to 120" for clear spans up to 240ft.

Special Joists

"SP" suffixes can be added to K, LH and DLH Series joists. Special Joists can handle 'non-uniform' loading situations. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Joist Girders

These are provided as an option to support steel joists. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Assumptions and limitations

The following assumptions and limitations currently apply to the design of steel joists in Tekla Structural Designer:

- All steel joists are considered as simply supported members.
- Steel joists cannot be released axially.
- Cantilevered joists cannot be defined.
- Joist Girders are able to support other joists and brace members, other joists are only able to support brace members.
- Joist inertia and area values are taken directly from the Steel Joist Institute tables, with the exception of Joist Girders for which you must provide the relevant data.
- For all joist types. any resulting load, other then those in the major axis, that exceeds the user specified Ignore Forces Below setting on Design Options is reported as a Warning in the results viewer along with the type and value of force detected.
- Design is currently beyond scope for SP joists and Joist Girders.
- For all steel joists, it is assumed that the top chord is sufficiently braced against lateral torsional buckling.
- There is no restriction on the minimum span for which a joist can be defined even though due to their open web nature joists can be almost impossible to fabricate for very small spans. The user should check the suitability of using such a joist in these situations.
- For steel joists which support a generic concrete slab, it is assumed that the minimum concrete slab thickness of 2 inches (50mm) is present. (SJI Steel Joist Specification 5.9.2). This is not checked by Tekla Structural Designer.

- In design, the user is expected to refer to the bridging requirements in the SJI Specification and decide the appropriate details for the relevant scenario. This is not checked by Tekla Structural Designer.
- ‘Non uniform’ loads are accommodated by KCS joists. E.g. parapet snow drift load, partition walls. If no KCS joists can be found then SP joists can be used for these loads but these are not designed/checked.
- Top chord extensions used as eaves and awnings are not designed.
- Camber of the joists is not shown in the graphics nor handled by Tekla Structural Designer.
- The design and specification of the joist seats (regular or sloping) is not handled by Tekla Structural Designer.
- Double joist configurations cannot be defined in Tekla Structural Designer.
- The design does not consider the minimum bearing requirements for K, KCS, LH or DLH joists – these are the responsibility of the designer/ engineer.
- Customisation of KCS joists to accommodate any applied concentrated loads is not considered byTekla Structural Designer.
- Loadings for accessories to the joist are not included in the standard load tables. An allowance for this should be included by the designer in the loading of the model.
- Steel joists are not designed for composite action and when supported by conventional composite a validation warning is issued. Composite Joists, CJ Series, cannot be defined in Tekla Structural Designer.
- Moment connections at steel joist ends are not allowed.
- Duct openings are stated in the standard Steel Joist Tables for standard panel sizes. Actual spacing and layout of the duct arrangements are beyond the scope of Tekla Structural Designer.
- Fire resistance requirements are not designed nor checked in Tekla Structural Designer, the designer must ensure suitable compliance.
- There is no design for net uplift currently although uplift can exist in a combination provided it is overcome by other gravity loads. The designer should ensure that any uplift due to Wind loads is accurate as the current version of the Wind Wizard does not determine wind forces for ‘Components and Cladding’.
- Sloping joists are permitted providing the loading is normal to the joist and the span will be taken as the sloped length. However, joists with sloping top chords are not allowed.
- SJI allows grades other than 50 ksi [345 MPa] to be used but the Safe Load Table values are based on 50 ksi [345 MPa]. Tekla Structural Designer **defaults to Grade A992 – 50**, it is the user's responsibility to check the

suitability of this or any other grade they wish to use. No adjustment is made for higher or lower grades and this is entirely the designer's call.

Loading

The loads on the joist are from 'one-way' load decomposition. The joist is analyzed as a pin ended beam, only loads in the plane of the web are considered.

Joist Girders should only be used to support steel joists, and should therefore have regularly spaced point loads of similar magnitude along their length. As they are not designed or checked various types of loading could be present, so the user should verify that they have been used appropriately.

Steel Joists are essentially used to support full length uniformly distributed loads, however it is acceptable for the joist to be designed to support other load configurations. This requires the loading pattern on the joist to be assessed and classified into one of the following loading types: Uniform (or near uniform), Equivalent Uniform, or Non-Uniform.

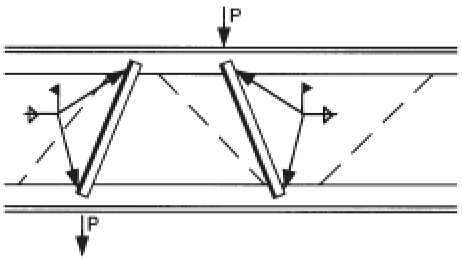
The classification of loading type is made in accordance with the following methods:

Loading Type	Method
Uniform	• No concentrated (point) loads are allowed. • Loads must be (member) Full UDL, UDL, VDL or Trapezoidal Load acting in the Major or Global Z directions. • Loads must be applied over the full length of the beam. • A percentage load tolerance is calculated and provided this is less than the **uniform load tolerance** specified in Design Settings, then the load combination is classified as 'Uniform'.
Equivalent Uniform	• Determine the position of the point of zero shear relative to the centre span point of the joist. • If the point of zero shear is located outside the **maximum eccentricity of zero shear** limit specified in Design Settings, then the procedure is ended and the load combination is classified as 'Non-uniform'. • If the point of zero shear is located within the maximum eccentricity of zero shear limit then an 'Equivalent Uniform' load is established. • The equivalent uniform load is then used to calculate maximum shear force, bending moment and deflection. The percentage variation of these values to the actual values from the beam analysis is then calculated and compared with the **equivalent load tolerance** limit specified in Design Settings.

Loading Type	Method
Non-Uniform	• All loads not qualifying as 'Uniform' or 'Equivalent Uniform' to the above methods are considered as 'Non-uniform'. **NOTE** It is possible for an individual loadcase e.g. Live to be classified as 'Non-uniform' which when combined with other loads becomes 'Equivalent uniform'. Thus, when designing for strength (comparing with the 'black value' in the SJI Tables) the loading is valid whereas when checking the deflection (comparing with the 'red value' in the SJI tables) the service design could be invalid (Fail).

Concentrated Loads

All joists supporting concentrated loads require special treatment by the manufacturer/fabricator even if the loading configuration can be configured as 'Uniform' or 'Equivalent Uniform' since the joist design usually presumes that all concentrated loads are applied at panel points. It is common practice for "field installed members" to be located at all concentrated loads not occurring at panel points as illustrated below.



In Tekla Structural Designer concentrated loads on K, LH and DLH joists are limited to the **maximum sum of concentrated loads** limit specified in Design Settings. This value is unfactored and in the event that the sum of all **unfactored** point loads in all loadcases within a combination exceeds this value, the relevant load combination is classified as 'Non-uniform'.

Uplift

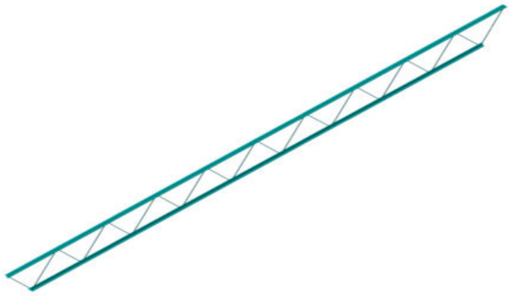
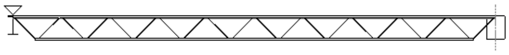
If (net) uplift is detected for a combination no design is performed for it and a warning message is displayed.

NOTE If uplift exists in a loadcase but is 'overcome' by positive loading in another loadcase then the design is valid and no warning is displayed.

Joist member reports

Joist design results can be viewed on the screen and incorporated into member design reports. Joists are also included in material listing reports.

The member design report for steel joists is configurable, but limited to the following chapters:

<table>
<tr><th>Chapter</th><th>Content</th></tr>
<tr><td>Picture</td><td></td></tr>
<tr><td>Drawing</td><td></td></tr>
<tr><td>Loading</td><td>
Loading
<table>
<tr><th>Loadcase</th><th>Source</th><th>Direction</th><th>In Proj.</th><th>Span/Stack</th><th>Type</th><th>Q₁ [klf]</th><th>Pos [ft, in]</th><th>Length [ft, in]</th></tr>
<tr><td>0 Self weight - excluding slabs</td><td>User</td><td>Global Z</td><td></td><td>1</td><td>Full UDL</td><td>0.01</td><td></td><td>26'9 3/8"</td></tr>
<tr><td>2 Dead</td><td>Decomposition 1-way</td><td>Global Z</td><td></td><td>1</td><td>UDL</td><td>0.10</td><td>0"</td><td>26'9 3/8"</td></tr>
<tr><td>4 Live</td><td>Decomposition 1-way</td><td>Global Z</td><td></td><td>1</td><td>UDL</td><td>0.08</td><td>0"</td><td>26'9 3/8"</td></tr>
</table>
Load Reductions
<table>
<tr><th>Loadcase</th><th>Imposed Load Reductions</th><th>Span</th><th>Reduction Factor</th><th>Tributary Area [ft²]</th><th>K_{LL} Factor</th></tr>
<tr><td>4 Live</td><td>No</td><td></td><td></td><td></td><td></td></tr>
</table>
</td></tr>
<tr><td>Design Summary</td><td>
<table>
<tr><th>Design Condition</th><th>#</th><th>Design Value</th><th>Design Capacity</th><th>Units</th><th>U.R.</th><th>Status</th></tr>
<tr><td>Min. Joist Depth</td><td>-</td><td>16</td><td>13 25/64</td><td>in</td><td>0.837</td><td>✓ Pass</td></tr>
<tr><td>Strength</td><td>62</td><td>0.19</td><td>0.23</td><td>klf</td><td>0.826</td><td>✓ Pass</td></tr>
<tr><td>Deflection Dead</td><td>62</td><td>0.11</td><td>-</td><td>klf</td><td>-</td><td>-</td></tr>
<tr><td>Deflection Live</td><td>62</td><td>0.08</td><td>0.14</td><td>klf</td><td>0.593</td><td>✓ Pass</td></tr>
<tr><td>Deflection Total</td><td>62</td><td>0.19</td><td>0.23</td><td>klf</td><td>0.826</td><td>✓ Pass</td></tr>
</table>
Head code: United States (ACI/AISC), design code: AISC360/341 ASD (2010)
</td></tr>
<tr><td>Design Calculations</td><td>
Min. Joist Depth
Span =26' 9 3/8" ft, in
Minimum depth =1' 1 3/8" ft, in
Joist depth =16 in
Ratio =0.837
✓ Pass
Strength
62 ASD$_2$-D+L - Critical
Required total load =0.19 klf
Design total load =0.23 klf
Ratio =0.826
✓ Pass
</td></tr>
</table>

Steel truss design

Steel truss design overview

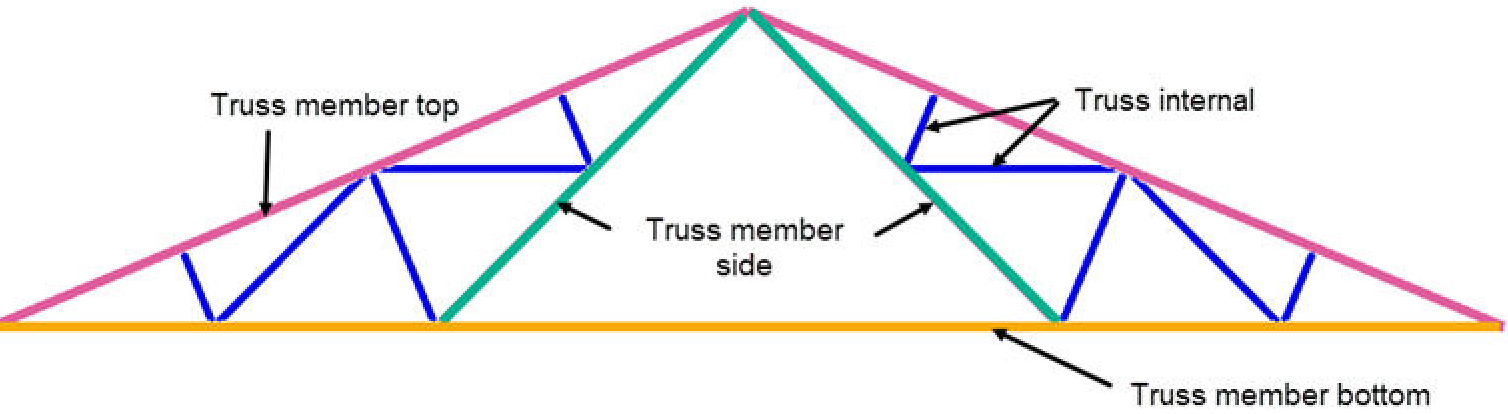
In Tekla Structural Designer although trusses can be defined in any material, design is restricted to steel and cold formed truss members only.

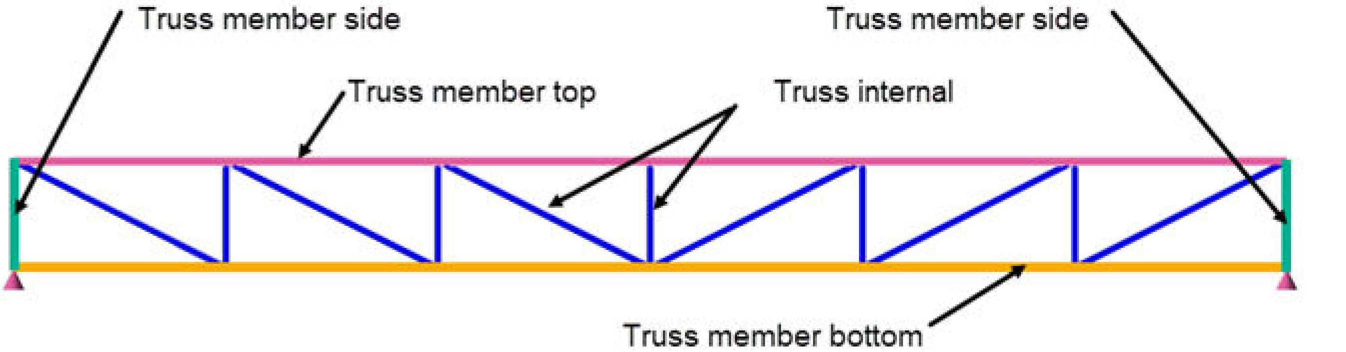
The Truss Wizard provides a wide range of standard truss shapes and configurations. This covers most practical layouts of truss. However, you can also modify a standard truss to produce one of slightly different configuration or manually 'stick build' your own truss from truss members if it is an unusual layout.

Trusses created by the Truss Wizard will comprise a mix of the following truss member types:

- Truss member top
- Truss member bottom
- Truss member side
- Truss internal

These are shown in two of the standard configurations below:





Solver elements are created directly between the member insertion points - they do not take into account major and minor snap points, or any offsets that might have been specified in the member properties. Consequently, all intersecting members 'node' at the same point – i.e. all internals meet along the set out line of the chords; this assumes that set out lines are coincident with the centroidal axes. Therefore, no in–plane eccentricities are considered in the analysis and design of the trusses.

A wide range of section types can be defined that includes all the common rolled sections. There is no restriction on the type and size of sections that can be connected within the truss. The practicality and efficiency of connections between members is your responsibility. A 'beyond scope' status is issued if a section type has been applied that cannot be checked for a given design condition.

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

The design checks performed depend on the truss member types as follows:

- **Truss member top and bottom** - these are continuous members with axial force and bending principally in the plane of the truss. They are designed as beams (page 178) for all internal forces (axial, major and minor axis bending and shear) depending upon the section type used. Where tension exists in a chord member, the tension capacity is based on the effective net area.
- **Truss internal** - these are restricted to be pin ended, and are designed to the appropriate clauses for tension and compression members. Primary bending moments due to self weight and secondary moments due to eccentricity of their connections are ignored. Effective lengths for compression and effective net area for bolted and welded connections can be taken into account via the properties of the truss members.
- **Truss member side** - depending on the truss type selected in the Truss Wizard these will either default to pin ended or fully fixed. If pin ended, the design checks are the same as for Truss internal. If fully fixed at both ends (as in the case of a Vierendeel truss), the design checks are the same as for Truss member top and bottom.

  NOTE If a (pin ended) side is intersected by another truss member then it is still designed for axial force only, but a warning is issued indicating the forces that have been ignored.

For top and bottom chords, conditions of restraint can be defined in and out-of-plane for strut buckling and, top and bottom flange for lateral torsional buckling (LTB). It is upon these that the buckling checks are based. Incoming members are identified by the program and sensible default values for whether these provide restraint or not are set up (see **Assumptions and Limitations** section below). Restraint cannot be added where no incoming member exists but full control of the effective length factors is provided. In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. You can also indicate chord sub-beams to be continuously restrained over their length where appropriate.

NOTE Effective length factors and continuous sub-beam restraints are edited by right clicking on individual chords and selecting Edit *chord reference* from the context menu - these can then be specified from the respective lateral and strut restraint pages of the Properties dialog. Continuous sub-beam restraints can also be edited in Show/Alter State > Restraints, as can restraint of internals to chord (defaults to in-plane only for strut and unrestrained for LTB).

Results of your truss design can be viewed on the screen and incorporated into a report. Truss members are listed as a separate type in the Material Listing report.

Assumptions and Limitations

Limitations

The following limitations apply:

- Web openings, plated sections including Fabsec beams (with or without openings) and Westok beams cannot be used as truss members
- Chord members cannot be placed vertically
- The arch member of a bowstring truss is drawn and designed as a series of facets and not as one continuous curved member
- Truss internals cannot be loaded directly and no loads from floors and roofs are decomposed to them - sides can be loaded, but forces other than axial are then ignored in the design if the pinned ends are not removed.
- Truss chords are not by default excluded from diaphragm action within a floor slab, but they can be deliberately excluded if required. See: https://teklastructuraldesigner.support.tekla.com/support-article/2816265

Assumptions - Restraints

- In both top and bottom chords the node points are assumed not to have incoming out of plane members unless you define such members in the model. Hence, at these positions only in-plane strut buckling restraint is assumed as a default. You can of course change these.
- In a top chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,
- - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out-of plane strut buckling restraint,

    - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- In a bottom chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,
    - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out of-plane strut buckling restraint,
    - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- Lateral restraints to the top or bottom flange of a chord are assumed to be capable of resisting restraint forces not less than those specified in the relevent section(s) of the design code.
- In all cases, for all member characteristics it is assumed that you will make a rational and 'correct' choice for the effective lengths between restraints *The default value for the effective length factor of 1.0 may be neither correct nor safe.*

Portal frame design

If portal frames are modelled and designed in Tekla Structural Designer they will be designed elastically for the forces determined from the 3D analysis, in the same manner as other steel beams and columns.

A more economic design can be obtained by exporting individual frames to **Tekla Portal Frame Designer**. This industry leading portal frame software performs an elastic-plastic analysis and design and undertakes member stability checks to EC3 or BS5950. Once designed, the resulting sections can then be returned to the Tekla Structural Designer model.

NOTE Tekla Portal Frame Designer is a separately purchasable product.

1.6 Concrete member and slab design handbook

To get started with designing concrete members in Tekla Structural Designer see:

- Concrete member design workflow (page 248)
- Concrete member autodesign (page 254)
- Cracked, partially cracked, and uncracked concrete members (page 255)
- Concrete beam and column groups (page 260)
- Concrete beam design properties (page 265)

- Concrete column design aspects (page 275)
- Concrete wall design aspects (page 281)
- Interactive concrete member design (page 285)

To get started with designing concrete slabs, see:

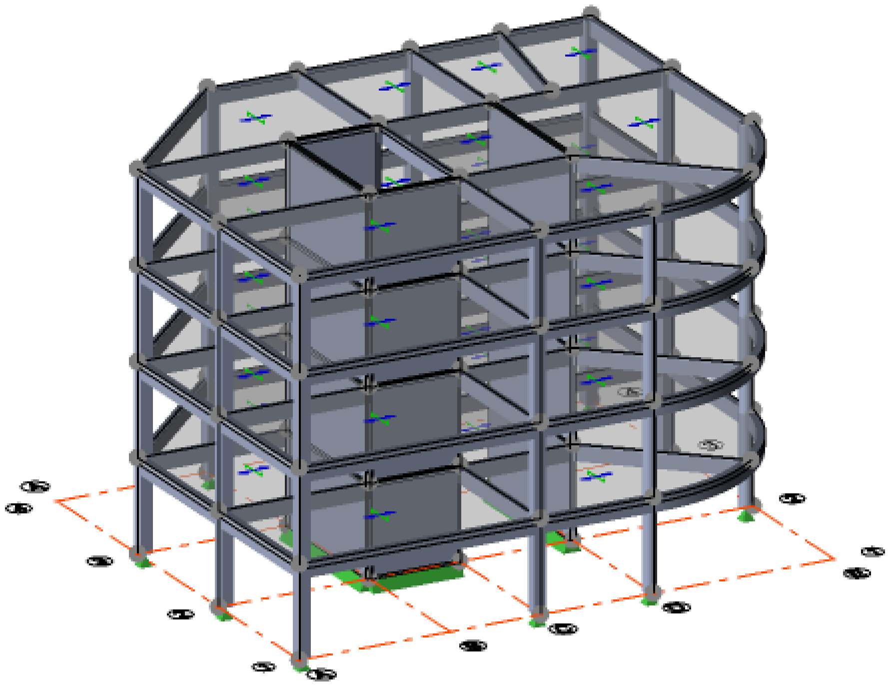
- Concrete slab design (page 329)

For guidance to help reduce the overall design time, see

- Working with large models

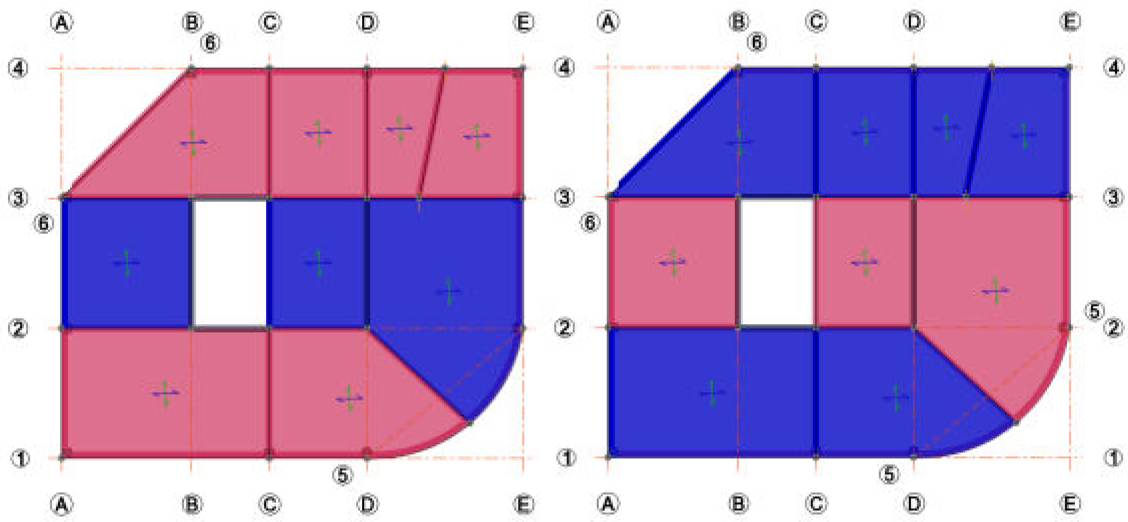
Concrete member design workflow

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



Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Set all beams columns and walls into autodesign mode

For the first pass, in order to get an efficient design at the outset, it is suggested that you set all members to "autodesign" with the option to select bars starting from Minima.

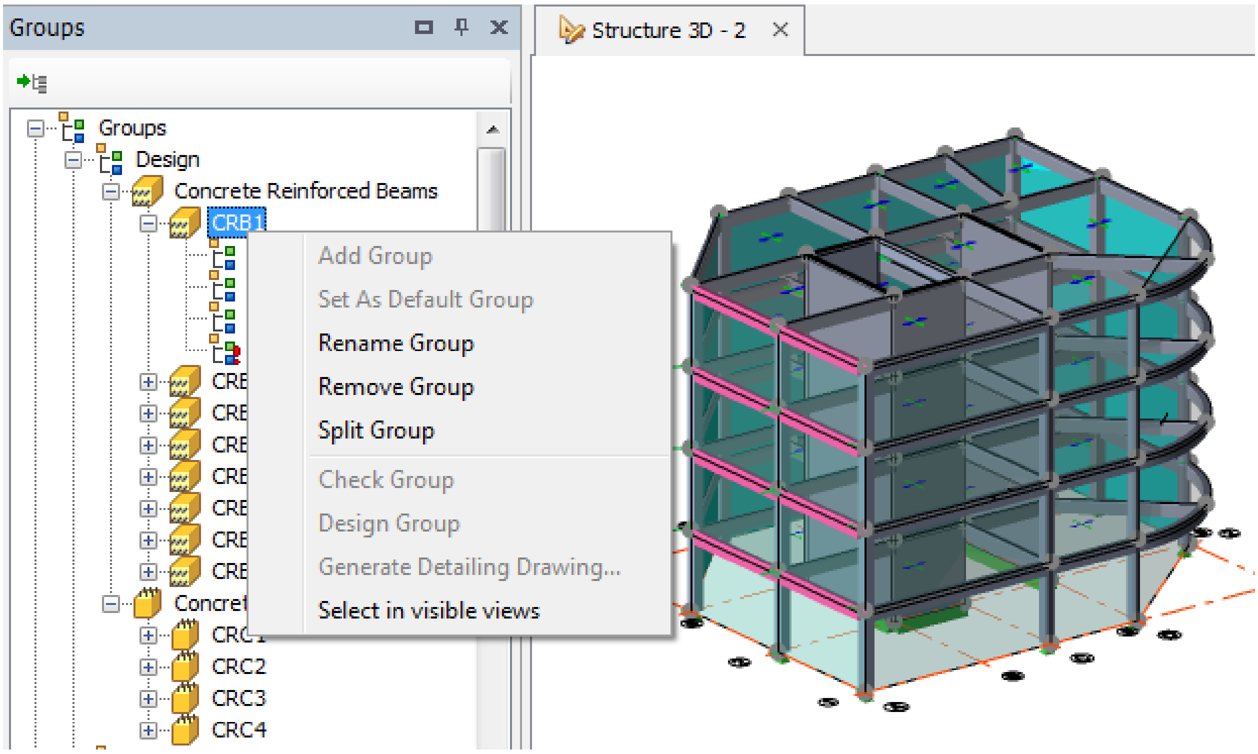
Related concept

The autodesign process for concrete members (page 254)

Review beam and column design groups

Provided that the concrete beam and column options are checked in Design Settings > Design Groups, the design groups shown in the Groups tab of the

Project Workspace will be applied in the beam and column autodesign processes.



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

Review beam, column and wall design parameters and reinforcement settings

The member design parameters and reinforcement settings should be carefully considered prior to running the design.

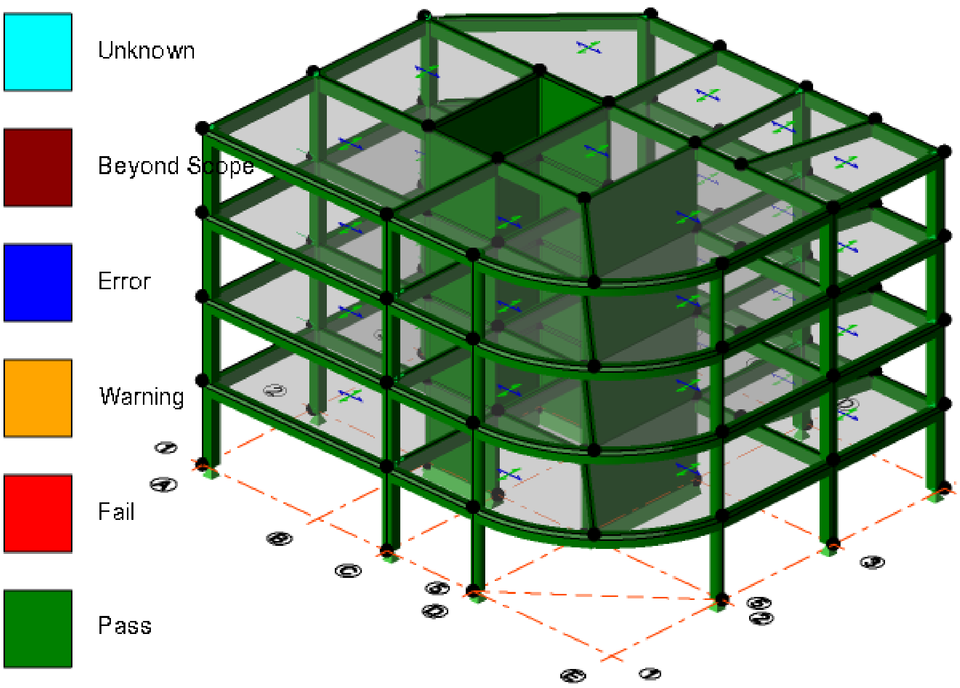
Perform the concrete design

By running Design Concrete (Static) from the Design ribbon, you effectively combine analysis and design (with the exception of slab design) into a single automated process.

Up to three separate analyses are automatically performed in order to generate the design forces required for the concrete beam, column, and wall design:

- 3D Analysis
- Grillage chasedown analysis
- FE chasedown analysis

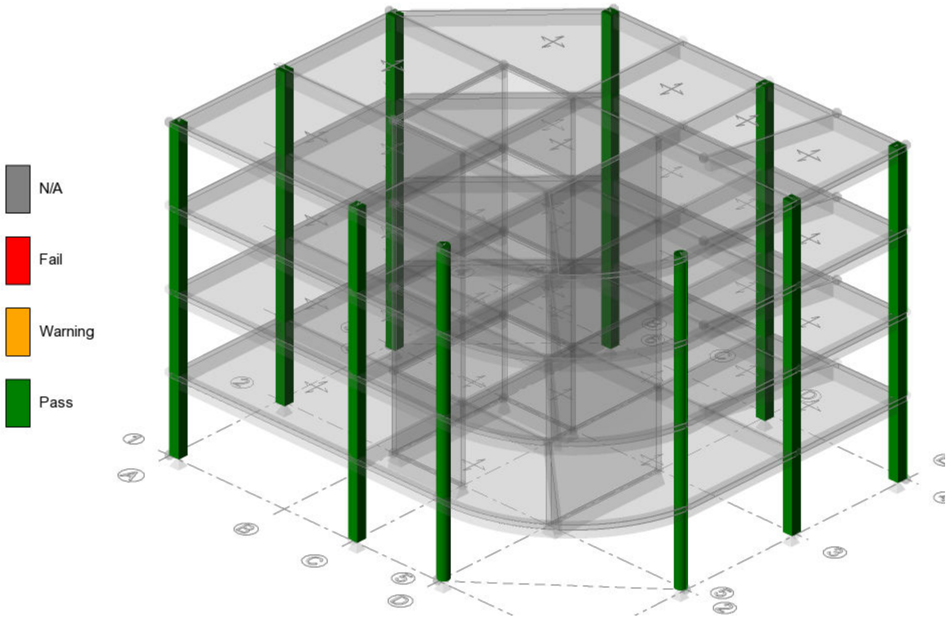
NOTE The sets of forces established from the FE chasedown analysis are considered by default for the design of each concrete member type. They can however be switched off should you decide that they are not required. The control for doing this for beams is located in Design Settings > Concrete > Beam > General Parameters. A similar control is provided for columns and walls also.



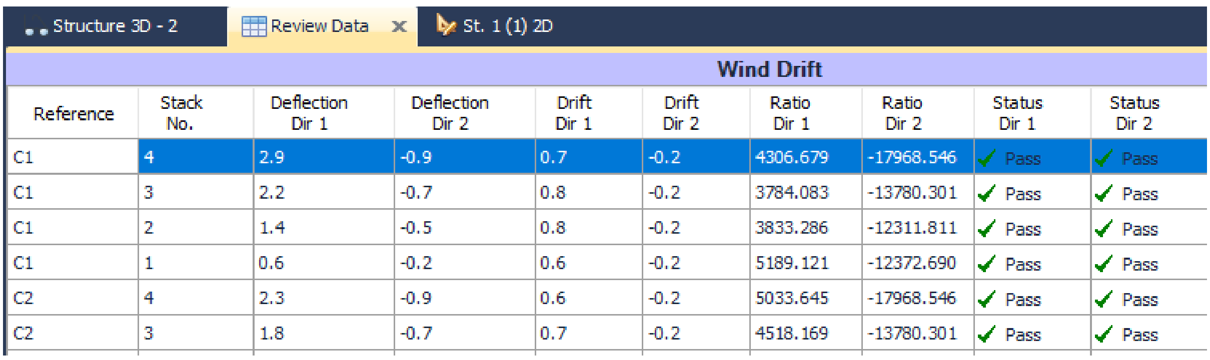
NOTE Reinforcement is designed, but member sizes are not changed during the design process.

Review stability issues

Issues relating to stability will be flagged in the Design branch of the Project Workspace Status tab. They can also be review graphically from Show/Alter State in the Review View.



Tabular results can be investigated from the Review View by clicking Tabular Data in the ribbon and then selecting the required View Type.

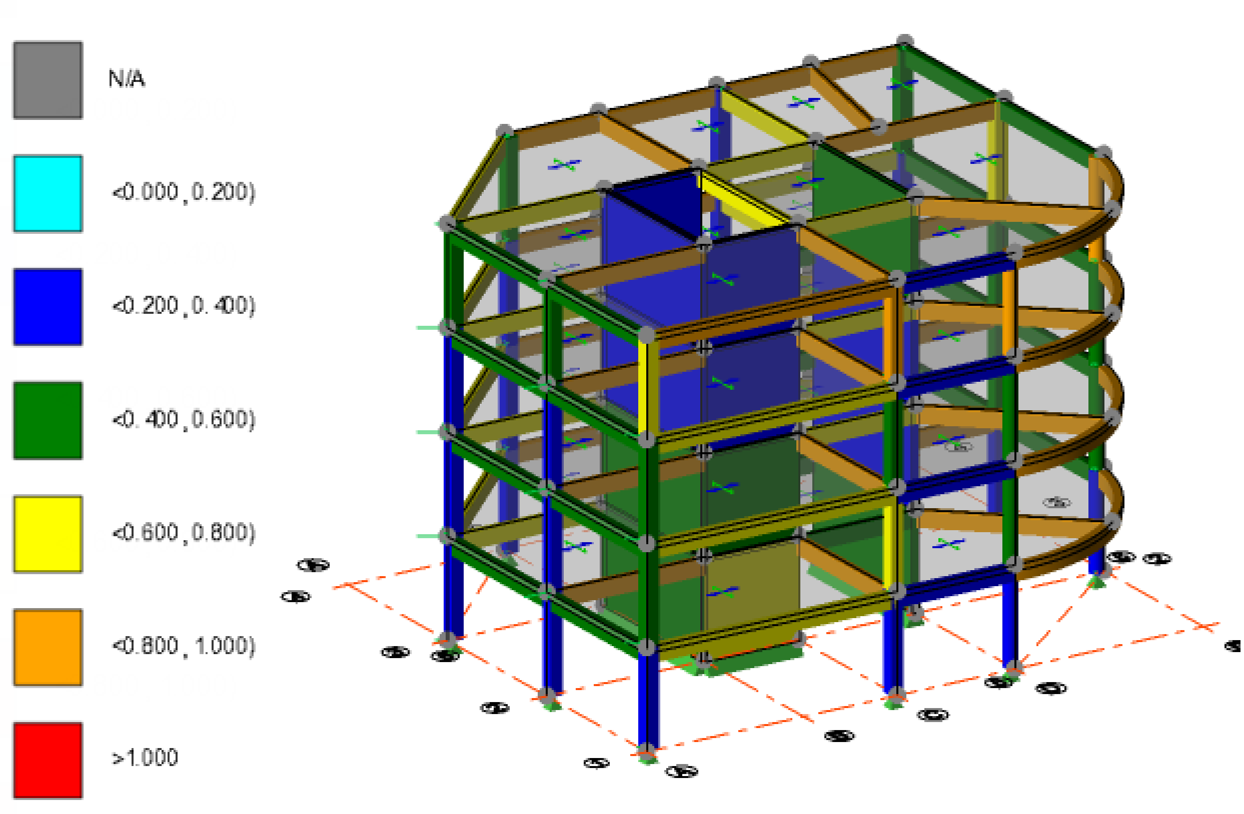


Wind Drift

Reference	Stack No.	Deflection Dir 1	Deflection Dir 2	Drift Dir 1	Drift Dir 2	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2
C1	4	2.9	-0.9	0.7	-0.2	4306.679	-17968.546	✓ Pass	✓ Pass
C1	3	2.2	-0.7	0.8	-0.2	3784.083	-13780.301	✓ Pass	✓ Pass
C1	2	1.4	-0.5	0.8	-0.2	3833.286	-12311.811	✓ Pass	✓ Pass
C1	1	0.6	-0.2	0.6	-0.2	5189.121	-12372.690	✓ Pass	✓ Pass
C2	4	2.3	-0.9	0.6	-0.2	5033.645	-17968.546	✓ Pass	✓ Pass
C2	3	1.8	-0.7	0.7	-0.2	4518.169	-13780.301	✓ Pass	✓ Pass

Review the design status and ratios

You can display the Design Status and Ratios from the Review View in order to determine if any remodelling is required.



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

If you make any changes, to see their effect simply re-run Design Concrete (Static) once more.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

Print calculations

Create a model report that includes the member design calculations that have been performed. (The default Member Design Calcs report includes these along with design calculations for other member types in the model).

Concrete member autodesign

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

Autodesign (concrete beam)

- When Autodesign is selected an iterative procedure is used to select longitudinal bars for each bending design region on the beam, both top and bottom. Similarly an iterative procedure is used to select links/stirrups for each shear design region on the beam.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete beams have been set to be designed using Design and detailing groups (concrete) (page 260), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Rationalization of Reinforcement

The Auto-design process returns a set of information about the reinforcement to be provided in each design region of the beam. The number and size of the longitudinal bars in the top and bottom of the beam is given as well as the size, number and spacing of the shear links/stirrups.

This information is then "rationalized" to give an arrangement of longitudinal reinforcement that provides a solution for the beam as a whole whilst still meeting the requirements of the individual design regions.

The rationalization process is carried out separately for the longitudinal bars in the top of the beam and those in the bottom of the beam.

The arrangement of shear links/stirrups is not rationalized. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

Autodesign (concrete column)

- When Autodesign is selected an iterative procedure is used to design both the longitudinal bars and links. This applies the spacing maximisation method which attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.

- For rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, autodesign will attempt a two layer solution. For circular sections, if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete columns have been set to be designed using Design and detailing groups (concrete) (page 260), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Autodesign (concrete wall)

- When Autodesign is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and links/ties. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

Select bars starting from

This option appears for concrete members when Autodesign is selected. It sets the autodesign start point for both longitudinal bars and links/stirrups/ties.

The options are:

- Minima (default)
- Current bars

Selecting Minima removes the current arrangement and begins with the minimum allowed bar size from the selection order.

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

Cracked, partially cracked, and uncracked concrete members

Assuming concrete sections (walls, columns and beams) are cracked has a direct affect on analysis - smaller Modification Factors are applied to cracked sections, (typically a cracked member is assumed to have half the stiffness of an uncracked member), causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.

The *Assume cracked* setting in the member properties is used to specify cracked/uncracked status. It can be set for individual concrete beam spans, column stacks and wall panels as follows:

- **Yes** - the section is fully (100%) cracked
- **No** - the section is uncracked (0%)
- **Partially** - the section is partially cracked (anywhere between 0 - 100%)

While it can be set directly in the member properties, *Assume cracked* can also be set or toggled graphically in a Show/Alter State Review View. For meshed walls a feature is provided to speed up this process.

Usage of cracked, partially cracked, and uncracked

This feature is of especial use in the design of tall buildings, for which the engineer needs to particularly assess and modify the stiffness of the lateral force resisting system (usually composed primarily of concrete (shear) walls) to control lateral displacements. The stiffness of concrete members is determined in large part by their state of cracking - which can vary between the 'extreme' conditions of entirely uncracked and fully cracked.

You have the option to specify the (fully) cracked and uncracked conditions for concrete members (beams, columns and walls) and control their stiffness accordingly and separately via Modification Factors (page 127).

NOTE The default values of these **for the Building Analysis** assume the loading is long term and should be modified when considering short term loading - for more see this TUA article Where do the default values for cracked and uncracked properties of concrete come from?.

You also have the option to specify an intermediate state, partially cracked i.e. neither uncracked nor fully cracked, which is applicable to beams, columns and walls (meshed). The partially cracked state is set by a simple degree of cracking % for which the stiffness modification factor is automatically calculated and applied.

Tall Building Companion Features - as mentioned, this feature adds to and can be considered in the context of a number of other Tekla Structural Designer features targeted particularly at the design of tall buildings. These are:

- Dynamic Analysis - Wind Tunnel Data Report Generation

- Design for high strength concrete grades (Eurocode) - for more on this see the Tekla Structural Designer 2019i Release Notes
- Stresses in 2D elements
- 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls
- Diaphragm loads and diaphragm load tables

Partially cracked modification factor

Assume cracked can be set to:

- Yes - the section is fully (100%) cracked
- No - the section is uncracked (0%)
- Partially - the section is partially cracked
  - Percentage - anywhere between 0 - 100%

The **partially cracked** modification factor ($ModFactor_{Partially\ Cracked}$) is automatically calculated by linear interpolation for the specified percentage (**x %**) between the factors set for uncracked ($ModFactor_{unCracked}$) and cracked ($ModFactor_{Cracked}$) as follows:

- $ModFactor_{Partially\ Cracked} = ModFactor_{unCracked} - \mathbf{x\%} * (ModFactor_{unCracked} - ModFactor_{Cracked})$
  - Thus for example for the default short term (Modal Analysis & RSA) concrete meshed wall modification factors of cracked = 0.5 and uncracked = 1.0, a Partially cracked value of 50% gives a modification factor = 1.0 - 0.5*(1.0 - 0.5) = 0.75.
  - The member stiffnesses are then adjusted accordingly using the calculated partially cracked modification factor values:
    - Meshed Walls - the factor is applied to properties E, G and/ or t as appropriate in the same manner as the cracked/ uncracked factors (specific to the factors currently set in the model)
    - Beams & Columns - the factor is applied to section properties in the same manner as the cracked/ uncracked factors (specific to the factors currently set in the model).

Determining wall cracked properties

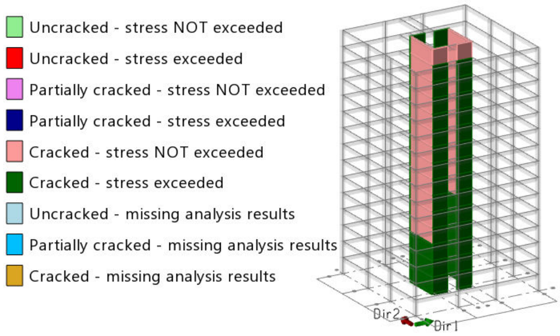
For engineers investigating the sway characteristics of taller buildings the cracked/uncracked status of walls is a complex topic of critical importance.

A couple of challenges faced by engineers are:

1. If using general stiffness adjustments - what are the realistic adjustment values for cracked and uncracked panels?

2. How can the cracked/uncracked status of wall panels be easily determined?

Tekla Structural Designer provides a feature in Show/Alter State to significantly assist with the latter task for meshed panels. This graphically displays the current cracked/uncracked status of each panel and indicates in which panels the stress threshold has and has not been exceeded. It then allows you to update the cracked status to be compatible with the threshold.



Use of the Review Wall Stress feature requires a degree of engineering judgment - you will need to make your own choices with regard to:

- strength or service level stress?
- basing the stress threshold on max instead of in-plane stress contours?
- whether the panel cracks for any level of stress, or could a higher threshold be considered?

You will also need to be familiar with the limitations and assumptions that apply.

Workflow for reviewing wall cracked properties

To determine an appropriate cracked status for wall panels, you could adjust the following basic workflow to suit your needs:

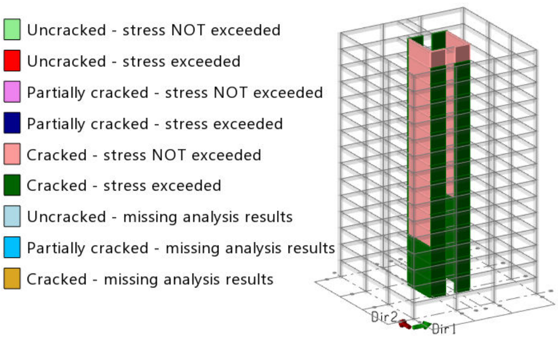
1. Analyze the model to determine some initial stresses.
2. Open a Review View and click Show/Alter State.
3. In the Show/Alter State **Properties** window:
    - Set the *Attribute* to **Assume cracked**
    - Set the *[M]ode* to
    - Set the *Result type* to **Strength** or **Service** as you deem appropriate.
    - Set the *Stress type* to **In-plane tension - Y** or **Max tension - Y** as you deem appropriate.

NOTE If using **In-plane tension - Y** contours:

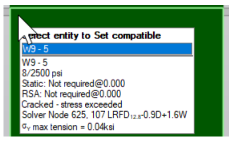
- these give an average stress through the thickness of the wall and therefore exclude local stress concentrations that exist as a result of out of plane bending
    - (engineers often choose to ignore out of plane bending when considering the overall cracked status of panels)

- Enter the *Stress threshold* that you want to work to.

In the below example, the most likely scenario where everything was initially set as cracked is shown; in lower panels this is a correct setting, in the higher ones they could be changed to uncracked.



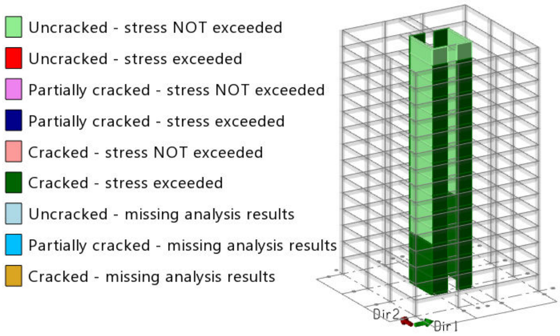
4. By hovering the cursor over an individual panel a tooltip is displayed. This reports the maximum stress value from all nodes in the panel and the case/combination in which it occurs. This value is used to determine if the stress threshold has been exceeded.



NOTE To investigate reported stresses in more detail you might choose to open a Results View to of tension stress contours for specific envelopes or combinations.

---

5. At this point you could choose to manually set or toggle the cracked/ uncracked status of the individual panels, however **Auto update** provides a quicker way to make all panels compatible.
6. To make all panels automatically compatible:
    - Click Auto update to expose the Auto update button.
    - Click the Auto update button



Each panel is updated as follows:

- uncracked panels in which the stress threshold has been not exceeded are unaffected,
- uncracked panels in which the stress threshold has been exceeded are set to cracked,
- cracked panels in which the stress threshold has been exceeded are unaffected,
- cracked panels in which the stress threshold has not been exceeded are set to uncracked.

7. Reanalyze the model to take into account the revised cracked/uncracked properties.
8. Review the results once more and make further adjustments if required.

Concrete beam and column groups

Why use design and detailing groups?

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

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design and detailing purposes - to standardise designs, reduce processing time, and reduce the volume of output created.

In a manual process, the Engineer might select a number of sufficiently similar members to form a "design group" to carry out a single design that is sufficient for all members in the group. Using this single set of design results, they would then create sub-groups of the members in the design group to produce a set of output details for each of these sub-groups.

In Tekla Structural Designer, concrete design groups are analogous to the manually created design groups described above. Concrete detailing groups are analogous to the sub-groups.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same number of stacks, bases must be of similar lengths in X and Y, and similar depths etc. The same rules also constrain manual group editing.

NOTE Grouped design and detailing is optional and can be deactivated if required:

From the **Design** tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

What happens in the group design process?

When the option to design a specific concrete member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the reinforcement so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the reinforcement is adequate for each and if this proves not to be the case, it is increased as necessary and the revised reinforcement is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

Design group requirements

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

Member type	Design group rules
Concrete beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties and nominal cover.

Member type	Design group rules
	• All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.
Concrete column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties and nominal cover.
Pad base	• A pad base may be in only **one** design group. • Each base in the group must have an identical cross section and depth. • Each base in the group must have identical eccentricities in X and Y. • Each base in the group must have identical material properties and nominal cover.
Pile cap	• A pile cap may be in only **one** design group. • Each pile cap in the group must have an identical cross section and depth. • Each pile cap in the group must have identical eccentricities in X and Y. • Each pile cap in the group must have identical material properties and nominal cover

Detailing group requirements

Each parent design group is sub-divided into one or more detailing groups.

Although there can be a "1 to 1" relationship between a design group and a detailing group, in practice there will often be a "1 to many" relationship as each design group is likely to require several detailing groups to allow for differences in the connected geometry.

Detailing groups are formed for the different concrete member types based on the following rules:

Member type	Design group rules
Concrete beam	• A detailing group may be associated with only **one** parent design group. • A beam element may be in only **one** detailing group.

Member type	Design group rules
	• Detailing groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • The cross section, including flange width where appropriate, span length and material properties in span • *"i"* of **all** beam elements in the group must be identical . • All beam elements in the group must have identical plan offsets. • All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans. • All beam elements in the group must have identical inclinations. • The support types and sizes, including the attached structure above and below the beam element, must be identical in all beam elements in the group however different support types and sizes in individual multi-span continuous beams are acceptable i.e. support *i* in beam element *j* must be identical to support *i* in all other beam elements in the group but supports *i* and *i+1* in beam element *j* may be different.
Concrete column	• A detailing group may be associated with only **one** parent design group, • A column may only be in **one** detailing group, • 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

Member type	Design group rules
	framing into it on all four sides, or not). These settings do not have to be identical for columns to be in the same group, but only if you have selected the option: Provide ties through floor depth for internal columns in Design Options > Concrete > Column > Detailing Options.
Pad base	• A detailing group may be associated with only **one** parent design group. • A base may be in only **one** detailing group. • The attached column cross-section above the base must be identical for all bases in the group however different support types are acceptable.
Pile cap	• A detailing group may be associated with only **one** parent design group. • A pile cap may be in only **one** detailing group. • The attached column cross-section above the base must be identical for all pile caps in the group however different support types are acceptable.

Group management

Automatic Grouping

Concrete beams and columns are grouped automatically.

In **Model Settings** > **Grouping** the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

Detailing groups cannot be edited manually.

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE	Any manually applied grouping will be lost if you elect to re-group!

Model Editing and Group Validity Checks

When new beam elements are created when a "split" or "join" command is run the resulting beam elements are automatically placed in existing design and detailing groups [or new groups created].

Concrete beam design aspects

Concrete type

While you can apply both normal and lightweight (LW) concrete in the beam properties, beam design using lightweight concrete is only available for Eurocodes.

When using other Head Codes beams can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE	LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

Deflection control (ACI/AISC)

Tekla Structural Designer implements both a simplified, and also a more rigorous method for controlling deflection.

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

When the rigorous method is applied, immediate short-term deflections, and also long-term deflections resulting from creep and shrinkage of flexural members are considered.

Structure supporting sensitive finishes

Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting Structure supporting sensitive finishes under the Design Control heading in the beam properties.

The deflection method applied to the beam depends on this setting as follows:

- beams not required to support sensitive finishes adopt the simplified method.
- beams required to support sensitive finishes adopt the rigorous method.

The simplified method

For those beams that have not being chosen to support sensitive finishes, deflection is controlled by the simplified method of limiting the span to depth ratio.

This check can be satisfied by providing a total beam depth which exceeds h_{min}, (where h_{min} depends on the clear span and beam end conditions).

If this check is not satisfied the beam is then rechecked by the rigorous method.

The rigorous method

For beams that do not meet minimum thickness requirements determined by the simplified method, or that support sensitive finishes, deflections should be calculated by rigorous method.

The parameters listed below directly affect the rigorous calculation method and hence require consideration:

1. Does the beam support sensitive finishes?
    - If the Structure supporting sensitive finishes beam property (located under the Design Control heading) is cleared the simplified method may suffice in which case rigorous calculations are not required.
2. Can the beam flanges be taken into account?
    - Selecting Consider flanges (located under the Design Control heading) can assist in reducing the calculated deflections.
3. Does the beam contain compression steel?
    - In the current release of the program it has been assumed that the beam always contains compression steel for the rigorous method deflection calculations.
4. 4. Are the deflection limits appropriate?
    - These limits are editable in the beam properties, the default ratios being span/360 for live load, and span/480 for total load affecting sensitive finishes.
    - The default span\over limits tend to produce conservative results.
    - You are also given the flexibility to specify absolute deflection limits as an alternative, or in addition to the above span\over limits.
5. Is the long term deflection period correctly specified?

- The default value is 5 years, but this can be edited via Design Options > Beam > General Parameters

6. Is the time at which brittle finishes are introduced correctly specified?
   - The default value is 1 month, but this can be edited via Design Options > Beam > General Parameters
7. Are the percentages of dead load applied prior to brittle finishes and long term live load correctly specified?
   - The deflection check takes account of these percentages, (defaulting to 50% of dead load applied prior to brittle finishes and 33% of long term live load). You are able to adjust each of these by selecting the loadcase name from the left hand side of the Loadcases dialog and then adjusting the value.

Deflection control (AS 3600)

Tekla Structural Designer controls deflections either by limiting span to depth ratios, or by applying the simplified method. The choice of method being set via Design Options > Concrete > Beam > General Parameters.

The simplified method

In the simplified deflection calculation procedure actual short-term deflection is calculated using the mean value of modulus of elasticity of concrete at appropriate age (Ecj) and an effective second moment of area of member (I_{ef}).

When the simplified method is applied, the shrinkage parameters that are required are specified on the Design Options > Concrete > Beam > General Parameters page.

Limiting span to depth ratios method

The deflection of reinforced concrete beams is not directly calculated and the serviceability of the beam is measured by comparing the calculated limiting effective span/effective depth ratio (L_{ef}/d)

Structure supporting sensitive finishes

Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting **Calculate deflection after installation of finishes** under the Design Limits heading in the beam properties.

When this option has been selected an additional span\over or absolute limit can be specified and checked against in the beam properties.

• beams not required to support sensitive finishes adopt the simplified method.

• beams required to support sensitive finishes adopt the rigorous method.

Parameters affecting deflection

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. 1. Increase reinforcement if deflection check fails
    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results
    - If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.
2. Consider flanges
    - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Deflection control (Eurocode BS and IS)

Tekla Structural Designer controls deflection by comparing the calculated limiting span/effective depth ratio L/d to the maximum allowable value $(L/d)_{max}$

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. Increase reinforcement if deflection check fails
    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results

    If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.
2. Structure supporting sensitive finishes (Eurocode only)
    - This beam property (located under the Design Control heading, and on by default) is used to control the value of the f2 parameter used in the deflection check. When unchecked f2 will be taken as 1.0.
3. Consider flanges
    - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Ignore lateral instability (Eurocode)

This option (located under the Design Control heading) allows you to consider or ignore lateral instability for slender spans to EC2 clause 5.9(1) (off by default). When the option is checked on the slender span check is excluded from design.

Consider flanges

Flanged beam properties can be specified under the Design Control heading in the beam properties, by selecting Consider Flanges.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

The characteristic behaviour of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Related concept

Flanged concrete beams (page 272)

Design parameters (Eurocode only)

Located under the Design parameters heading in the beam properties, the following parameters relating to shrinkage and creep can be specified for individual members.

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

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

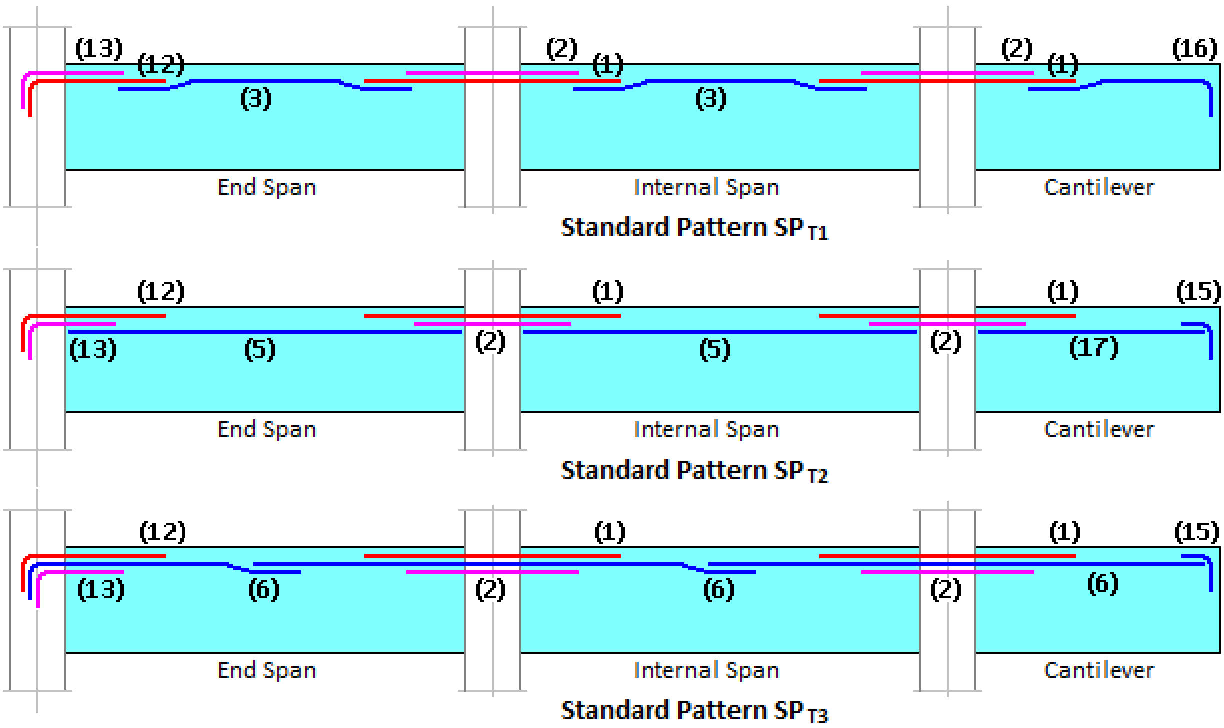
Nominal cover

In the beam properties, the nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement - longitudinal bar patterns

Under the reinforcement heading in the beam properties, there are three Standard Patterns available for top reinforcement, SP_{T1}, SP_{T2} and SP_{T3} and two Standard Patterns available for bottom reinforcement, SP_{B1} & SP_{B2} as illustrated in the figures below.

Standard Patterns of Top Reinforcement



The bars used in the Standard Top Patterns are:

(1) Straight bar extending to approximately 25% or 33% of each span (end points of this bar are determined by the design region settings)

(2) Straight bar extending to approximately 10% of each span (end points of this bar are determined by the design region settings) - if required by the design

(3) Double cranked bar lapped with bar (1)

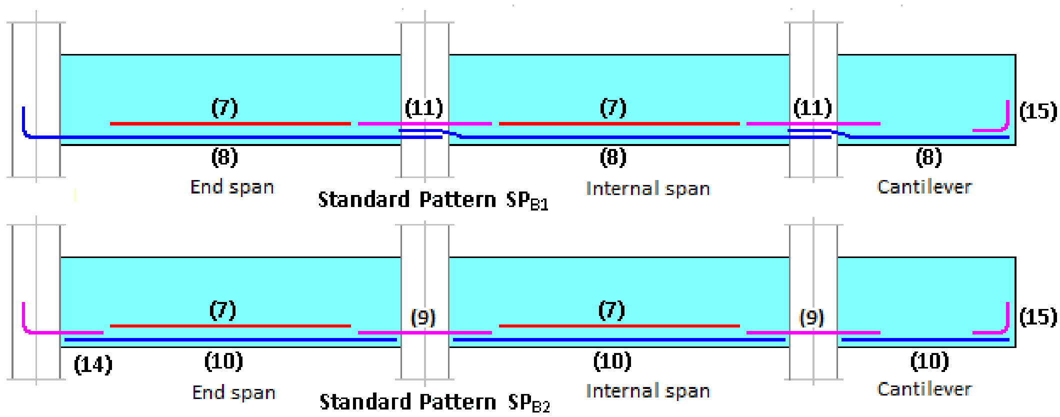
(5) Straight bar running approximately from face to face of beam supports

(6) Single cranked bar running from center span to center span with the option to merge bars if they are the same size and number to extend the bar over several spans

(12) Bob bar

(13) Bob bar

Standard Patterns of Bottom Reinforcement



The bars used in the Standard Bottom Patterns are:

(4) Bar with a bob at each end

(7) Straight bar with a length approximately 70% of span - if required by the design

(8) Single cranked bar extending over several spans or over one span only and lapped within a support - with bob if it continues over an end span.

(9) Straight bar

(10) Straight bar running approximately from face to face of beam supports

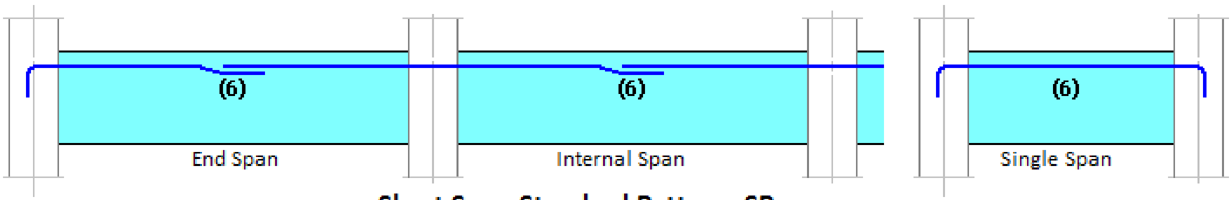
(11) Straight bar

(14) Bob bar

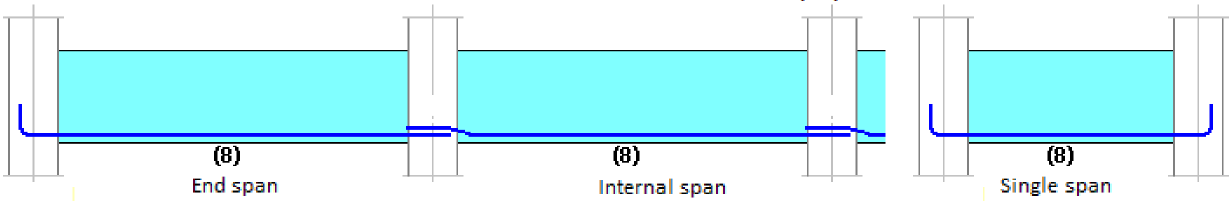
Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in **Design Options > Beam > Reinforcement Settings** and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams



Short Span Standard Patterns SP $_{T1, T2, \& T3}$



Short Span Standard Patterns SP $_{B1 \& B2}$

Flanged concrete beams

Consider flanges

Flanged beam properties can be specified under the Design Control heading in beam properties, by selecting Consider Flanges.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

The characteristic behaviour of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Validation of slabs for use in the flange effective width calculations

If a slab is present (and provided that a user defined flange has not been specified), the program automatically validates the slab as a potential candidate for being a beam flange using a number of criteria, the main ones being;

• the slab can be on one or both sides of the beam but

• it must extend for a distance ≥ the slab depth from the vertical face of the beam and

• it must extend for the full span length of the beam

• the slab must be a reinforced concrete slab

• if there are slabs on both sides of the beam, they may be of different depths and these depths may vary along the length of the beam

The effective width of any **valid** slab on each side of the beam, $b_{eff,i}$, is calculated and the results that are appropriate at the mid-span length point are displayed along with the flange depth, under **Design control** in the Beam Property dialog.

NOTE When automatically calculated, the flange width and depth are only displayed in the Beam Property dialog and not in the Beam Properties window, (because the width and depth could vary if multiple beams were to be selected).

Include flanges in analysis

Selecting this option allows the flanged beam section properties to be considered in the analysis, stiffening the beam and reducing the deflection.

Consider as isolated (ACI/AISC)

ACI 318 clause 8.12.4 states:

"Isolated beams, in which the T-shape is used to provide a flange for additional compression area shall have a flange thickness not less than one-half the width of web and an effective flange width no more than four times the width of web."

When the **Consider flanges** check box is selected, an **Isolated Beam** check box is displayed to control whether or not the above code limit is applied. When the check is performed, if the flange geometry does not meet the above requirements the flanges are ignored.

NOTE Our understanding is that while this limit usually applies to precast beams, it is not usually applied to in-situ construction. Therefore, by the default the Isolated Beam check box is cleared, which means that the above check is not performed.

Effective Width of flanges (ACI/AISC)

For ACI 318-08 and ACI 318-11

For "T" shaped flanged beams the effective flange width, b_{eff}, is given by :

b_{eff} = MIN(L/4, 16*h_f + b_w, b_1 + b_2 + b_w) - O_{wi}

For beams with slab one side only, the effective flange width, b_{eff} is given by :

b_{eff} = MIN(L/12+ b_w, 6*h_f + b_w, b_i + b_w) - O_{wi}

where

L = span length

IF construction is continuous:

= distance of center-to-center of supports

ELSE

= MAX(clear span + h, distance between centers of supports)

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centerline of any supporting steel beam

O_{wi} = the user specified allowance for an opening

For ACI 318-14

All limits on the flange width apply to the overhangs on each side of the beam. It is also clarified in this version that the clear span should be used in these calculations.

$b_{eff,i} = MIN(l_n/8, 8*h_f , b_i) - O_{wi}$

Effective Width of flanges (Eurocode)

The effective width of the compression flange is based on L_0, the distance between points of zero bending moment.

For flanged beams the following values of L_0 are to be used;

For a simply supported beam $L_0 = L$

For a continuous beam, the value of L_0 may be obtained using the following simplified rules;

End span of a continuous beam with a pinned end support $L_0 = 0.85*L$

End span of a continuous beam with a fixed end support $L_0 = 0.70*L$

Internal span of a continuous beam $L_0 = 0.70*L$

where

L = the clear length of the span under consideration

The effective flange width, b_{eff}, is given by;

$b_{eff} = b_w + \sum b_{eff,i}$

where

$b_{eff,i}$ = the effective width of the flange on side i of the beam

$= MIN[0.2*L_0, b_i,(0.2*b_i + 0.1*L_0)] - O_{w\,i}$

where

L_0 = the distance between points of zero moment as defined above

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centreline of any supporting steel beam

b_w = the width of the beam

O_{wi} = the user specified allowance for an opening

NOTE If the slab thickness varies on each side of the beam, the thinner value is used in calculating the beam properties.

The above calculation for b_{eff} is also used for "Γ" beams with a slab on only one side although in this case, b_1 or b_2 as appropriate is = 0.

Concrete column design aspects

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Apply rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the Rigid zones not applied option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the Apply rigid zones option that is provided in the column properties under the Design control heading.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the positions of releases in analysis model are affected, and member start and end points for design are also adjusted.

The Apply rigid zones setting is located under the Design control heading in the column properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the column properties, the following parameters relating to shrinkage and creep can be specified for individual members.

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

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Confinement reinforcement

Located under the Confinement reinforcement heading in the column properties, the **Provide support regions** setting determines the way each stack is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.
- Cleared - the same confinement reinforcement is designed for the whole stack.

Slenderness

Located under the Slenderness heading in the column properties, the significant parameter within the slenderness criteria is a choice of how the column is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) which is the clear dimension between the restraining beams at the bottom of the stack and the restraining beams at the top of the stack. The unsupported length may be different in each direction.

Effective concrete beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the column. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the stack if it is within the depth of the stack section from the end of the stack, and if its centre is nearer to this end of the stack than the far end. Therefore, at a node at a stack join, if the top of the beam is below the node by a dimension greater than the depth of the stack below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the stack above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a stack, Tekla Structural Designer considers whether there is a flat slab restraining the stack at that end. The Use slab for calculation... upper/ lower, major/minor options, (which are located under the Stiffness heading in the column properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the stack if it is within half the slab depth from the end of the stack, and if its centre is nearer to this end of the stack than the far end.

If, at an end of the stack, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the stack beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next stack (and so on). If there is no stack beyond this restraint (i.e. this is the end of the column), the unsupported length ends at the end of the column.

Sway/Drift Checks

By default all stacks of all columns are taken into account in the sway/drift, wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the column properties, these parameters provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.

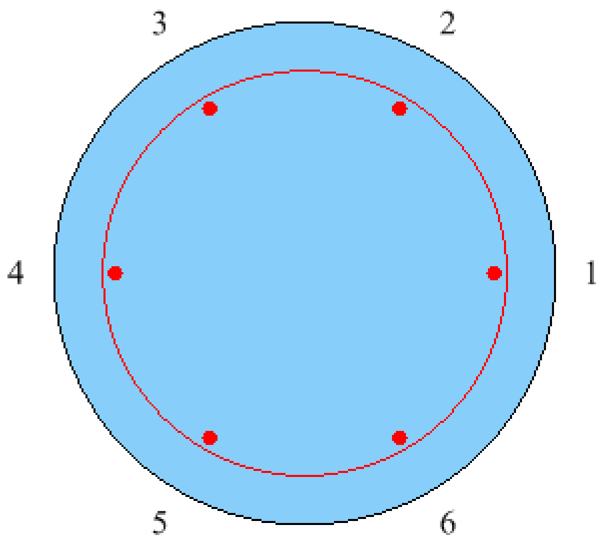
Nominal cover

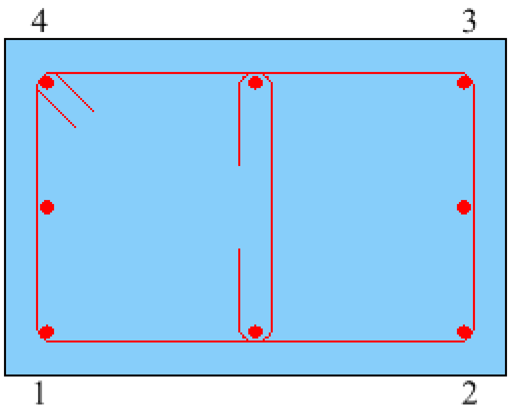
In the column properties, the nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

The type and grade of vertical and confinement reinforcement to be considered for the design are specified under the Reinforcement heading in the column properties.

Where possible bars are arranged in a single layer:

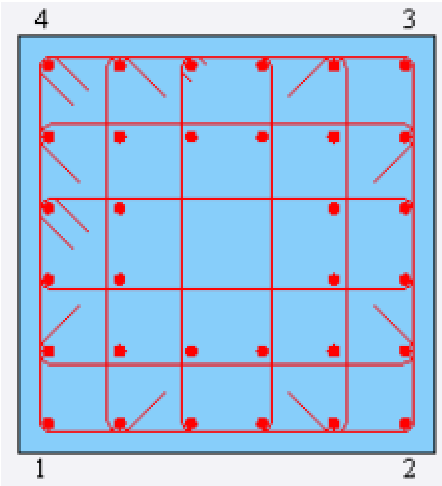


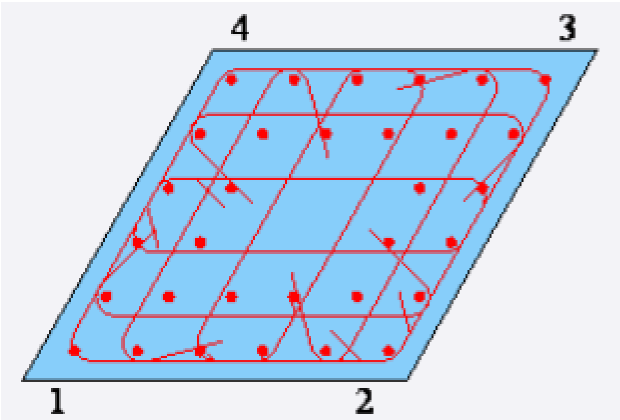


Using a 2nd layer of reinforcement

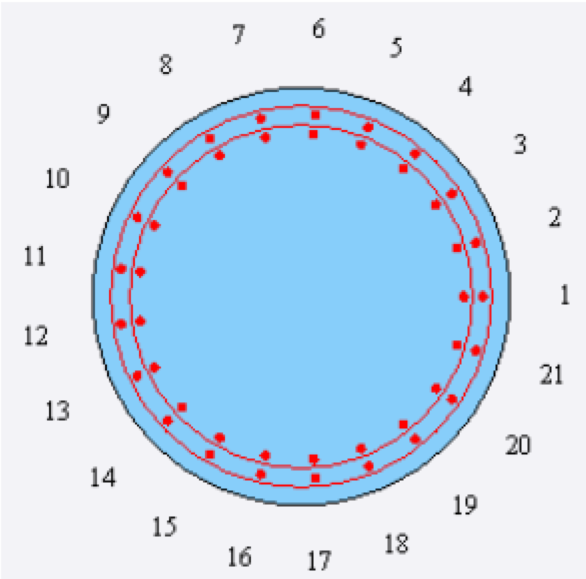
- In Autodesign: for rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, design will attempt a two layer solution.
    - Rectangular and parallelogram sections: a second layer is added in line with the first bar along the adjacent length.

NOTE At least 2 intermediate bars are needed in each direction for a 2nd layer to be added.





- Circular sections: if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).



- A link is added along the second layer for all 3 section shapes.
- In Interactive Design: you can choose to toggle the second layer on or off in the Interactive Design dialog.
- When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

Concrete wall design aspects

Concrete type

While you can apply both normal and lightweight concrete in the wall properties, wall design using lightweight concrete is currently beyond scope.

End 1 and End 2 extensions

Wall extensions (End 1/End 2) can be applied in the wall properties in order to remove physical overlaps with adjoining walls and columns without compromising the integrity of the underlying analysis model.

Negative extensions can be created automatically where appropriate. Extensions can also be defined manually if required, in which case they can be input with either positive or negative values:

- A positive extension extends the wall length beyond its insertion point.
- A negative extension trims the wall back from the insertion point.

Although the length of the wall used in the analysis model (L_{wall}) is unchanged, the wall length that is used in the design, quantity reporting and drawings changes to $L_{wall,d}$

Reinforcement layers

Either one or two layers of reinforcement can be specified in the wall properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the wall properties, the following parameters relating to shrinkage and creep can be specified for individual walls.

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

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Sway/Drift Checks

By default all panels of all walls are taken into account in the sway/drift , wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the wall properties, these parameters provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All panels' to exclude the entire wall, or you can exclude a particular panel by clearing the check box located under that panel only.

Confinement reinforcement

Located under the Confinement reinforcement heading in the wall properties, the **Provide support regions** setting determines the way each panel is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.
- Cleared - the same confinement reinforcement is designed for the whole panel.

Slenderness

Located under the Slenderness heading in the wall properties, the significant parameter within the slenderness criteria is a choice of how the wall is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the head code being worked to) which is the clear dimension between the restraining beams at the bottom of the panel and the restraining beams at the top of the panel. The unsupported length may be different in each direction.

Effective Concrete Beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the wall. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the panel if it is within the depth of the panel section from the end of the stack, and if its centre is nearer to this end of the panel than the far end. Therefore, at a node at a panel join, if the top of the beam is below the node by a dimension greater than the depth of the panel below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the panel above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a panel, Tekla Structural Designer considers whether there is a flat slab restraining the panel at that end. The Use slab for calculation... upper/ lower, major/minor options, (which are located under the Stiffness heading in the wall properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the panel if it is within half the slab depth from the end of the panel, and if its centre is nearer to this end of the panel than the far end.

If, at an end of the panel, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the panel beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next panel (and so on). If there is no panel beyond this restraint (i.e. this is the end of the wall), the unsupported length ends at the end of the wall.

Nominal cover

Nominal concrete cover is specified in the wall properties.

For walls, it is measured as follows:

- For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.
- For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/confinement transverse reinforcement that may be present.

Reinforcement

Under the Reinforcement heading in the wall properties, the Reinforcement layers, Form and Include end zones properties can be combined as required in order to obtain a range of reinforcement patterns, e.g:

- Single layer, using mesh reinforcement
- Two layers, using mesh reinforcement
- Single layer, using loose bars

- Two layers, using loose bars
- End zones, with a single layer of mesh in the mid zone
- End zones, with two layers of mesh in the mid zone
- End zones, with a single layer of loose bars in the mid zone
- End zones, with two layers of loose bars in the mid zone

Interactive concrete member design

The combined analysis and design processes, **Design Concrete (Static)**, **Design All (Static)** etc. are complemented by the program's interactive member design facilities. These allow you to interact with the concrete member designs to override the results arising from the auto-design process.

The following interactive member designs are provided:

- Interactive concrete beam design (page 285)
- Interactive concrete column design (page 290)
- Interactive concrete wall design (page 310)

Generally you are advised to perform interactive member designs only after the Design All process has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.

The Interactive design dialogs display a limited selection of the relevant critical design results including bar details and allow you to make changes to the number, size and spacing (for links/ties only) of the selected bars.

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

Interactive concrete beam design

Opening the Interactive Beam Design Dialog

The Interactive Beam Design dialog (page 286) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Beam Design dialog is displayed.
3. Interactively adjust the reinforcement as required until the design is satisfactory.

Changing the bar pattern used in the Interactive Beam Design Dialog

The interactive dialog can be opened from any of the 2D or 3D Views as follows:

1. If the Interactive Beam Design Dialog is open, click Cancel to close it.
2. If necessary, re-select the beam to be designed.
3. Change the Top and Bottom longitudinal bar pattern in the Properties Window as required.
4. Hover the cursor over the beam until its outline is highlighted, then right-click.
5. From the context menu select Interactive Design...

The Interactive Beam Design Dialog opens and reinforcement is automatically re-selected for the beam based on the new bar pattern.

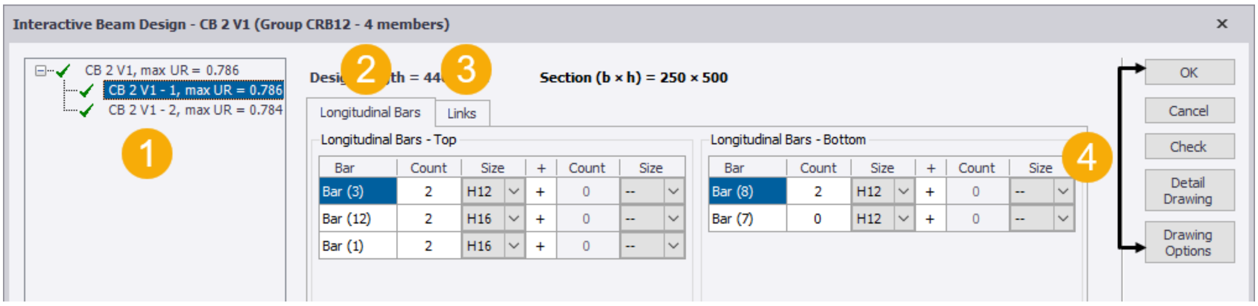
Interactive Beam Design dialog

The **Interactive Beam Design dialog** shows the current reinforcement and check results for each beam span in the selected beam line. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

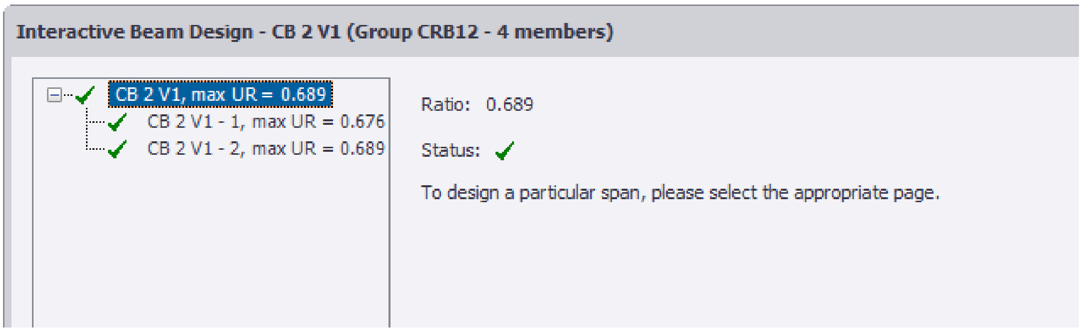
To display the dialog:

1. Right click on an existing concrete beam.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Beam/span summary pane

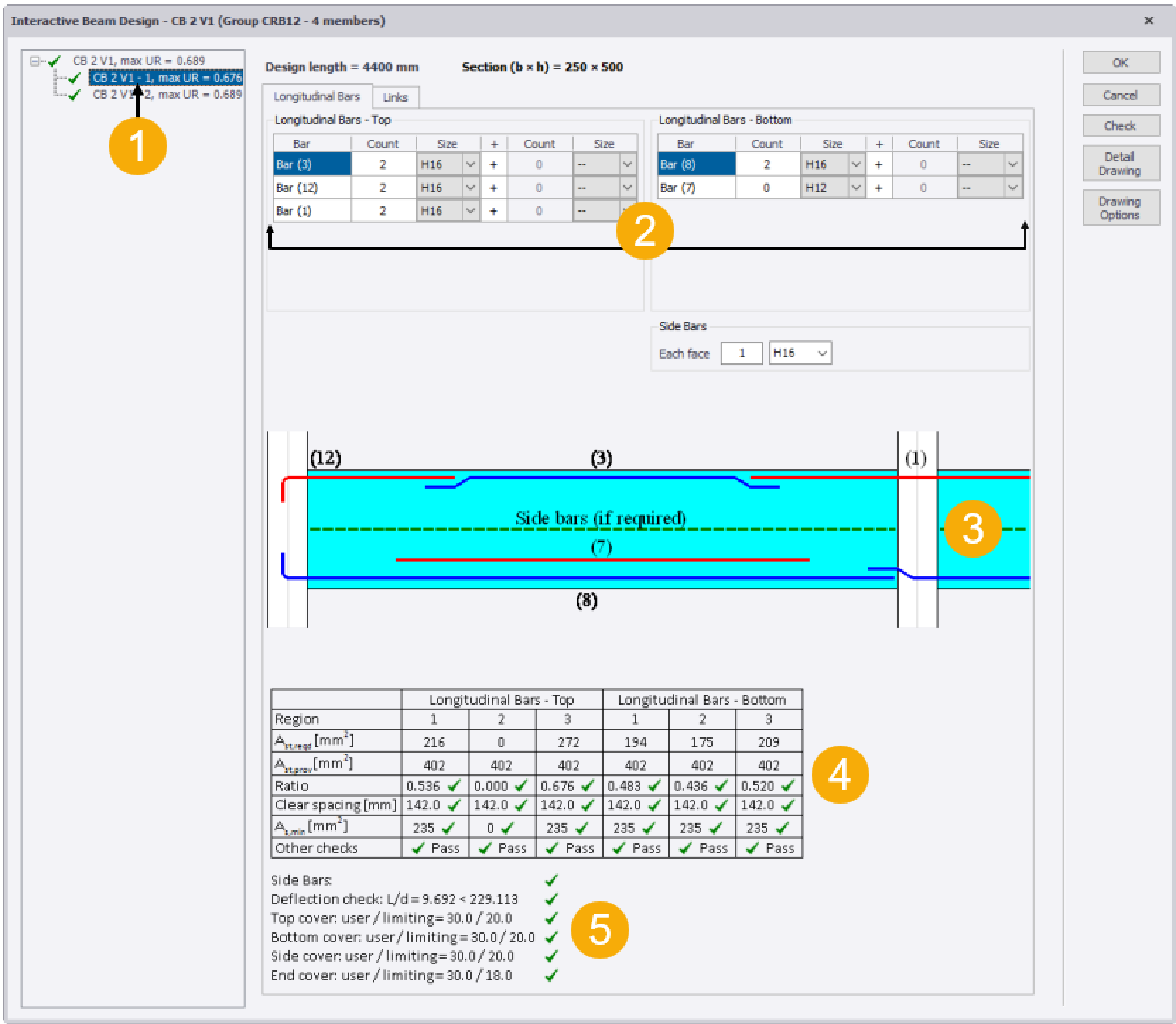


The top row in this pane shows the beam line summary, consisting of the overall utilization ratio and design status.

Subsequent rows show the design status of each span and associated utilization ratio.

- To design a particular span, click on the corresponding row for that span in the summary pane.

2. Longitudinal Bars tab

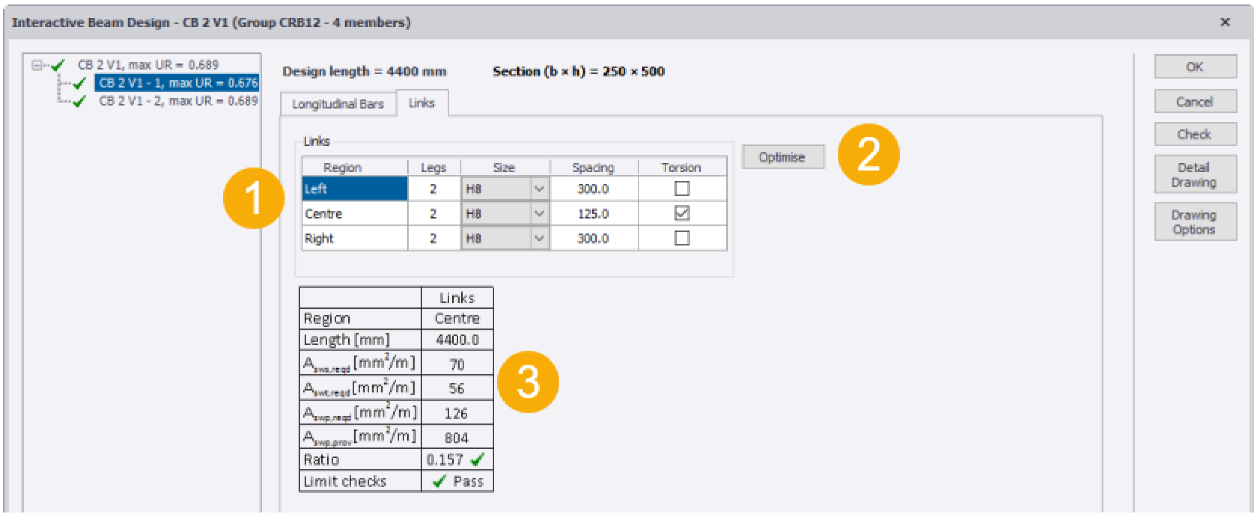


1. **Span Summary**: Displays the design status of the selected span and the associated utilization ratio.
2. **Bar Selection Tables**: Used for editing the longitudinal bars into the beam.
    - Each row in the table is labeled with a specific “bar number” (taken from the standard patterns applied to the beam in the Properties Window); these represent bar locations within the beam.
    - Two different bar sizes can be defined in each row, the only restriction being that the second bar must always be smaller than the first.
    - The number of bars of each size is defined using the **Count** field.

- When bars are joined to the adjacent span, changing those bars within this span has the effect of changing those bars in the adjacent span, as they are effectively the same bar. (This is only done when the spans are "matching" in terms of their alignment and dimensions.)

3. **Bar Pattern Layout**: a schematic diagram representing the top and bottom patterns assigned to the beam.
4. **Design Summary Table**: Displays critical results for each of the design regions from all combinations:
   - Area of reinforcement required, $A_{s,reqd}$
   - Area of reinforcement provided, $A_{s,prov}$
   - Reinforcement area utilization ratio
   - Smallest clear spacing between bars
   - Minimum required reinforcement area, $A_{s,min}$
5. **Additional checks**: Side bar, deflection and cover check results are displayed below the design summary table.

3. Links/Stirrups tab



1. **Link/Stirrup Selection Table**: Specifies the number of link/stirrup legs, size and spacing in each of the regions.
2. **Optimise Button** : This calculates the optimum length of the central region given the reinforcement that you have selected. The button is not be visible when the beam is in a design group with other beams, and is also not visible when the span is a cantilever.
3. **Link/Stirrup Design Summary Table**: Displays the most critical result from all combinations:

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

4. **Buttons**: (See separate section below.)

Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog

See also

Interactive concrete member design (page 285)

Interactive concrete column design

Opening the Interactive Column Design Dialog

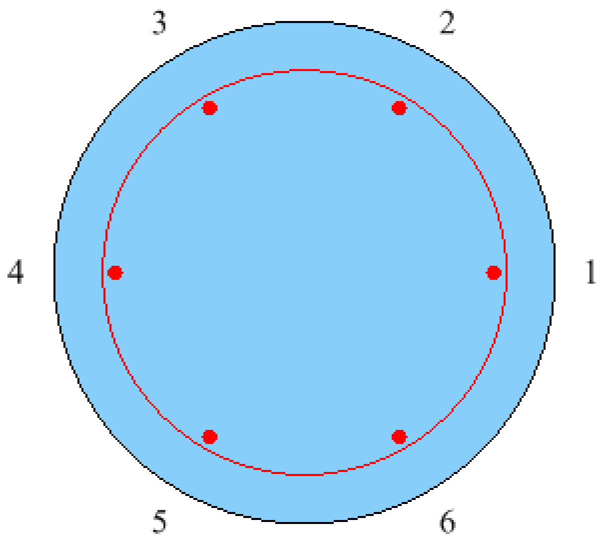
The Interactive Column Design dialog (page 302) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Column Design dialog is displayed.
3. Click on an individual stack in the Column/stack summary pane (page 303).
4. Interactively adjust the reinforcement as required for the chosen stack until the design is satisfactory.

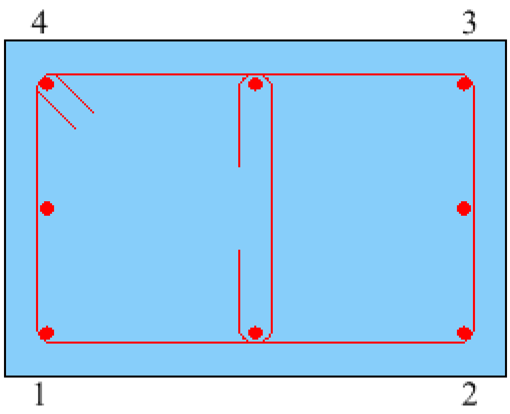
Arranging bars in the Interactive Column Design Dialog

The way in which bars are arranged depends on the column shape.

In circular columns, bars are arranged simply by modifying the bar size and count fields.



In rectangular columns bars are arranged as described below.



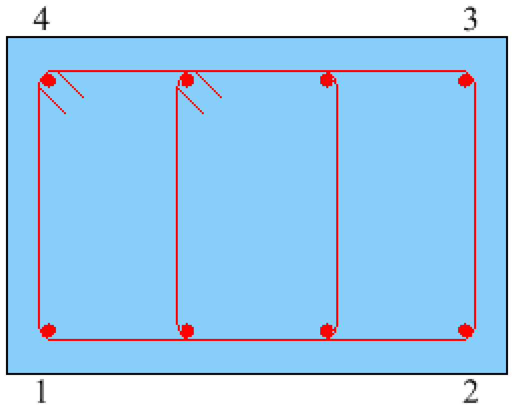
Principal bars exist at fixed locations; they are labeled with numbers in the cross-section, as shown above. You can only change the principal bar sizes, not their locations.

Intermediate bars are the unnumbered bars in the cross-section. You can change both their size and number. They are defined in the bar location table by reference to the principal bars between which they lie.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	1	249.0	✓	3-4	1	249.0	✓
2-3	1	149.0	✓	4-5	1	149.0	✓

A count of "1" for each intermediate length in the bar location table indicates that a single intermediate bar is positioned between each of the principal bars.

If the count is increased to "2" for Int. length 1-2, but reduced to "0" for Int. length 2-3, the following arrangement is achieved. (Two intermediate bars are positioned between principal bars 1 and 2, but there are now no intermediate bars between principal bars 2 and 3.)



Note that Int. lengths 3-4 and 4-5 are adjusted automatically in the table to match.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	2	166.0	✓	3-4	2	166.0	✓
2-3	0	298.0	✓	4-5	0	298.0	✓

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

- Single links/ties,
- Double links/ties,
- Triple links/ties,
- Cross links/ties.

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

Using a 2nd layer of bars

It is sometimes not possible to find a solution using a single-layer of reinforcement - (a problem more common in hi-rise structures with large columns in lower stacks).

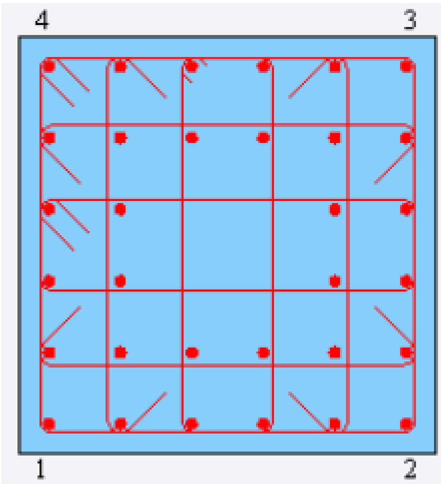
Selecting the **Use 2nd layer** option in the interactive design provides you with more flexibility to find a solution in these cases.

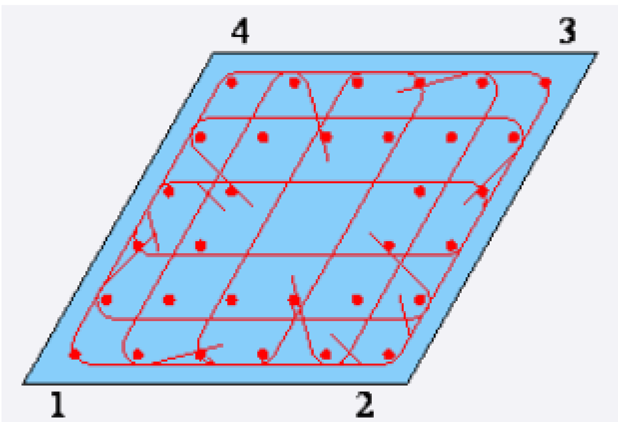
A 2nd layer can be applied to rectangular/parallelogram and circular sections.

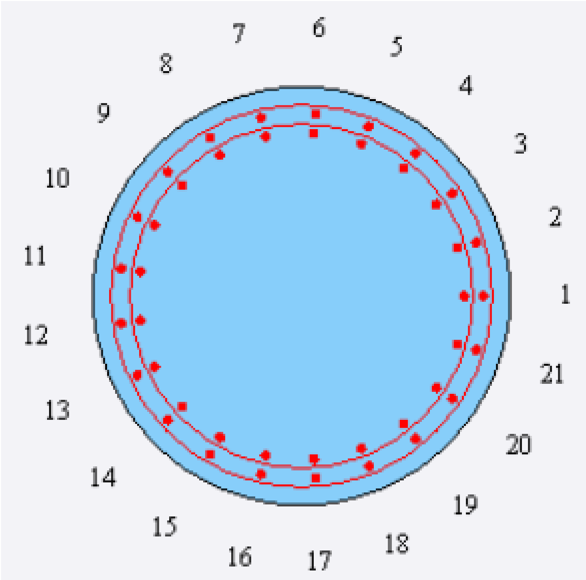
Rectangular/parallelogram: a second layer in line with the first bar along the adjacent length

- At least 2 intermediate bars are needed in each direction

A link is added along the second layer for all 3 section shapes.







When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

Column interaction diagrams (US customary units)

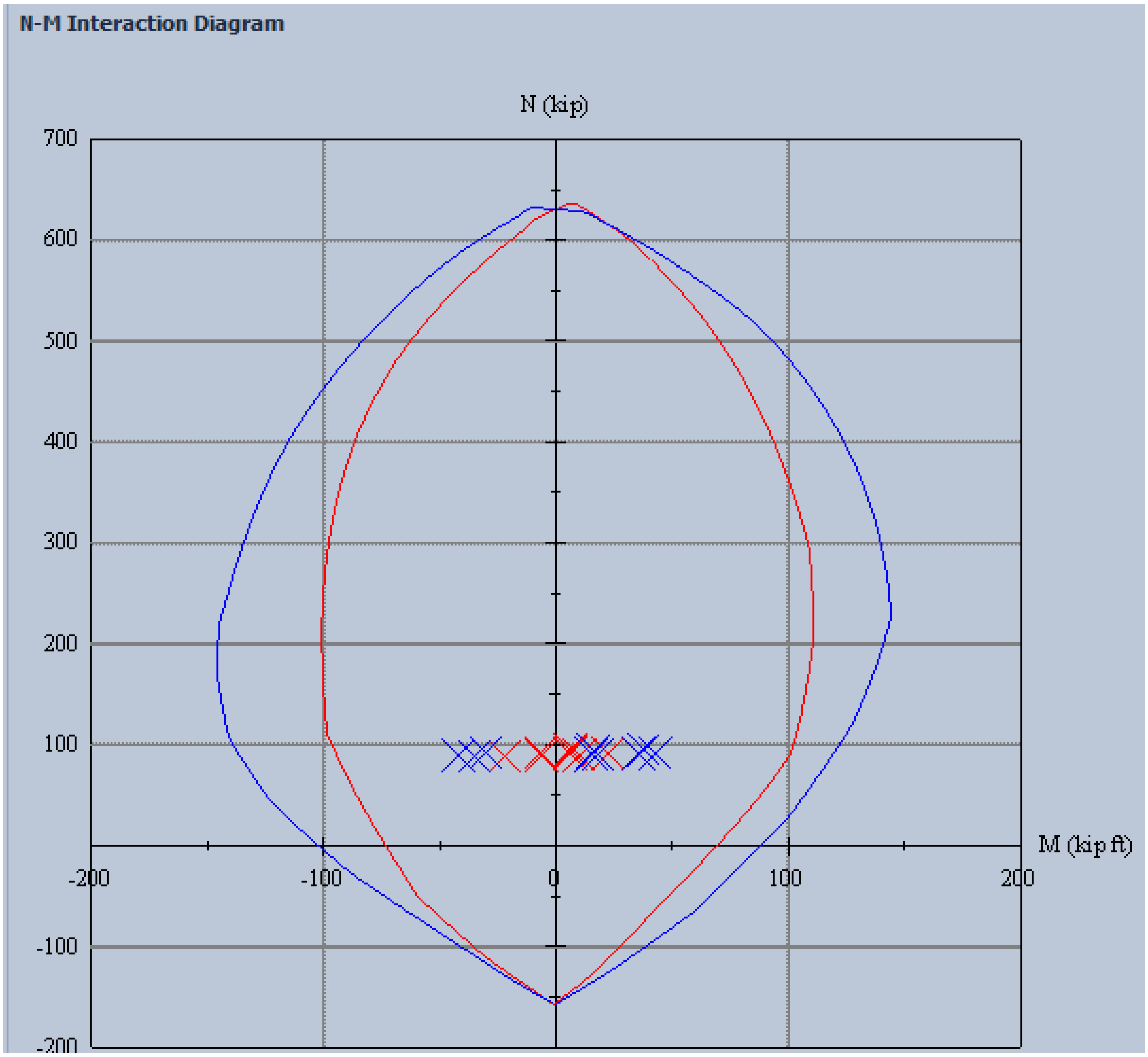
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

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

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

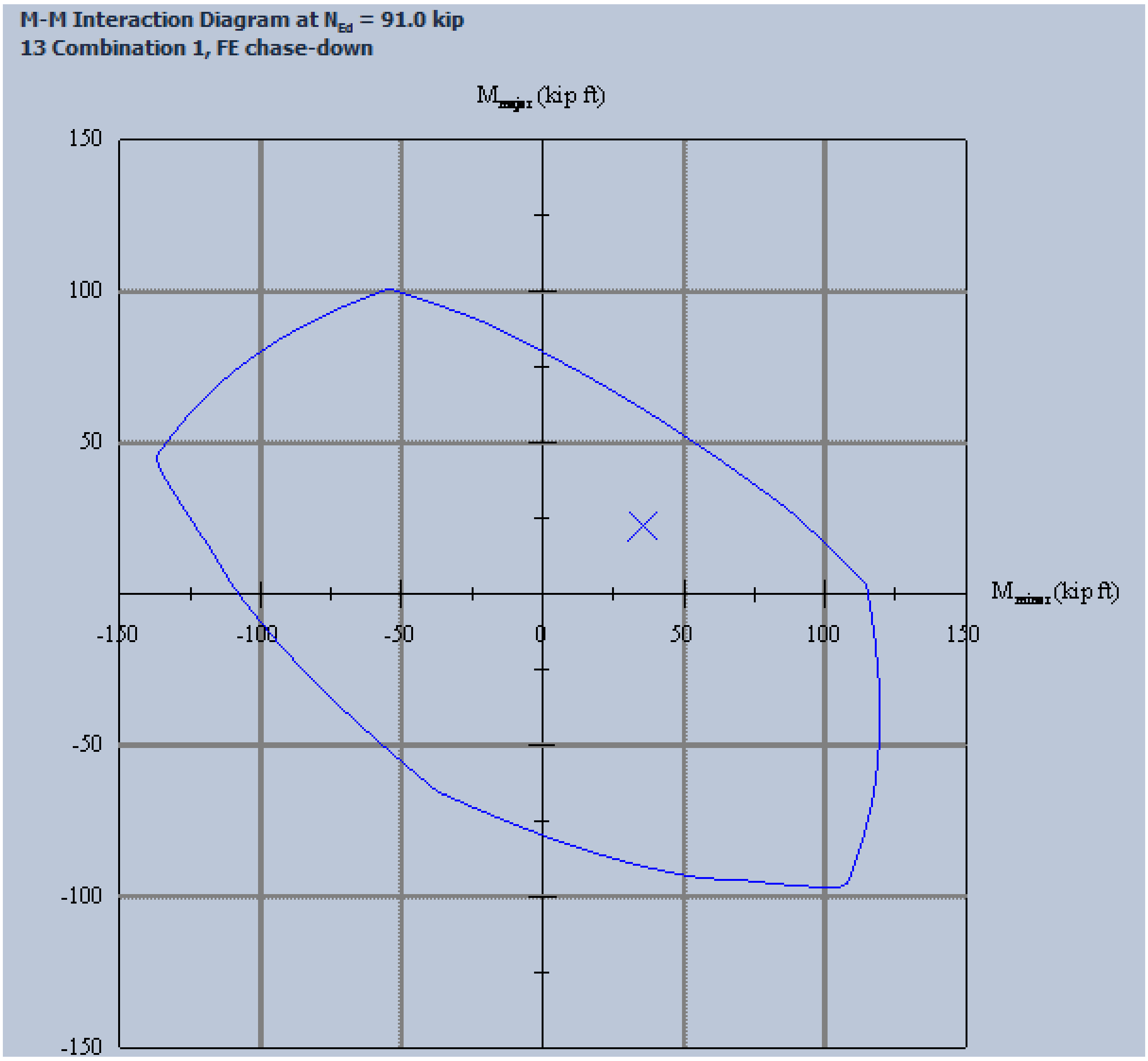


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

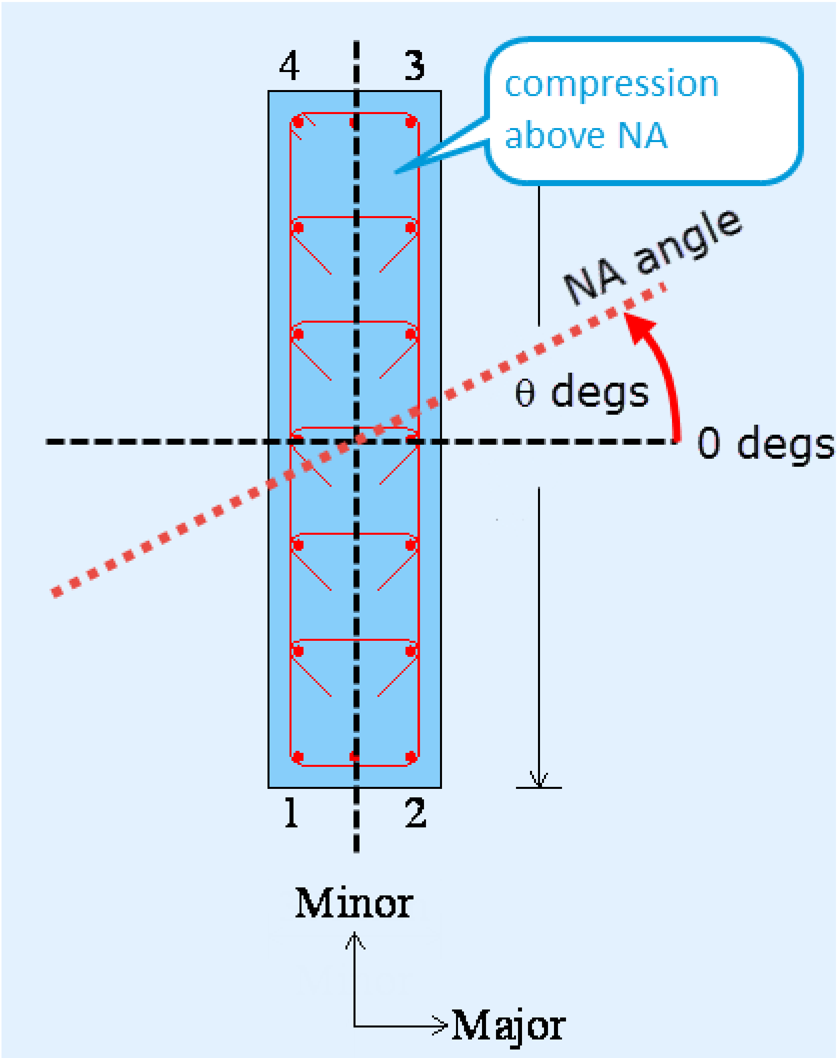
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla

Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Column interaction diagrams (metric units)

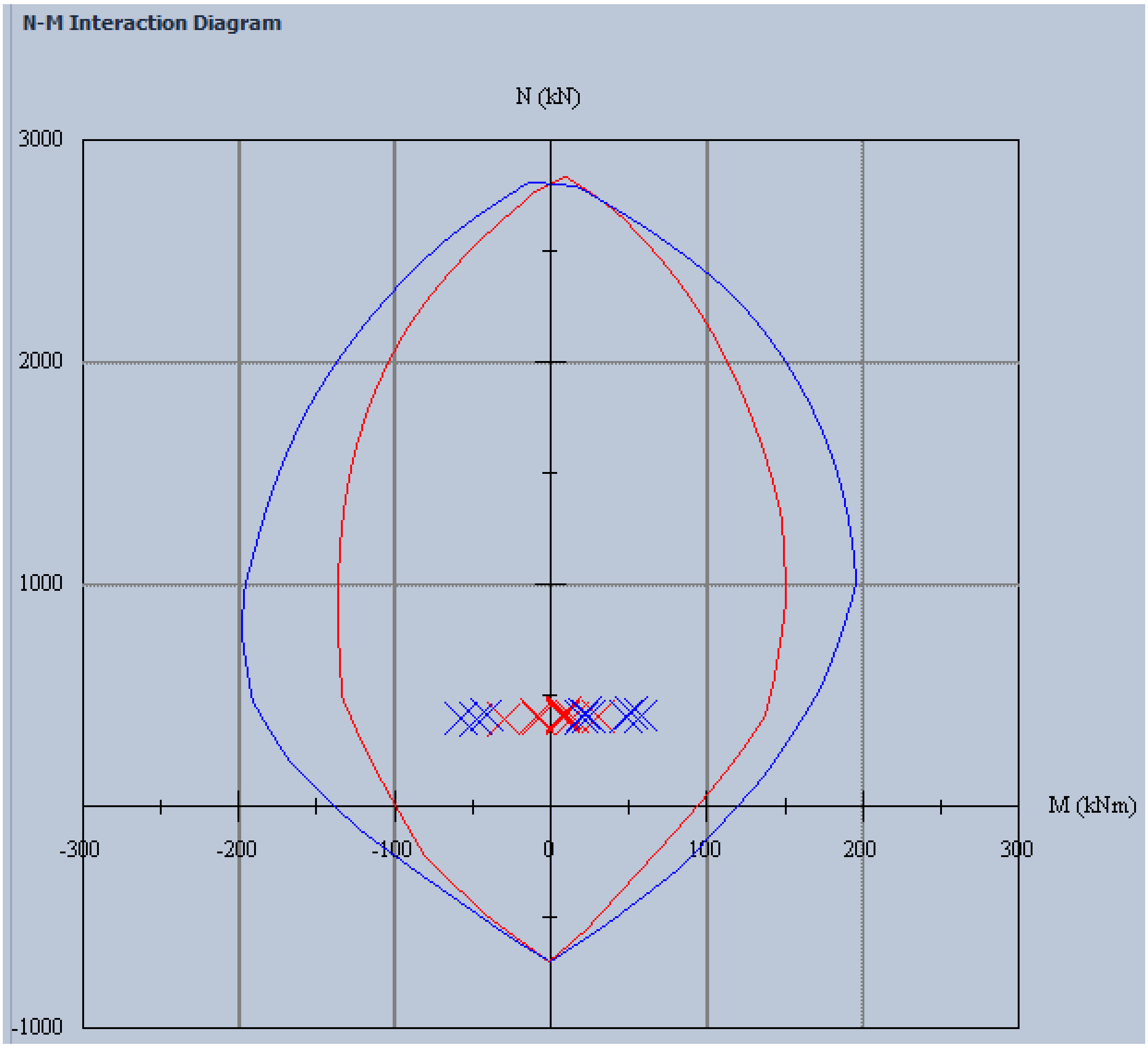
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

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

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

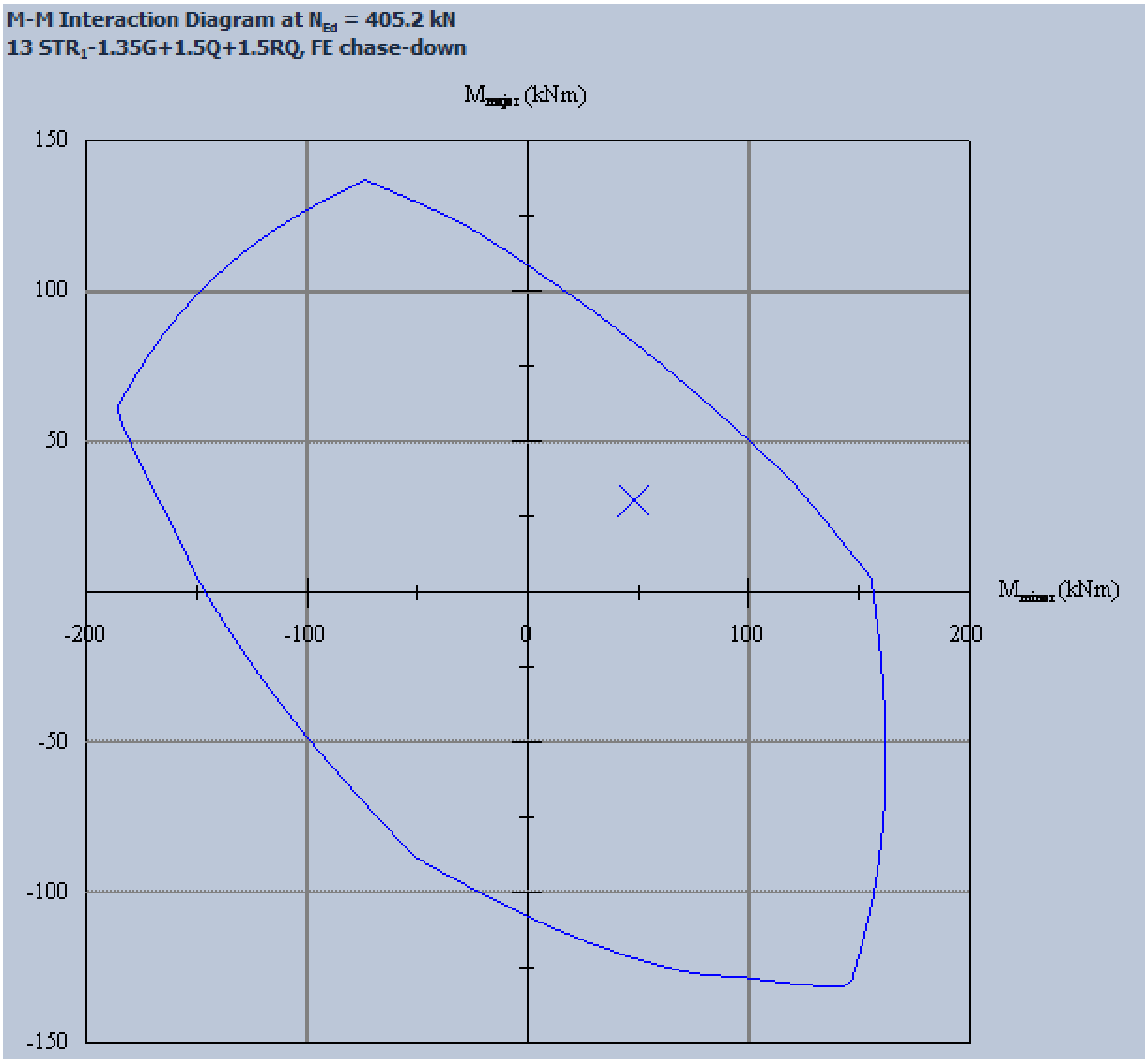


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

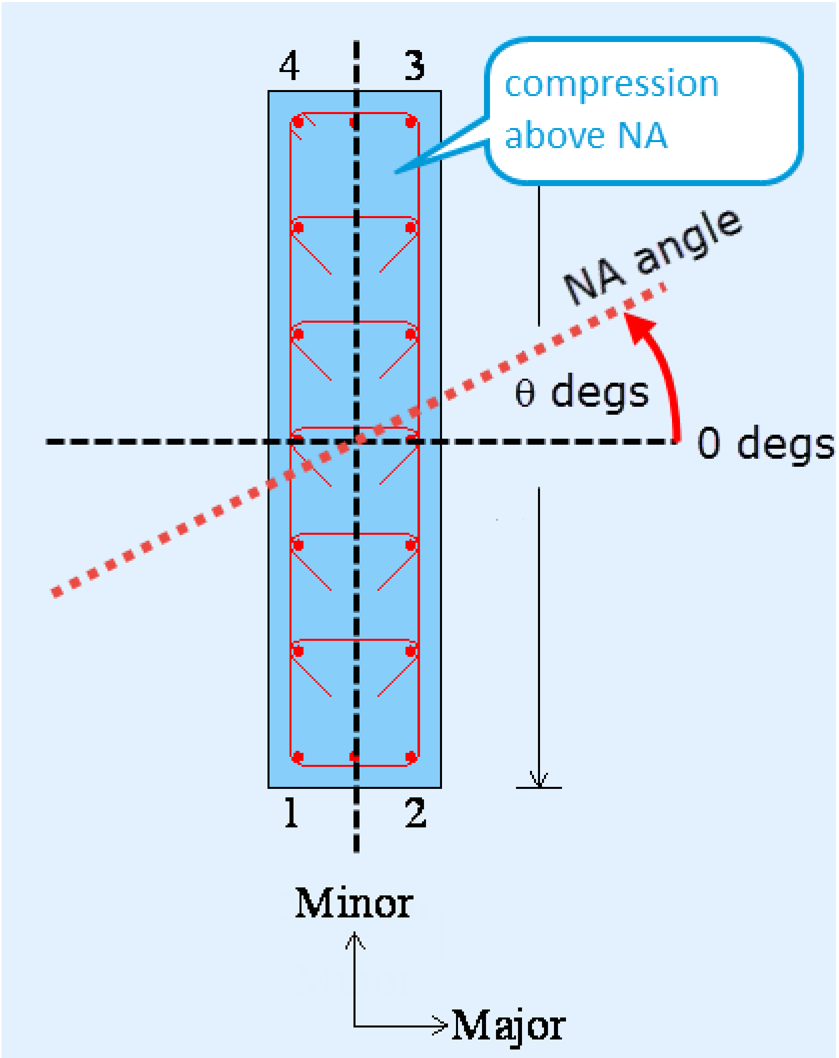
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla

Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional column design cases for user defined forces

Additional design cases can be specified in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected stack on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Column Dialog, select Additional Design Cases (page 308) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all column stacks where appropriate.

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

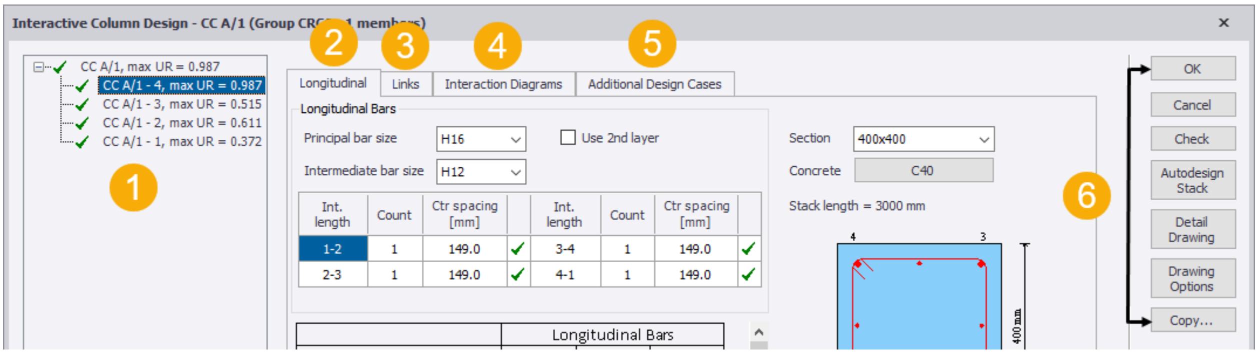
Interactive Column Design dialog

The **Interactive Column Design dialog** shows the current reinforcement and check results for each stack in the selected column. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

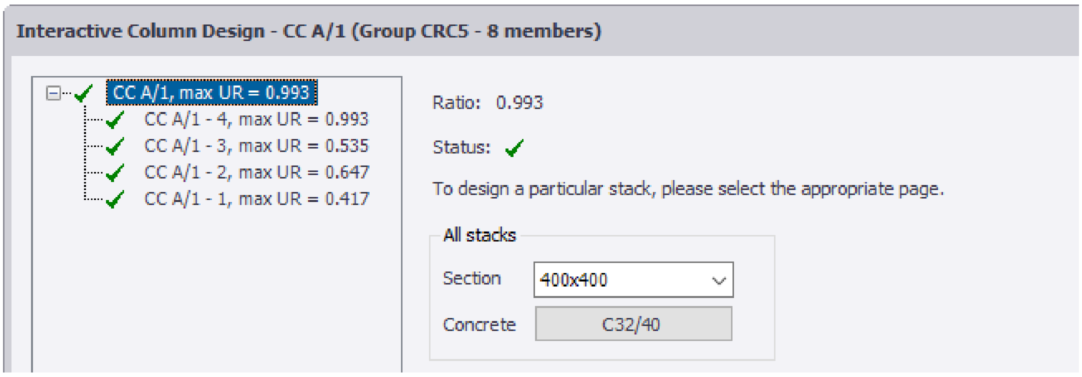
To display the dialog:

1. Right click on an existing concrete column.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Column/stack summary pane



The top row in this pane shows the column summary, consisting of the overall utilization ratio and design status.

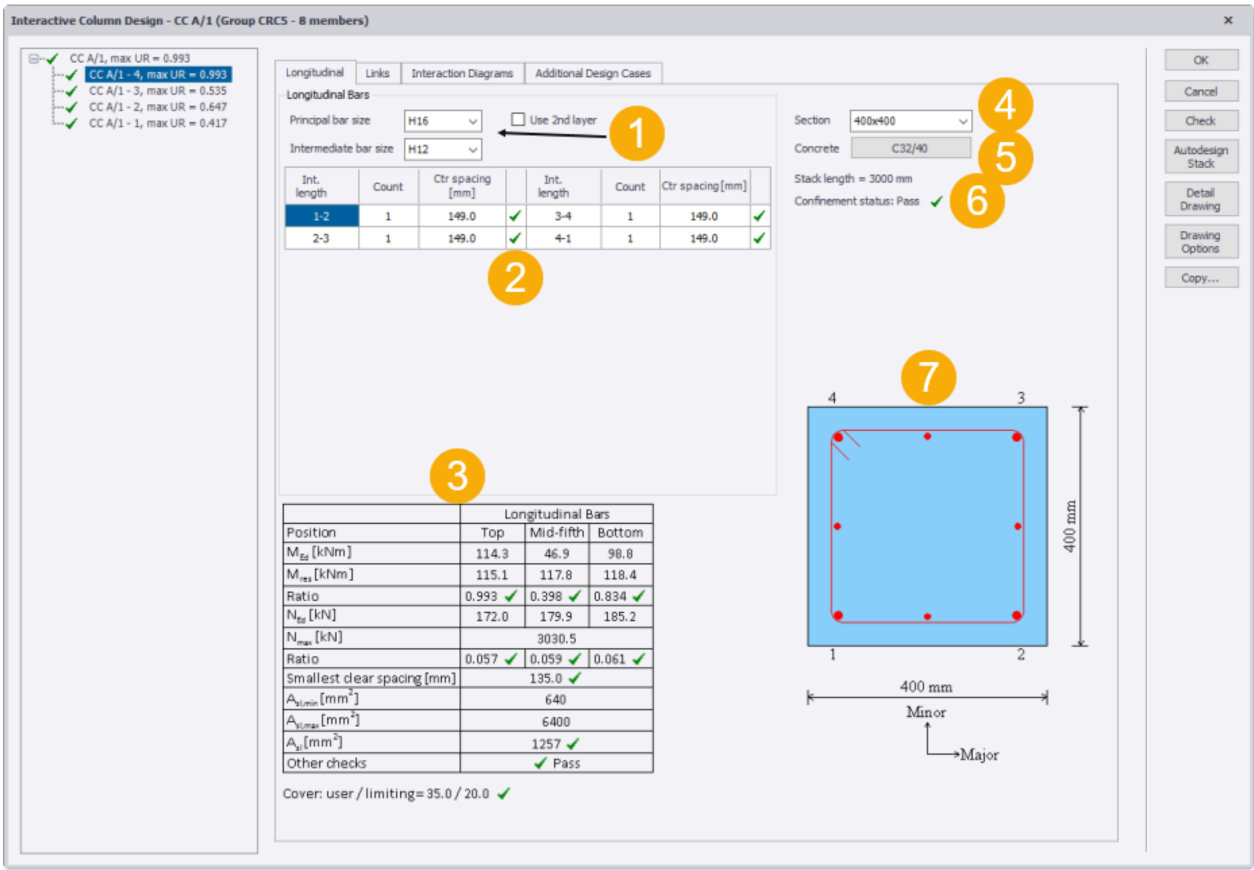
- With this row selected you can edit the section size and grade for all stacks simultaneously.

Subsequent rows show the design status of each stack and associated utilization ratio.

- To design a particular stack, click on the corresponding row in the summary pane.

2. Longitudinal tab

All straight-edged cross sections have “Principal” bars located at shear tie corners. Between these, evenly spaced identical “Intermediate” bars can be located. Circular sections have 6 or more evenly spaced bars around the edge of the section.



1. **Longitudinal Bars**:
   - **Principal bar size**: Used to change the size of **all** principal bars.
     - **Use 2nd layer**: Check the box to allow a second bar layer if required.
       - **Spacing**: For circular sections the user can control the layer spacing.
   - **Intermediate bar size**: (Not displayed for circular columns) Used to change the size of **all** intermediate bars.
2. **Bar Location Table**: Used for adding intermediate bars into the cross-section:
   - Int. length - identifies the edge along which the bars are positioned
   - Count - for changing the number of intermediate bars along the length

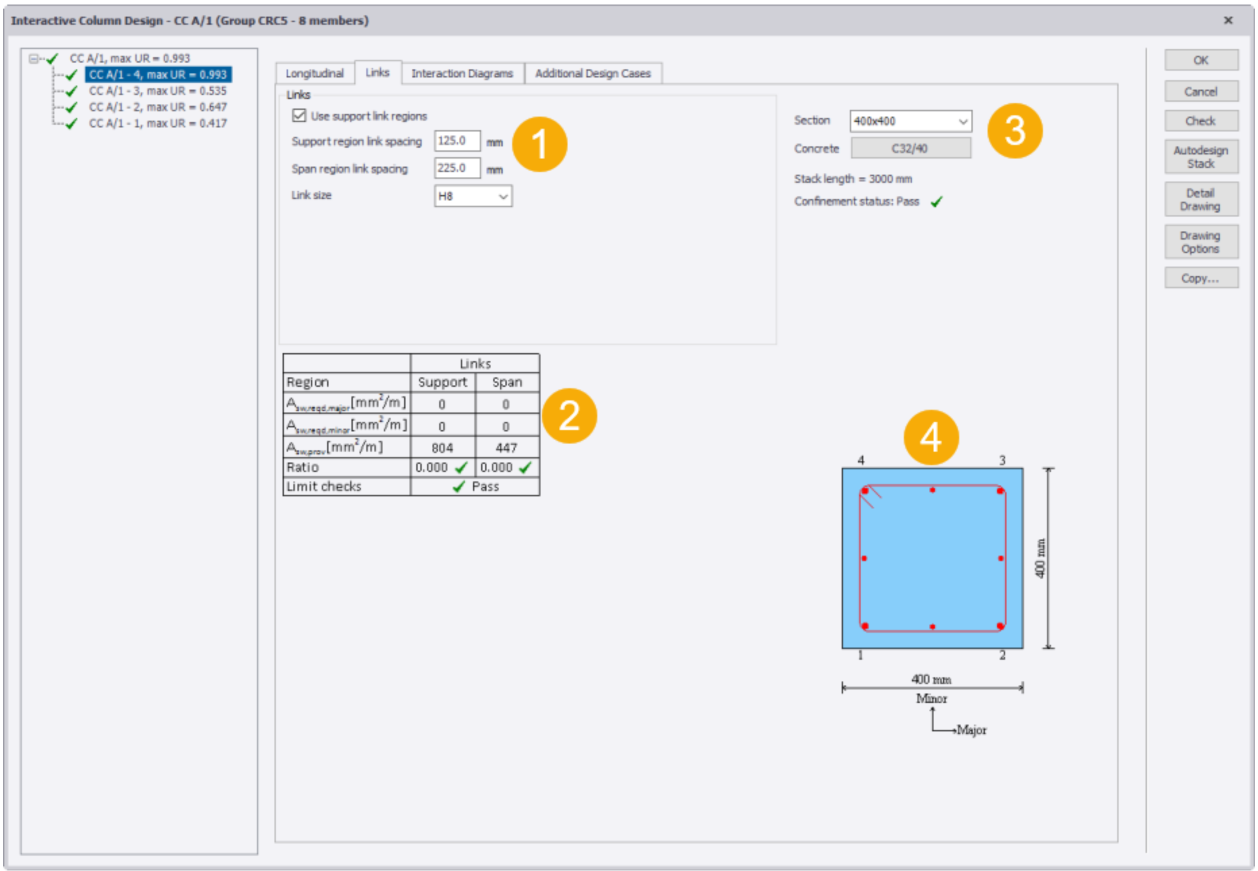
- Ctr spacing - the centerline spacing for the current number of bars along the length
- Status - indicates when the maximum bar spacing limit has been exceeded. (When the minimum bar spacing limit is exceeded this is displayed elsewhere in the Design Summary Table).

3. **Design Summary Table**: Displays the most critical result from all combinations:
   - Design and Resistance Moments and Moment Ratios
   - Axial Force, Axial Resistance and Axial Ratios
   - Smallest clear bar spacing
   - Minimum area of steel
   - Area of steel provided
4. **Section Droplist**: Used for changing the section size for the current stack.

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

5. **Concrete Droplist**: Used for changing the concrete grade for the current stack.
6. **Confinement status** : This status is determined based on the requirements for bars being tied.
7. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Tie locations
   - Section dimensions
   - Principal bar labels

3. Links/Ties tab



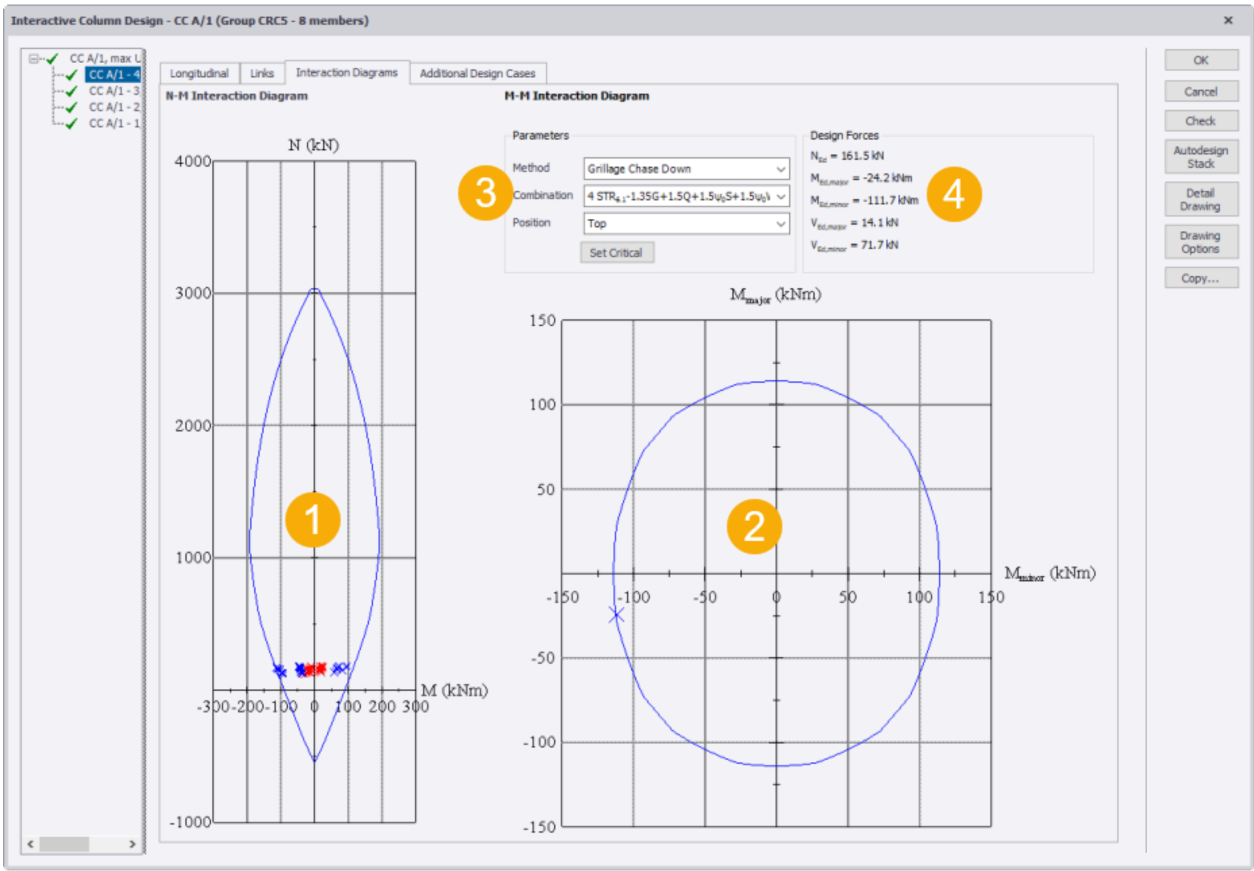
1. **Links**:
   - **Use support region links/ties** : Select to design support regions for the links/ties.
   - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
   - **Link/Tie size**: Used to change the size of link/tiebars (all must have the same size).
2. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
   - Region length
   - Link/Stirrup area over spacing required, major
   - Link/Stirrup area over spacing required, minor
   - Link/Tie area over spacing provided
   - Link/Tie utilization ratio

3. **Section Droplist**: Used for changing the section size for the current stack.

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

4. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Link/Tie locations
    - Section dimensions
    - Principal bar labels

4. Interaction Diagram tab

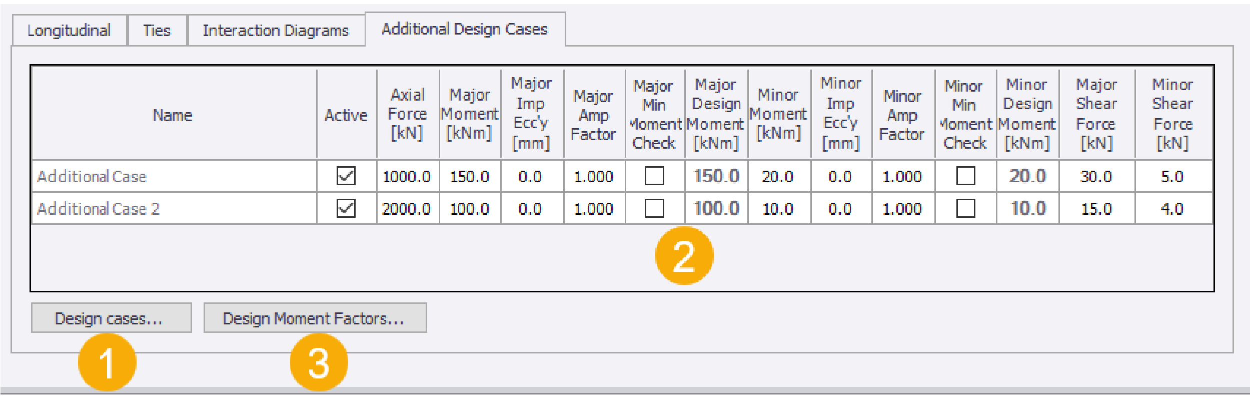


1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
    - The curves for bending about the major axis are shown in red
    - The curves for bending about the minor axis are shown in blue

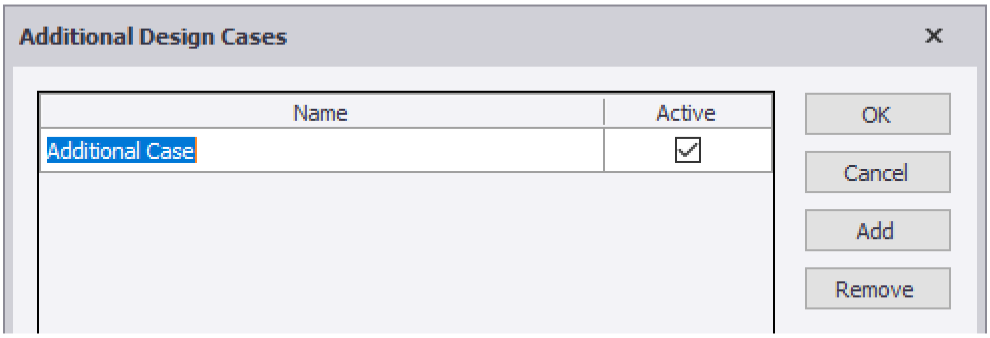
2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.
   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.
4. **Design Forces**: The design forces applicable for the selected parameters.

5. Additional Design Cases tab

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



1. **Design cases...**button: Click to open a dialog in which to add any additional design cases.



The design case names defined here are available to all columns and walls in the model.

The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given stack you must:

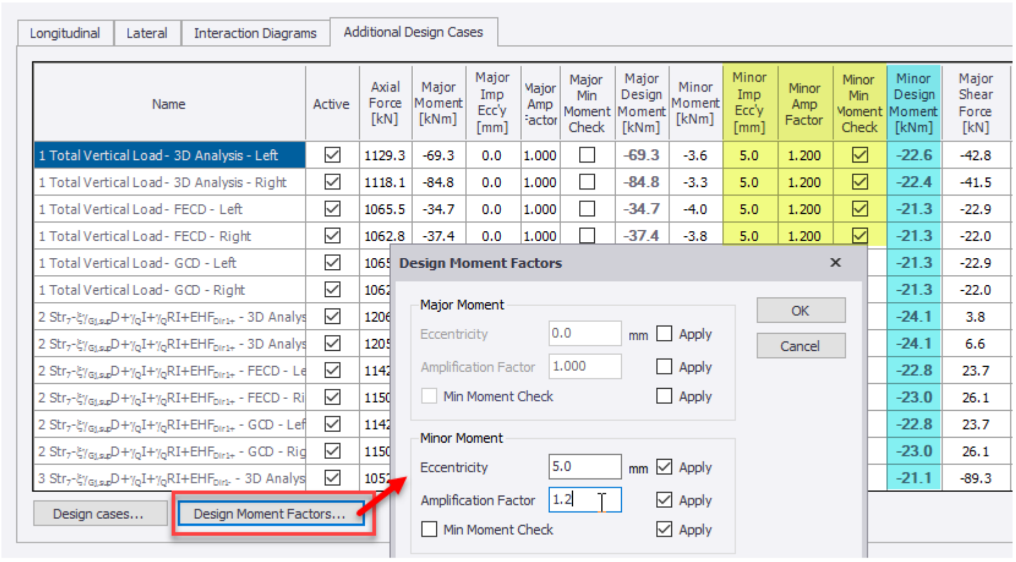
   - Select the required stack in the stack summary
   - Click the Active box for the design case
   - Enter the design forces

3. **Design Moment Factors...** button: Offers three potential "Design Moment" adjustments for each direction:

   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
   - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
   - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension "h" in the direction considered).

   When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

   The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with "Active" option checked on) as shown below:



6. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Autodesign Stack	Performs an autodesign reselecting bars for the current stack
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog
Copy	Opens a new dialog enabling the selective coping of section/concrete grade/reinforcement from the selected stack to other stacks in the same column.

See also

Interactive concrete member design (page 285)

Interactive concrete wall design

Opening the Interactive Wall Design Dialog

The Interactive Wall Design dialog (page 320) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the wall you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Wall Design dialog is displayed.
3. Click on an individual panel in the wall/panel summary pane (page 320).
4. Interactively adjust the reinforcement as required until the panel design is satisfactory.

Wall interaction diagrams (US customary units)

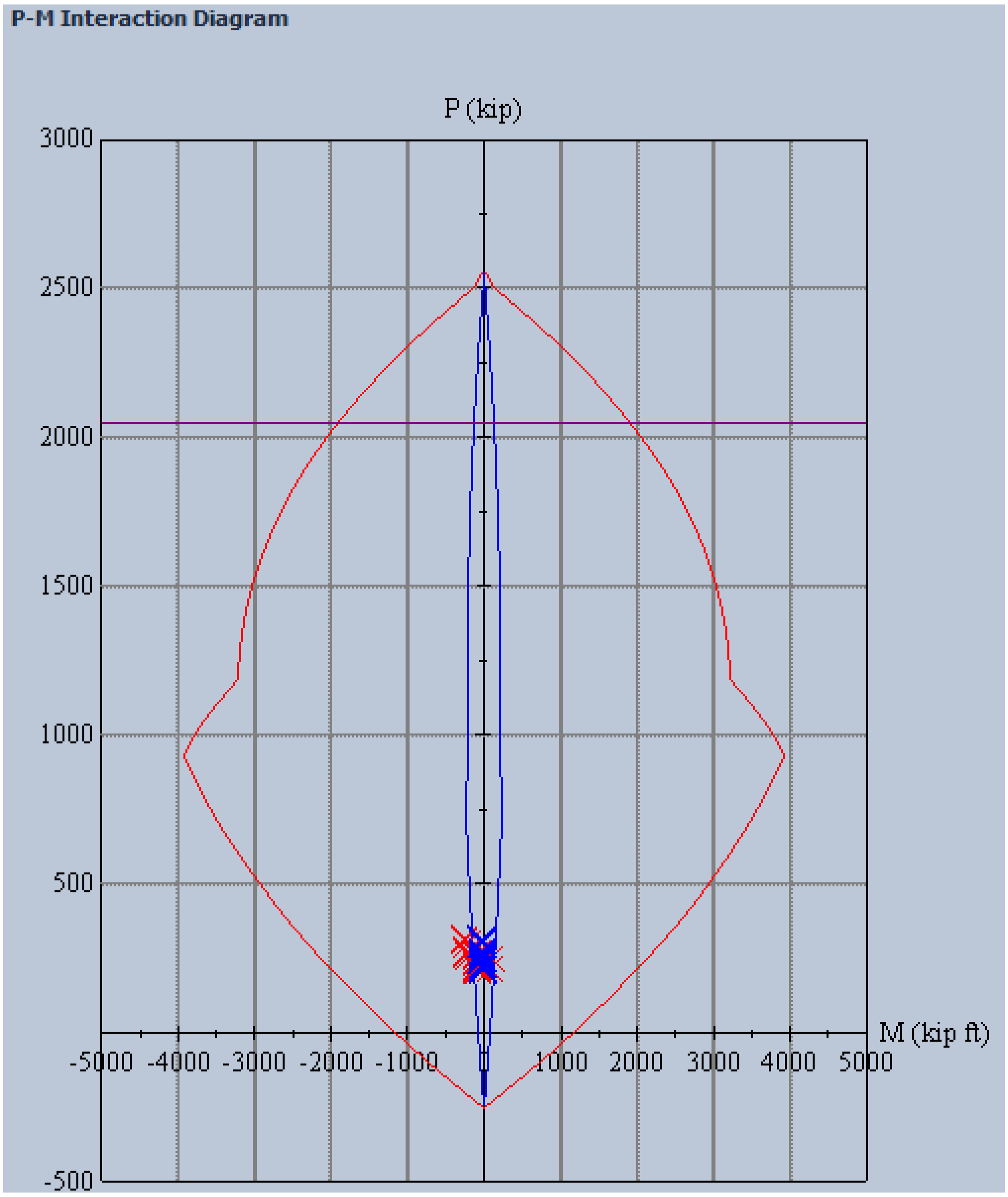
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

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

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

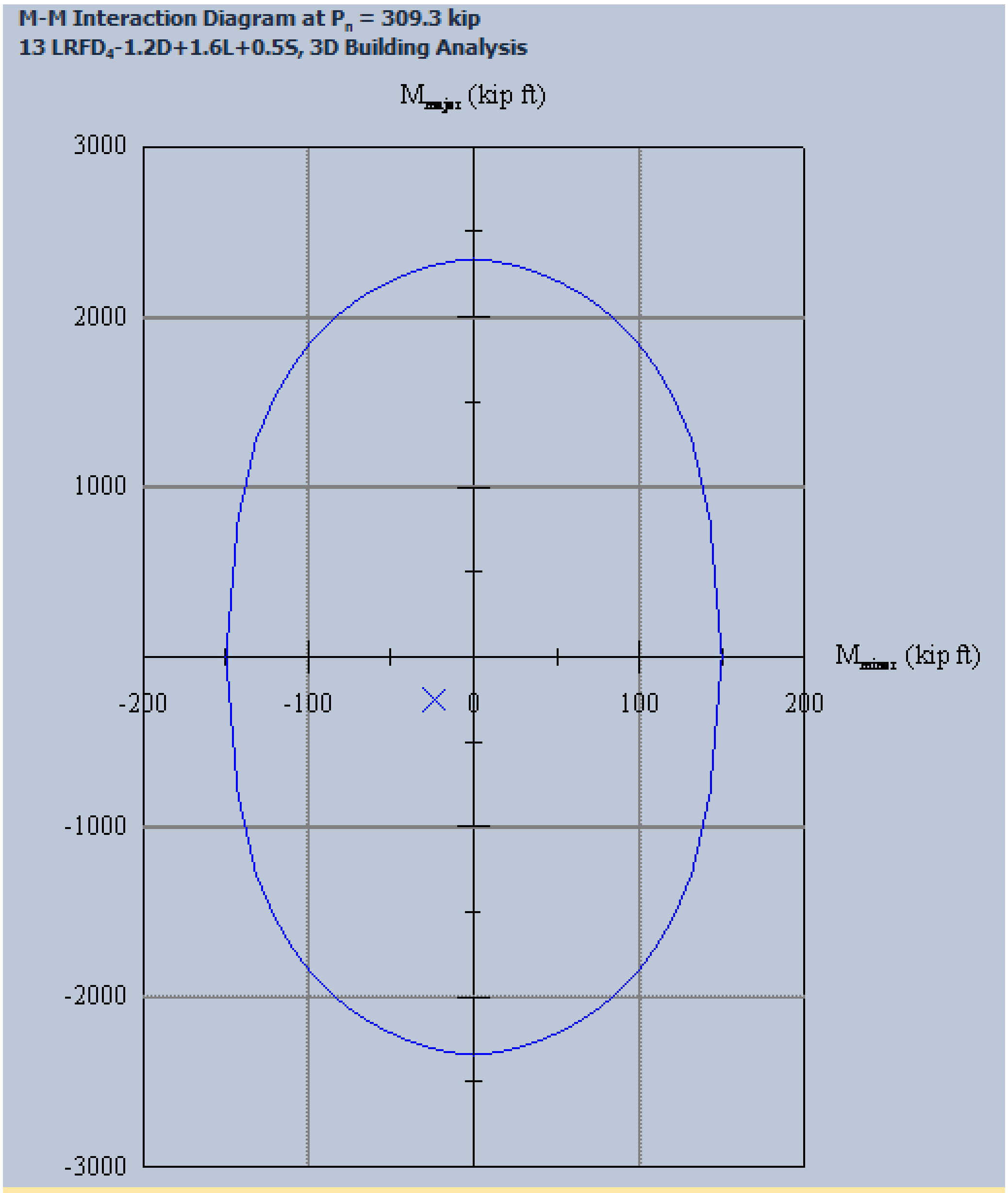
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



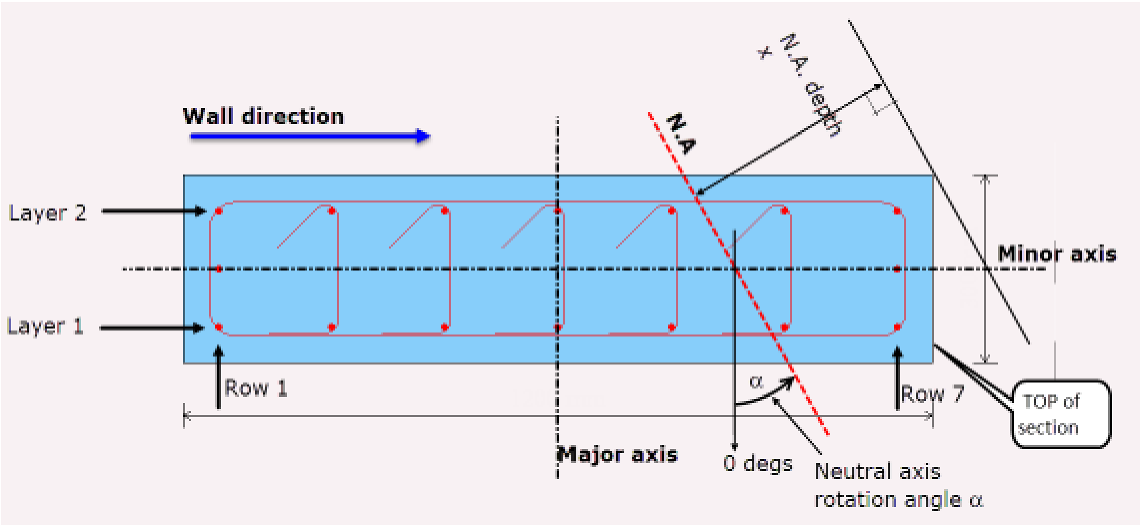
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Wall interaction diagrams (metric units)

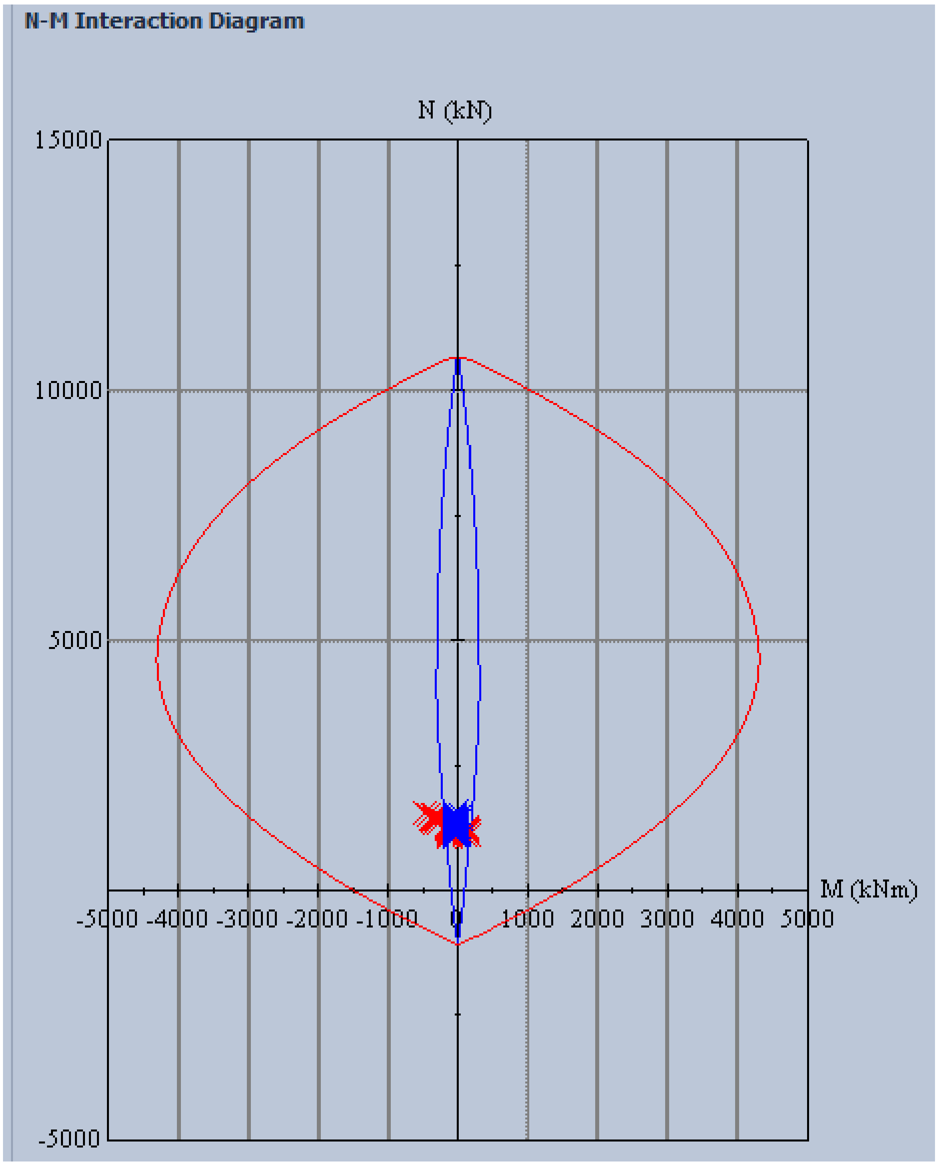
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

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

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

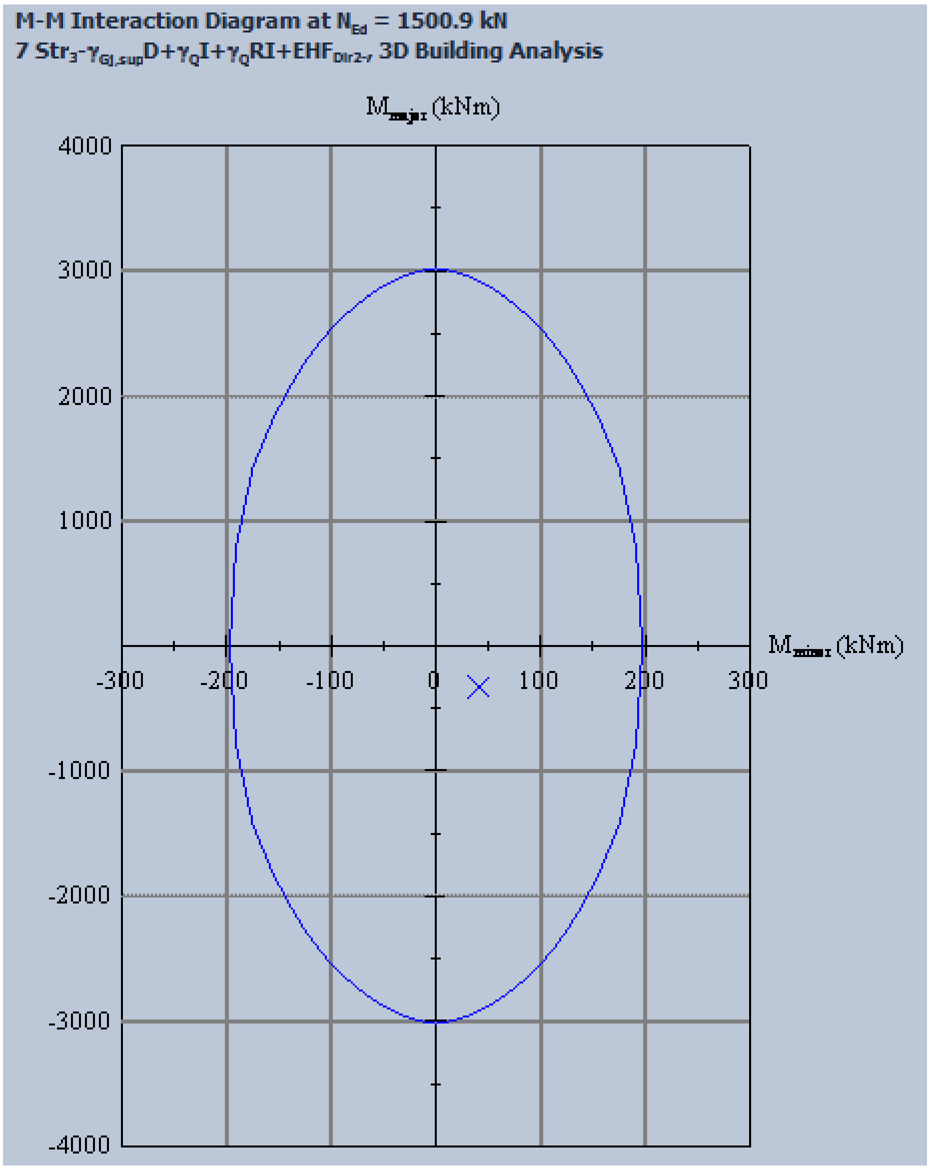
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



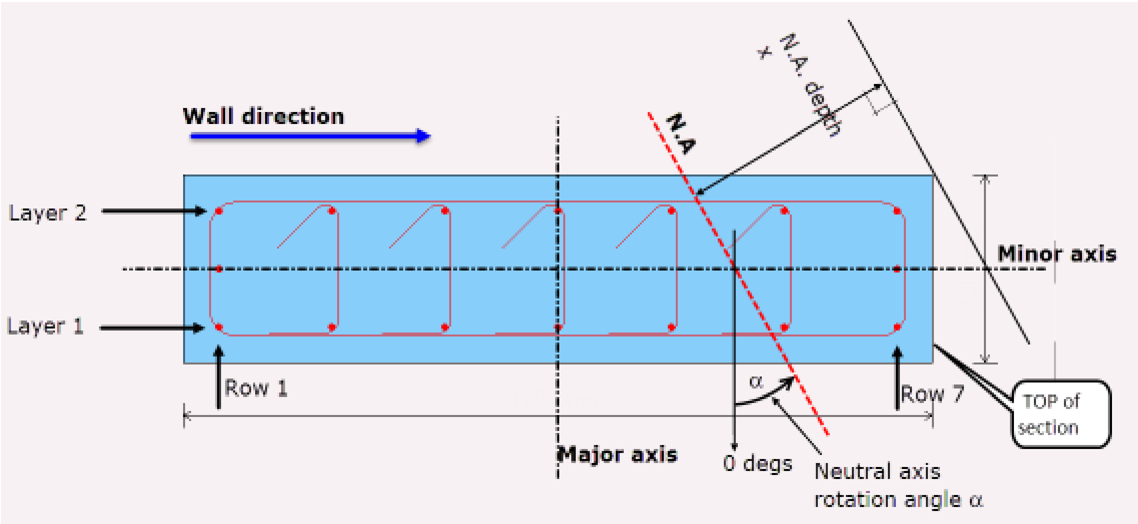
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional wall design cases for user defined forces

Additional design cases can be specified typically in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected panel on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Wall Dialog, select Additional Design Cases (page 327) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all wall panels where appropriate.

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

NOTE Additional design cases can also be set up directly from **Result Lines** in order to facilitate local section design around openings - see the following topic.

Using result lines for local section design around openings

Interactive design of local sections around/between openings is possible using **Result Lines**.

The design is performed either as a column or wall section (as specified by the user), using those design forces determined along the lengths cut by the result lines. Engineering judgement is therefore required when positioning the lines to ensure suitable design forces are obtained.

For further details of the process, see:

Related video

Interactive design using Result lines

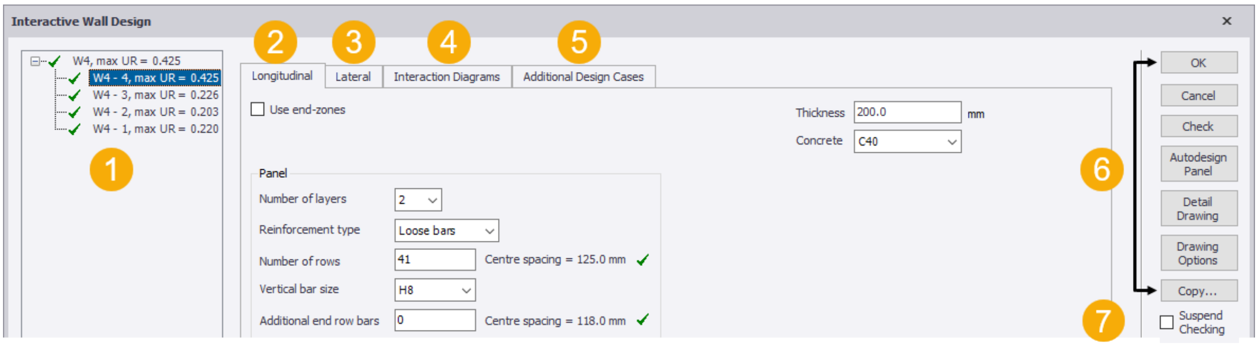
Interactive Wall Design dialog

The **Interactive Wall Design dialog** shows the current reinforcement and check results for each panel in the selected wall. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

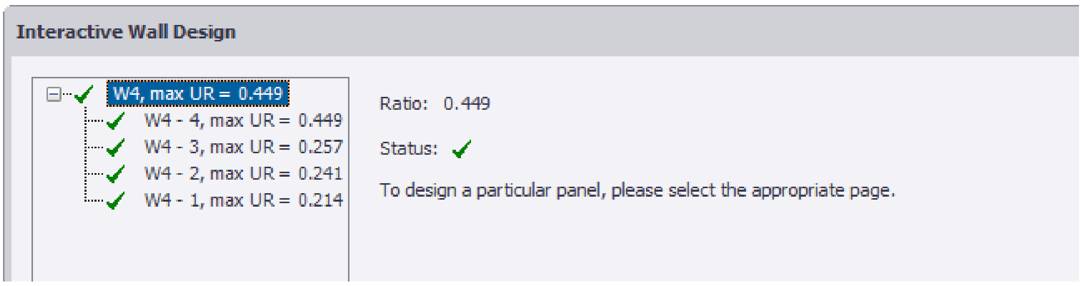
To display the dialog:

1. Right click on an existing concrete beam.
2. In the context menu, select **Interactive Design...**.

The dialog content is described below.



1. Wall/panel summary pane



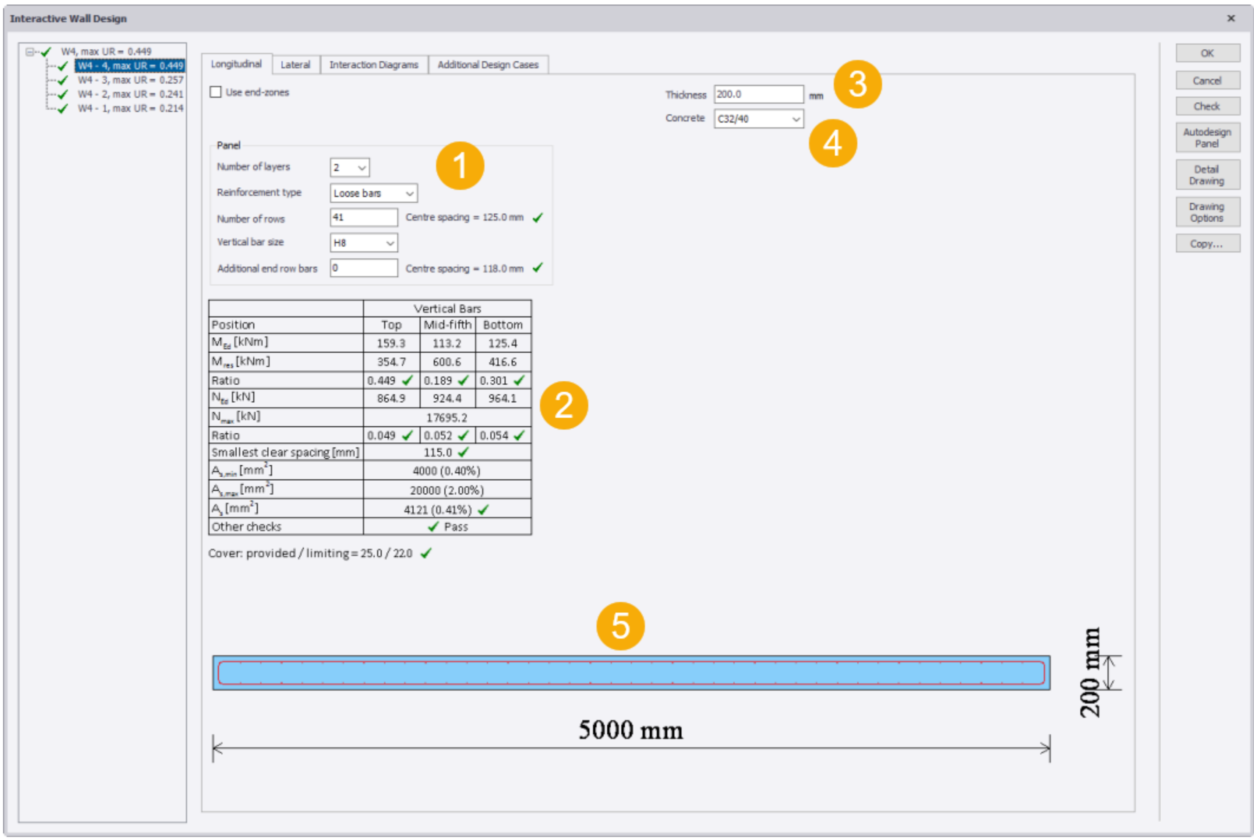
The top row in this pane shows the Wall summary, consisting of the overall utilization ratio and design status.

- With this row selected you can edit the thickness and grade for all stacks simultaneously.

Subsequent rows show the design status of each panel and associated utilization ratio.

- To design a particular panel, click on the corresponding row for the panel in the summary pane.

2. Longitudinal tab (no end-zones)



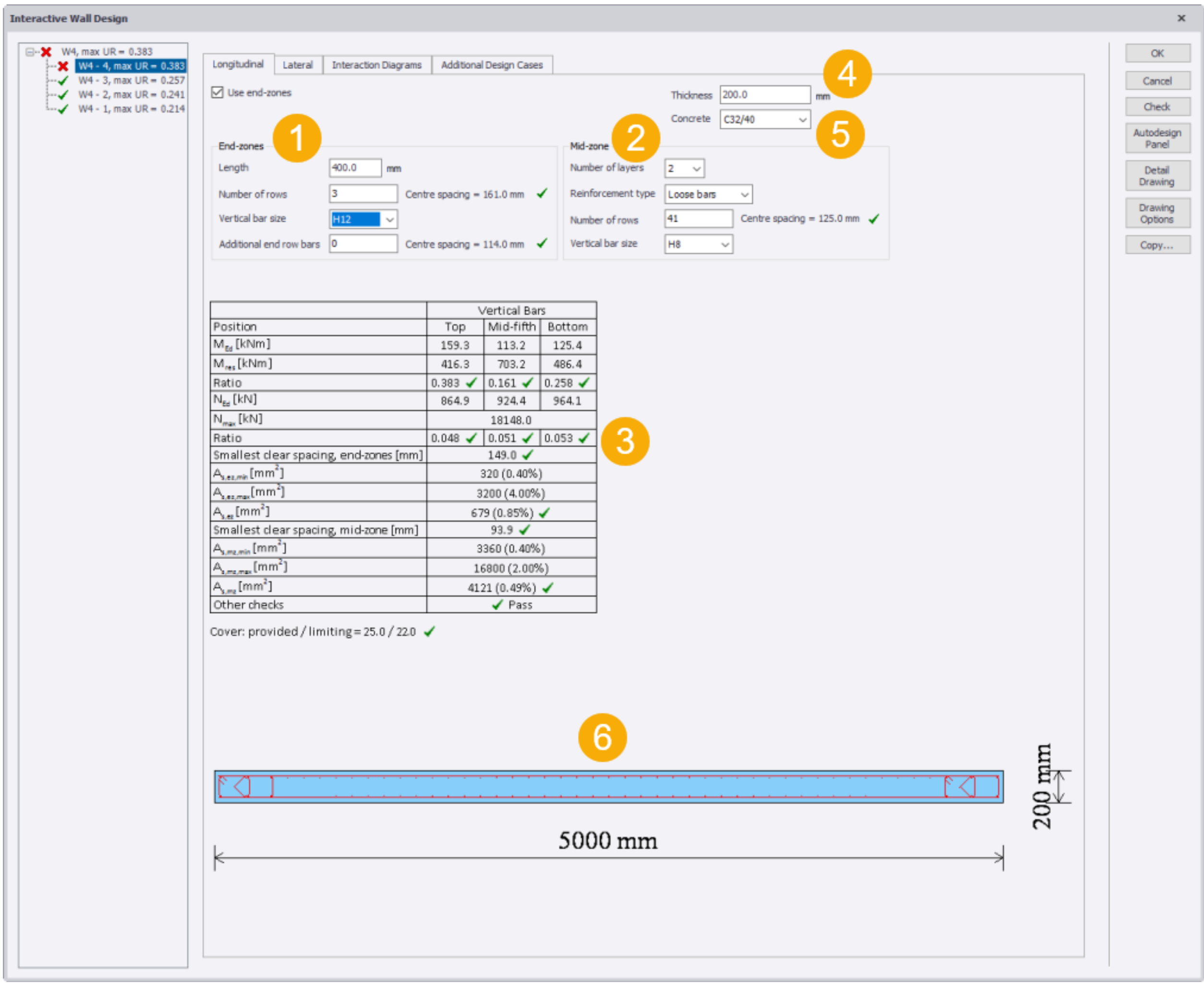
	Vertical Bars		
Position	Top	Mid-fifth	Bottom
M_{Ed} [kNm]	159.3	113.2	125.4
M_{res} [kNm]	354.7	600.6	416.6
Ratio	0.449 ✓	0.189 ✓	0.301 ✓
N_{Ed} [kN]	864.9	924.4	964.1
N_{max} [kN]	17695.2		
Ratio	0.049 ✓	0.052 ✓	0.054 ✓
Smallest clear spacing [mm]	115.0 ✓		
$A_{s,min}$ [mm^2]	4000 (0.40%)		
$A_{s,max}$ [mm^2]	20000 (2.00%)		
A_s [mm^2]	4121 (0.41%) ✓		
Other checks	✓ Pass		

Use end-zones : With this box unchecked end-zones are not used.

1. **Panel**: Used for adding either one or two layers of bars in the panel.
    - Number of layers - (1, or 2)
    - If Reinforcement type = loose bars
        - Number of rows - the number of bars in each layer
        - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
        - Vertical bar size - specifies the size to be checked
        - Additional end row bars - specifies the number of additional bars in the end row
        - Centre spacing (end rows) - the spacing between layers (measured centre to centre)
    - If Reinforcement type = mesh
        - Mesh size - specifies the mesh size to be checked

        - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
    - Additional end row bars - specifies the number of additional bars in the end row
    - Centre spacing (end rows) - the spacing between layers (measured centre to centre)
2. **Design Summary Table**: Displays the most critical result from all combinations:
    - Design and Resistance Moments and Moment Ratios
    - Axial Force, Axial Resistance and Axial Ratios
    - Smallest clear bar spacing
    - Minimum area of steel
    - Area of steel provided
3. **Thickness**: Used for changing the thickness of the current panel.
4. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
5. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Link/Tie locations
    - Section dimensions

2. Longitudinal tab (with end-zones)

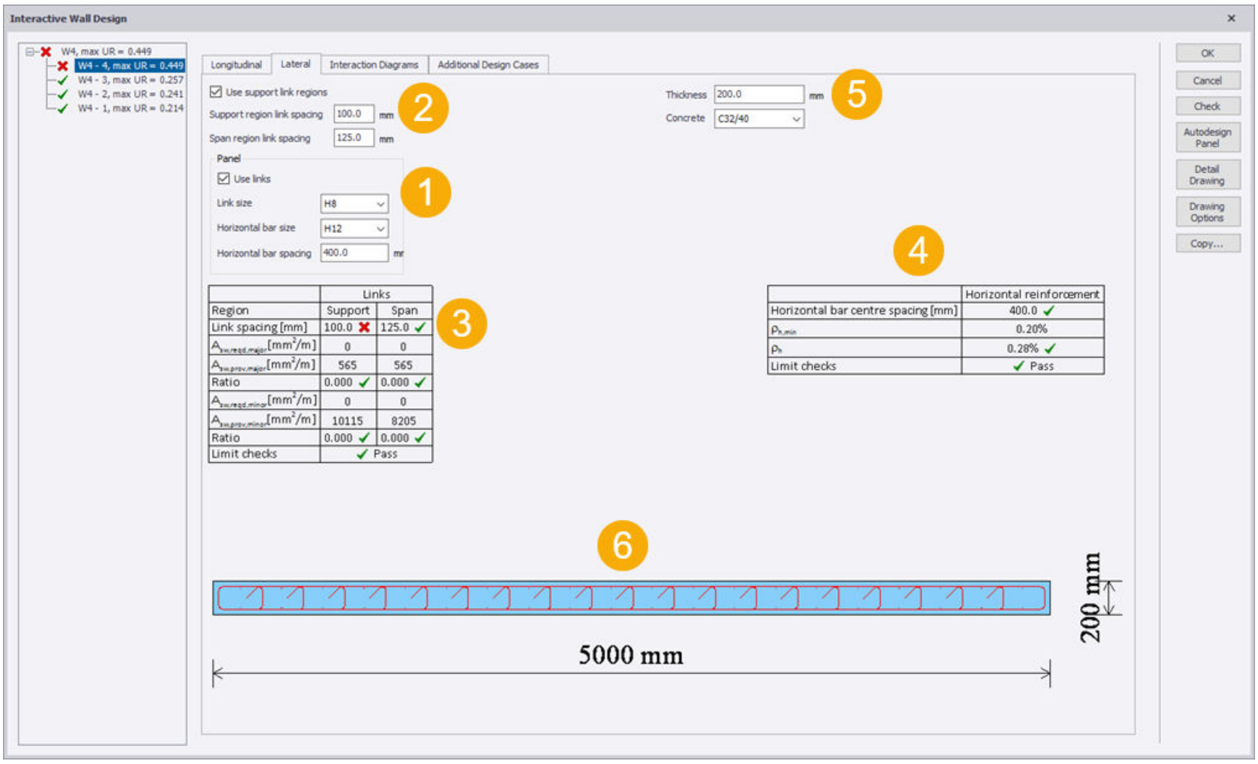


Use end-zones : With this box checked end-zones are used.

1. **End-zones**: Used for adding bars in the end-zones.
   - Length - length of each end-zone
   - Number of rows - the number of loose bars in each end-zone
   - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
   - Vertical bar size - specifies the size to be checked
   - Additional end row bars - specifies the number of additional bars in the end row
   - Centre spacing (end rows) - the spacing between layers in the end-zone (measured centre to centre)

2. **Mid-zone**: Used for adding either one or two layers of bars in the panel between the end-zones.
    - Number of layers - (1, or 2)
    - If Reinforcement type = loose bars
        - Number of rows - the number of bars in each layer
        - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
        - Vertical bar size - specifies the size to be checked
    - If Reinforcement type = mesh
        - Mesh size - specifies the mesh size to be checked
        - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
3. **Design Summary Table**: Displays the most critical result from all combinations:
    - Design and Resistance Moments and Moment Ratios
    - Axial Force, Axial Resistance and Axial Ratios
    - Smallest clear bar spacing
    - Minimum area of steel
    - Area of steel provided
4. **Thickness**: Used for changing the thickness of the current panel.
5. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
6. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Link/Tie locations
    - Section dimensions

3. Lateral tab



1. **Panel**:
   - **Use links/ties** : Select to specify links/ties.
     - **Link/Tie size**: Used to change the size of link/ties (all must have the same size).
   - **Horizontal bar size**: Used to specify the size of horizontal bars.
   - **Horizontal bar spacing**: Used to specify the vertical spacing of horizontal bars.
2. **Use support region links/ties**: Select to design support regions for the links/ties.

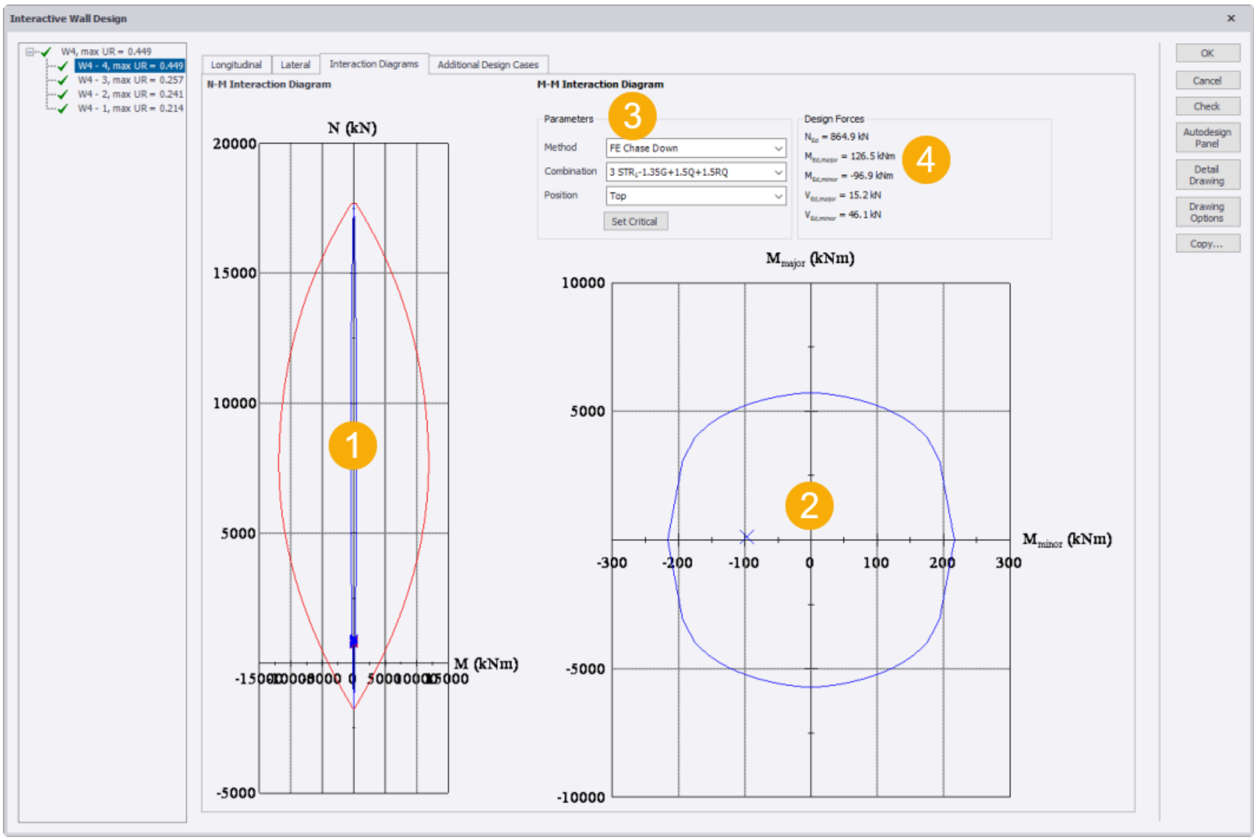
   NOTE Only displayed provided Use links/ties box is also checked.

   - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
3. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
   - Link/Tie area over spacing required, major
   - Link/Tie area over spacing provided, major
   - Link/Tie utilization ratio, majo

- Link/Tie area over spacing required, mino
- Link/Tie area over spacing required, mino
- Link/Tie utilization ratio, minor

4. **Horizontal Reinforcement Summary Table**: The table displays the horizontal bar spacing and reinforcement ratios.
5. **Thickness**: Used for changing the thickness of the current panel.
6. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions

4. Interaction Diagrams tab



1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
   - The curves for bending about the major axis are shown in red
   - The curves for bending about the minor axis are shown in blue

2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.

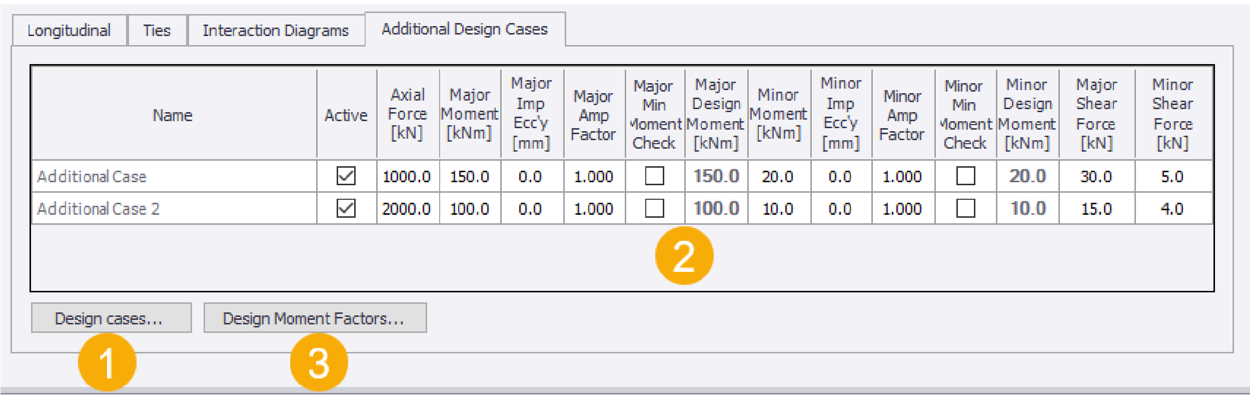
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.

   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.

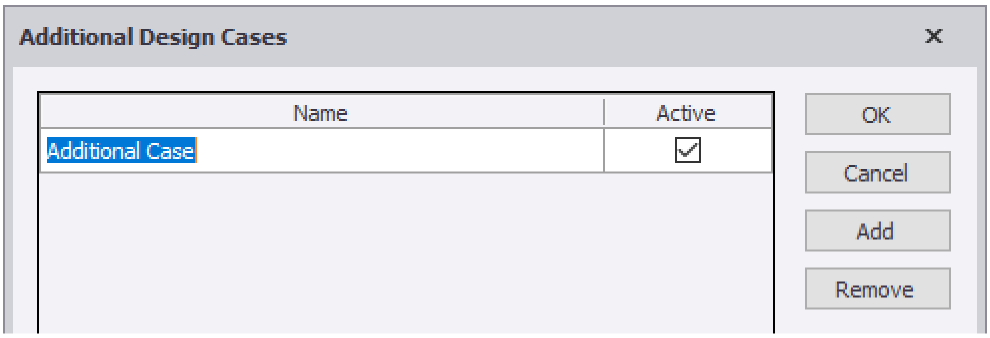
4. **Design Forces**: The design forces applicable for the selected parameters.

5. Additional Design Cases tab

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



1. **Design cases...**button: Click to open a dialog in which to add any additional design cases.



The design case names defined here are available to all columns and walls in the model.

The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given panel you must:

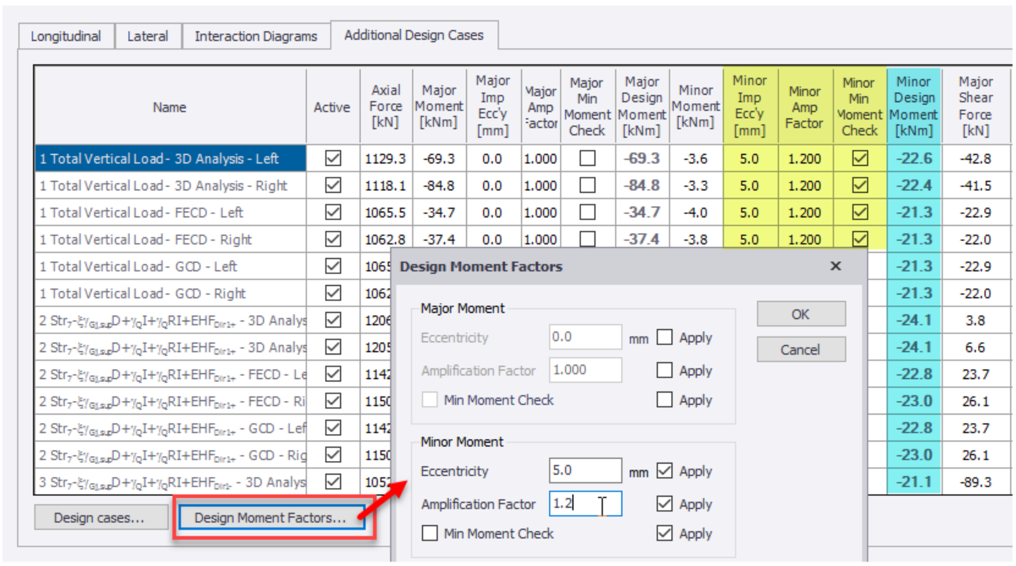
   - Select the required panel in the panel summary
   - Click the Active box for the design case
   - Enter the design forces

3. **Design Moment Factors...** button: Offers three potential “Design Moment” adjustments for each direction:

   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
   - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
   - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension “h” in the direction considered).

   When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

   The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with “Active” option checked on) as shown below:



6. Buttons

Button/option	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.
Autodesign Panel	Performs an autodesign reselecting bars for the current panel
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog
Copy	Opens a new dialog enabling the selective coping of wall thickness/ concrete grade/reinforcement from the selected panel to other panels in the same wall.

7. Suspend Checking option

By default this option is unselected and checks are made after every edit. For large structures with many combinations interactive checking takes time which can be frustrating if you want to make several changes.

When this option is selected, checking is suspended. (The option also changes to red to highlight that this is the case).

Checks are only carried out and results updated when the option is unselected once more.

See also

Interactive concrete member design (page 285)

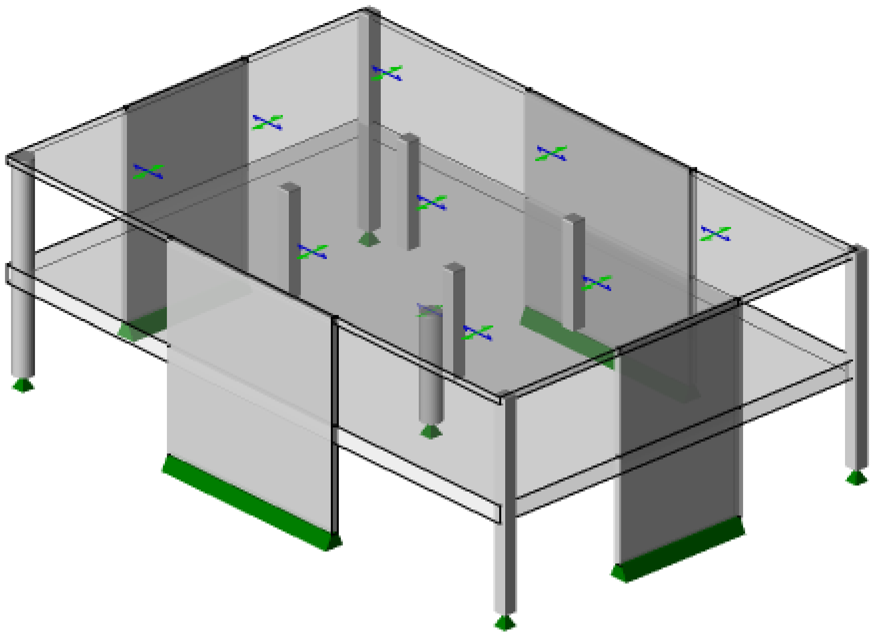
Concrete slab design

To get familiar with the concrete slab design processes. click the following links:

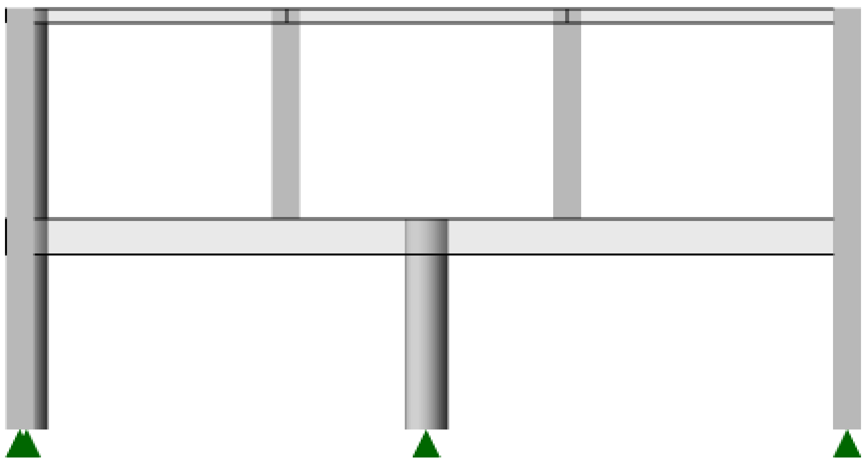
- Flat slab design workflow (page 330)
- Slab on beams design workflow (page 342)
- Concrete slab design aspects (page 348)

Flat slab design workflow

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

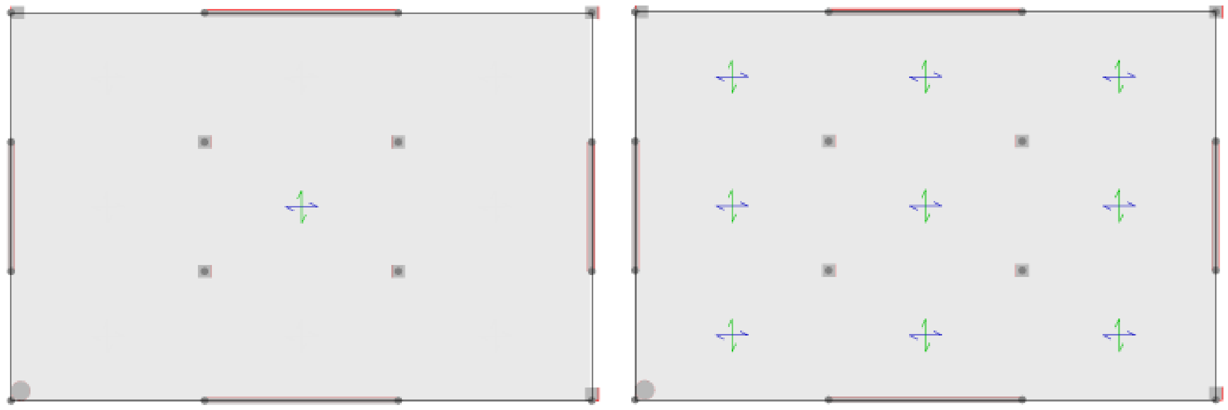


Note that there is a transfer level at the first floor:

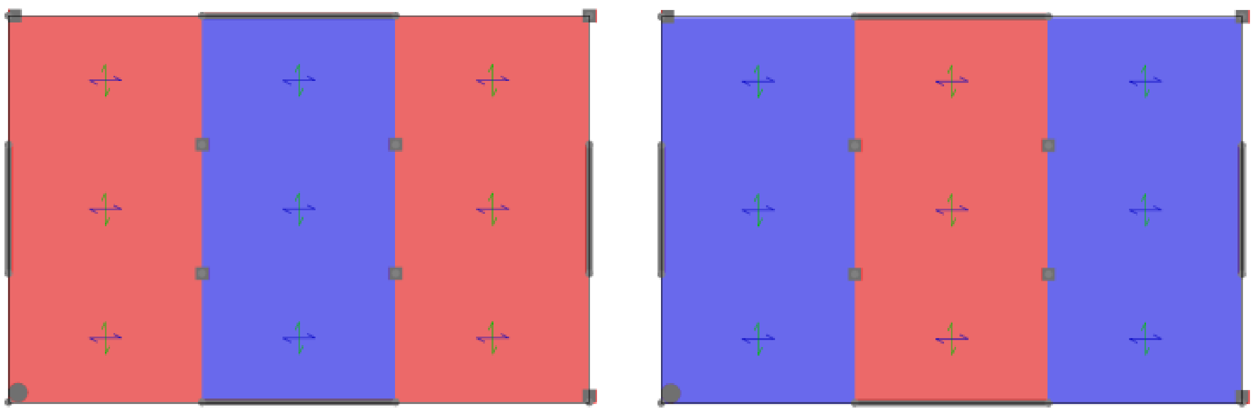


Set up pattern loading

If necessary you should consider manually splitting and joining slab panels to facilitate management of the pattern loading process.



By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Establish slab design moments

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

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

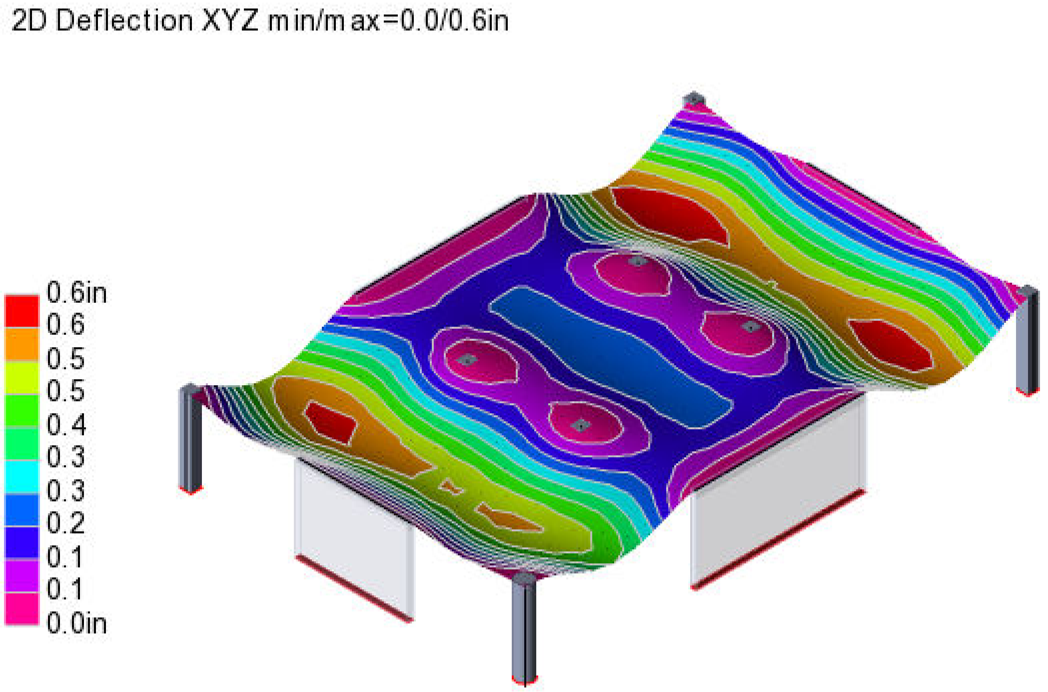
If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Consider simple (linear) deflection (US customary units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

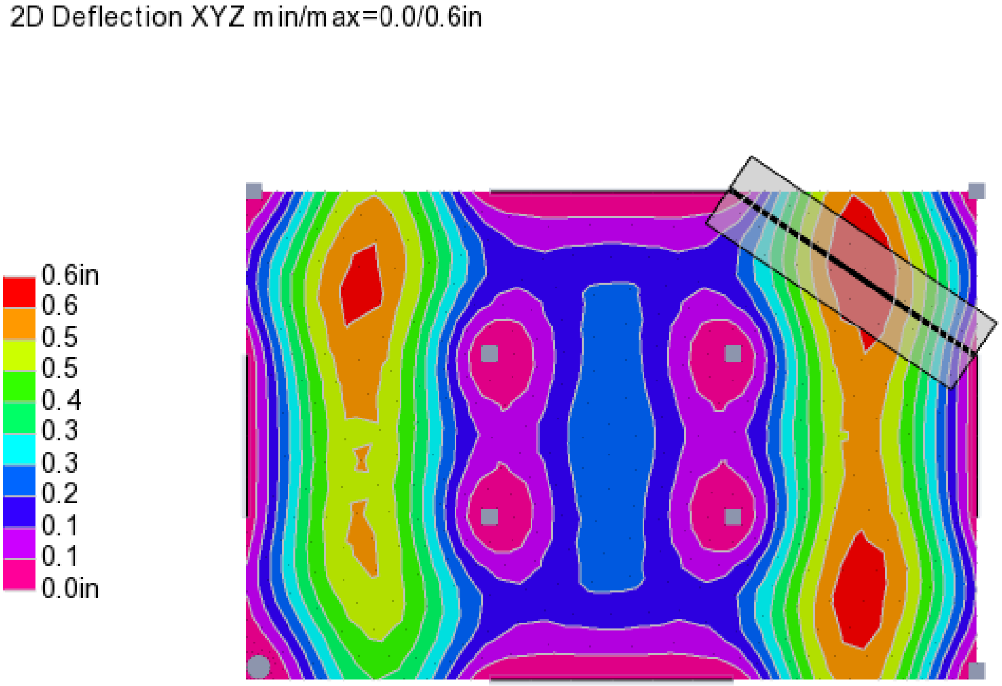
NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.



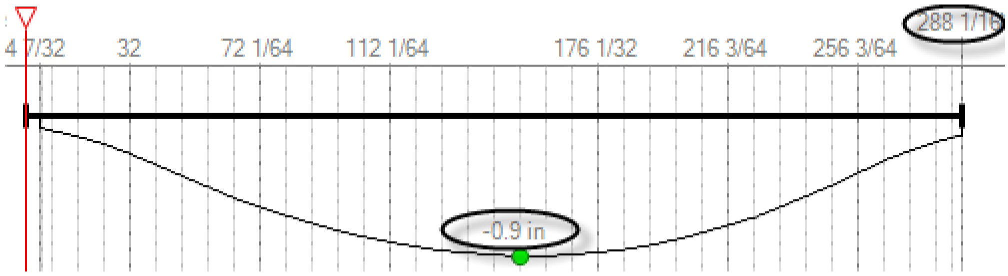
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

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

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



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



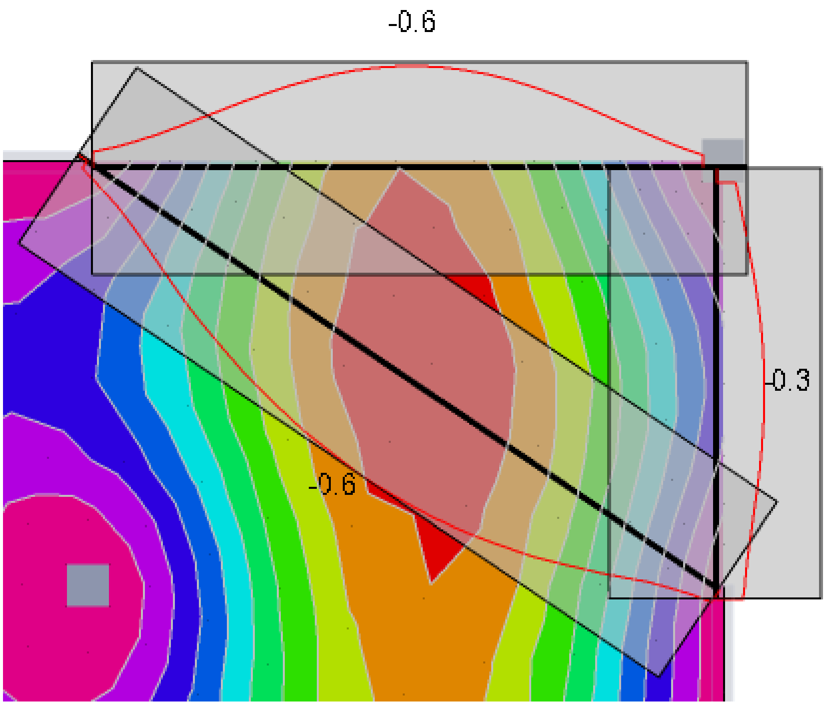
In this example there is a requirement to limit total deflection to span/240

Taking the span as the length of the slab strip: 288in

Deflection limit: 288/240 = 1.2 in

Actual deflection: 0.9 in - the check passes

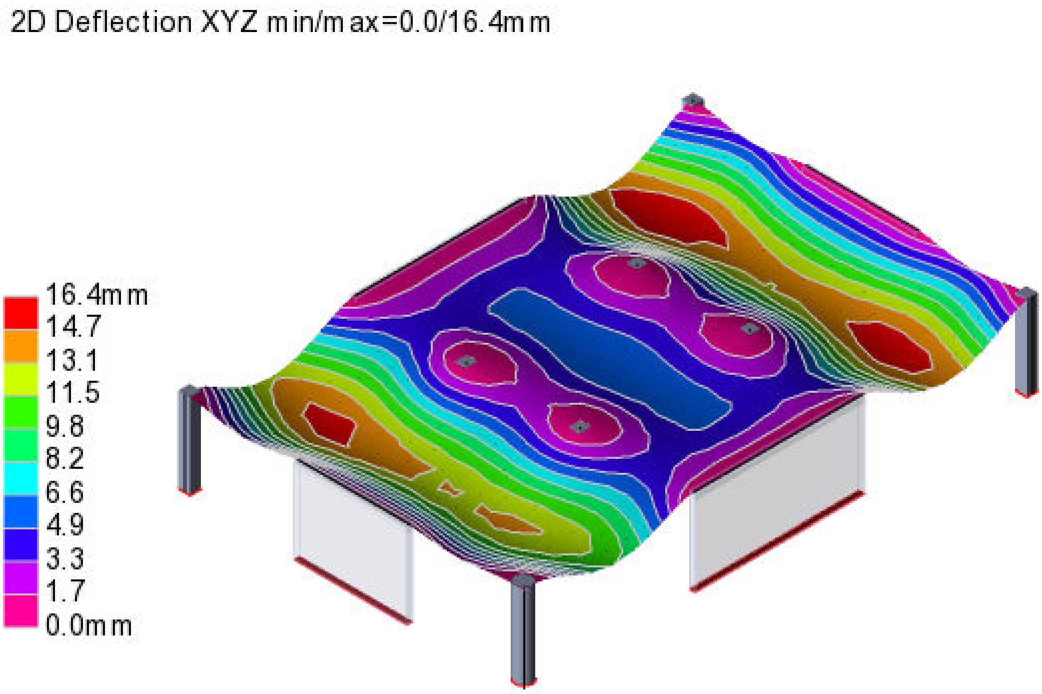
The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Consider simple (linear) deflection (metric units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

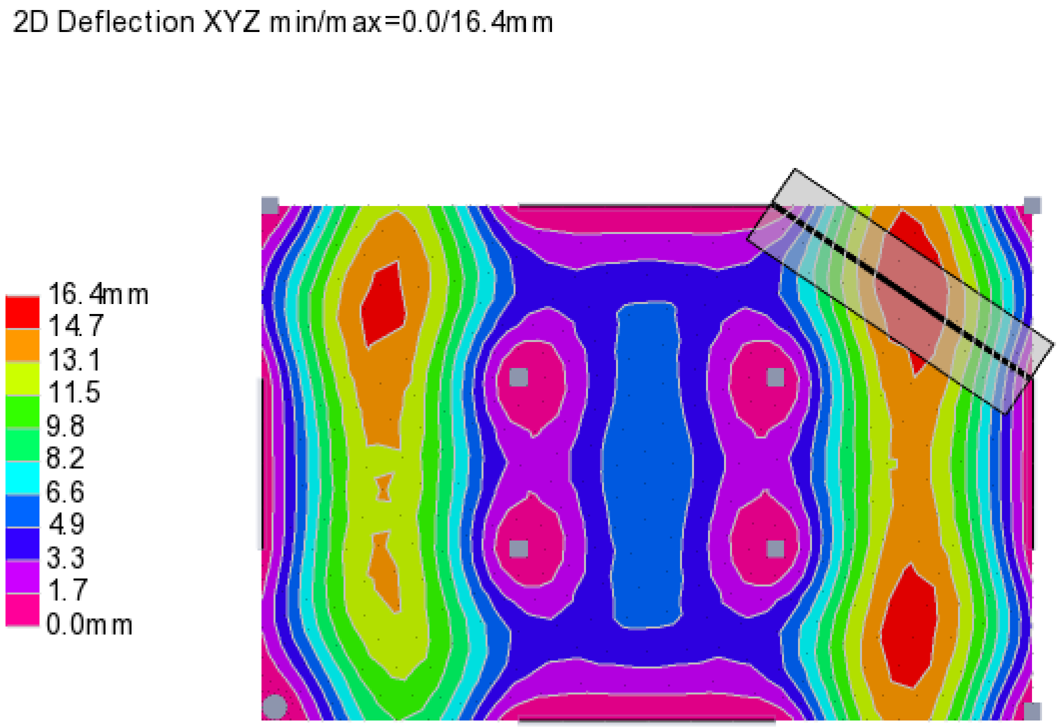
NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.



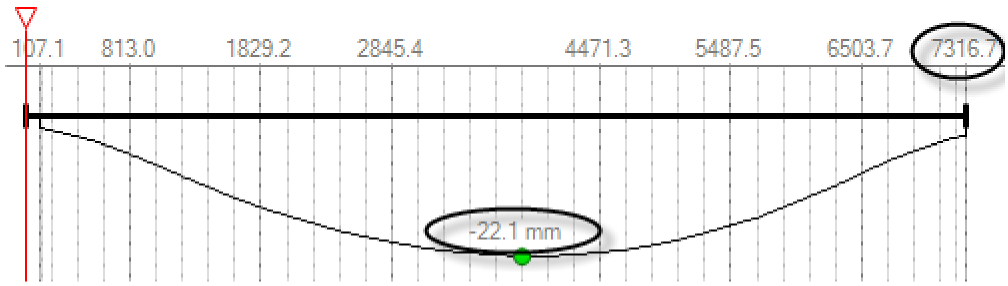
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

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

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



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



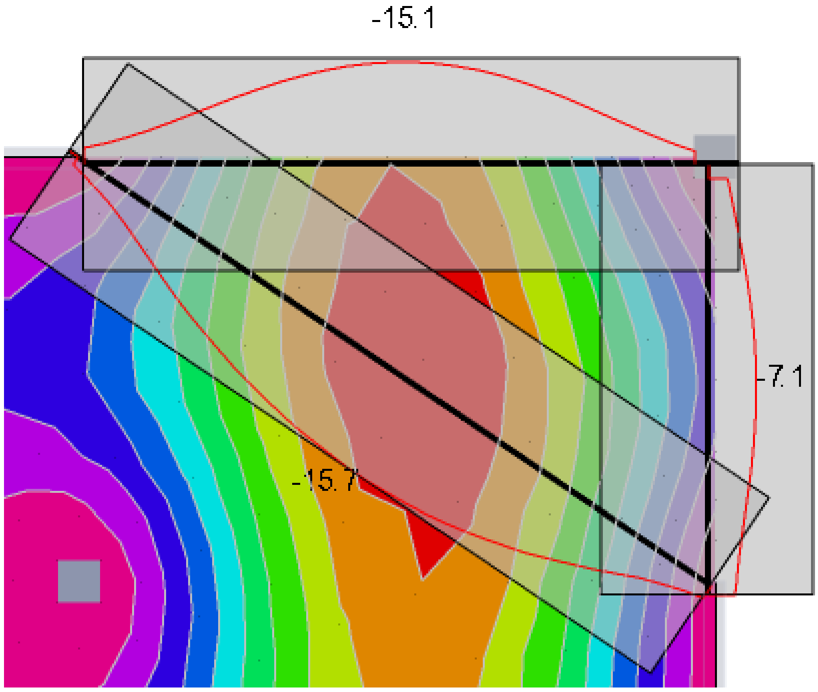
In this example there is a requirement to limit total deflection to span/250

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

Deflection limit: 7317/250 = 29mm

Actual deflection: 22mm - the check passes

The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Select a Level, or sub-model

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

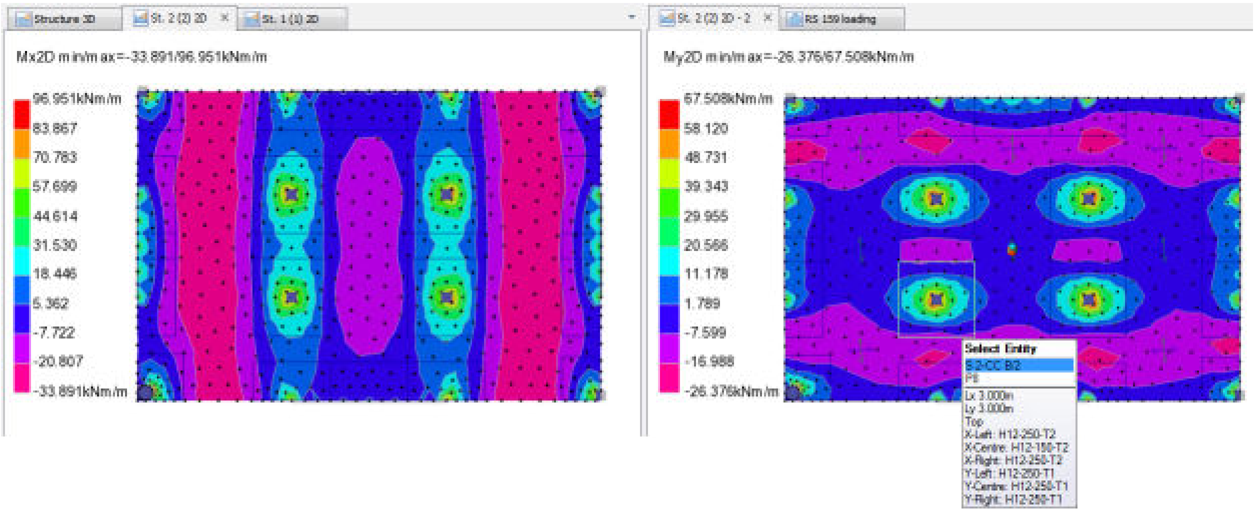
NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mx moments on in one view on the left and My moments in a second view on the right, as below:



By doing this, it is possible to see how patches extend over the peaks.

Typically, at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

In a "slab on beam" situation, you may want to add beam and wall patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

Design panels

NOTE Panel design is dependent on the areas of patches (patch areas which are excluded from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the Design ribbon run Design Slabs in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Slabs or Check Slabs. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

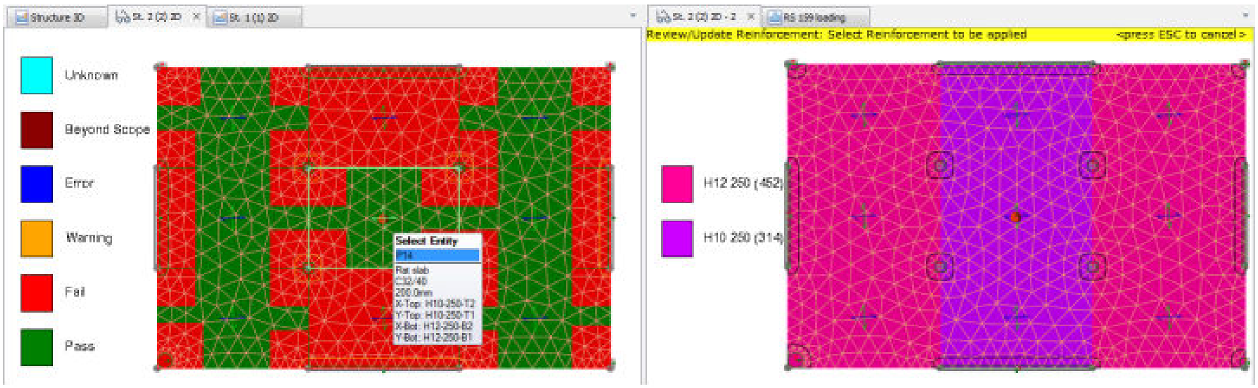
NOTE When accessed from the right-click menu, these commands operate as follows:

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

- Check Slabs will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

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



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

The view on the left shows Slab Design Status, the view on the right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a flat slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Design ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

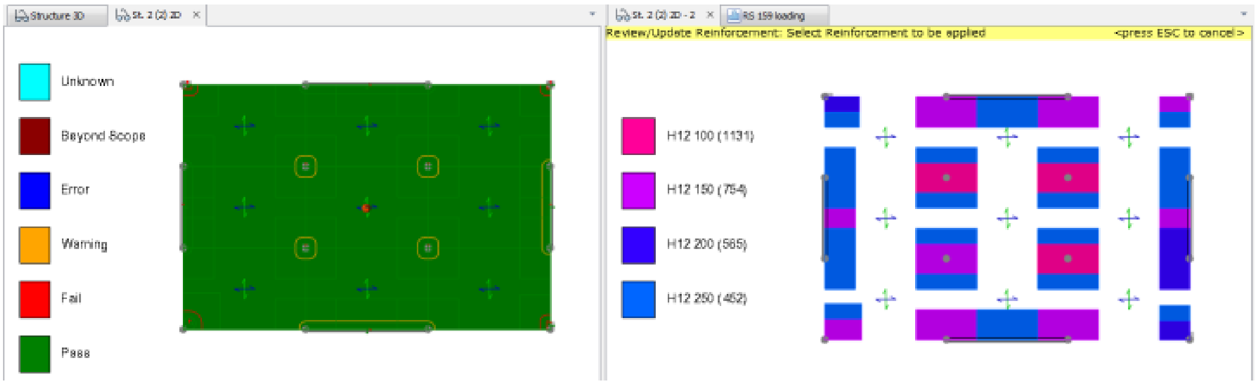
- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Column patches - Is the size reasonable - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the slab reinforcement to be matched (if the slab has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.



Add and run punching checks

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

Punching checks can be added over the entire floor, or structure by windowing it. You can then select any check and review the properties assigned to it. Internal/edge/corner locations are automatically determined (with a user override if you require). Once added click Design Punching Shear and the checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the slab

NOTE In order to see the punching check status in the Review View you might need to first switch off Slab Items and Slab Patches in Scene Content.

Rigorous deflection check

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis. For futher details, see:

Create drawings and quantity estimations

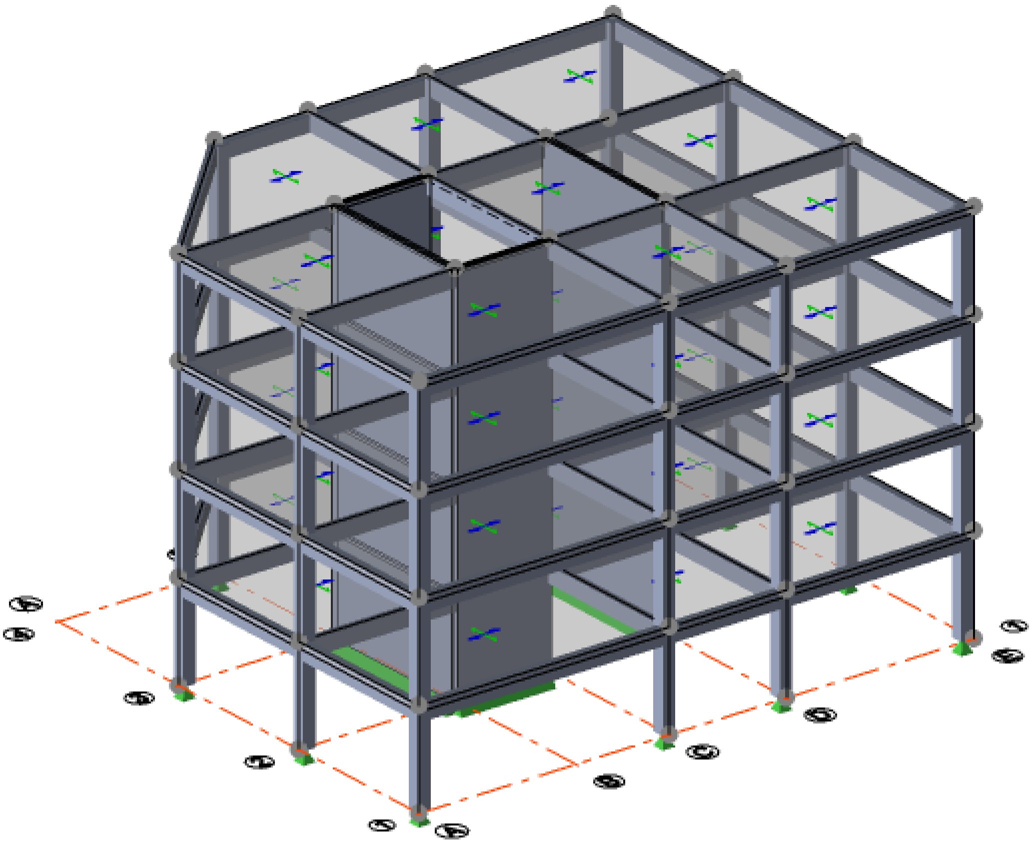
Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

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).

Slab on beams design workflow

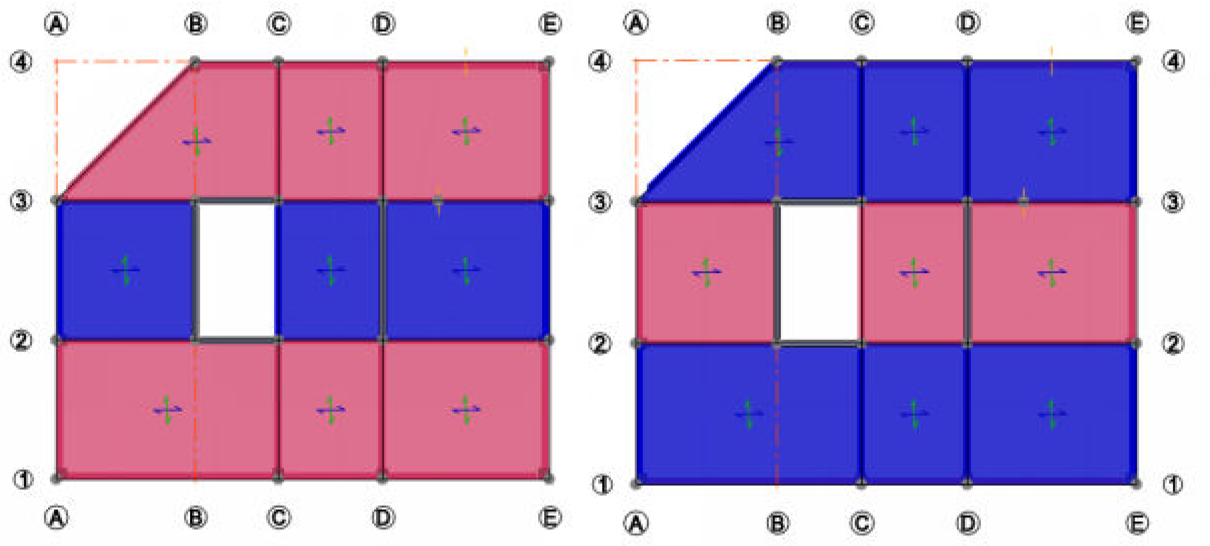
A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.



Note that not all the slab panels are rectangular.

Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Establish slab design moments

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

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Select a Level

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

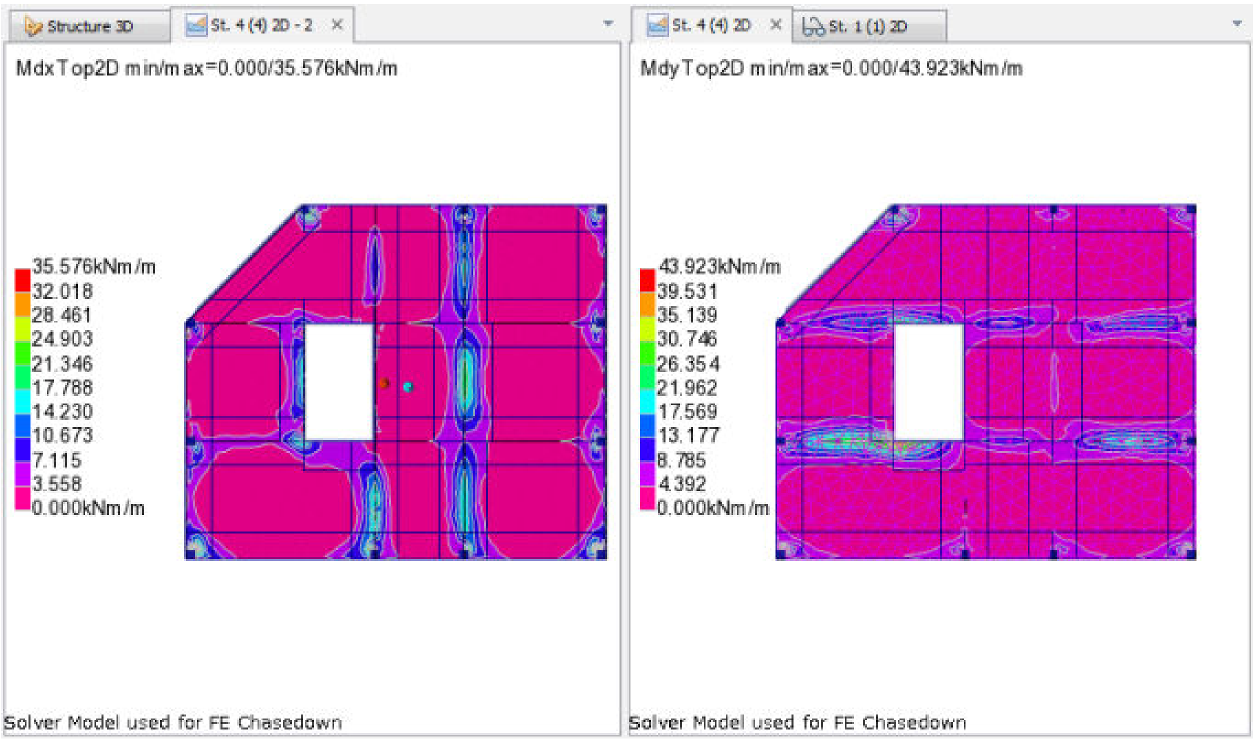
Add beam and wall top patches

You may optionally want to add beam and wall "top surface" patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating beam patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mdx top moments on in one view on the left and Mdy top moments in a second view on the right, as below



By doing this, it is possible to see how patches extend over the moment contours.

It is suggested that at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

Design panels

NOTE Slab on beams panel design takes account of any beam or wall patches (by excluding the patch areas from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the Design ribbon run Design Slabs in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Slabs or Check Slabs. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

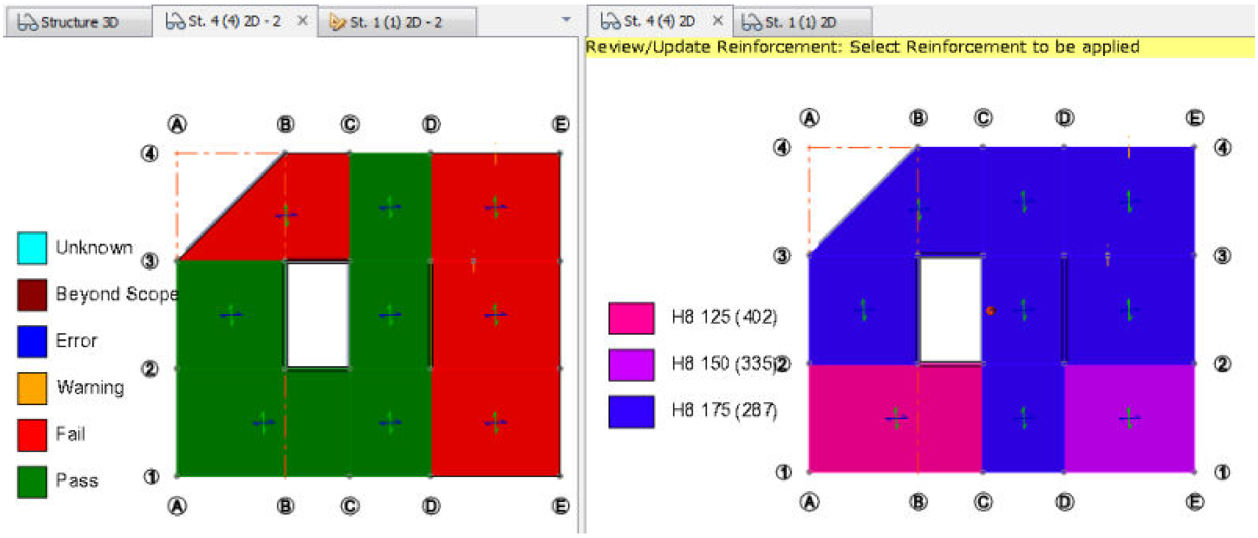
NOTE When accessed from the right-click menu, these commands operate as follows:

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

- Check Slabs will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

It is suggested that you use split Review Views to examine the results as indicated below.



The view on left shows Slab Design Status, (with slab patches turned off in Scene Content to assist clarity), the view on right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.
- If the span-effective depth check is failing, review the panel properties to confirm that edge categories are set correctly, (and if the panel shape is irregular consider whether it is being suitably idealized as a rectangular panel). See: Slab on beam idealized panels (page 353)

Design beam and wall patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design any beam or wall patches that you may have defined.

To do this, either:

- From the Design ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Concrete slab design aspects

Concrete type

While you can apply both normal and lightweight (LW) concrete in the slab properties, slab design using lightweight concrete is only available for Eurocodes.

When using other Head Codes slabs can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

How the load decomposition method affects slab design

Only slabs that have been specified with two-way load decomposition are designed.

One-way load decomposition in Tekla Structural Designer is a simple procedure that does not determine slab design forces. When a slab's decomposition is set as one-way it is assumed that it is some form of precast slab (presumably designed by safe load tables).

- A flat slab panel always uses two-way load decomposition.

- A slab on beams panel can either be specified to use one-way or two-way decomposition - however if it is specified as one-way it cannot then be designed in Tekla Structural Designer.

It should be noted that any in-situ slab is capable of two-way decomposition:

- When a slab is set as two-way it will only effectively span in 2 directions if its proportions and support conditions mean that there will be a two-way effect.
- For example - If a slab that has a span of 6 units in one direction and 50 units in the other is set to two-way decomposition, then although it is two-way the FE analysis will still inherently take the load one-way.

Two-way spanning slab panel design moments

When a slab panel is specified with two-way decomposition, a general FE based approach is used to determine the design moments. If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis results are also considered:

- The worst design moment (per unit width) is found in each direction of the slab - if the design moment is zero in one of the directions then the analysis has shown that the slab is effectively spanning one-way and the supplied reinforcement in this secondary direction will be selected to suit the minimum requirements of secondary reinforcement.
- Note that this FE based approach inherently caters for point loads, line loads, openings, etc and for the possibility of variable adjacent span lengths in a continuous "1-way" slab (and of course it can still be applied to the simple case of a "1-way" slab with a uniform UDL applied and uniform span lengths).

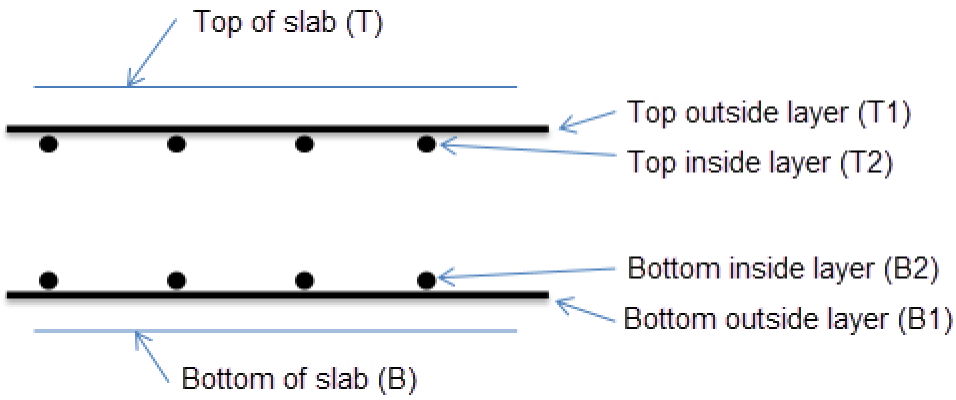
Alignment of slab panel reinforcement

X direction reinforcement is aligned to the panel span, which is determined by the slab's **rotation angle** property. Y direction reinforcement is always perpendicular to the X direction reinforcement.

For further details, see: Rotation angle for slab items and panels (page 355)

Slab panel reinforcement

In a slab you have two surfaces: Top (T) and Bottom (B).



In each surface you have two layers of steel in orthogonal directions - X direction steel and Y direction steel. Layer 1 is the outside layer - the one closest to the surface. Layer 2 is the inside layer. Which direction is the outer layer is controlled by this slab panel setting: Outside layer in X direction.

Any of the four layers can be set to "none" if required.

Slab patch reinforcement

Additional rectangular reinforcement patches can be applied to slab panels:

- column patch - at column stack heads
- beam patch - along beams
- wall patch - along walls
- panel patch - at the panel centre
    - typically positioned centrally - but not restricted to this location and also not restricted to existing purely within one panel
    - can also be positioned under loads

These patches are either in the top or the bottom of the slab and may or may not have reinforcement defined in them. If no reinforcement is defined then the background reinforcement is used. If patch reinforcement is defined then it will either be used on its own, or if you select the "Combine with Panel Reinforcement" option, the sum of the background + patch reinforcement will be used. Note that this option is only selectable when the "Align to Panel Reinforcement" option is also selected since combining in this way would only be valid provided the reinforcement is reasonably aligned. Choosing the reinforcement to be combined also forces the "Cover as Panel" option to be selected as the program assumes the reinforcement to be in a single layer. If the reinforcement is not combined you can specify the cover to the patch reinforcement by turning off the "Cover as Panel" option.

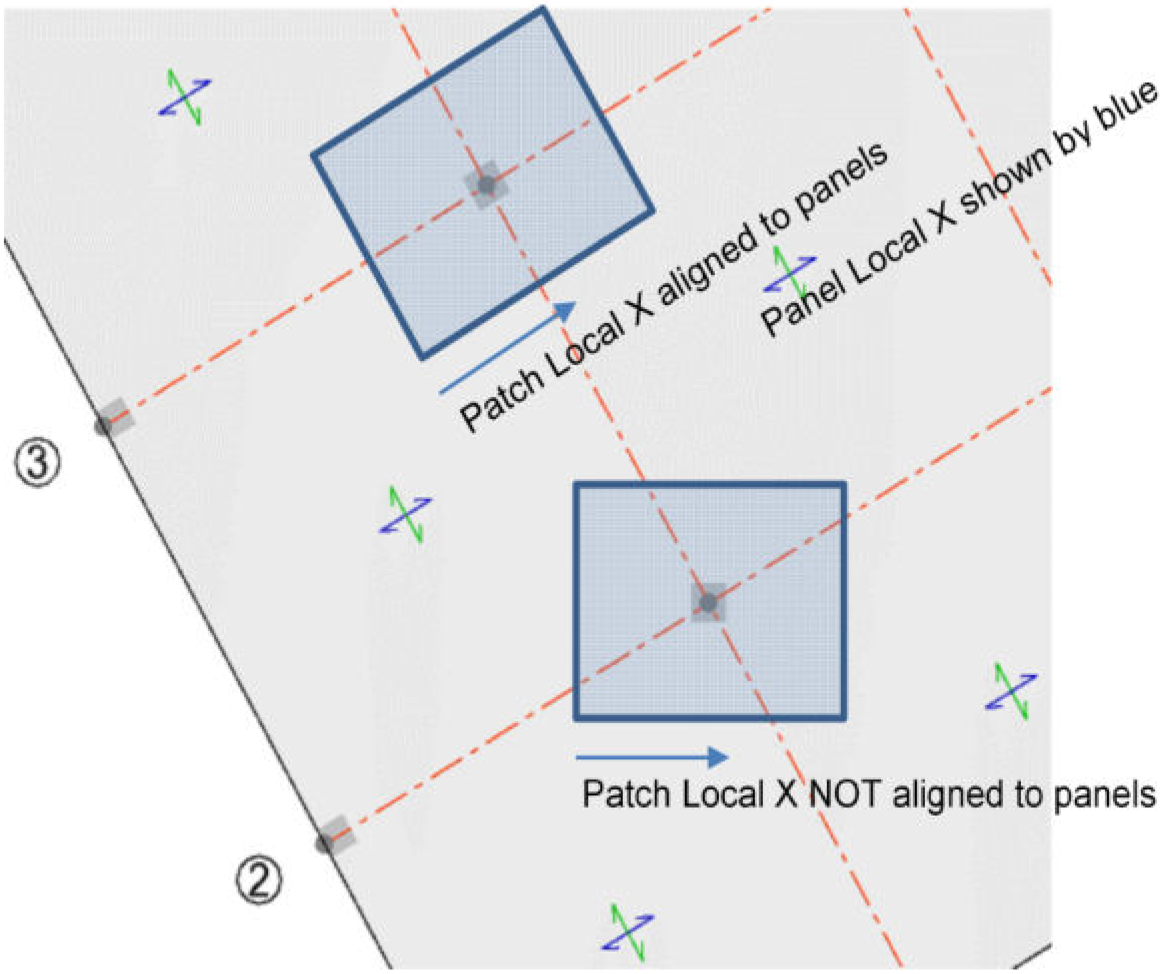
Note that patches may overlap on the plan view, and there is no restriction on this, even patches relating to the same layer of reinforcement are allowed to overlap. This situation is handled conservatively during design by simply ignoring the overlap.

Each patch manages reinforcement and the reinforcement design using a number of slab design strips. Some key points to bear in mind considering patch design are:

- A patch only manages data in the top or the bottom layers of a slab, not both.
- There can be up to 3 design strips in each direction of a patch
- There is no requirement to have design strips in both directions - there can be one design strip in one direction and none in the other.
- Within any strip there might be patch reinforcement to consider but note that:
    - The underlying panel reinforcement can be none
    - The added patch reinforcement can be none.
- If there is patch reinforcement to consider this can be considered instead of, or in addition to, relevant slab reinforcement

Slab patch strip design

For the strip designs within each patch it is necessary to establish which bar layer is to be designed and work out if and how the patch reinforcement combines with the panel reinforcement.



As shown above there are 2 distinct options:

1. Patch aligned to panel
2. Patch NOT aligned to panel:

In both cases the reinforcement that is determined for use in the design checks is some combination of the panel and patch reinforcement. Expanding upon this the cases considered are:

Patch aligned to panel:

1. Patch reinforcement type = NONE then the panel reinforcement is used for all layers
2. Combine with Panel Reinforcement = No Then the patch reinforcement is used in the patch surface and the panel reinforcement is used in the opposite surface.

3. Combine with Panel Reinforcement = YES Then the patch reinforcement is combined with panel reinforcement and used in the patch surface and the panel reinforcement is used in the opposite surface.

Patch NOT aligned to panel:

Patch reinforcement is used in the surface to which the patch applies. Reinforcement in the other surface is taken from the panel using the most aligned possibility.

Patches to both surfaces

As stated above, patch reinforcement is only ever used in the surface to which a patch applies, reinforcement in the other surface is taken from the panel.

In rare situations you may have separate patches on both surfaces; in which case you would want the patch reinforcement from the top patch to be considered on the top surface and patch reinforcement from the bottom patch to be considered on the bottom surface.

In this specific situation, for both patches the other surface doesn't necessarily need to be designed to only consider reinforcement in the panel; you can avoid this by selecting the patch property **Consider patch surface moments only**.

Span effective depth checks for irregular shaped panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

For further details, see: Slab on beam idealized panels (page 353)

Slab on beam idealized panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

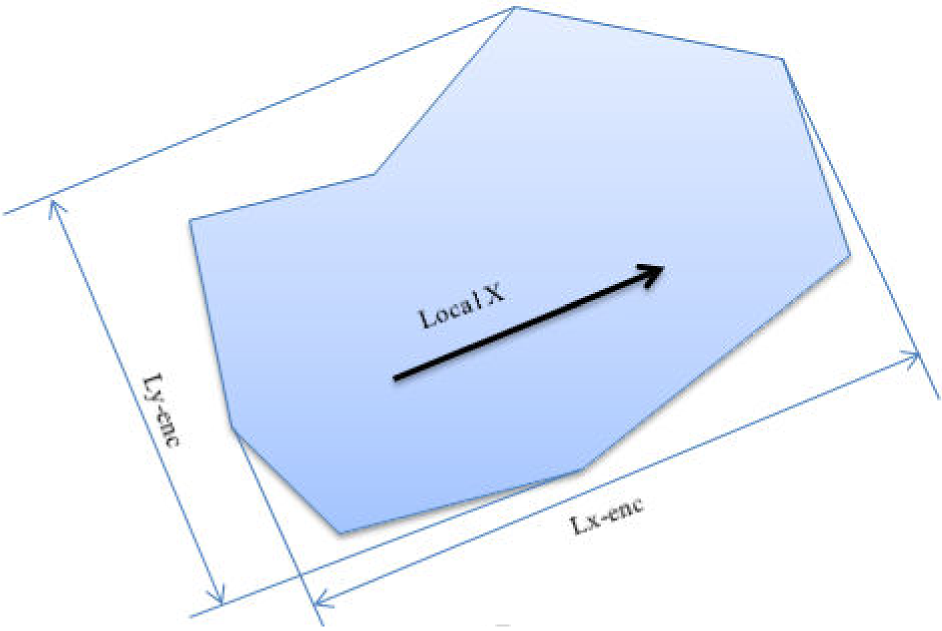
To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel rotation angle):

Lx-enc = maximum overall length of the panel measured parallel to local X

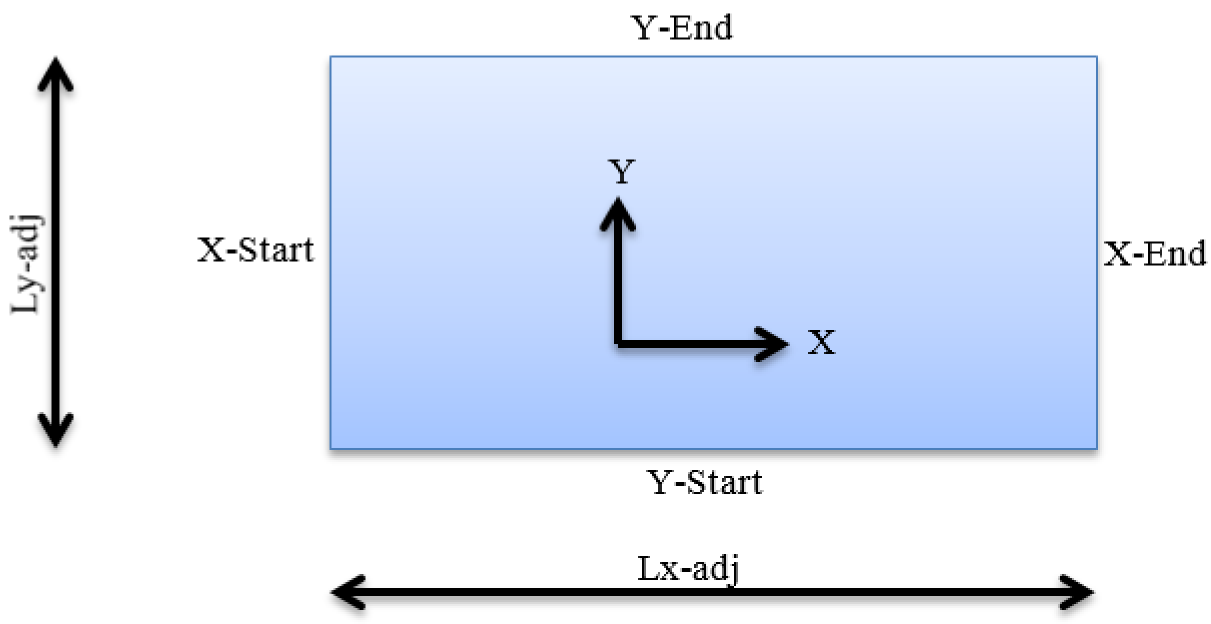
Ly-enc = maximum overall length of the panel measured perpendicular to local X

A user specified adjustment ratio is then applied to these lengths to determine the adjusted lengths. Conservatively the adjustment ratio defaults to 1.0 in both directions.

In situations where the panel does not have 4 sides, (such as the one shown below), some engineering judgement might be required when deciding on appropriate values of the adjustment ratios in each direction.



The resulting idealized panel with dimensions in X and Y is illustrated below:.



Edge Category

For the span-effective depth check, the edge categories in each direction have to be manually assigned to the idealized slab panel. The three edge categories being:

- Unsupported
- Continuous Support
- Dis-continuous Support (default)

Rotation angle for slab items and panels

The rotation angle in the slab item/panel properties is used to control the span direction. For reinforced slabs the X direction reinforcement is aligned to the span.

NOTE When SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow.

Slab items and panels defined by a level

When the rotation angle/X direction = 0

- Span direction aligns with global X

When the rotation angle/X direction = 90

- Span direction aligns with global Y

Panels defined by a frame

When the rotation angle = 0

- Span direction is horizontal

When the rotation angle = 90

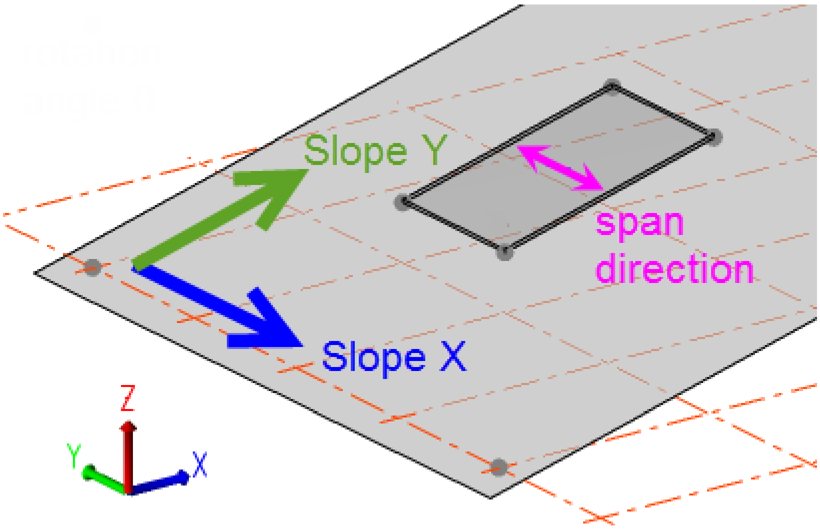
- Span direction is vertical

Slab items and panels defined by a sloped plane

NOTE In a sloped plane positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

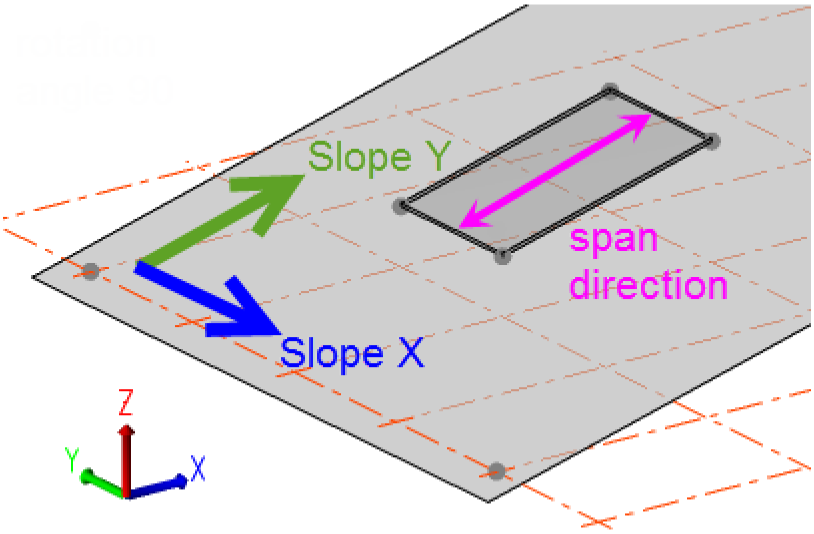
When the rotation angle/X direction = 0

- Span direction aligns with X direction of the slope (i.e. perpendicular to the slope, as shown below)



When the rotation angle/X direction = 90

- Span direction aligns with Y direction of slope (i.e. parallel to the slope, as shown below)



1.7 Slab deflection handbook

This handbook provides an overview of how iterative cracked section analysis is applied in Tekla Structural Designer for the purpose of obtaining a better estimate of slab deflections.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

Related video

Rigorous Slab Deflection (Eurocode)

Rigorous Slab Deflection Design (ACI)

Slab deflection methods

Concrete is considered a durable and economic material for floors systems. However, reinforced concrete slabs deflect. The magnitude of the deflection is more complicated for concrete as deflection increases with time. It's long term behavior is characterized by cracking caused by flexure, shrinkage and creep. If this is not taken into consideration by allowing adequate tolerances to glass facades and internal partitions for example then problems can arise.

Tekla Structural Designer provides two alternative methods for checking deflections. Either:

- Deemed-to-satisfy checks, or
- A rigorous theoretical deflection estimation using iterative cracked section analysis

Deemed-to-Satisfy Checks

A couple of deemed-to-satisfy checks are presented here.

1. The use of a limiting span-to-depth ratio (L/d) method. This method is assumed to 'be adequate for avoiding deflection problems in normal circumstances'. It can only be considered as a rough deflection estimate and is not intended to predict how much a member will actually deflect. Total deflection is expected to be < span / 250 (Eurocode) or < span / 250 (US) and deflection affecting sensitive finishes is expected to be < span / 500 (Eurocode) or < span / 480 (US)
2. A linear analysis with adjusted analysis properties.

It is important to appreciate that these deemed-to-satisfy methods do not predict actual deflections even though the linear analysis method provides a total deflection that can be checked against the span / 250 (Eurocode) or < span / 250 (US) limit mentioned above.

When normal deflection limits do not apply, for example, due to stricter usage limits, glazed cladding systems or where a faster pace of construction is applied then the 'deemed-to-satisfy' checks are no longer applicable - the alternative is a rigorous deflection estimation which is the primary topic for this guide.

See also

- Deemed to satisfy slab deflection checks example (Eurocode) (page 402)
- Deemed to satisfy slab deflection checks example (ACI) (page 438)

Rigorous theoretical deflection estimation

The rigorous theoretical deflection assessment takes into account cracking, creep and shrinkage over time.

In the UK, rigorous deflection estimation is taken to mean deflection estimation in accordance with the Concrete Society Technical Report 58.

The principle of assessing deflections rigorously involves assessing the curvatures induced by both load and shrinkage, adding them together and then the total curvature is translated into a deflection.

The Technical Report discusses the importance of construction events. Total deflection at the end of every event comprises:

- An instantaneous deflection which is influenced by the extent of cracking
- An additional accumulated creep deflection
- An additional accumulated shrinkage deflection.

Once these totals are known, differential deflections between any two events can be calculated.

The Technical Report gives detailed guidance on some very complex looking calculations - it all seems very "rigorous". However, we must not lose sight that the material - Concrete is a very variable material. Furthermore, how accurately can we really predict input parameters such as event loads and timings? The report advises that deflection accuracy can only be considered an estimate in the range +15 to -30%.

In the US, the basic approach described in ACI 318 has a similar approach to cracking, interpolating between the fully cracked and the uncracked states, (although it doesn't recognize the reduction in tension stiffening). For creep and shrinkage, in ACI 318 there is a single multiplier for the deflection calculated from the cracked flexural rigidity. There are additionally two ACI Committees 435 and 209 which go into more detail about creep and shrinkage calculations. The US user can therefore either adopt the basic ACI 318 approach, or, take on board the ACI Committee 435 & 209 guidance. In special situations the TR58 approach could even be considered.

Expectations - It is incorrect to think that rigorous methods will provide greater economy. i.e. by allowing the engineer to reduce slab thickness or the quantity of reinforcement. The end result is greatly influenced by the various input parameters which each can impact on the deflection.

See also

- Rigorous slab deflection workflow (page 359)
- Rigorous slab deflection analysis example (Eurocode) (page 405)
- Rigorous slab deflection analysis examples (ACI) (page 440)

Rigorous slab deflection workflow

You perform rigorous slab deflection analysis in Tekla Structural Designer from the **Slab Deflection** toolbar.

The suggested workflow is as follows:

1. Define the slab loading event sequences (page 368).
2. and review deflections for the final load event.

   You can analyze the current or selected level (sub-model) or choose to analyze every slab in the model. Obviously your choice has an effect on the time necessary to undertake the iterative slab deflection analysis.
3. Define the slab deflection checks via the **Slab Deflection Check Catalogue** and then place Check lines (page 396) as required.
4. Graphically review the Slab deflection status and utilization (page 399).
5. Make adjustments as necessary until the status passes. For more information, see .

NOTE A reasonable level of slab reinforcement should be established prior to running the Slab Deflection Analysis as this will have a significant impact on the deflection calculations.

Factors that affect rigorous slab deflection estimates

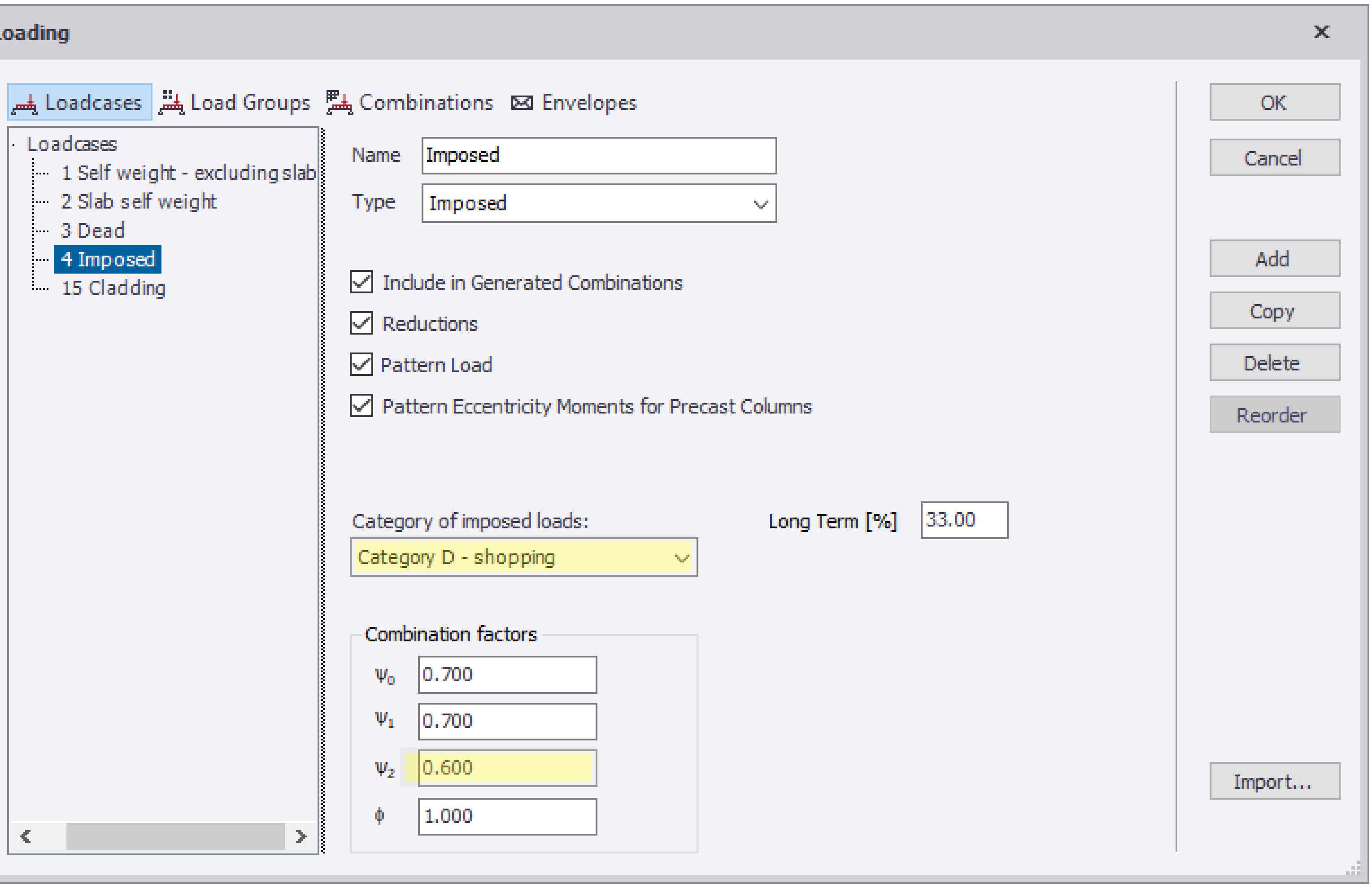
Many input parameters have an impact on the rigorous deflection estimates. These include:

- Level of Restraint
- Concrete properties
- Stiffness adjustments
- Allowance for shrinkage effects
- Event sequence parameters
- and even the assumed analysis properties of connected columns and walls

Some of the key factors are described below, these vary depend on the head code being worked to.

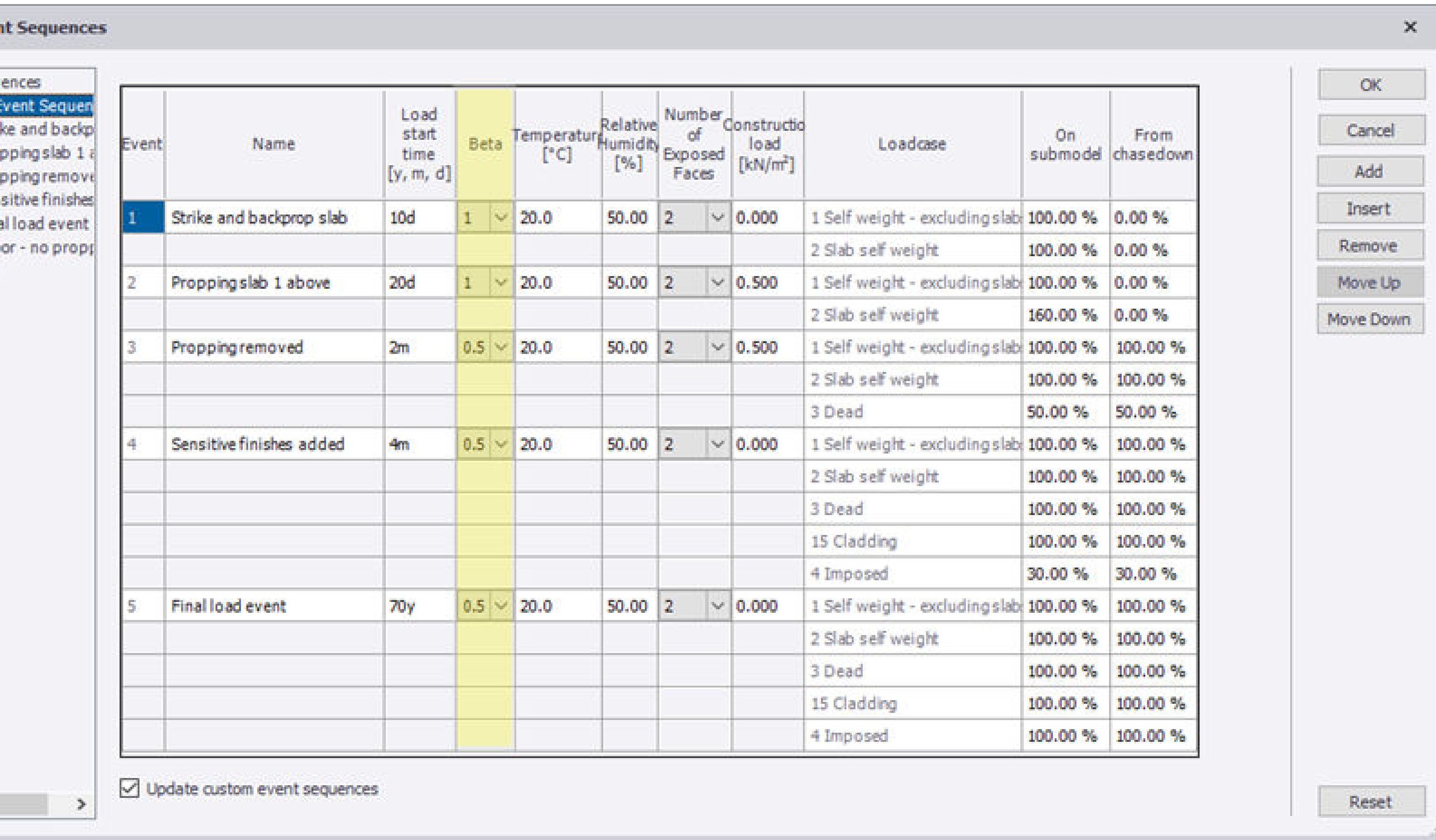
Quasi-permanent load factors (EC2)

An accurate prediction of deflection requires a realistic assessment of the loading. To the Eurocodes long term loads are termed quasi-permanent and a ψ_2 factor is applied to the imposed load which varies based on the use of the structure. These are dealt with when defining the imposed load case.



Beta coefficient (EC2)

This property is set in the Event Sequence dialog.



Event	Name	Load start time [y, m, d]	Beta	Temperatur [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Beta relates to the duration of the load and tension stiffening effects.

NOTE Tension stiffening occurs when the concrete is not fully cracked - because there is still concrete in the tension zone that transfers some tensile forces, the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness).

Due to phenomena such as increased cracking or local bond failure, tension stiffening effects reduce over time - Beta is used to account for these, (see EC2 Clause 7.4.3 and TR58).

- For loads with a short duration, and for cyclic loads, Beta should be set to 1.0.
- For loads with a long duration, Beta should be set to 0.5.

NOTE In Tekla Structural Designer, Beta defaults to 1.0 where the start event time is <= 30 days and 0.5 if >30 days, but may be changed for any event.

Since these phenomena are irreversible, it is not recommended that Events have a value of Beta greater than that set for any previous Event and a warning will be issued if you do this.

However, there are circumstances where you may wish to have a value of Beta=0.5 in the earlier events, and Beta=1.0 in a later Event. For example, TR58 suggests that Beta=1.0 be used for the variable part of imposed load (if you wish to consider that at all).

In this case the analysis will permit you to enter these values. However, because the reduction in tension stiffening is carried forward and is irreversible in the analysis, caution is advised.

To explain this issue more fully, the impact of choosing Beta=1.0 at a later event, for different extents of cracking, is explored below.

Consider 3 possible configurations of a model, where:

1. There are many elements that are uncracked at the end of earlier events. For a later event where the duration is short (which implies you should set Beta=1), setting Beta=0.5 could lead to an overestimation of incremental cracking, and subsequently overestimation of deflection.
2. There are many elements that are cracked in an earlier event, for which you have chosen Beta=0.5. For a later event where the duration is short (which implies you should set Beta=1), if cracking increases in that Event, setting Beta=1.0 could lead to an underestimate of incremental cracking, and subsequently an underestimate of deflection for this later event.
3. The majority of cracking occurs in an earlier event, for which Beta=0.5, if cracking does not increase in a later Event, the value of Beta in that later Event will have minimal impact on deflection. To ascertain whether the cracking increases, you may run separate analyses, with values of Beta=0.5 & 1.0, and compare the results.

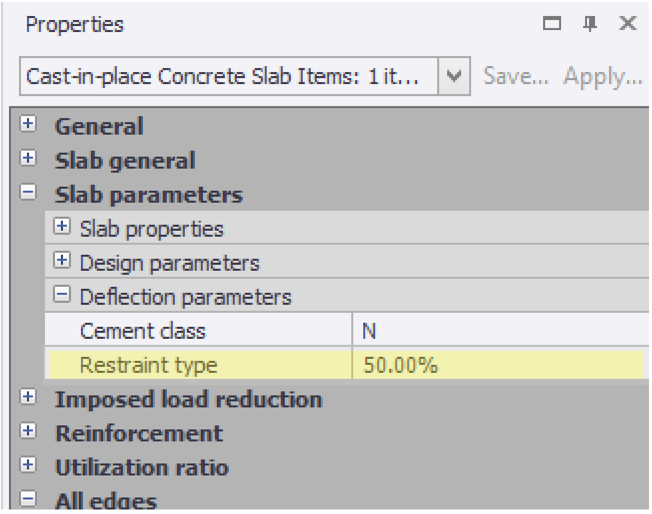
In summary, it is prudent to consider whether cracking is likely to increase in later events for which you wish to set Beta=1.0.

However, it is likely that you will observe minimal difference between the total deflection estimates, and that other assumed values will be of much greater significance.

It is straightforward to run the analysis with different values of Beta to determine whether this is the case.

Restraint type (EC2)

This property is located under the Deflection parameters heading in the slab item Properties Window.



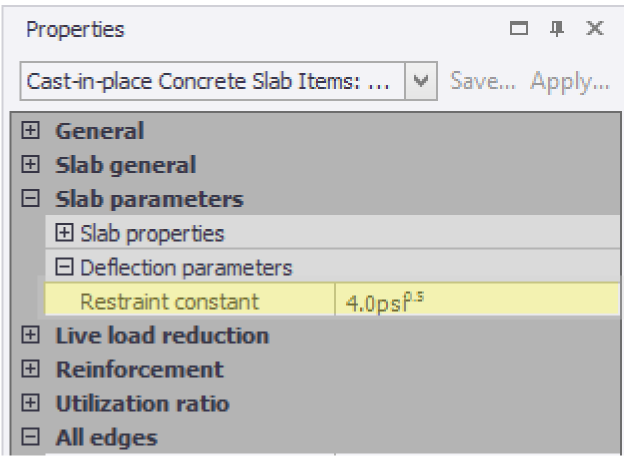
NOTE The Restraint type is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

The value to adopt is a matter of judgment - for guidance on this refer to EC2 cl 7.4.3(4) and TR58

Changing the value will affect tensile strength (f_{ct}) and hence cracking moment.

Restraint constant (ACI)

This property is located under the Deflection parameters heading in the slab item Properties Window.



NOTE The Restraint constant is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

For US customary units, restraint constant values from ACI 435 are:

- For situations with significant restraint - 4.0
- For insignificant restraint - 7.5

The default is conservatively set to 4.0.

The allowable input range is between 1 and 10 and is a user specified value by the engineer.

The restraint constant is used to determine the Modulus of Rupture, f_r = "restraint constant" x λ x √(f_c')

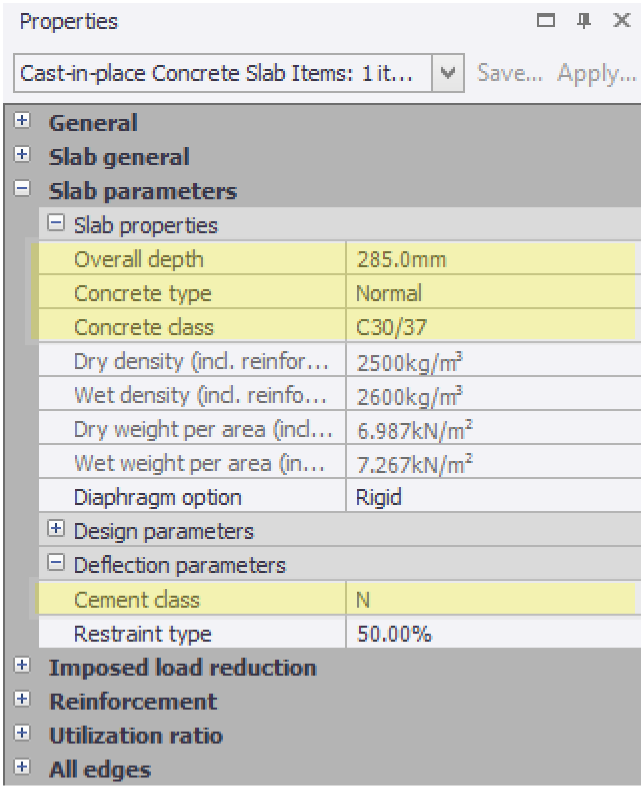
Where:

f_c' = specified compressive strength (psi)

λ = 1.0 (for normal weight concrete)

Concrete Properties (Eurocode)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent. The Cement class can also affect deflection.



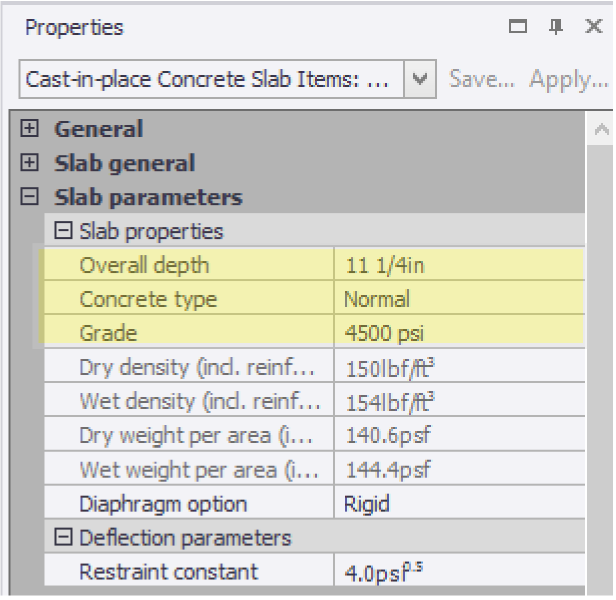
The elastic deformation of concrete largely depends on its composition (especially the aggregates). For C35/45 concrete, Eurocode 1992-1-1:2004 Table 3.1 provides a modulus of elasticity, E of 34GPa (34000 N/mm^2) as a mid range value using quartzite aggregate. For different aggregates this can range between -30% and +20%.

What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Concrete Properties (ACI)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent.



What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Stiffness Adjustments

The relative stiffness of the interconnecting elements will play a role in the force distribution and hence the deflection results.

For existing models the values can be reviewed and amended by clicking **Settings > Modification Factors** on the Slab Deflection ribbon. These are user defined values with assumed defaults.

As mentioned in the above **Concrete Properties (Eurocode)** and **Concrete Properties (ACI)** sections, an alternative to assessing the impact of a different grade could be to alter the Modification Factors specified for flat slab E and G values. i.e. Concrete material property of slab = 34000 N/mm2. What impact would using a value of 32000 N/mm^2 make? Flat slab stiffness adjustment = 32000/34000 = 0.9412

Shrinkage

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

Shrinkage is taken into consideration by making an overall adjustment to the total deflection. This approach is in line with many other software products.

The adjustment is specified in the **Slab Deflection Settings** dialog.

We recommend an allowance with an upper limit of 30%. The default is set as 0.25 (25%).

Event sequences

An Event sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

By default a model event sequence is initially applied to all sub models in the structure; if required custom event sequences can also be defined to override the model event sequence for specific sub models.

NOTE Event sequences are NOT structure event sequences, i.e. they do not describe all the events starting from the first day of the overall construction.

Click the links below to find out more:

- Construction stage events (page 368)
- A typical model event sequence (page 369)
- Custom event sequences (page 377)
- Understanding event sequence deflections (page 380)

Construction stage events

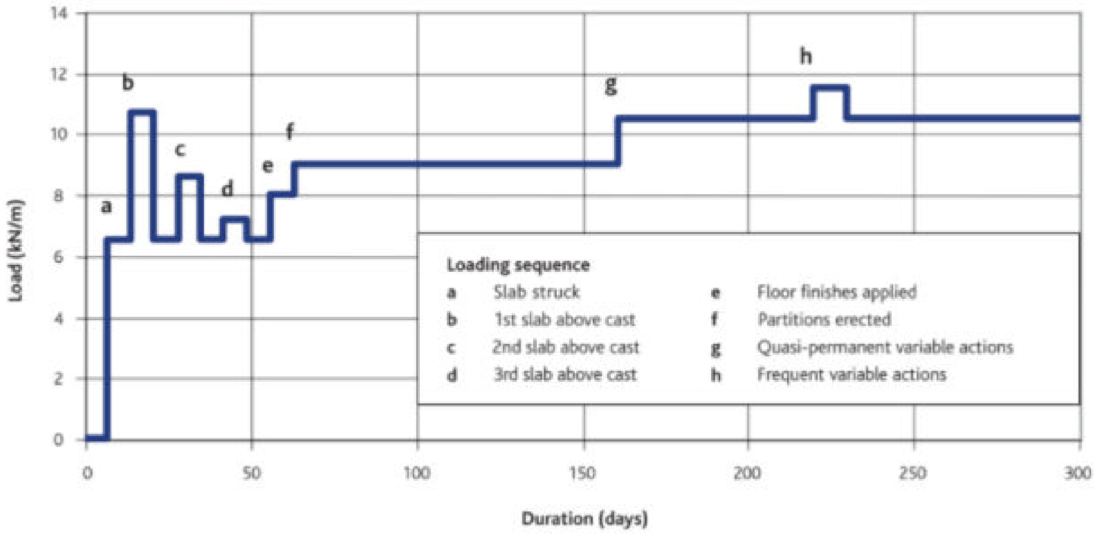
An Event Sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

Deflection is very dependant on the extent of cracking. Once the slab has cracked then it is assumed to remain cracked. The tensile stress of concrete varies with time so careful consideration of load events are required so predicted stresses can be compared with allowable at the appropriate age. Propping loads from slabs being cast above are acknowledged to be significant early loading events which will cause cracking.

In addition to the Final load event, construction stages that could be considered are:

1. Striking
2. Casting the floor above

NOTE Propping loads from casting slabs above have an impact on already struck slabs - dependant upon the extent of back-propping adopted.

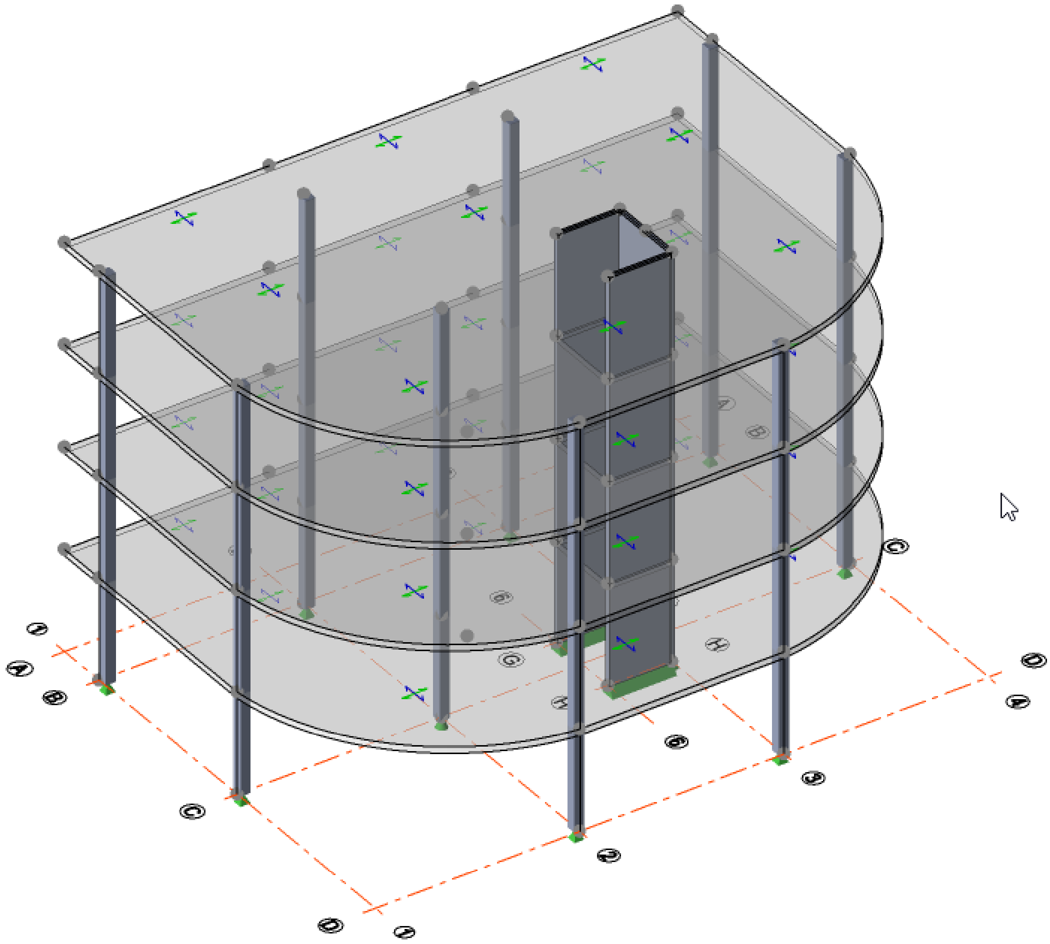


Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

3. Adding Partitions
4. Finishes

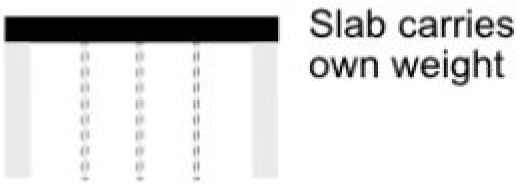
A typical model event sequence

Let us consider a multistory building where the slab layout is the same on each level.

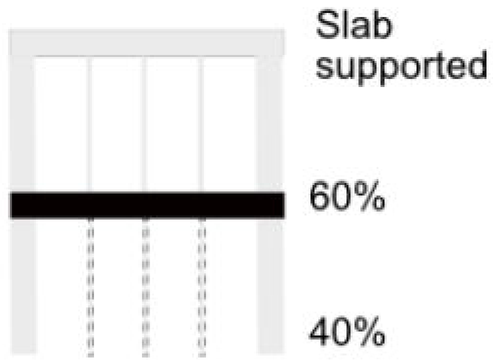


If we think about the slab model event sequence that occurs for the slab at level 1 it could be something like this:

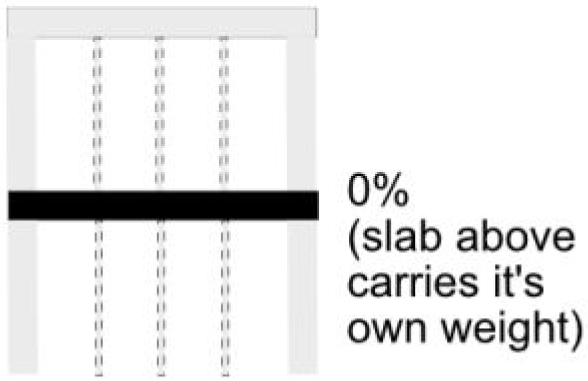
1. Strike and backprop slab (slab carries it's own weight)



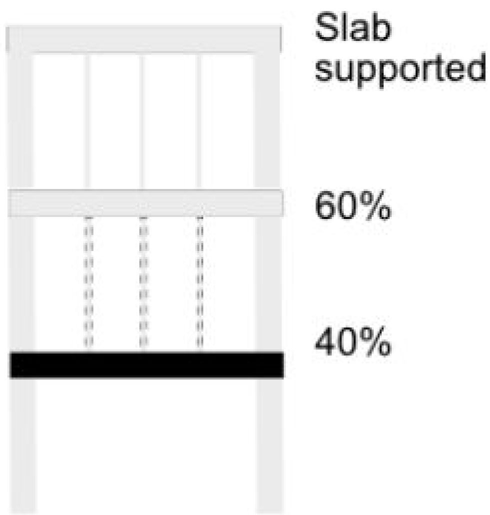
2. Cast Slab above (slab carries a proportion of the weight of slab above - the proportion is dependent on the number of levels of backpropping and there is also some debate about the efficiency of load sharing between the supporting levels. In this example we will assume 2 levels of propping and that the level directly below supports 60% and level 2 below supports 40%.)



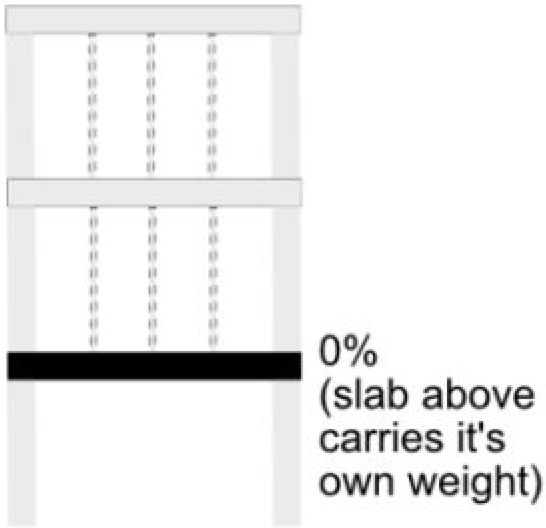
3. Strike and backprop slab above (slab above now carries its own weight)



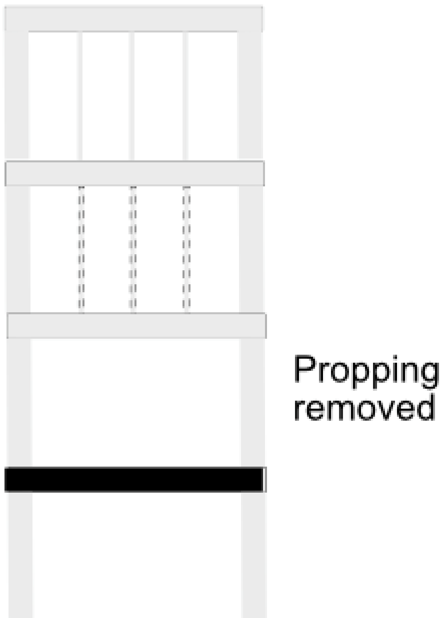
4. Cast slab 2 above (slab carries a proportion of the weight of the slab 2 above)



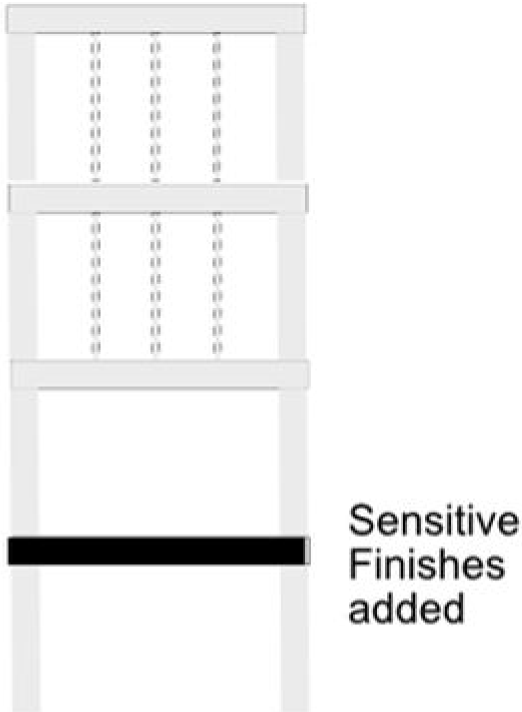
5. Strike and backprop slab 2 above (slab 2 above now carries its own weight)



6. If propping extends through 3 levels then there can be events for casting and striking the slab 3 above. In this example, the propping is removed and used two levels above.



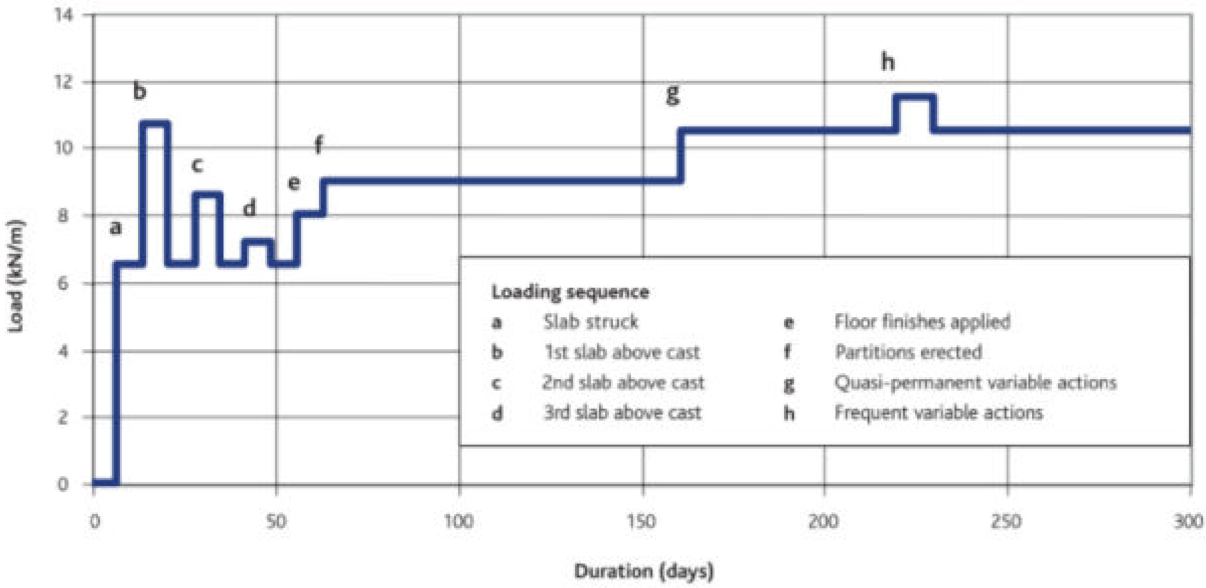
7. Additional load from finishes (in particular from sensitive finishes)



8. Start of occupation.
9. Final Load Event

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

The view shown below is a graphical representation of the above but in this case is recognizing 3 levels of backpropping



Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

This sort of event sequence can be described within the Event Sequences dialog as shown below.

NOTE Some of the event parameters vary between headcodes, so both the ACI and Eurocode versions of the dialog are shown.

Model Event Sequence ACI (US Imperial units)

:vent	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Constructio load [psf]	Loadcase	On submodel	From :hasedown
1	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	160.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Model Event Sequence Eurocode (metric)

:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m^2]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	160.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Note the increased slab self weight when casting slab 1 above and slab 2 above where an additional 60% and 40% of the slab weight is being supported respectively. This is defined in the On submodel % column.

A slab model event loading sequence for any other internal slab i.e. level 2 would be identical to that described above.

If we now consider the uppermost slab - the roof.

- Is the slab event sequence for the roof any different?
- Can we use the slab model event sequence above for the roof?

The event sequences for casting any slabs above are not required, since there is no need to make an allowance for additional propping loads. This means that a different event sequence is necessary to deal with the roof. In Tekla Structural Designer differences in event sequences are dealt with using a Custom Event Sequence.

Technically, we could deal with the propping events in one of two ways.

1. Delete the event
2. Keep the event, but adjust the included slab self weight to allow only for the roof load as the previous event. i.e the event has no change in load to that of the previous load event.

In Tekla Structural Designer, we must deal with the change using method 2 above. This is due to the way Tekla Structural Designer deals with Custom event sequences (page 377).

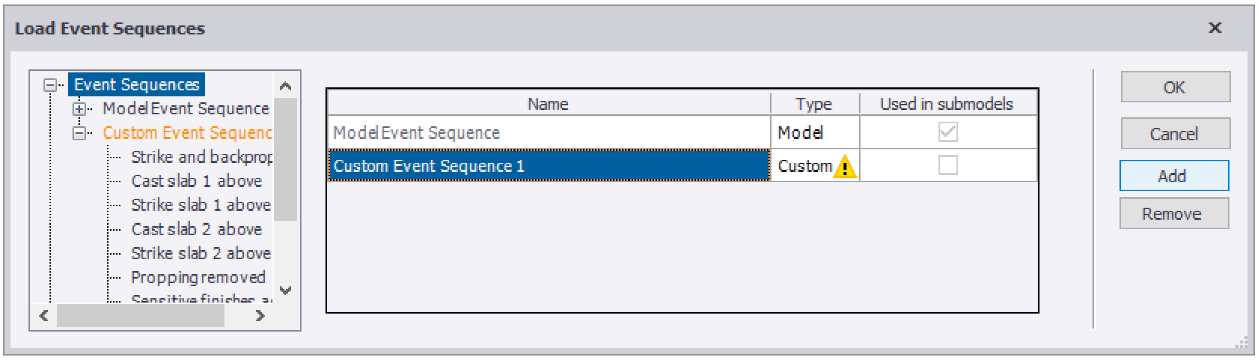
See also

Custom event sequences (page 377)

Custom event sequences

Custom event sequences are required to deal with different slab loading sequences, such as for the roof slab and special cases like transfer slabs.

A custom event sequence can be created via the Add button on the Event Sequences page (highlighted below).



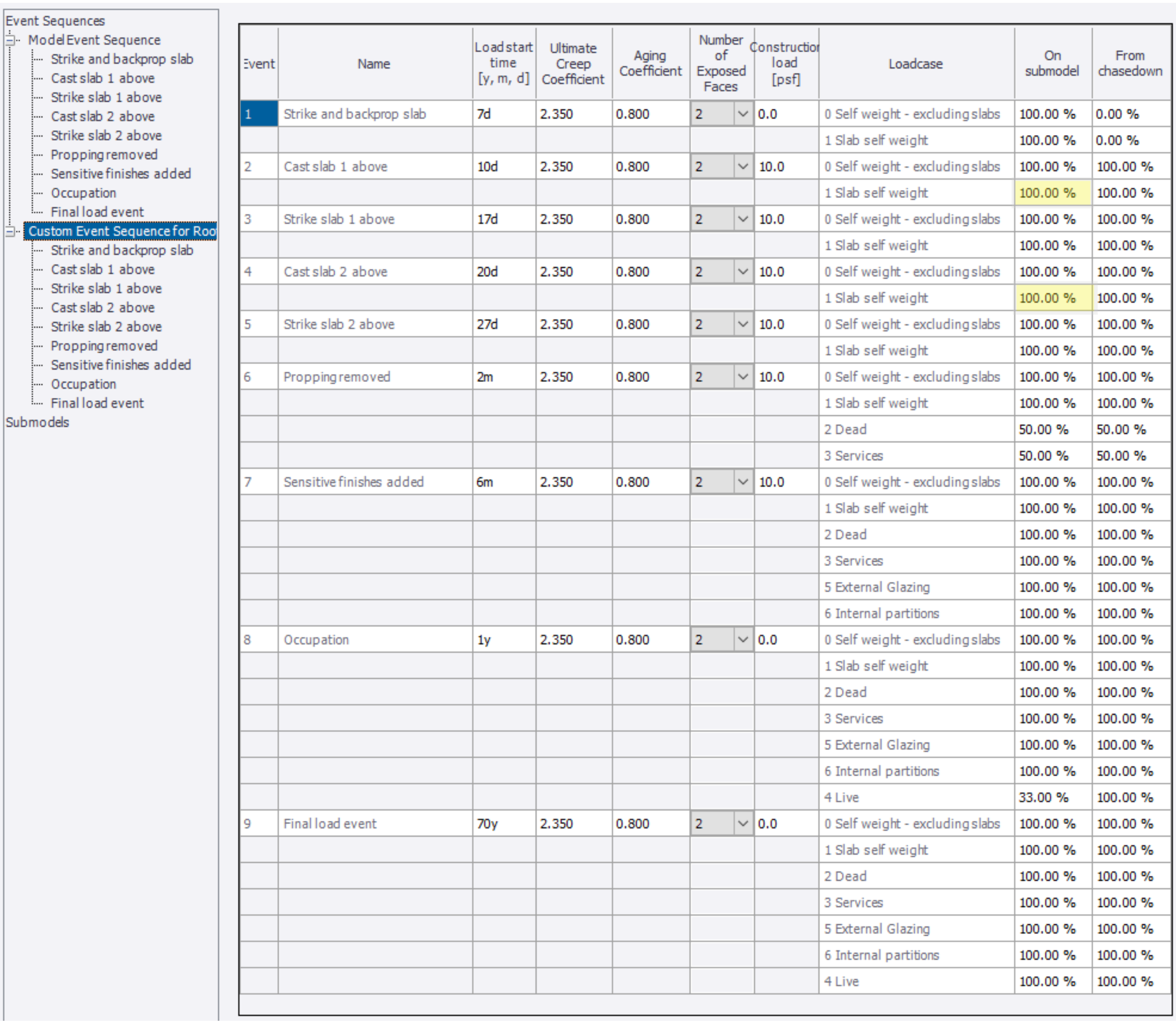
Once it has been added and given a name you can then edit the Custom event sequence by selecting it in the list.

Editing a custom event sequence

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

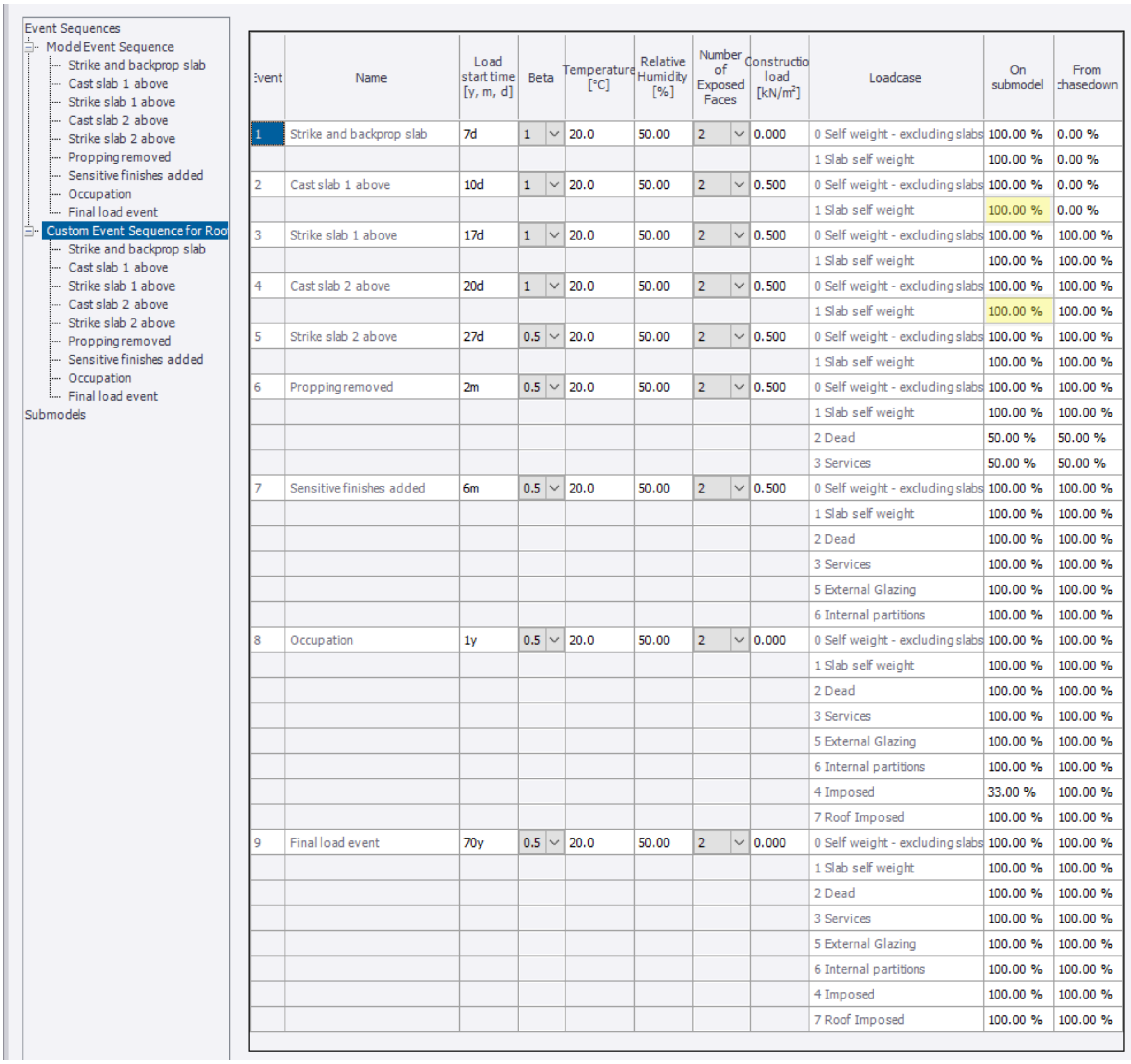
In the screenshot below, to create a custom event sequence for the roof we have edited the slab self weight load in the Cast slab 1 above and Cast slab 2 above events to 100% i.e. it is only supporting the roof slab self weight and not any extra propping load from any slabs above.

Custom Event Sequence ACI (US Imperial units)



Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Custom Event Sequence Eurocode (metric)



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %

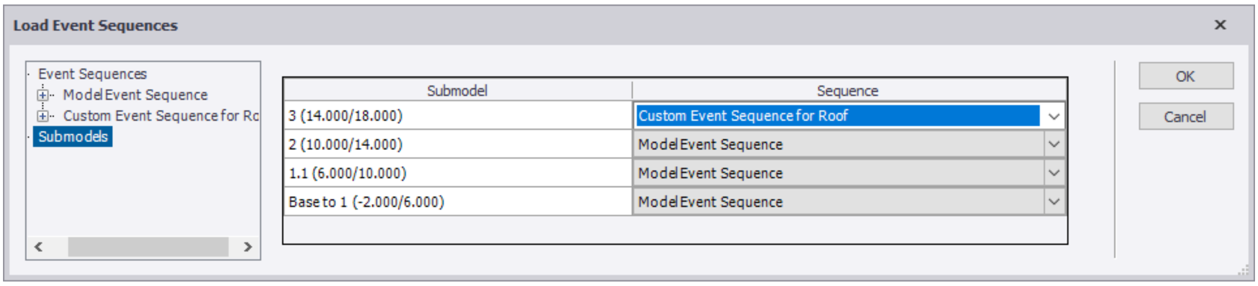
It should be noted that it is not possible to Add, Insert, Remove or change the order of Custom event sequences. It is also not possible to alter any item that is greyed out within the table. This includes events and loadcases. You can, however, edit the Load start time, Number of Exposed faces, Construction loads and the % of load to apply in the combinations (and if working to ACI: Ultimate Creep and Aging Coefficients; or if working to Eurocodes: Beta, Temperature, Relative Humidity). This could mean that careful consideration of the slab model event sequence is necessary to ensure any necessary events are included.

Assigning a custom event sequence to a submodel

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

You can assign different event sequences to different submodels (slabs) using the Submodels page of the Load Event Sequence.

In the screenshot below, the slab model event sequence is assigned to all slabs, except the roof, which has its own custom event sequence defined.



Hence, when you run a rigorous slab deflection estimate on a selected slab, it runs the Load Event Sequence specified on the Submodel page above.

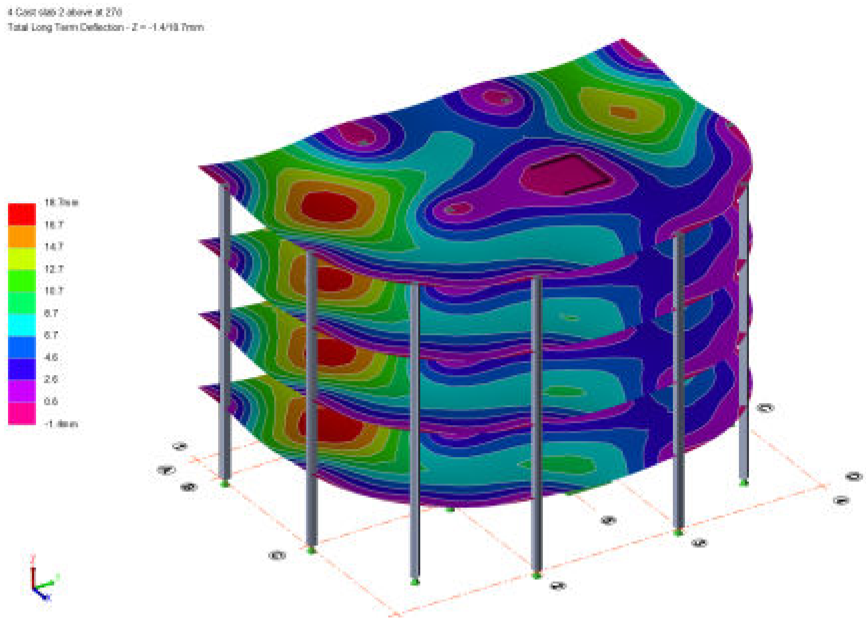
Understanding event sequence deflections

The model described in A typical model event sequence (page 369) consists 3 identical levels and then a roof level with different loading applied. To cater for this a Custom Event Sequence has been applied to the roof submodel.

When you run **Slab Deflection > Analyse All** for this model, 4 sub-models are dealt with one after the other:

- Each one considers 9 events
- Each one does an iterative cracked section analysis at every one of these events
    - assume say, average of 20 iterations in each cracked section analysis
    - That's 180 analyses for each sub-model - over 700 analyses for the whole structure.
- There is an additional instantaneous analysis for each event.

After analysis the deflections can be reviewed for individual events. The view below shows the deflections for Event 4 “Cast slab 2 above”.



Event 4 ends when the slab is 27 days old. The view above is the deflection estimate when each of the slabs are this age. These things don't happen at the same time but it's a very convenient way to display things.

Initially, this has the potential to be confusing, however it should be easy to understand provided you remember that:

- Event Sequences are “Slab Event Sequences”. They describe the events a slab goes through where Day 0 = the day the slab is cast.
- Event Sequences are NOT structure event sequences. They do not describe the all the events starting on day 1 of the overall construction.

Slab deflection analysis sequence

The same basic process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

Having run a Slab Deflection Analysis the following analysis results are then available to review for either the chosen level (sub-model) or the entire structure dependent upon your chosen analysis

- Deflections (page 382) - Three deflection types are available: Total, Differential, and Instantaneous.
- Extent of Cracking (page 386) - You can also review contours to display the extent of cracking at any load event.
- Relative Stiffness (page 388) - You can also review the relative stiffness in a particular result direction for any specified event.
- Effective Reinforcement (page 390) - You can also review the area of effective reinforcement for a chosen result direction for each FE element.

Total, differential, and instantaneous deflection types

Once the slab deflection analysis has been run, three deflection types are available for review:

- **Total** deflection at the end of any event.
- **Differential** deflection between any two events (Start of Event and End of Event).
- **Instantaneous** deflection (not actually needed for TR 58).
    - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
    - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

Understanding differential deflections

When looking at differential deflections it is important to appreciate that an event is not a single point in time, it is a time period which has different deflections at the start and the end - so it is logical that there can be a differential deflection for a single event.

To clarify the selections requested in the ribbon:

- **Start Event** defines the **start** of the start event
- **Event** defines the **end** of the event

Therefore, if you specify the same event as the start and the end event, you will still see a differential deflection.

To clarify further, consider the scenario below:

- Event 1 - Construction loading

- Event 2 - Application of Sensitive Finishes
- Event 3 - Application of Additional Finishes
- Event 4 - Occupation (all loading with live load set at long term factor)
- Event 5 - Final loading event (as above with live load set at 100%)

To determine the maximum differential deflection relevant to the application of sensitive finishes you would define this as the deflection between Events 2 and 5 (Start of Event 2 to End of Event 5). If you opted to pick Events 1 and 5 you would get an un-conservative deflection reported since the effect of construction loading is also considered.

Slab deflection calculations in depth

Interrogating slab deflection calculations

Tekla Structural Designer's Slab Deflection Analysis is not a "black box" - the calculated results are all exposed for interrogation as required.

Several different Results views are available: Deflection, Extent of Cracking, Relative Stiffness and Effective Reinforcement. There is also a Composite modulus report available for each slab.

To help pull all the results together, the items noted above are tied together in the following way:

1. Every element is part of a slab item. Each slab item can have different effective concrete properties.

   The best way to understand this is to look at the summary of information from the Composite Modulus report for each event and consider things like:

   • Adjusted event times due to temperature and cement class

   • Adjusted creep properties due to the number of exposed faces

   • Incremental loading factors

   • The Composite Modulus Calculation

2. Every shell element can have different Effective Reinforcement:

   • This is determined automatically

   • The Effective Reinforcement results view allows you to confirm the values used.

3. Cracking and Stiffness Calculations for each event:

• This is dependent upon the effective concrete properties, effective reinforcement, and the forces that develop

• So this calculation is unique for each direction of each shell for each event.

• The stiffness of cracked sections is dependent on the degree of cracking - so the procedure is iterative (force -> stiffness -> new force, etc)

• At the converged conclusion of this you can see (and check):

- the extent of cracking via the Extent of Cracking results view

- the stiffnesses determined via the Relative Stiffness results view

Composite creep

Design codes typically provide a way of calculating an effective creep modulus at time, t for a constant load/stress applied at time t_0. Typically this is presented as $\phi(t, t_0)$.

An effective Young's Modulus is then calculated as $E_{c, eff}(t, t_0) = E_{c,28} / [1 + \phi(t, t_0)]$

However, codes typically do not give guidance on how to deal with a loading history where loads vary over the time period being considered.

Technical Report 58 introduces guidance on this topic and proposes a method by which the loading history can be taken into account. (Reference Section 8.4.1, equation 8.37 and also the example on page 36):

$$\left(\frac{\Sigma w}{E_{comp}}\right)n = \frac{w_1}{E_{eff,1}} + \frac{w_2}{E_{eff,2}} + \ldots\ldots + \frac{w_i}{E_{eff,i}} + \ldots\ldots + \frac{w_n}{E_{eff,n}}$$

Where:

n = event under consideration

w_i = incremental load in event i (= load in event i - load in event (i-1)) (note that this will be a negative value when load is removed)

$E_{eff,i} = E_{c, eff}(t_{end,n}, t_i)$ (i.e. covers period from start of event i to end of event n)

The above is logical when you consider a single member subjected to a constant loading arrangement that is increased or decreased at each event. However, when you consider an entire slab with many panels receiving different loading increments in different events it does not seem reasonable to consider all the panels together. Two examples of this are:

1. Why should the addition of cladding loads affect internal panels to the same extent as edge panels?
2. In a transfer slab why should panels that don't support columns be affected to the same extent as those which do?

The aim of TR58 is clear, in that loading on a span/panel is taken as an indication of stress. What this fails to consider is that loading on another span/ panel can also induce stress (although in most situations this will be a secondary effect). It is also clear that you do not actually need "loads", you just need "relative loads" or some other measure of the relative work done in each event. With this in mind a more general approach has been developed where the relative "work done" by each panel is determined by considering the strain energy in each event:

1. Calculate total strain energy 'Q_0' for each Slab Item for a unit loadcase

   • 'Q_0' = sum of 2D Element strain energies

2. For each Load Event 'i'

   • Calculate incremental strain energy 'Q_i' 'Q_i' = sum of incremental 2D Element strain energies

   • Calculate equivalent incremental load factor 'λ_i

   ' $\lambda_i = Q_i / Q_0$

3. E_{comp} can be established from the equation below where "incremental work done" replaces "incremental load" in the TR58 equation.

$$\left(\frac{\Sigma\lambda}{E_{comp}}\right)n = \frac{\lambda_1}{E_{eff,1}} + \frac{\lambda_2}{E_{eff,2}} + \ldots\ldots + \frac{\lambda_i}{E_{eff,i}} + \ldots\ldots + \frac{\lambda_n}{E_{eff,n}}$$

There is an array of intermediate values which lie behind the calculation of the composite modulus, E_{comp} for each slab item, for each event. The composite modulus calculation is provided as an excel spreadsheet report.

You can either generate a report for a chosen slab, selected slabs or all slabs, dependant upon your selection method.

- To obtain slab modulus reports for all slab items, right click anywhere in a scene view and choose Export Eff. Modulus report to Excel > For all slab items
- To obtain slab modulus reports for selected slab items, select the slabs in the structure view regime, right click anywhere in the scene view and choose Export Eff. Modulus report to Excel > For selected slab items
- To obtain slab modulus reports for a chosen slab items, right click a slab panel in the structure view regime and choose Export Eff. Modulus report to Excel > For current slab items

A typical composite modulus report for a slab panel is shown below.

Composite Modulus Calculation

Event	Start [d]	Adjusted Start Temp. and Cement [d]	Incremental load factor, λ	To end of Event 1			To end of Event 2			To end of Event 3			To end of Event 4			To end of Event 5		
				φ	$E_{c,eff}$ [N/mm²]	$\lambda / E_{c,eff}$ [mm²/N]	φ	$E_{c,eff}$ [N/mm²]	$\lambda / E_{c,eff}$ [mm²/N]	φ	$E_{c,eff}$ [N/mm²]	$\lambda / E_{c,eff}$ [mm²/N]	φ	$E_{c,eff}$ [N/mm²]	$\lambda / E_{c,eff}$ [mm²/N]	φ	$E_{c,eff}$ [N/mm²]	$\lambda / E_{c,eff}$ [mm²/N]
1 Strike and backprop slab	10	10	6.983	0.772	19558	0.00036	1.235	15503	0.00045	1.526	13716	0.00051	2.714	9330	0.00075	2.714	9330	0.00075
2 Propping slab 1 above	20	20	4.690	-	-	-	1.019	17164	0.00027	1.307	15020	0.00031	2.381	10249	0.00046	2.381	10249	0.00046
3 Propping removed	61	61	-3.443	-	-	-	-	-	-	0.921	18036	-0.00019	1.926	11843	-0.00029	1.926	11843	-0.00029
4 Sensitive finishes added	122	121	2.734	-	-	-	-	-	-	-	-	-	1.686	12902	0.00021	1.686	12902	0.00021
5 Final load event	25550	25502	3.499	-	-	-	-	-	-	-	-	-	-	-	-	0.000	34650	0.00010
Total of $\lambda / E_{c,eff}$ [mm²/N]						0.00036			0.00072			0.00063			0.00113			0.00123
E_c to end of Event [N/mm²]						19558			16130			13054			9727			11776

It should be noted that Tekla Structural Designer takes into account the cement class when determining the temperature adjusted age of loading so minor variations will occur.

The effective modulus is used in determining the properties for each load event.

Extent of cracking

The effective modulus, and the effective reinforcement are used to determine the cracked or uncracked state of each shell for each event, for each direction.

Eurocode 2 provides an expression that predicts the behavior between the cracked and uncracked states. This expression uses a Distribution factor, ζ that apportions the behavior between a fully-cracked state (±1.0) and an uncracked state (0) for interpolating the stiffness when a state of partial cracking exists.

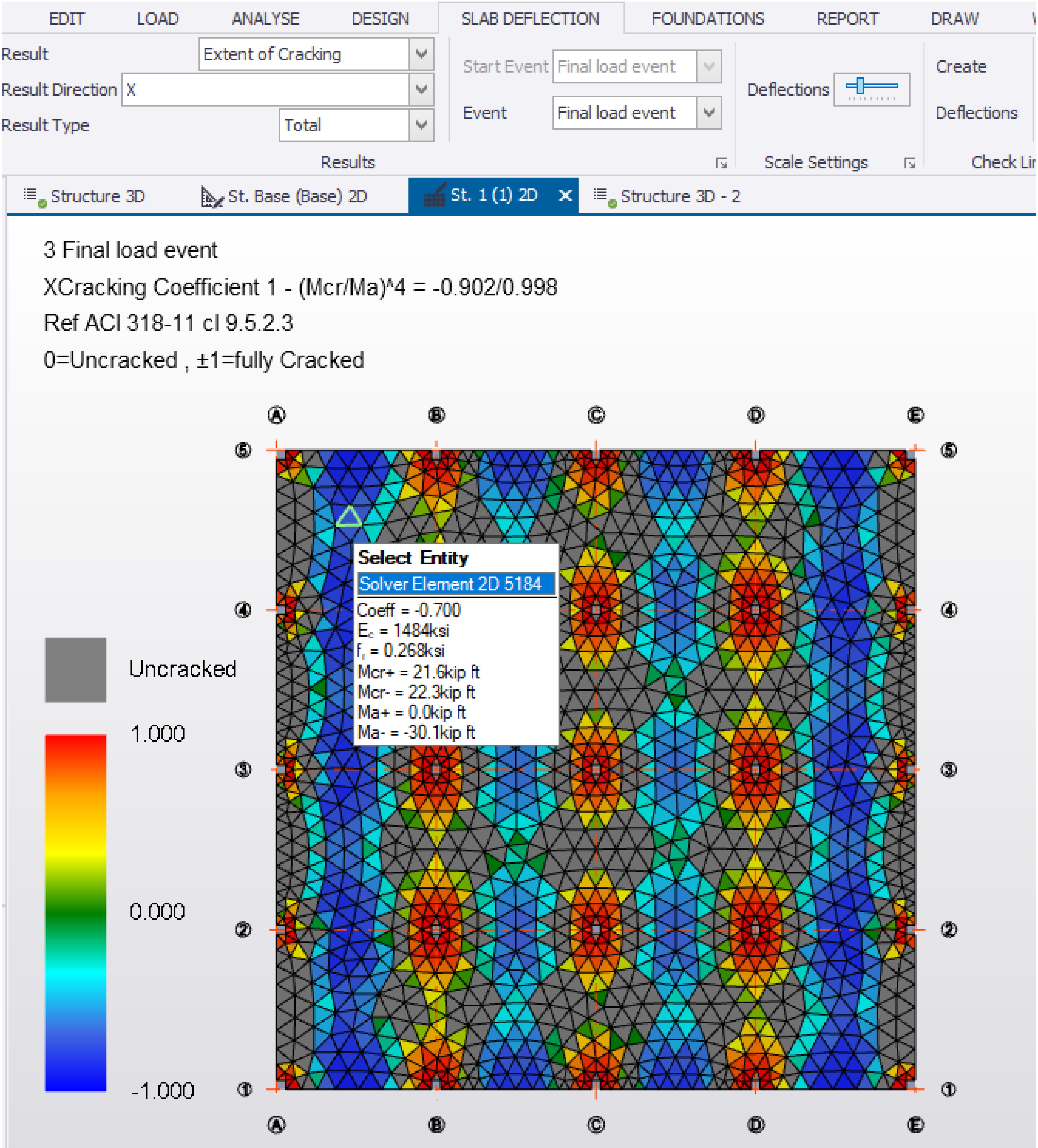
$\zeta = 1 - ß (M_{cr} / M)^2$

Where:

- ß is a user defined value specified in the Event Sequences and is either: 1.0 for single short-term loading 0.5 for sustained loads or many cycles of repeated loading
- M_{cr} is the hogging (positive) or sagging (negative) cracking moment.
- $M_{a(+|-)}$ is the relevant Wood-Armer moment in the direction for which the display is shown (X or Y). This is calculated from M_x, M_y & M_{xy} in the usual way, when determining the extent of cracking for a shell element for each iteration for each Event.

If you view Extent of Cracking results for a chosen result direction and cycle through the events you will see each of the FE elements shaded to indicate the extent of cracking.

At the first sign of cracking, $M_{cr} > M_a$. If you hover over an FE element the tooltip provides some intermediate calculation results to verify this distribution factor.

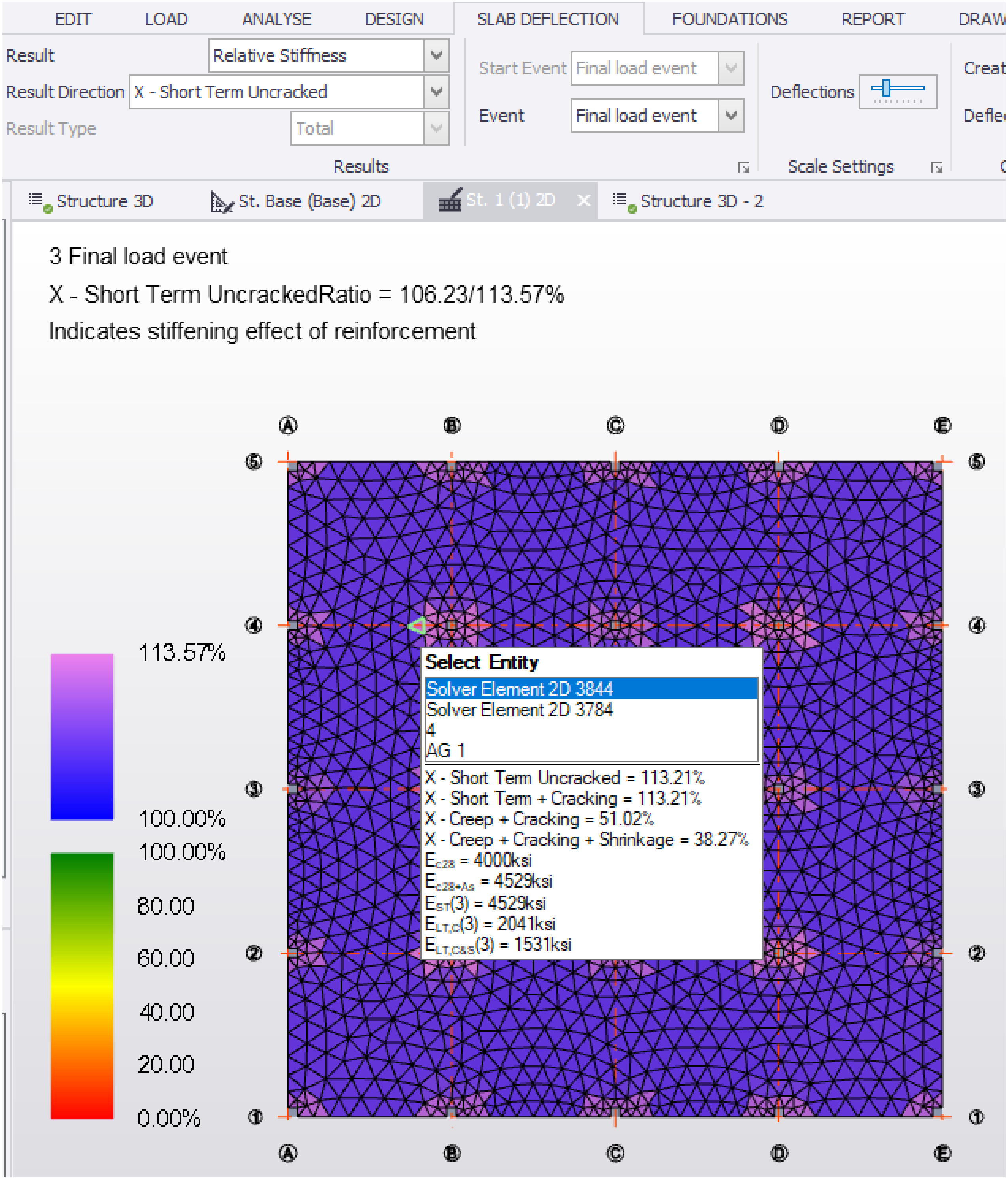


The tooltip shows the applied moment for the Event in question, rather than the worst moment of all Events up to and including the one being looked at -

this allows engineers to deduce that a greater level of cracking was caused by an earlier Event.

Relative stiffness

The tooltip provides detailed information on the relative stiffness calculations.



The tooltip lists:

- E_{c28} : E_{db}*1.05*stiffness adjustment factor. (Where E_{db} = the short term modulus from the concrete materials database). The stiffness adjustment factor used is determined from the Slab Deflection ribbon > Settings >, Modification Factors page. Provided for information - not directly used in any analysis.

- Ec_{28+As} : the short term modulus including for reinforcement. Provided for information - not directly used in any analysis.
- $E_{ST}(i)$: the short term modulus used in the instantaneous analysis (i.e. includes area of reinforcement and cracking if cracking has occurred) for the selected event.
- $E_{LT,C}(i)$: the modulus used in the final iteration of long term deflection estimation (i.e. includes area of reinforcement and cracking if cracking has occurred, and effective creep) for the selected event.
- $E_{LT,C\&S}(i)$: $E_{LT,C}(i)$ with further adjustment to allow for effect of shrinkage (= $E_{LT,C}$ / multiplier) for the selected event. The shrinkage multiplier to determine the shrinkage contribution is determined for the chosen event based upon the ratio of the maximum panel Z deflection (including shrinkage) / total Z deflection (excluding shrinkage). This provides an indication of the overall effective stiffness adjustment. Provided for information - not directly used in any analysis.

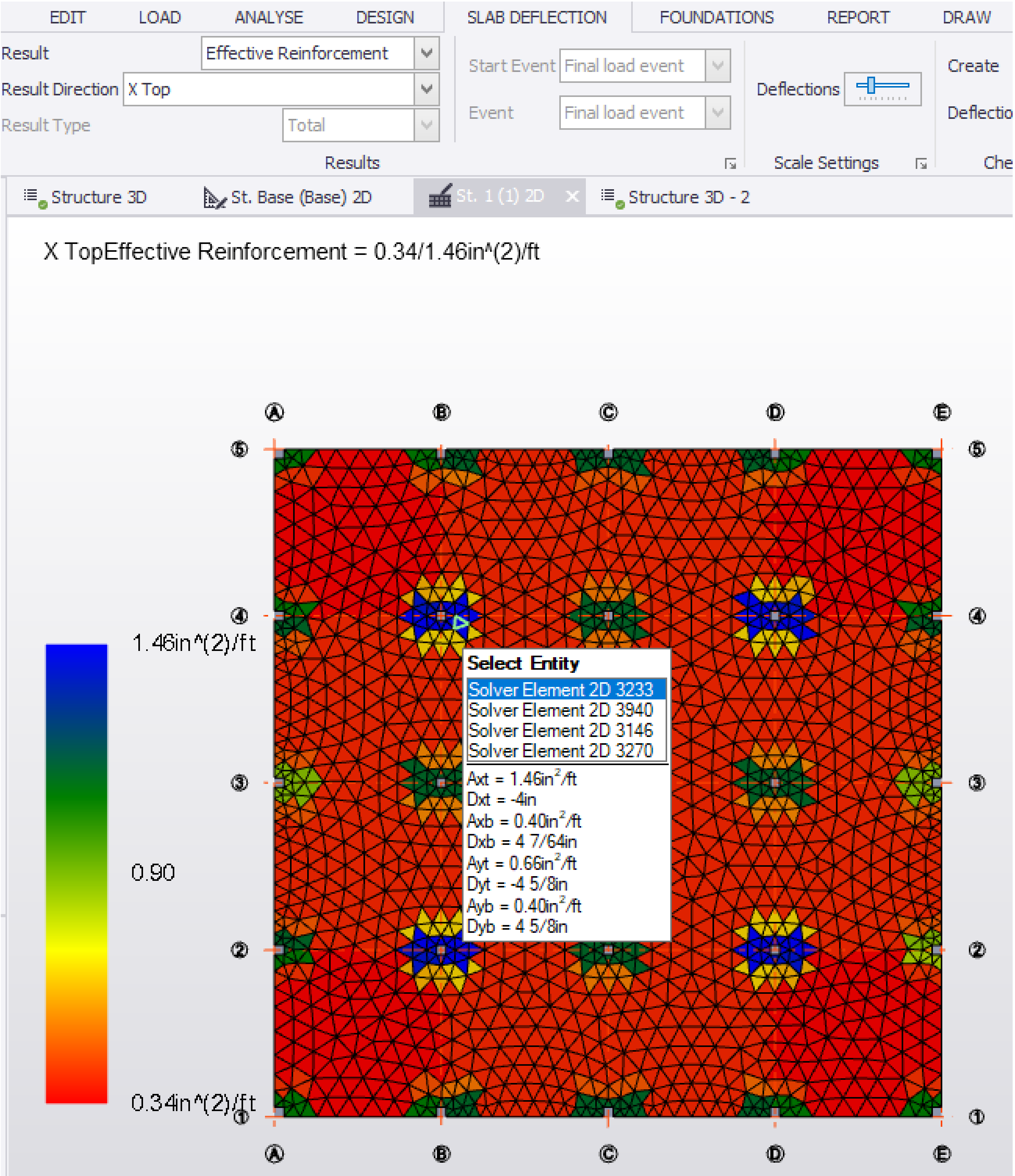
Based on the modulus, E defined above, a number of ratios are provided in the tooltip for the chosen result direction.

- Short Term Uncracked = E_{c28+As} / E_{c28}
- Short Term + Cracking = E_{ST} / E_{c28}
- Creep + Cracking = $E_{LT,C} / E_{c28}$
- Creep + Cracking + Shrinkage = $E_{LT,C\&S} / E_{c28}$

Effective reinforcement

Effective reinforcement for each shell element is also required for the determination of the shell's effective properties at the end of each load event.

The effective reinforcement used in the property calculations are reported to you in the 4 layers for every shell element. The information is provided as a color coded shell display from minimum to maximum reinforcement area.



If you hover over a shell the tooltip provides the following information:

- A_{xt} : X Top Effective Reinforcement.

- D_{xt} : Distance from the section centroid to the center of X top reinforcement.
- A_{xb} : X Bottom Effective Reinforcement.
- D_{xb} : Distance from the section centroid to the center of X bottom reinforcement.
- A_{yt} : Y Top Effective Reinforcement.
- D_{yt} : Distance from the section centroid to the center of Y top reinforcement.
- A_{yb} : Y Bottom Effective Reinforcement.
- D_{yb} : Distance from the section centroid to the center of Y bottom reinforcement.

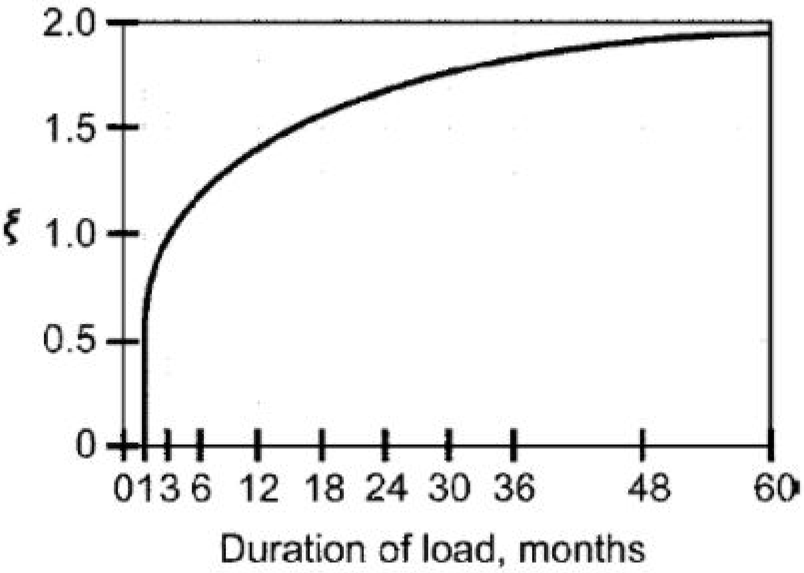
Shrinkage allowance

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

- Eurocode 2 provides a method to estimate shrinkage strains and curvatures based on exposed surface area, member size, relative humidity and reinforcement quantity and position.
- Asymmetry of reinforcement leads to curvature which leads to deflection. It is estimated this effect can contribute up to 30% to the long-term deflection.
- Technical Report 58 provides a theoretical method of estimating the additional shrinkage deflection effect in the analysis

At this time the TR58 method has not been implemented within Tekla Structural Designer. Shrinkage is taken into consideration using a multiplier, by making an overall adjustment to the total deflection (excluding shrinkage) in line with simpler adjustment proposals of the ACI code. This approach is in line with many other software products.

The simplest ACI approach makes an allowance for all long term effects (creep and shrinkage) by using an adjustment factor. This is based on the graph below and also provides some specific values.



Time	Multiplier for long-term deflections
5 years or more	2.0
12 months	1.4
6 months	1.2
3 months	1.0

Note that we said shrinkage effects and not creep and shrinkage. Creep is dealt with rigorously in Tekla Structural Designer so we need to ascertain the proportional effect of shrinkage only. ACI 435 provides some indication of the separate contribution of creep and shrinkage.

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ 4 $\sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

Comparing the different sources the ratio of shrinkage is as follows:

- Sbarounis 1.2 / 5 = 24%
- Branson 1 / 4 = 25%
- Graham and Scanlon 1 / 3.5 = 28%

  (ignore higher modulus of rupture because reduced values are considered automatically in the cracked section analysis).

The above provides a shrinkage ratio of between 24% and 28%. Hence we recommend a value of between 20%-30% is used. A 25% default is provided via the Slab deflection ribbon > Options dialog and the Creep and Shrinkage page.

The total deflection due to shrinkage effect is determined based on an identified "Total Shrinkage Event" towards the end of the event sequence. The event sequence with the latest load start time is used for calculating the shrinkage adjustment. If multiple events exist with the latest load start time then the first one is considered.

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

Using this "total shrinkage effect" we can then assign a proportion of the total shrinkage to each event.

With reference to earlier versions of ACI 435 (1966) on which the values in the graph above are based, additional values for 1 month and 3 years can be obtained. This follows the case where As' = 0 (because compression steel is allowed for differently in the code). By comparison with the graph above, we can see closely matched values.

Duration of loading	**Factor F**		
	As' = 0	**As' = 0.5As**	**As' = As**
1 month	0.58	0.42	0.27
3 months	0.95	0.77	0.55
6 months	1.17	0.95	0.69
1 year	1.42	1.08	0.78
3 years	1.78	1.18	0.81
5 years	1.95	1.21	0.82

The values we have adopted for considering shrinkage effects are as tabulated below. The final column provides the proportion of the total shrinkage at a given time.

Time	Long Term Effects Factor	Proportion of Total Shrinkage
0	0	0.00
1 month	0.6	0.30
3 months	1	0.50
6 months	1.2	0.60
1 year	1.4	0.70
3 years	1.8	0.90
5 years and above	2	1.00

From the above, for any event, the end of event time is used to calculate a "Proportion of total shrinkage" using linear interpolation between the values discussed in the table above.

Deflection calculations on the Z deformation are then adjusted to account for shrinkage effects.

As an example, let's assume the following event sequence.

- Event 1 = 7 days, Event 2 = 10 days, Event 3 = 17 days, Event 4 = 20 days, Event 5 = 27 days, Event 6 = 2 months, Event 7 = 6 months, Event 8 = 1 year and Event 9 = 70 years

Assuming a shrinkage factor of 25% (user defined input value), a basic multiplier can be determined = 1/(1-25%) = 1.333

Event 9 Final event at 70 years analysis deflection (excluding shrinkage) = 32.4mm

Therefore, the Total deflection (including shrinkage) = 32.4 x 1.333 = 43.2 mm

Total Deflection from shrinkage alone is 43.2 - 32.4 = 10.8 mm

We can now apportion this deflection due to shrinkage, to each event based upon the event time and a proportion value.

i.e.

At 0 days proportion of total shrinkage is 0, At 1 month proportion is 0.3. Therefore using linear interpolation between these values;

- Event 1 (7 days) Shrinkage multiplier = 0.3 * 7/30 = 0.07
- Event 2 (10 days) Shrinkage multiplier = 0.3 * 10/30 = 0.1
- Event 3 (17 days) Shrinkage multiplier = 0.3 * 17/30 = 0.17
- Event 4 (20 days) Shrinkage multiplier = 0.3 * 20/30 = 0.2
- Event 5 (27 days) Shrinkage multiplier = 0.3 * 27/30 = 0.27

At 1 month proportion of total shrinkage is 0.3, At 3 month proportion is 0.5. Therefore using linear interpolation

- Event 6 (2 months) Shrinkage multiplier = 0.4

At 6 month proportion of total shrinkage is 0.6

- Event 7 (6 months) Shrinkage Multiplier = 0.6

At 1 year proportion of total shrinkage is 0.7

- Event 8 (1 year) Shrinkage Multiplier = 0.7

At 70 years proportion of total shrinkage is 1.0

- Event 9 (70 years) Shrinkage Multiplier = 1.0

The Shrinkage deflection that occurs at each event is then the total shrinkage 10.8 mm x the shrinkage multiplier calculated above.

- Event 1 (7 days) Shrinkage = 0.07 x 10.8 = 0.76 mm
- Event 2 (10 days) Shrinkage = 0.1 x 10.8 = 1.08 mm
- Event 3 (17 days) Shrinkage = 0.17 x 10.8 = 1.84 mm
- Event 4 (20 days) Shrinkage = 0.2 x 10.8 = 2.16 mm
- Event 5 (27 days) Shrinkage = 0.27 x 10.8 = 2.92 mm
- Event 6 (2 months) Shrinkage = 0.4 x 10.8 = 4.32 mm
- Event 7 (6 months) Shrinkage = 0.6 x 10.8 = 6.48 mm
- Event 8 (1 year) Shrinkage = 0.7 x 10.8 = 7.56 mm
- Event 9 (70 years) Shrinkage = 1.0 x 10.8 = 10.8 mm

For each event, the total deflection (including shrinkage) reported in the Slab deflection view regime and the tooltips is the event analysis deflection (excluding shrinkage) + the proportion calculated above using the shrinkage multiplier.

Check lines

Check lines - a unique feature inTekla Structural Designer - provide a practical way to automate deflection checking and reduce the possibility of errors.

Each check line defines a line along which deflection checks are required. Multiple check lines can be created and each line can have several different deflection checks performed (against either a total, instantaneous, or differential deflection limit).

Whilst the check lines have to initially be positioned using engineering judgment, once they are in place they provide an instantaneous means to evaluate revised deflections following changes to the model parameters and re-analysis.

Assessing slab deflections without check lines would be quite an arduous task, since deflection limits are of the form (span / fixed value) and the spans can vary on an irregular slab, hence the permissible limit would also vary. Furthermore, you may wish to check total or differential deflections at or between different load events. Would the critical location be obvious?

Setting up the checks in advance (via the slab deflection check catalogue)

The deflection checks to be performed for the check lines are set up in the Slab Deflection Check Catalogue before the check lines are positioned.

Each check in the catalogue is specified as a Total, Differential or Instantaneous deflection limit that is checked for a specified event, (or in the case of a Differential check between the start of an event and the end of an event).

Related task

Application of check lines

The application of check lines is an iterative activity. They should be created at all locations where you deem deflection checks to be required.

Check lines can only be created in a 2D view. If the Create command is greyed out ensure you switch to a 2D view.

When you click Create, the Properties Window automatically includes those slab deflection checks from the Slab Deflection Check Catalogue where "Use in new Check Lines" was checked. Note that a maximum of six checks can be assigned to a single check line; if you require more than six checks, then multiple check lines can be applied at the same location.

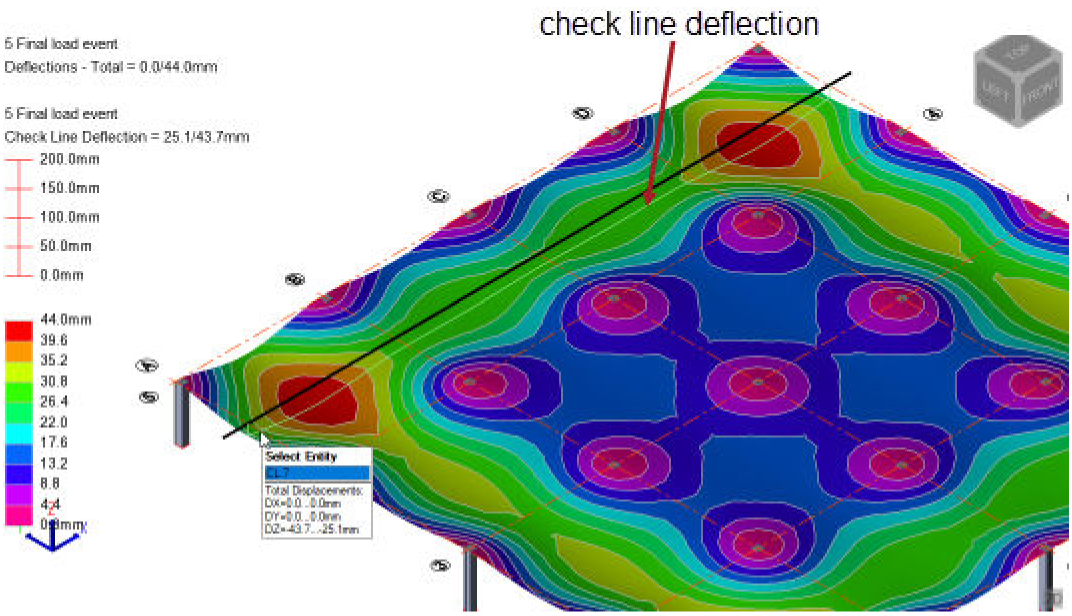
You can add further checks from the catalogue or add new checks directly from the Properties Window.

You can also edit the "Use in New Check Lines" option in the catalogue prior to running the Create command to change the default checks automatically assigned to each new check line.

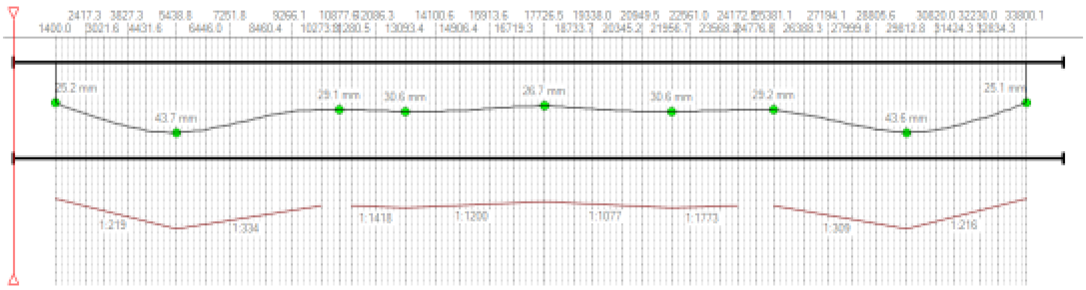
Related task

Displaying check line results

The Check Line deflection can be overlayed on the slab by using the Deflections command in the ribbon, it is perhaps easier to visualize if you switch the view to a 3D view of the slab using the 2D/3D toggle button in the bottom right of the window.



You can also right click on a check line and open the deflections check view. This displays a cut line through the slab, showing the deflected shape with the maximum and minimum deflection values. Beneath this it also draws the average slopes between maximum and minimum points and reports the average slope ratios.



The above deflection check view is controlled from the Loading Analysis ribbon which has droplists to enable you choose the Result Type (Total, Instantaneous, or Differential) and the Event(s).

The average slope ratios (for the appropriate result type/event) are checked against the deflection limits that have requested for the check line.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Related task

Check line reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

You can also generate a Slab deflection check line report for the entire structure, selected level, planes or sub structures via the Model Report command on the Report ribbon.

Related task View slab deflection reports

Slab deflection status and utilization

The status and the utilization can be graphically displayed for both Check lines and slabs.

Check lines Status and Utilization

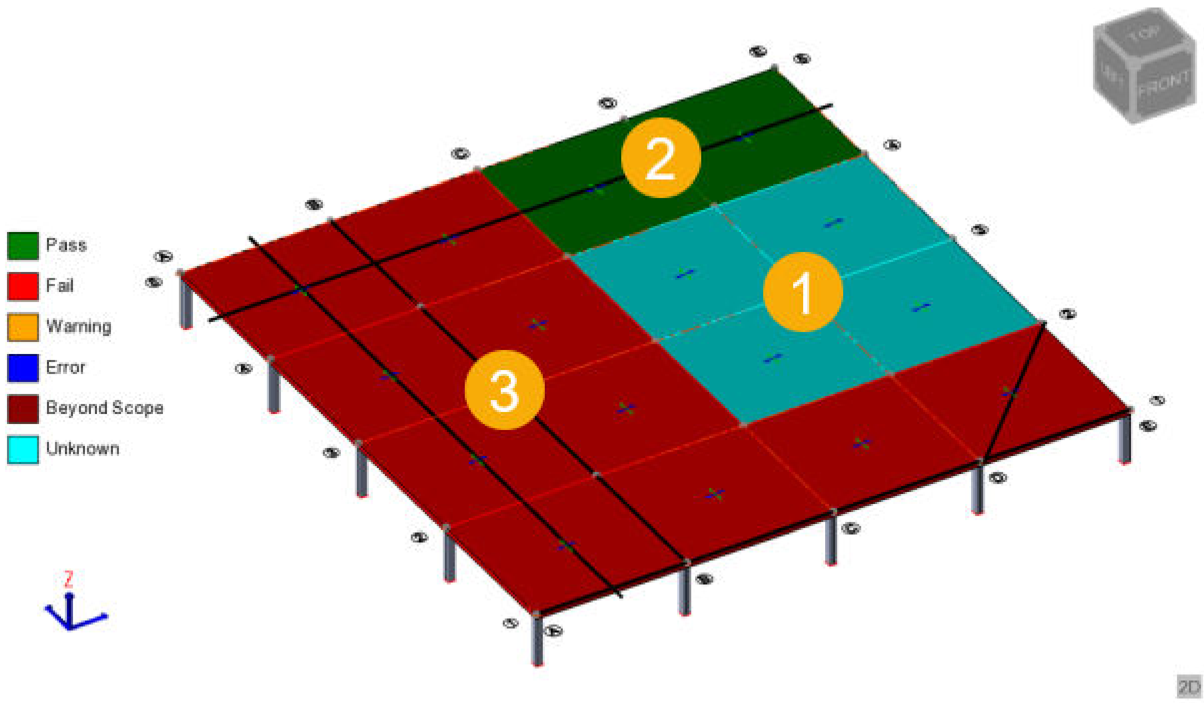
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical utilization ratio

Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization.

Related task:

Slab Deflection Status and Utilization

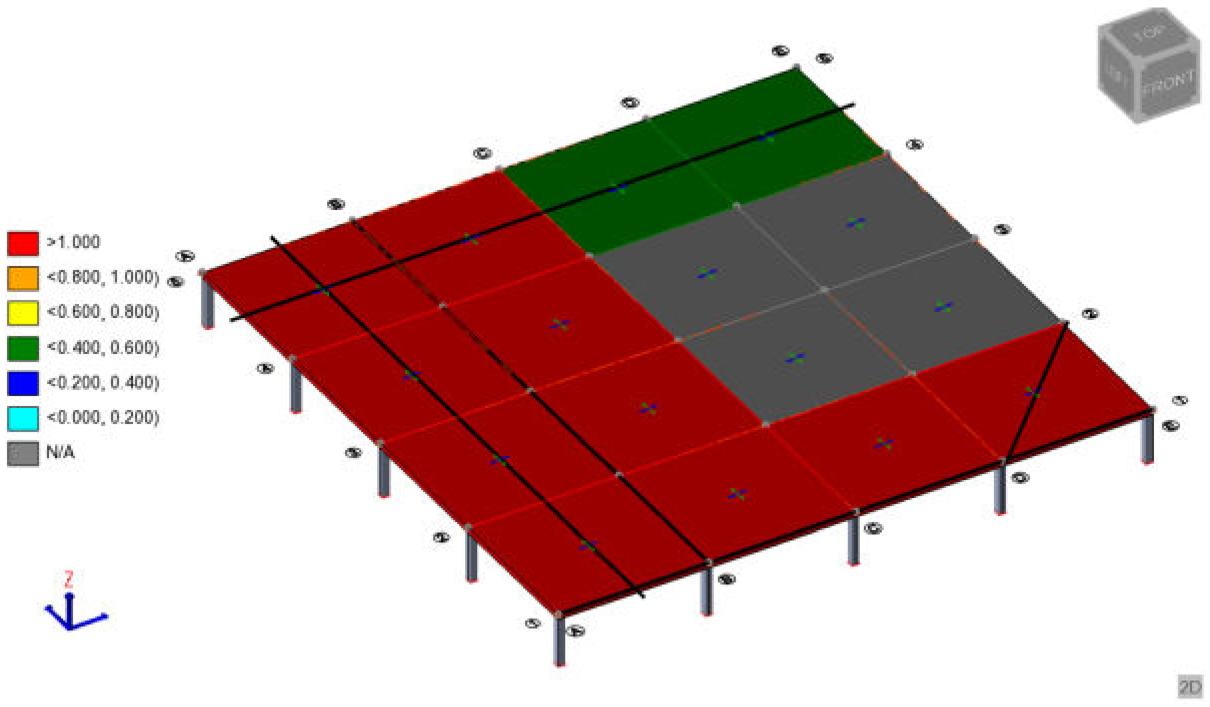
Slab deflection status is determined for each slab item as the worst status from all associated check lines detected within the slab item or touching the slab item boundary.



In the above status view:

1. No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
2. One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
3. A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Slab deflection utilization is similarly determined for each slab item as the worst utilization from all associated check lines.

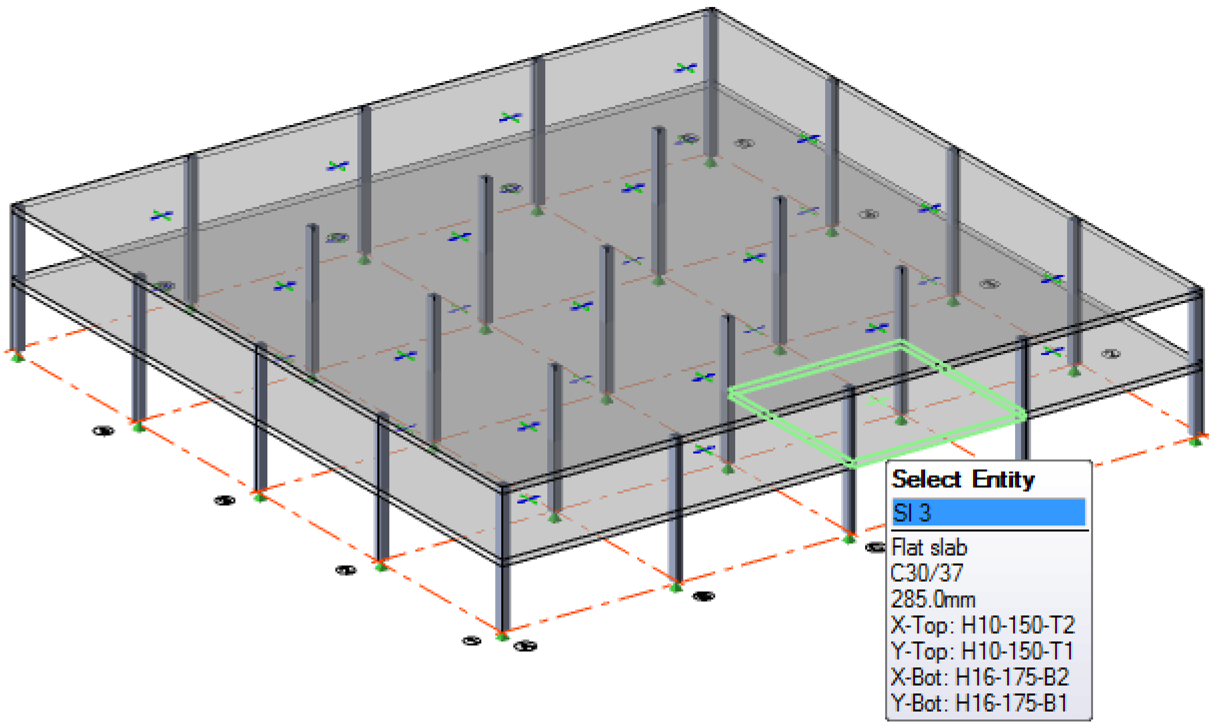


Related task:

Slab deflection example (Eurocode)

In the following exercise slab deflections will be checked for the tutorial model shown below.

It is a simple multi-bay flat slab structure on an 8m square grid of columns. The slabs have been sized based on deemed-to-satisfy rules taken from "Economic Concrete Frame Elements to Eurocode 2 - The Concrete Society".



Geometry:

- 8m grid
- 285 thick C30/37 slab
- 400 square columns

Loading:

- Finishes - 1.5kN/m^2
- Imposed - 5.0kN/m^2
- Perimeter Cladding - 10.0kN/m

Deemed to satisfy checks (page 402) provide one method of checking, however the main focus of the exercise will be to investigate rigorous slab deflection checks (page 405).

Deemed to satisfy slab deflection checks example (Eurocode)

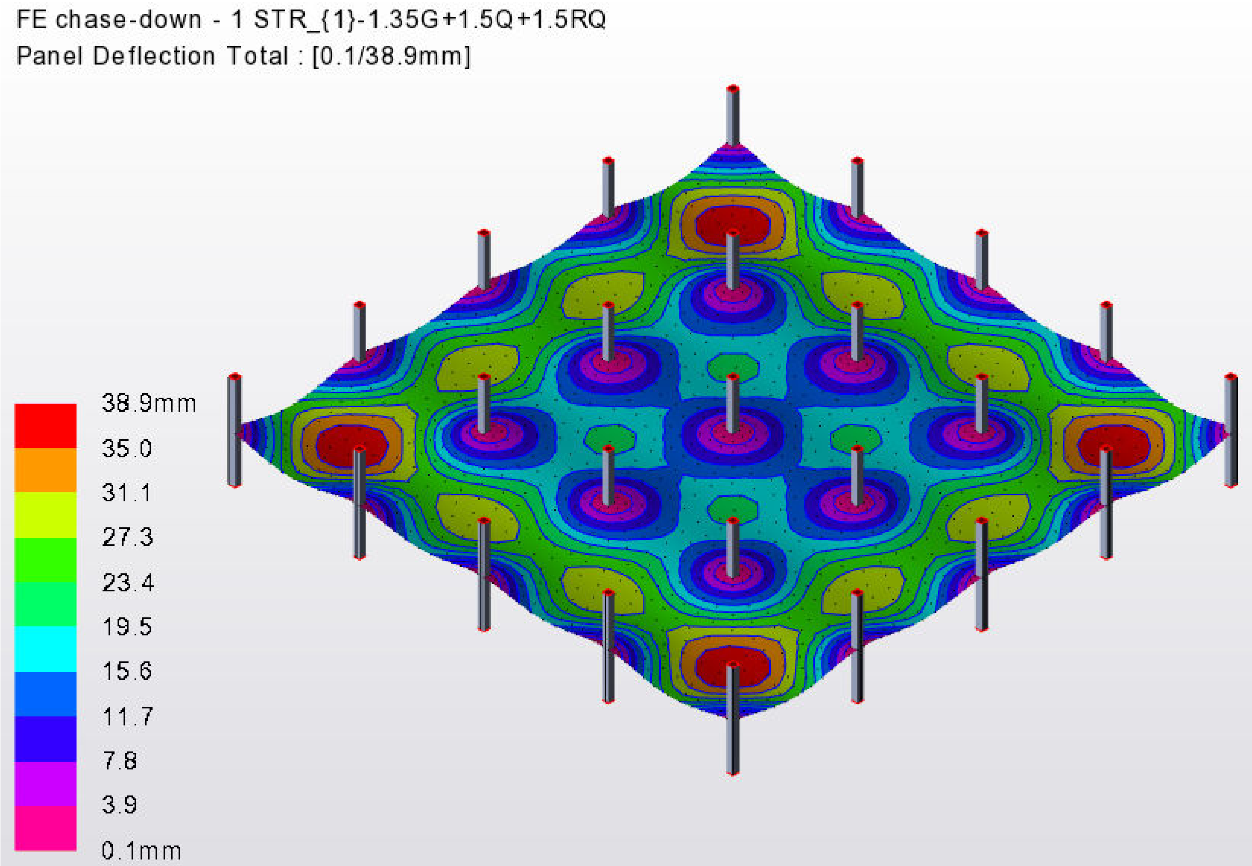
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results



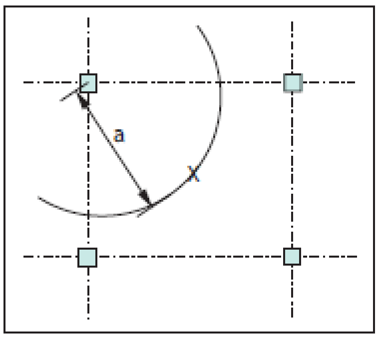
Identify critical check locations

We can see that the maximum reported deflection is 38.9mm, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs some engineering judgment might be required when assessing which deflections and span lengths require checking.

Guidance exists in How to Design Concrete Structures using Eurocode 2, The Concrete Centre, Figure 9

Figure 9
Recommended acceptance criteria for flat slabs



Notes

If maximum permitted $\delta = L/n$ and X is the position of maximum δ

where

L = Span

n = Limiting span-to-depth ratio, e.g. 250

then the deflection at X should not be greater than $2a/n$.

(Maximum deflection on gridlines may be more critical.)

In our example, taking the diagonal dimension across the columns, the deemed-to-satisfy span / 250 rule provides a deflection of $[\sqrt{(8000^2+8000^2)}]$ / 250 = 45.3mm.

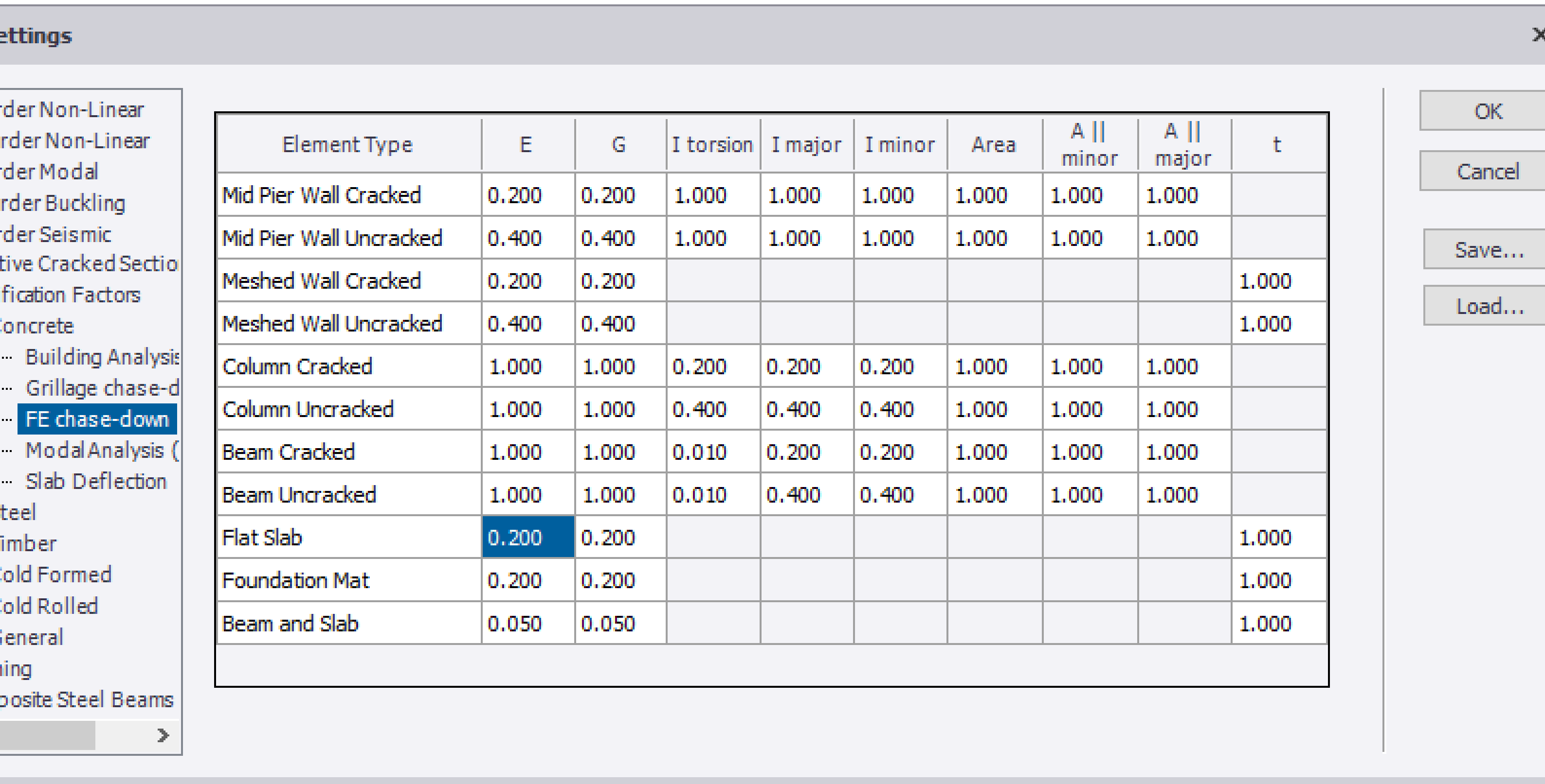
This compares favorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 250.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.2.



Element Type	E	G	I torsion	I major	I minor	Area	A || minor	A || major	t
Mid Pier Wall Cracked	0.200	0.200	1.000	1.000	1.000	1.000	1.000	1.000	
Mid Pier Wall Uncracked	0.400	0.400	1.000	1.000	1.000	1.000	1.000	1.000	
Meshed Wall Cracked	0.200	0.200							1.000
Meshed Wall Uncracked	0.400	0.400							1.000
Column Cracked	1.000	1.000	0.200	0.200	0.200	1.000	1.000	1.000	
Column Uncracked	1.000	1.000	0.400	0.400	0.400	1.000	1.000	1.000	
Beam Cracked	1.000	1.000	0.010	0.200	0.200	1.000	1.000	1.000	
Beam Uncracked	1.000	1.000	0.010	0.400	0.400	1.000	1.000	1.000	
Flat Slab	0.200	0.200							1.000
Foundation Mat	0.200	0.200							1.000
Beam and Slab	0.050	0.050							1.000

Rigorous slab deflection analysis example (Eurocode)

There are many input parameters that will have an impact on the rigorous deflection estimates and would therefore need to be considered. For details see: Factors that affect rigorous slab deflection estimates (page 360)

For this exercise, initially it will be assumed that the default settings have already been reviewed and set as required.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**

2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Model Event Sequence

To review the model event sequence:

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

   This has already been setup for this example as shown below:

Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	C...	20.0	50.00	2	0.500	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	C...	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	C...	20.0	50.00	2	0.000	1 Self weight - excluding slab	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Update custom event sequences

Deflections to the end of each of the above event periods will be calculated by the analysis.

3. Click **OK** to close the dialog.

Perform Iterative Slab Deflection Analysis

To establish some initial results (with all parameters left as default values) you could just click **Analyse All** from the Slab Deflection toolbar, however, in a real model rather than analysing all levels at once, it can be more efficient to work on just the current level, or a selected level. Obviously considering just a single level reduces the time necessary to undertake the iterative slab deflection analysis.

1. Open a 2D plan view of the level **Typical floor**
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.

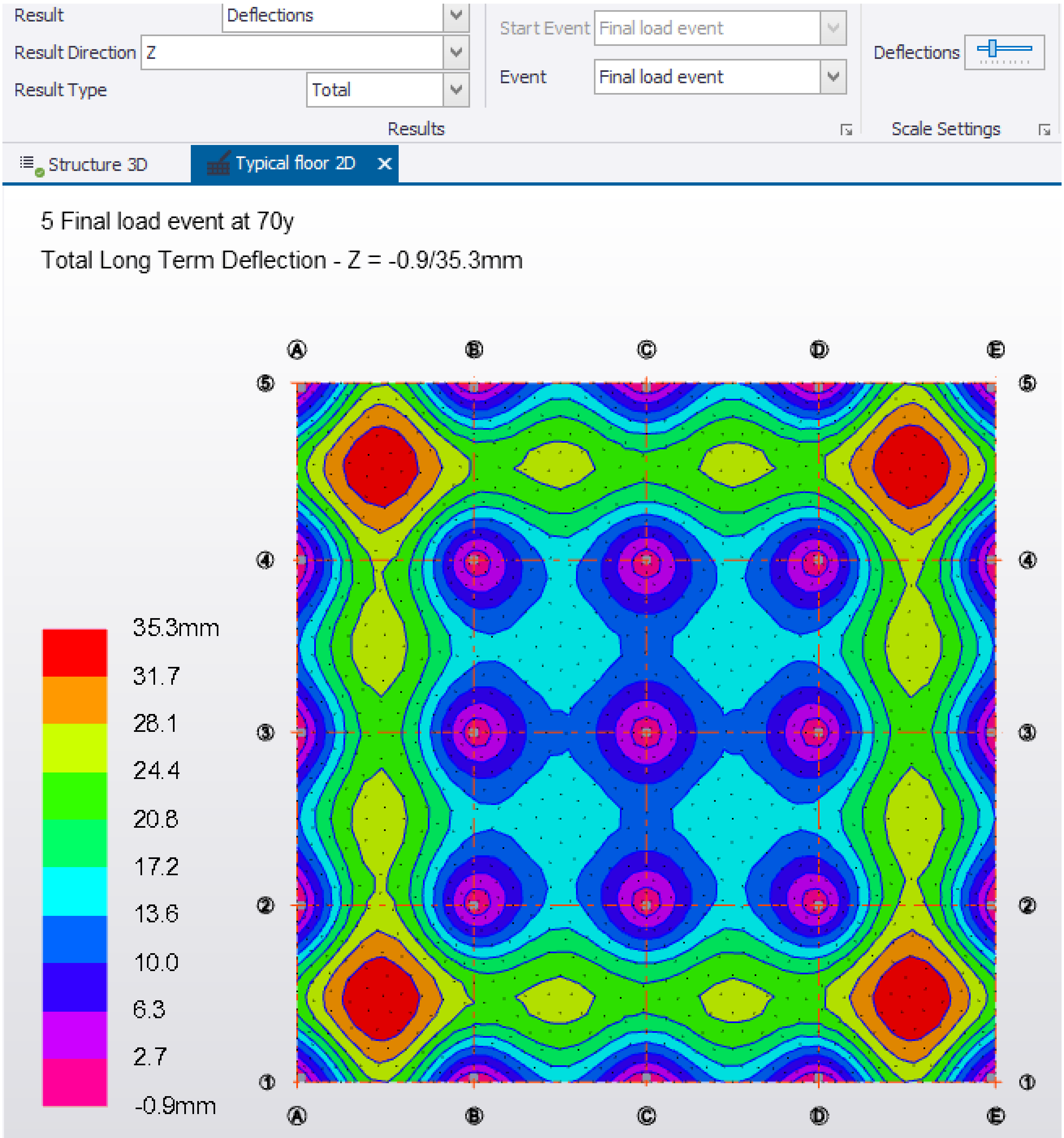
Review Deflections for Events

Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon.

You are able to review:

- Total deflection at the end of any event.
- Differential deflection between any two events (Start of Event and End of Event).
- Instantaneous deflection (not actually needed for TR 58).

The image below shows total deflection contours for the final load event.



NOTE To see the image as shown, you may need to adjust some of the selections in Scene Content, for example **Slab Patches** should be de-selected.

As a comparison with the simple approach earlier (39.7mm), the Total deflection at the final load event for the chosen location is 35.2mm.

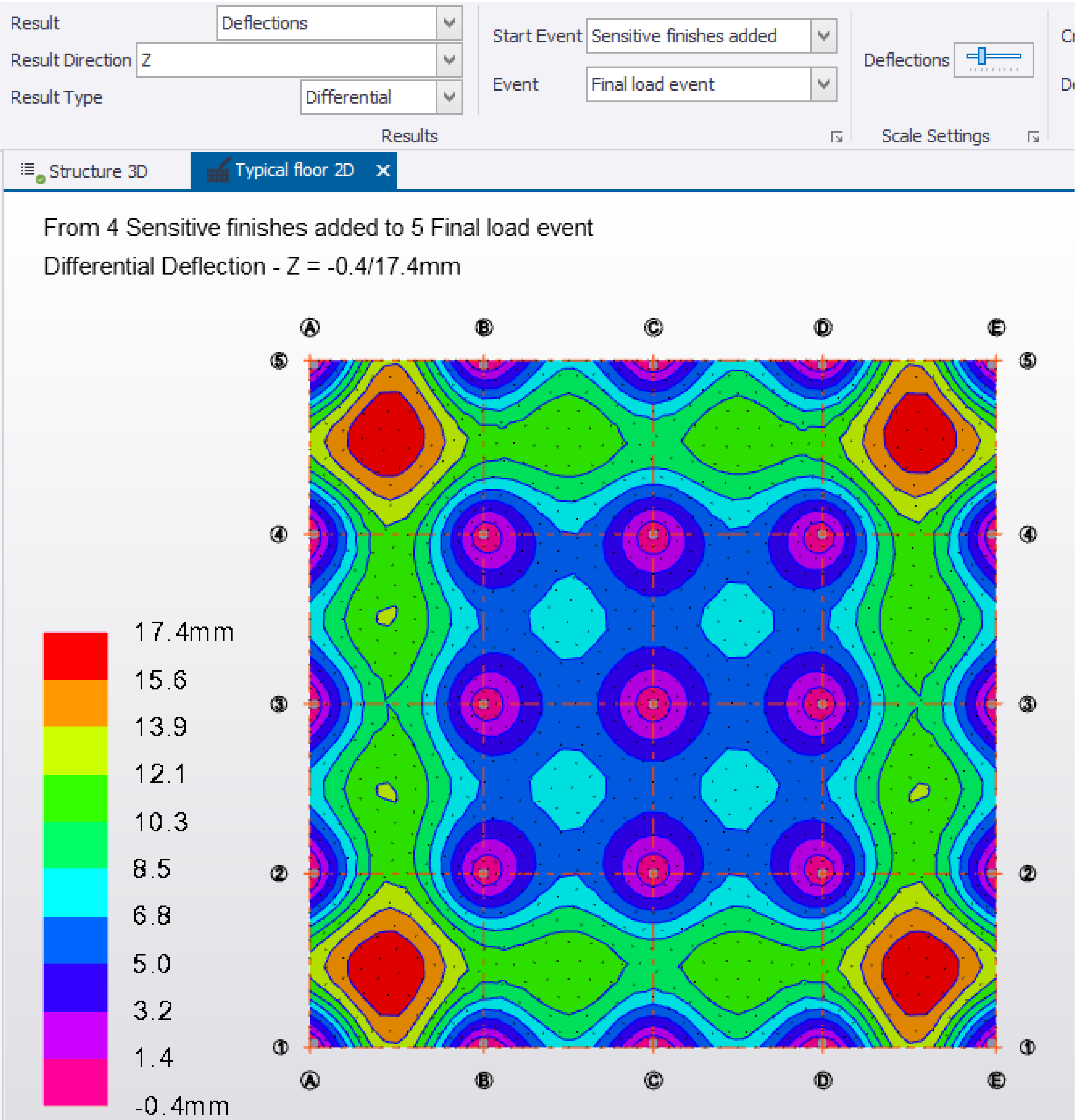
NOTE In the above contour plot the deflections are not exactly symmetrical - this is because reinforcement is in the outer layer in the Y direction making the slab stiffer in that direction.

Total deflections to the end of each of the event periods in the Event Sequence are available and could also be displayed as required.

In the Event Sequence there is an event for “Sensitive Finishes added” - we shall now show differential deflection between this and the final event.

1. From the **Slab Deflection** toolbar, change the Result Type to **Differential**
2. Select the Start Event as **Sensitive Finishes added**
3. Select the Event as **Final load event**

   The 2D view now displays the differential deflections between “Sensitive Finishes added” and the final event.


Result
Deflections
Result Direction
Z
Result Type
Differential
Results
Start Event
Sensitive finishes added
Event
Final load event
Deflections
Scale Settings
Structure 3D
Typical floor 2D
From 4 Sensitive finishes added to 5 Final load event
Differential Deflection - Z = -0.4/17.4mm
17.4mm
15.6
13.9
12.1
10.3
8.5
6.8
5.0
3.2
1.4
-0.4mm

Review Other Results

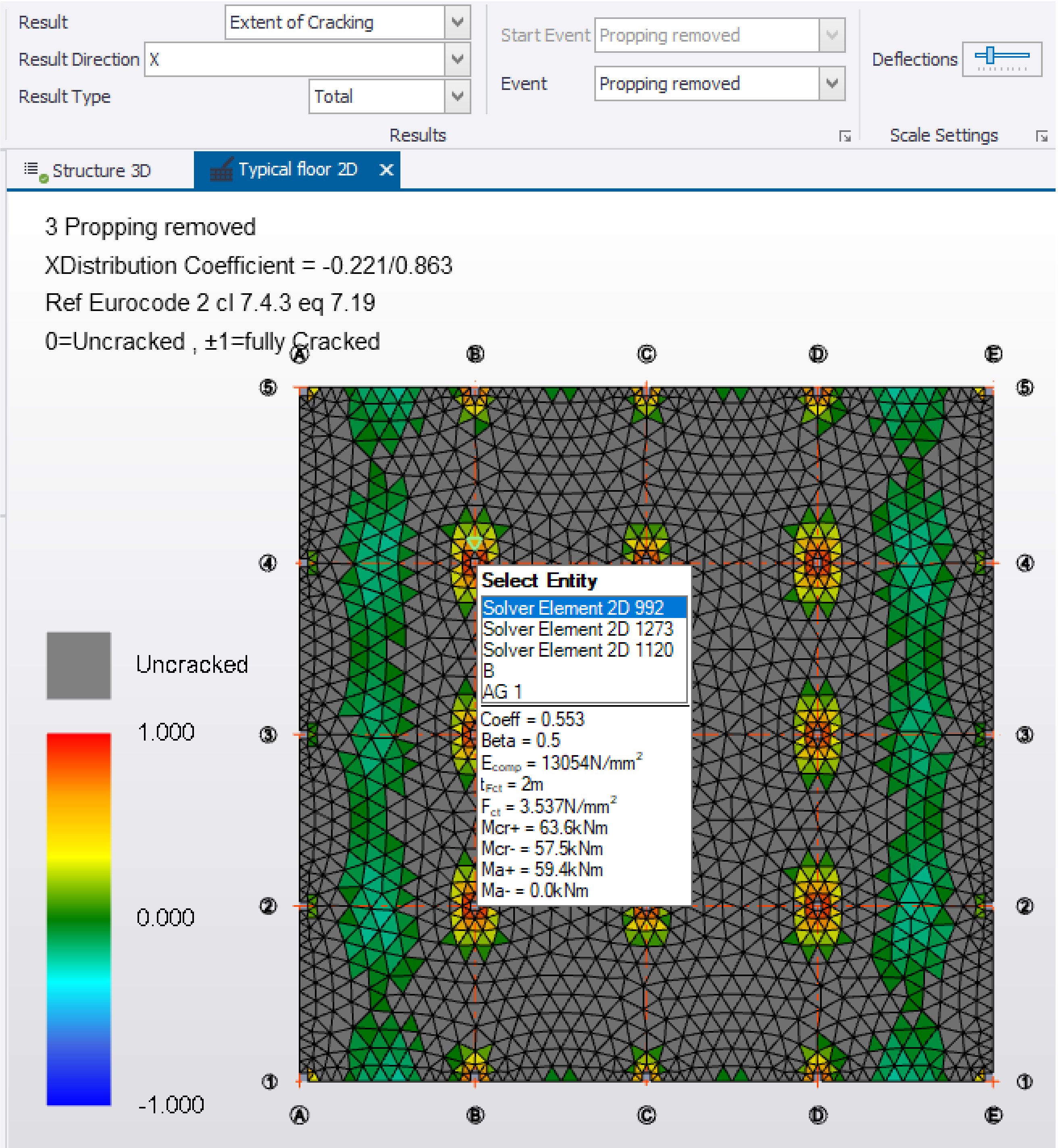
In addition to the deflections you can display the extent of cracking at any load event.

You can also review the relative stiffness in a particular result direction for any specified event.

You can also review the area of effective reinforcement for a chosen result direction for each FE element.

1. Select the Result as **Extent of Cracking**, and then:
   a. Select the Result Direction as **X**,
   b. Select the Result Type as **Total**,
   c. Select the Event as **Propping removed**,

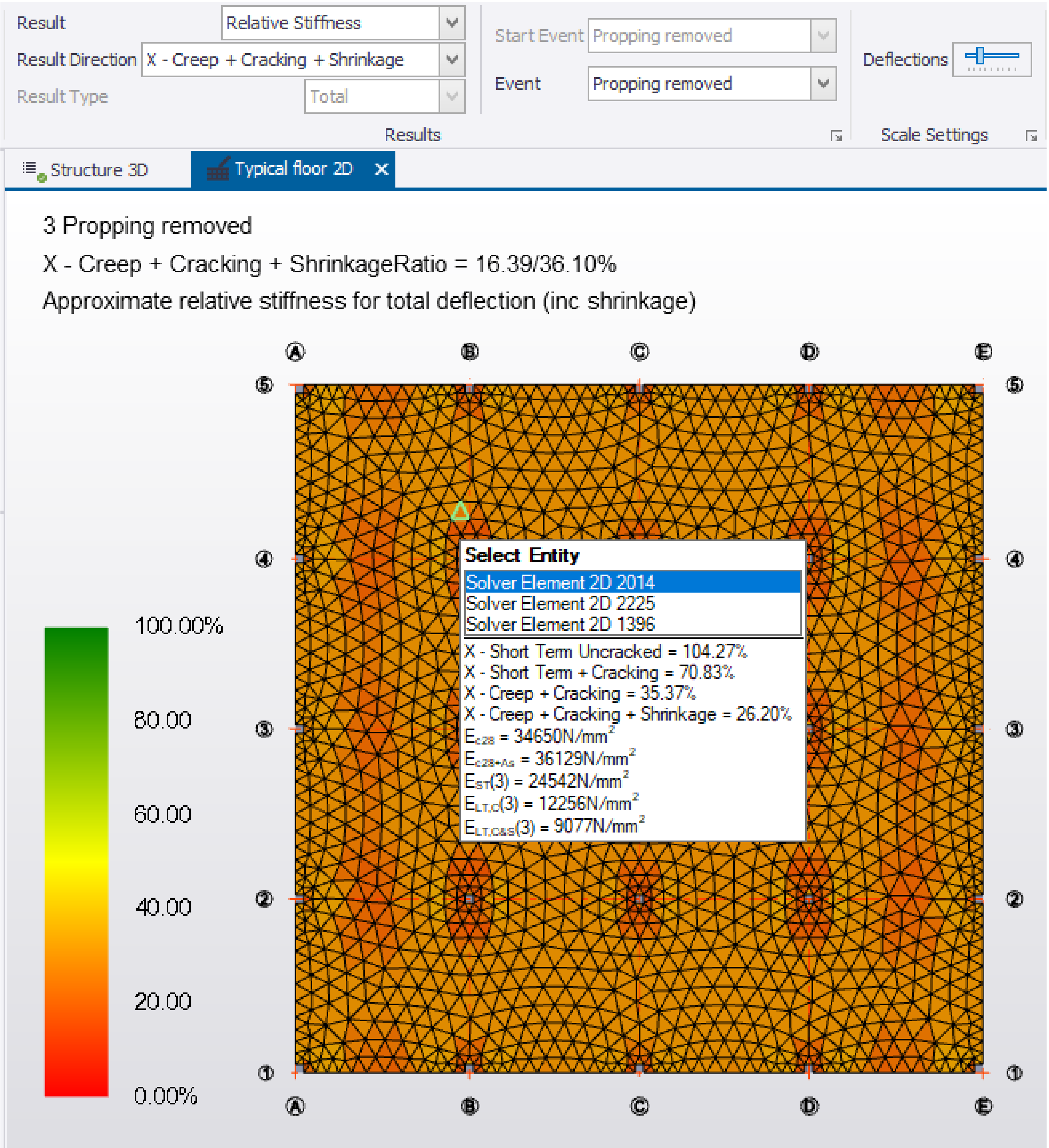
   The extent of cracking for the **Propping removed** load event is displayed.



2. Select the Result as **Relative Stiffness**, and then:

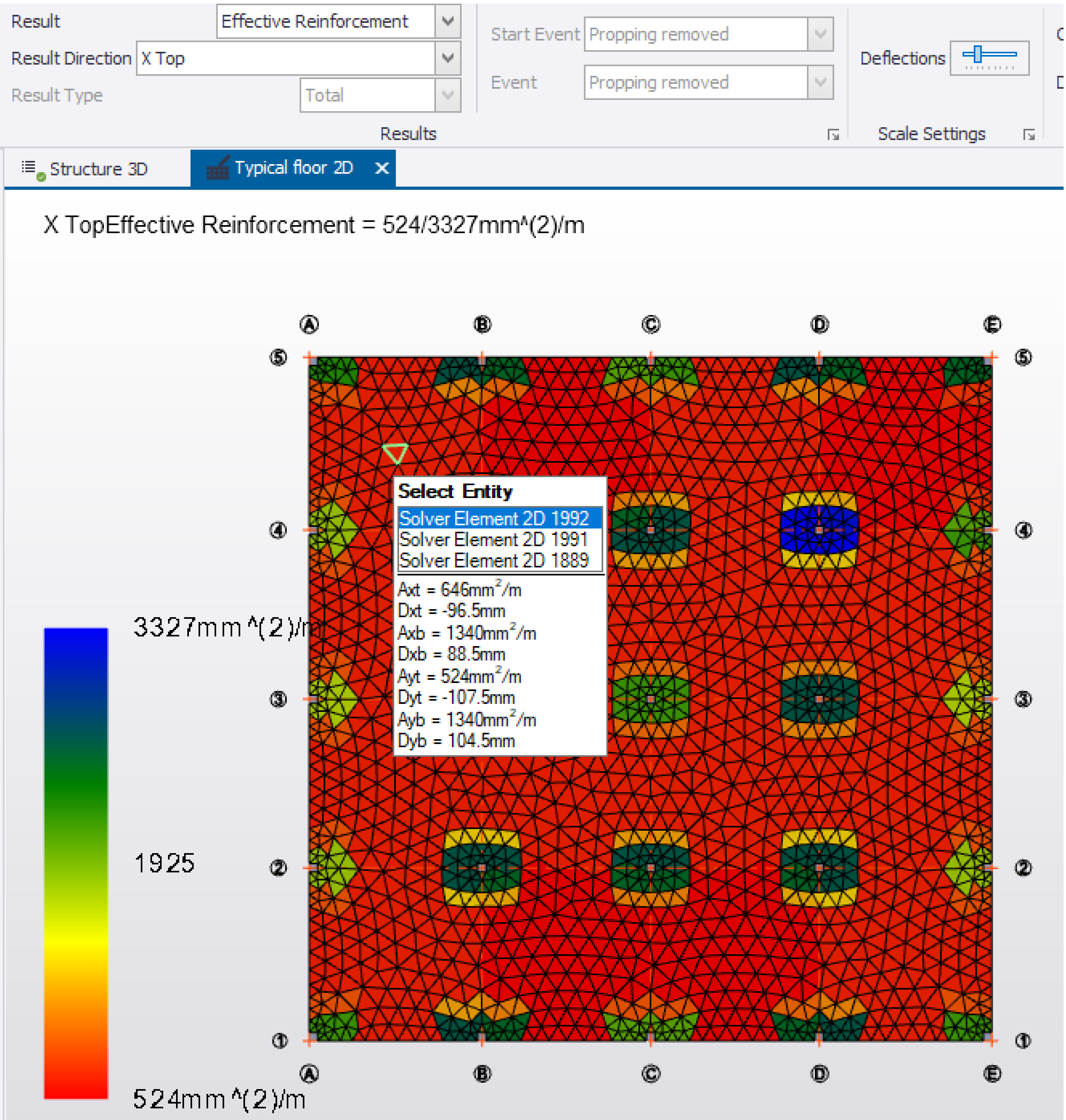
   a. Select the Result Direction as **X - Creep + Cracking + Shrinkage**,

   The relative stiffness for the **Propping removed** load event is displayed.



3. Select the Result as **Effective Reinforcement**, and then:
   a. Select the Result Direction as **X Top**,

   The effective reinforcement for the **X Top** direction is displayed.



NOTE When you hover over any FE element in the slab deflection view regime - values are provided within the tooltip.

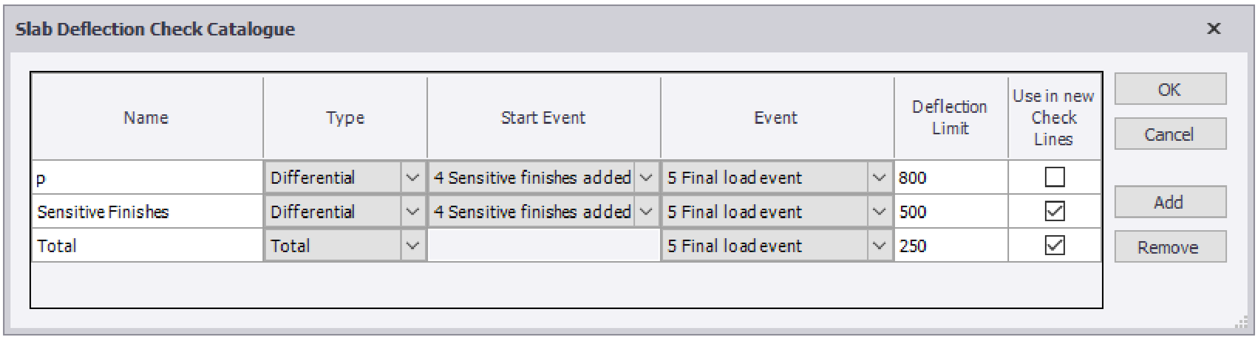
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the **Slab Deflection** toolbar, click **Deflection Checks**



   Whilst three checks have been defined above, only two of these have been set to be used in new Check Lines:

   - **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/500
   - **Total** will check the total deflections to the final load event against a deflection limit of 1/250

2. Click **OK** to close the dialog.

Place Check Lines

NOTE	Check lines can only be created in a 2D view.

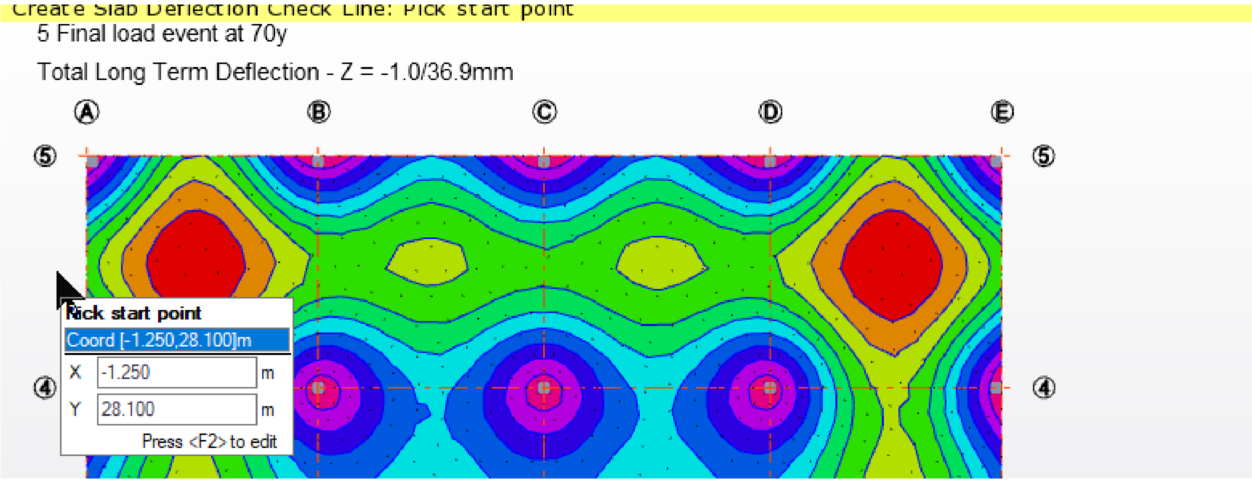
We will define several check lines in this example:

1. Ensure you are in the **Typical floor** 2D plan view and if necessary change:
   a. the Result back to **Deflections**,
   b. the Result Type to **Total**,
   c. the Event to **Final load event**
2. Click **Create**

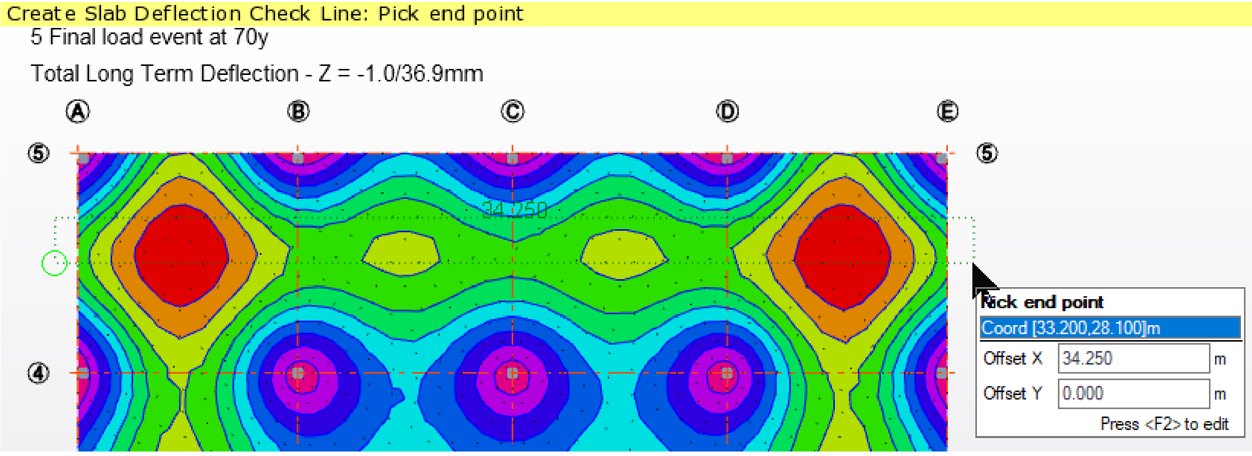
NOTE	When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which “Use in new Check Lines” was checked.

3. To place the check line at approximately mid-span between grid lines 4-5 from grid line A to E:

   a. Pick the start point to the left of grid line A as shown:

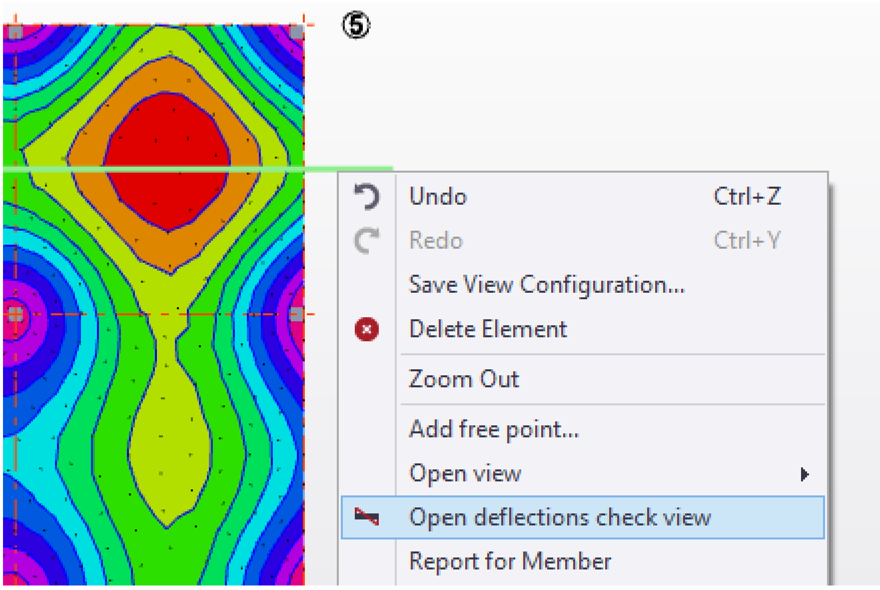


   b. Pick the end point to the right of grid line E as shown:

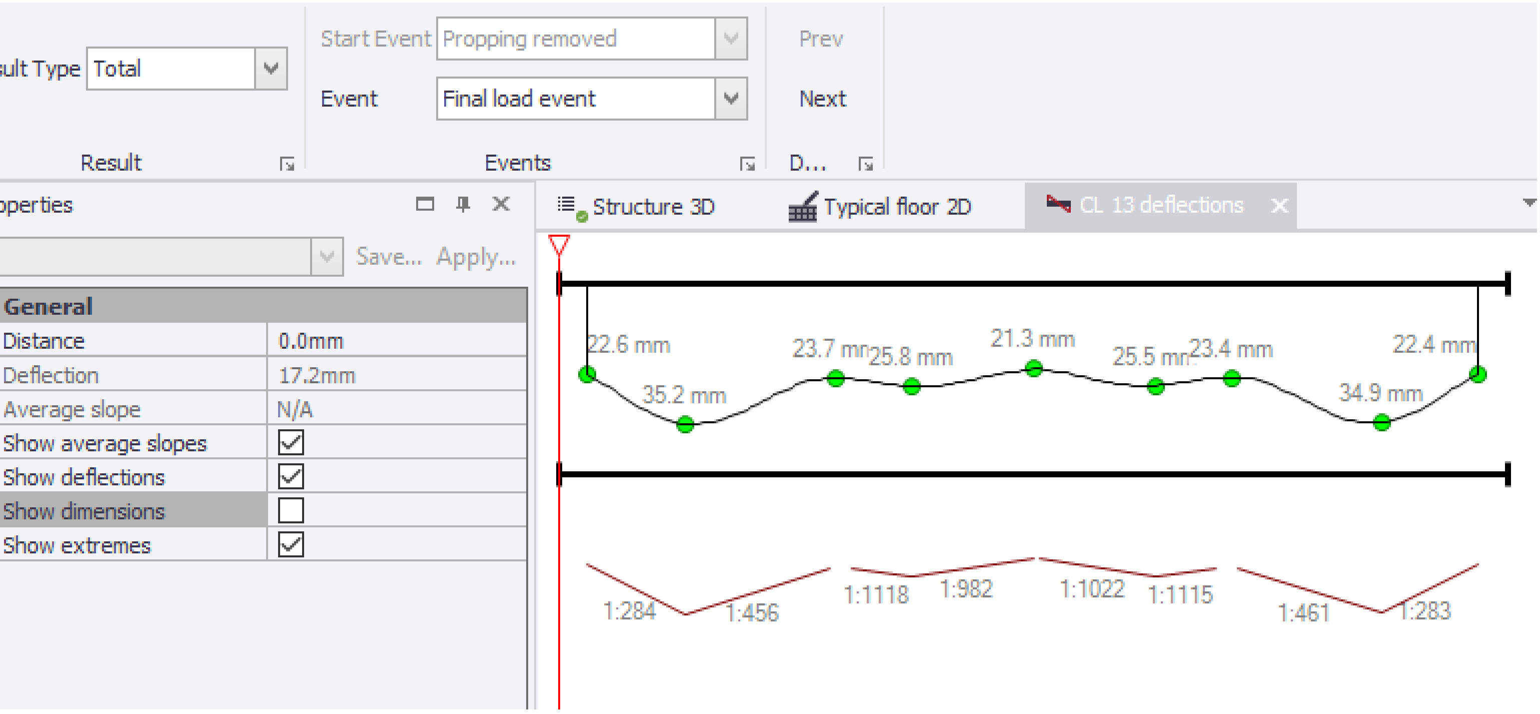


   c. Press **Esc** to end the command.

4. Right click on the check line and choose **Open deflections check view** from the context menu.



The deflection results along the length of the check line are displayed.

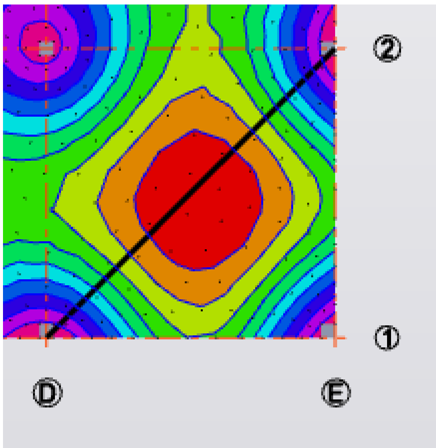


The **Slab Deflection** toolbar allows you to specify the total (as shown above), or differential or instantaneous results for the selected events.

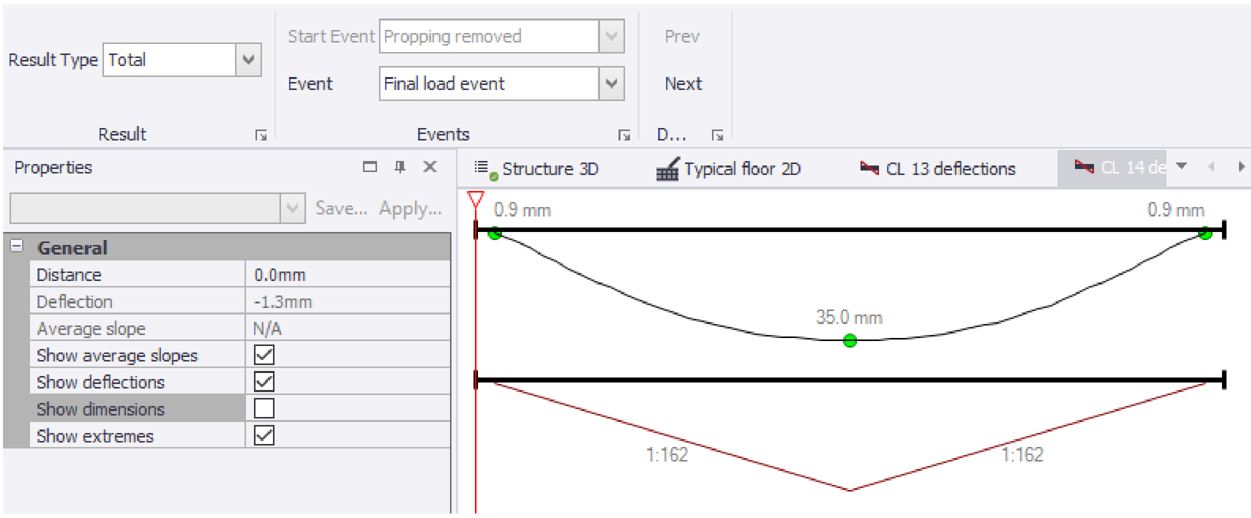
Tekla Structural Designer then draws average slopes between maximum and minimum points.

If we return to the original deemed-to-satisfy check - this was performed diagonally between columns in bottom right corner panel of the slab - we will now revisit this using a check line.

5. Create a check line running diagonally between columns in bottom right corner panel (from D/1 to E/2) where the peak deflection occurs.



6. Right click to open the deflections check view for the new check line.



A total deflection limit of Span / 250 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 125. In the view above the average slope between these points is 1 / 155, i.e. less than the limit.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Generate Check Line Reports

A tabulated report is available for each check line which itemises each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **Typical floor 2D** view, right click on the diagonal check line and select **Report for Member**

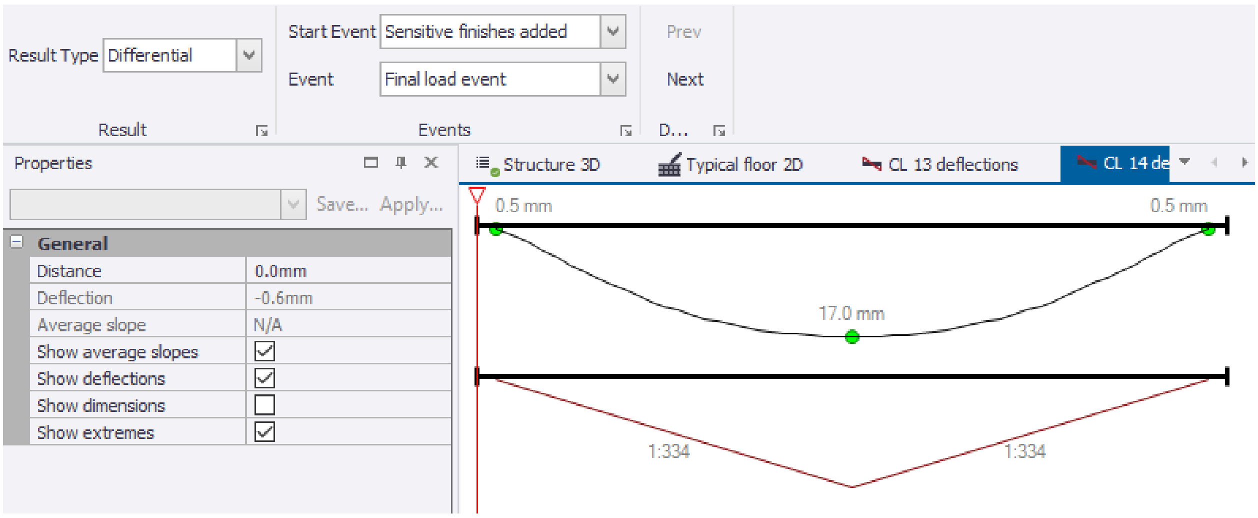
Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	16.5	5515.4	1 : 334	✓ Pass	0.749
Total	250	1 : 125	34.1	5515.4	1 : 162	✓ Pass	0.772

As previously noted, the slope above is reported as 1:155 which is not less than the allowable slope limit of 1:125 and hence a Pass.

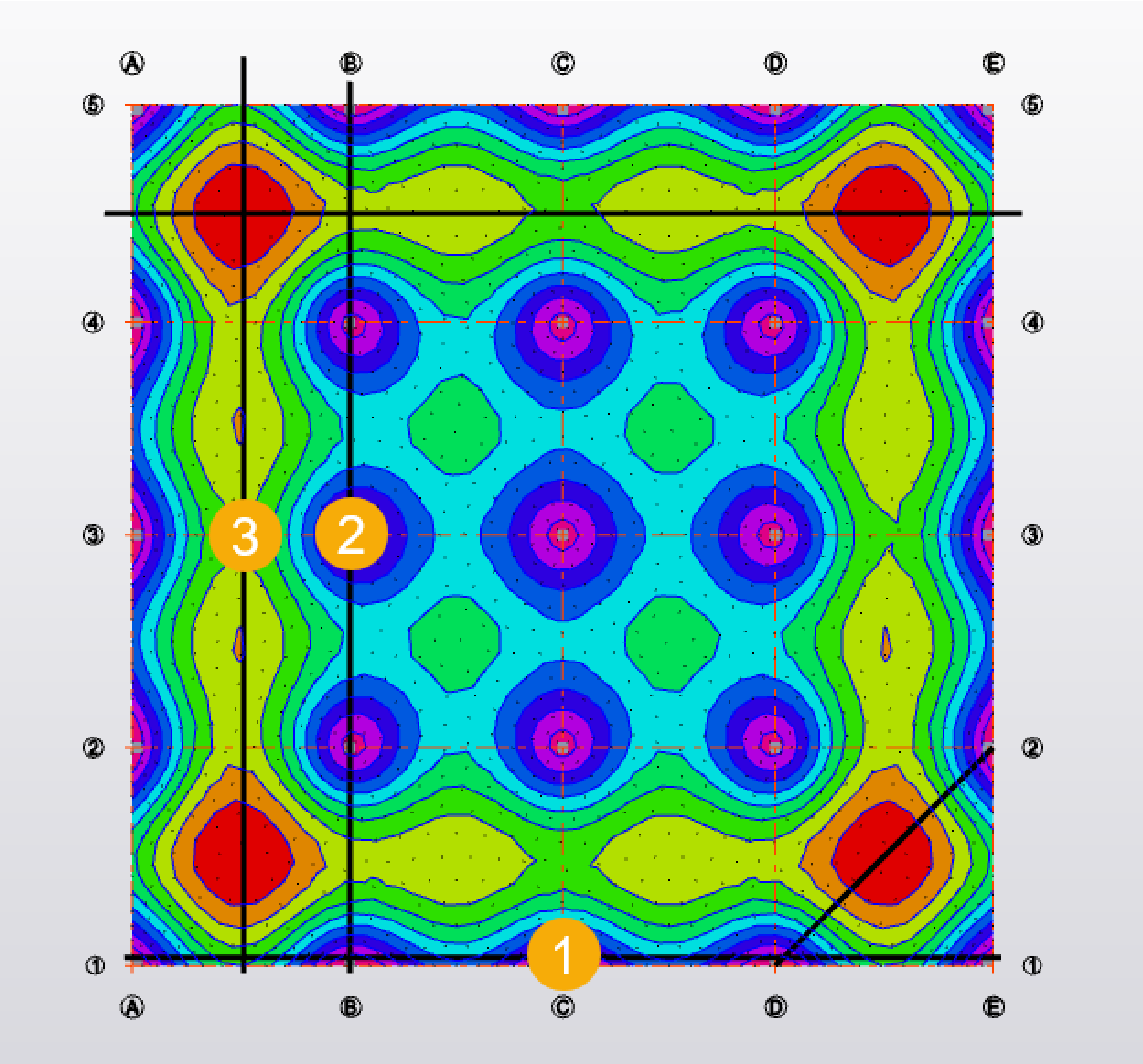
If you click within the Load Analysis Deflection view, the ribbon changes to allow you to display deflection results and slopes for the Result Type - Total or Instantaneous for a chosen event, or Differential between chosen events.

2. Switch to the Load Analysis Deflection view for the diagonal check line.
3. From the **Slab Deflection** toolbar, change the Result type to **Differential** and check deflection and slopes between the**Sensitive finishes added** and **Final load event**.



We can add as many check lines to the model as we consider appropriate.

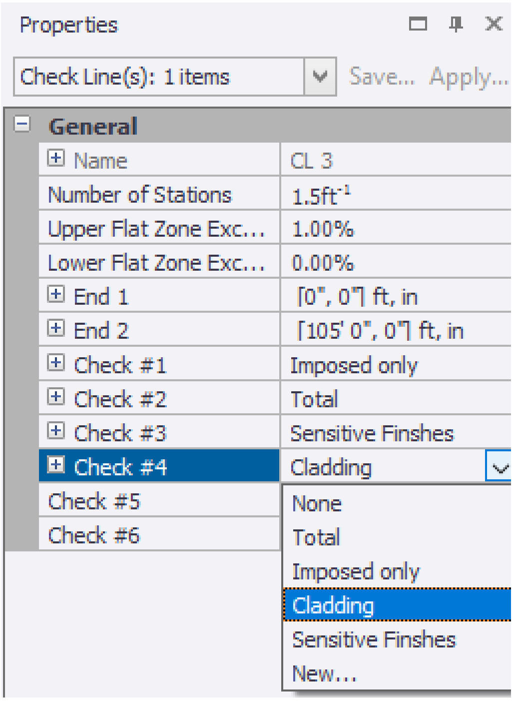
4. Return to the **Typical floor 2D** view.
5. Add three further check lines using the default deflection checks in the catalogue as follows:
    a. along grid line 1, from A/1 to E/1
    b. along grid line B, from B/1 to B/5
    c. half way between grid line A and B, starting at grid line 1 and finishing at grid line 5



6. Press **Esc** to end the command.

    If you select each Check Line in turn you are able to edit the deflection checks associated with it in the Properties Window.

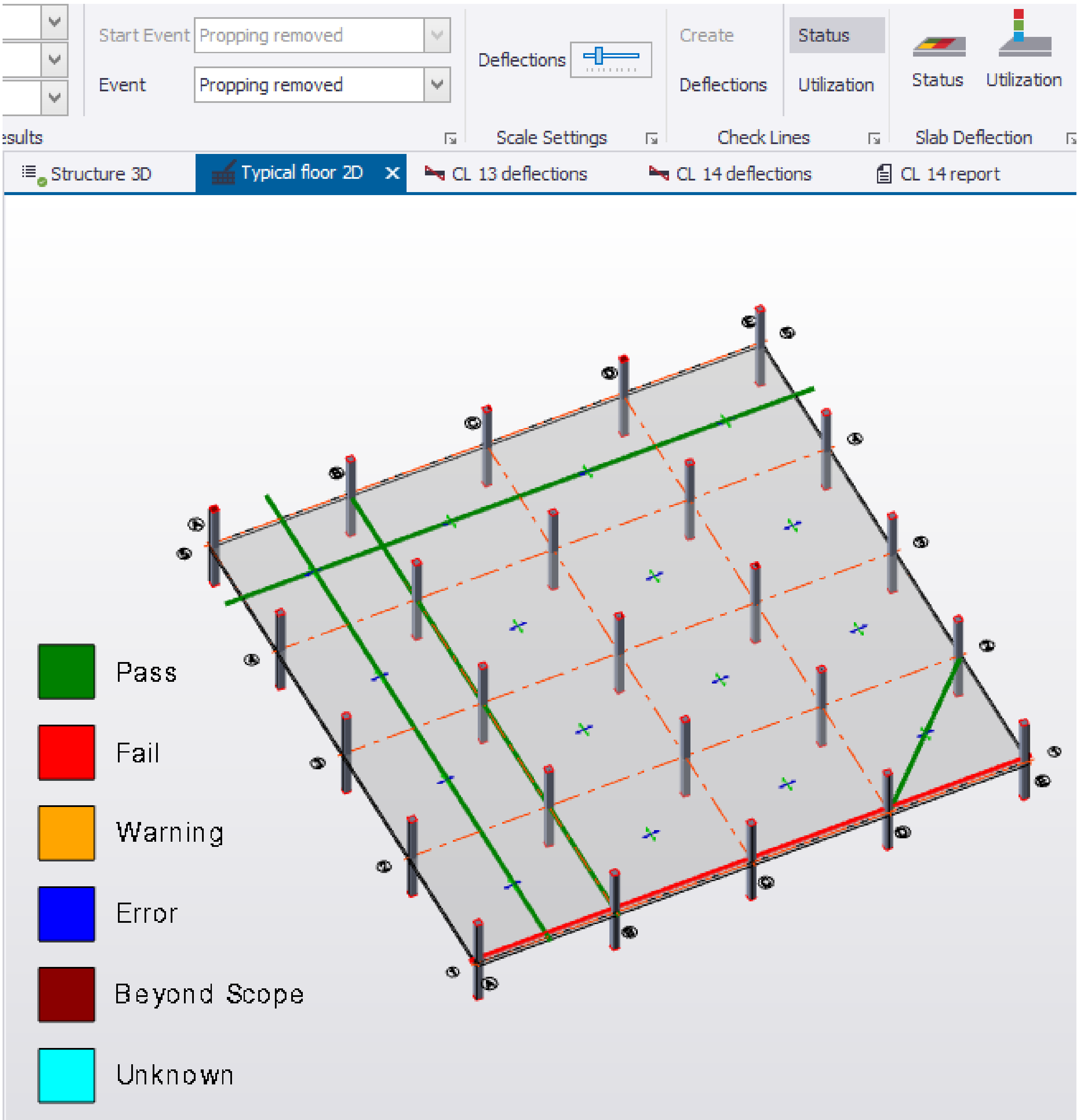
7. Select the check along grid line that runs along grid line 1 and ensure that it also has a **Check #3** defined as **Cladding**.



Review Check Line Status and Utilization

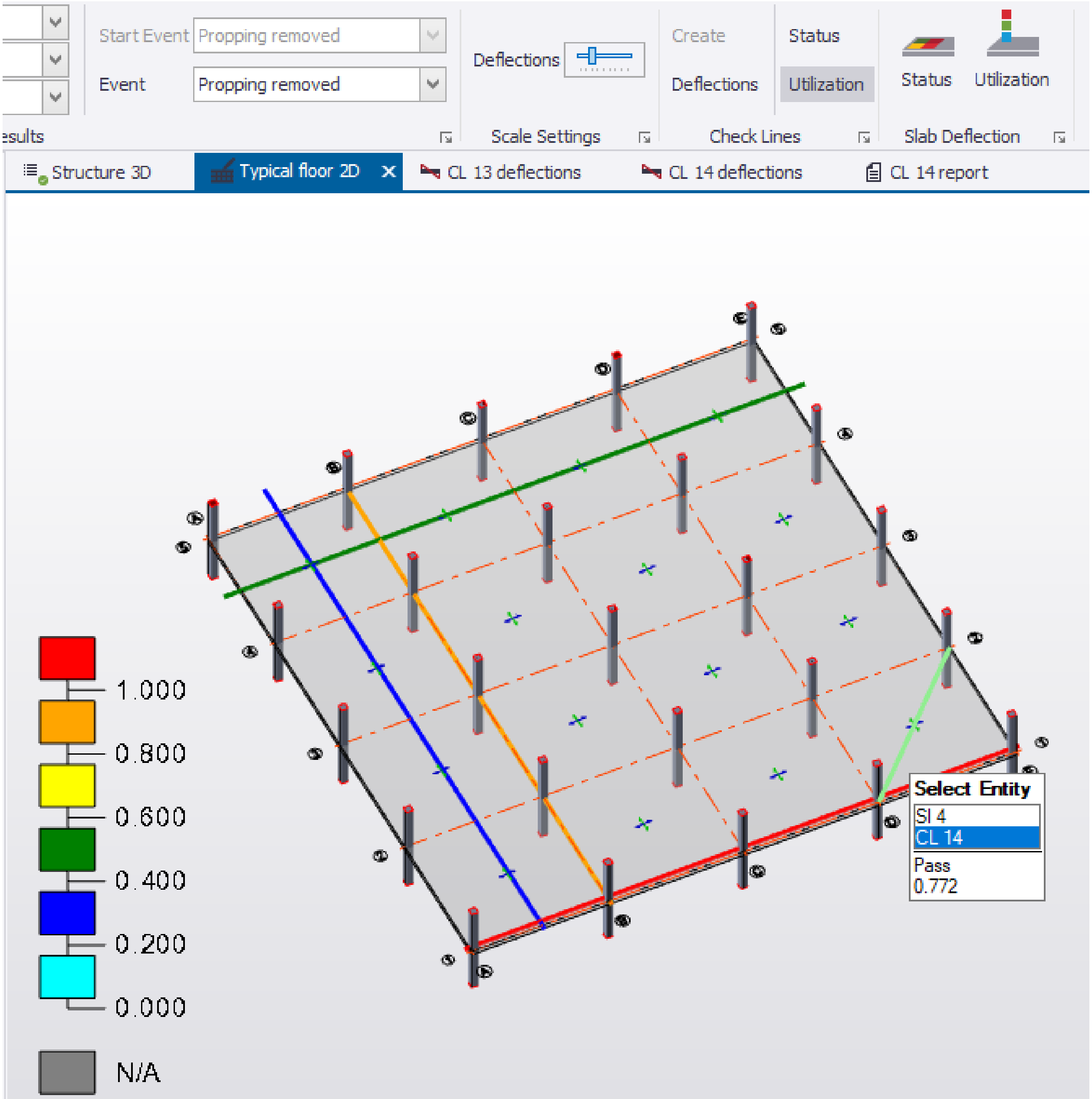
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the Typical floor 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



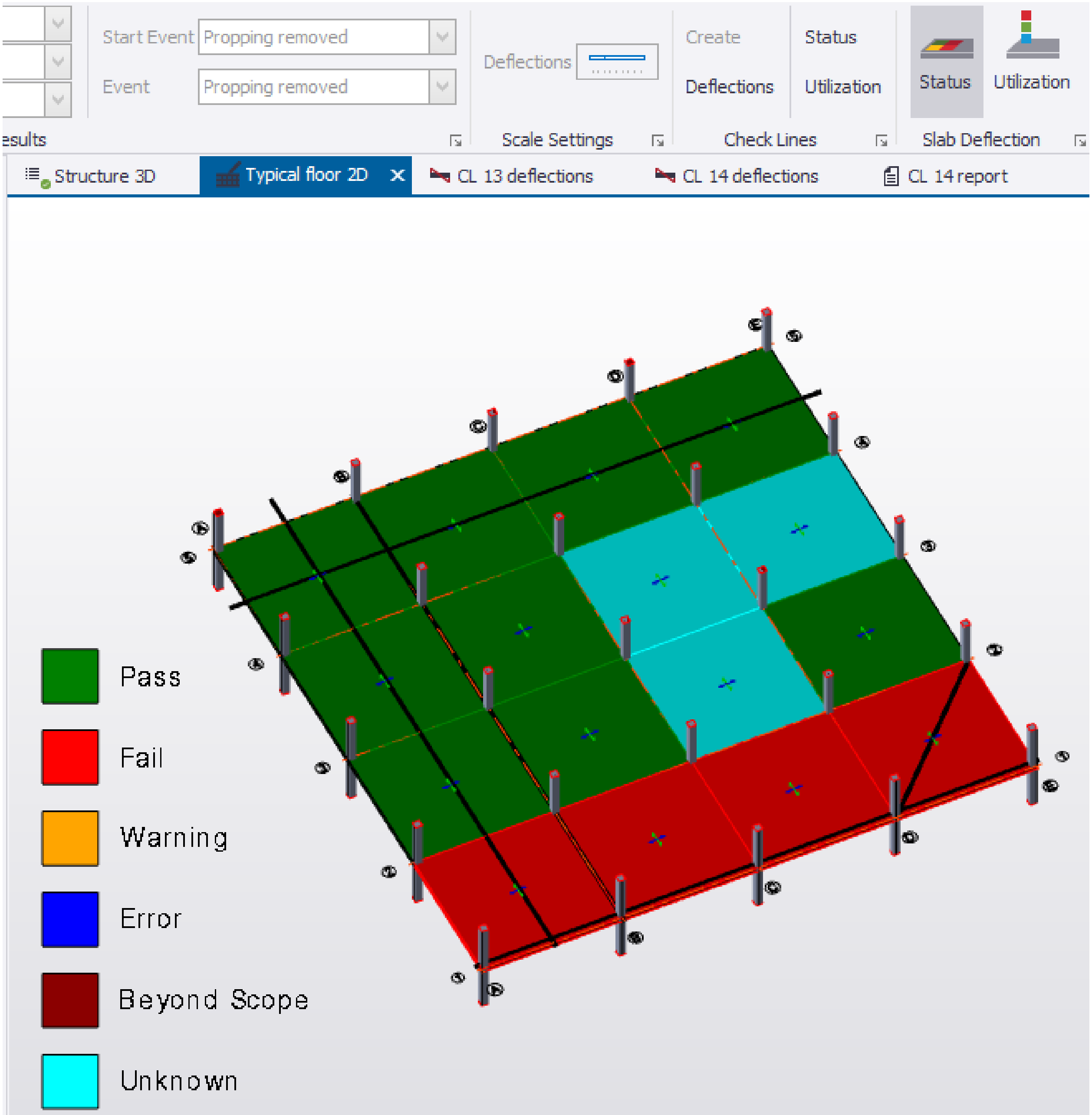
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/fail status of each slab.

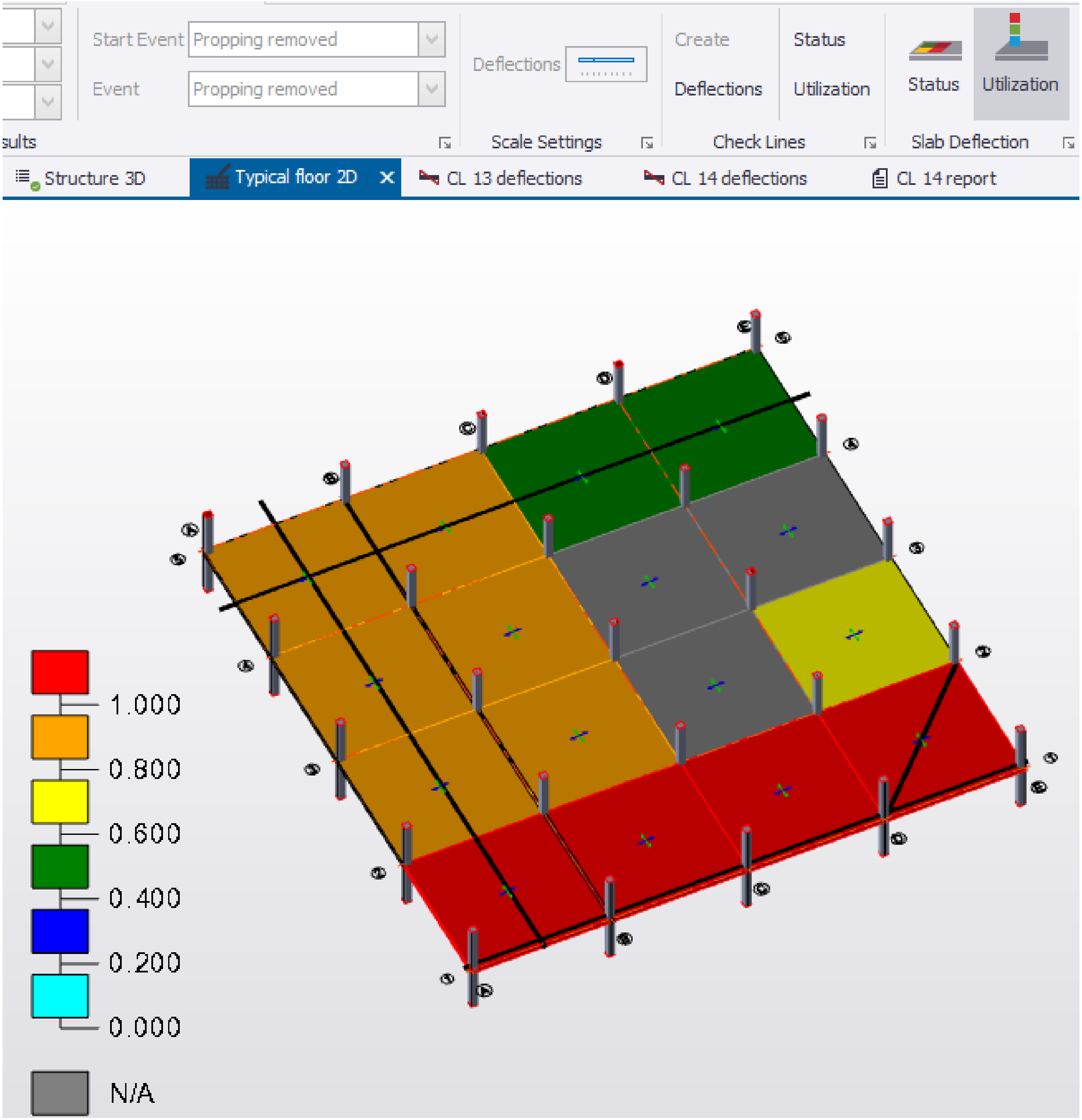


- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed.
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.

- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the slab items Fail where the check lines runs along A/1-E/1.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

   This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

The analysis is extremely quick and since everything is contained within one model file, it allows “What If” scenarios to be considered to find the optimum solution.

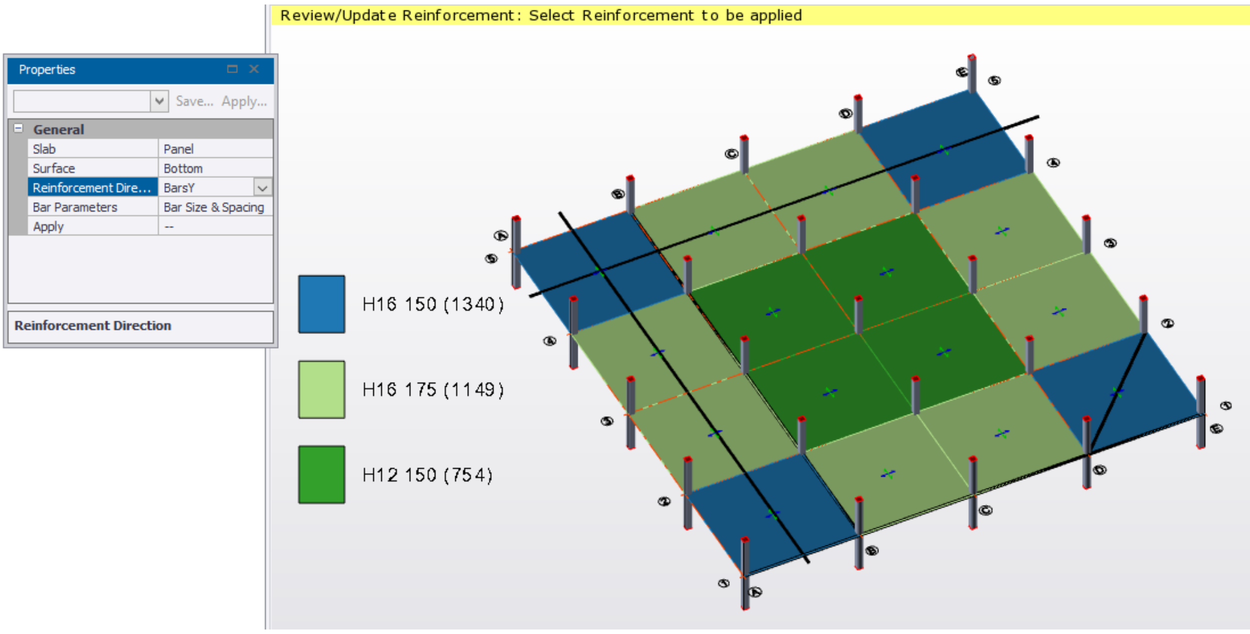
In this exercise we will start by looking at the impact of adjusting the reinforcement.

1. Highlight the check line running along grid line B and make a note of its utilization.

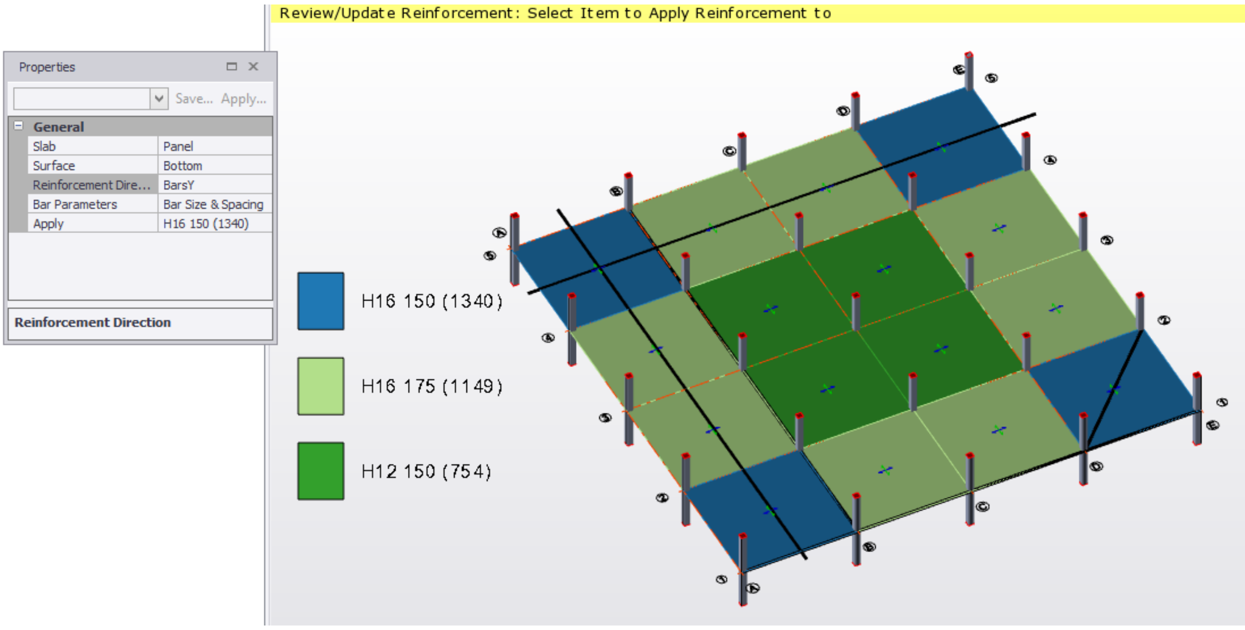
   TIP Press the tab key if necessary to highlight the check line when it is directly under the mouse cursor.

   ---

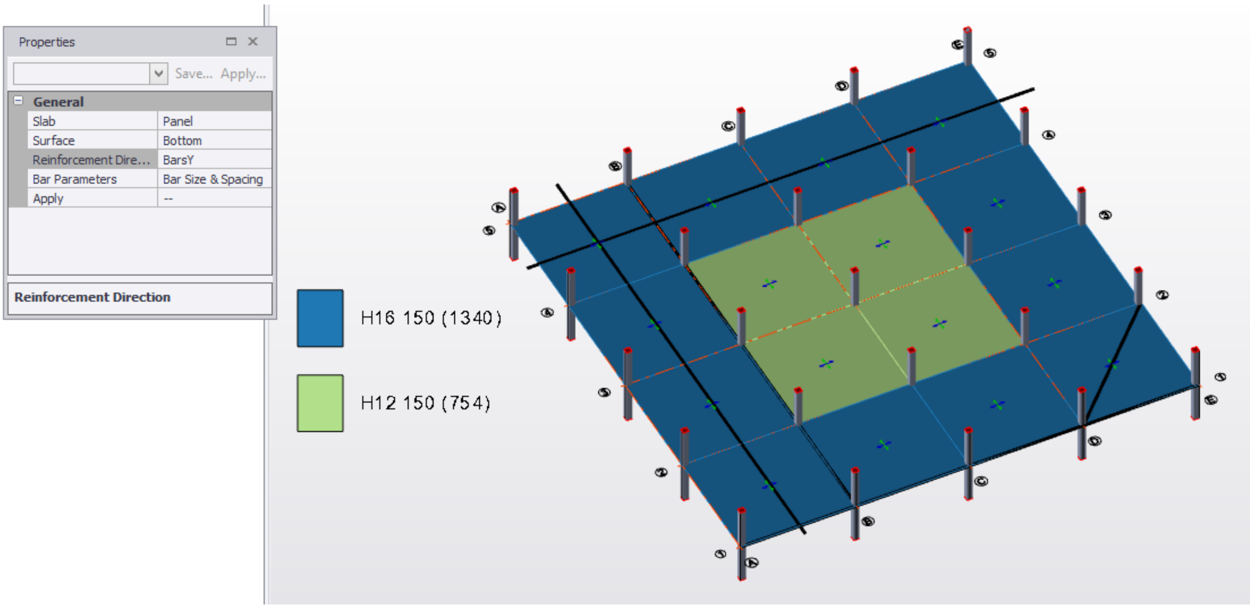
2. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.
3. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



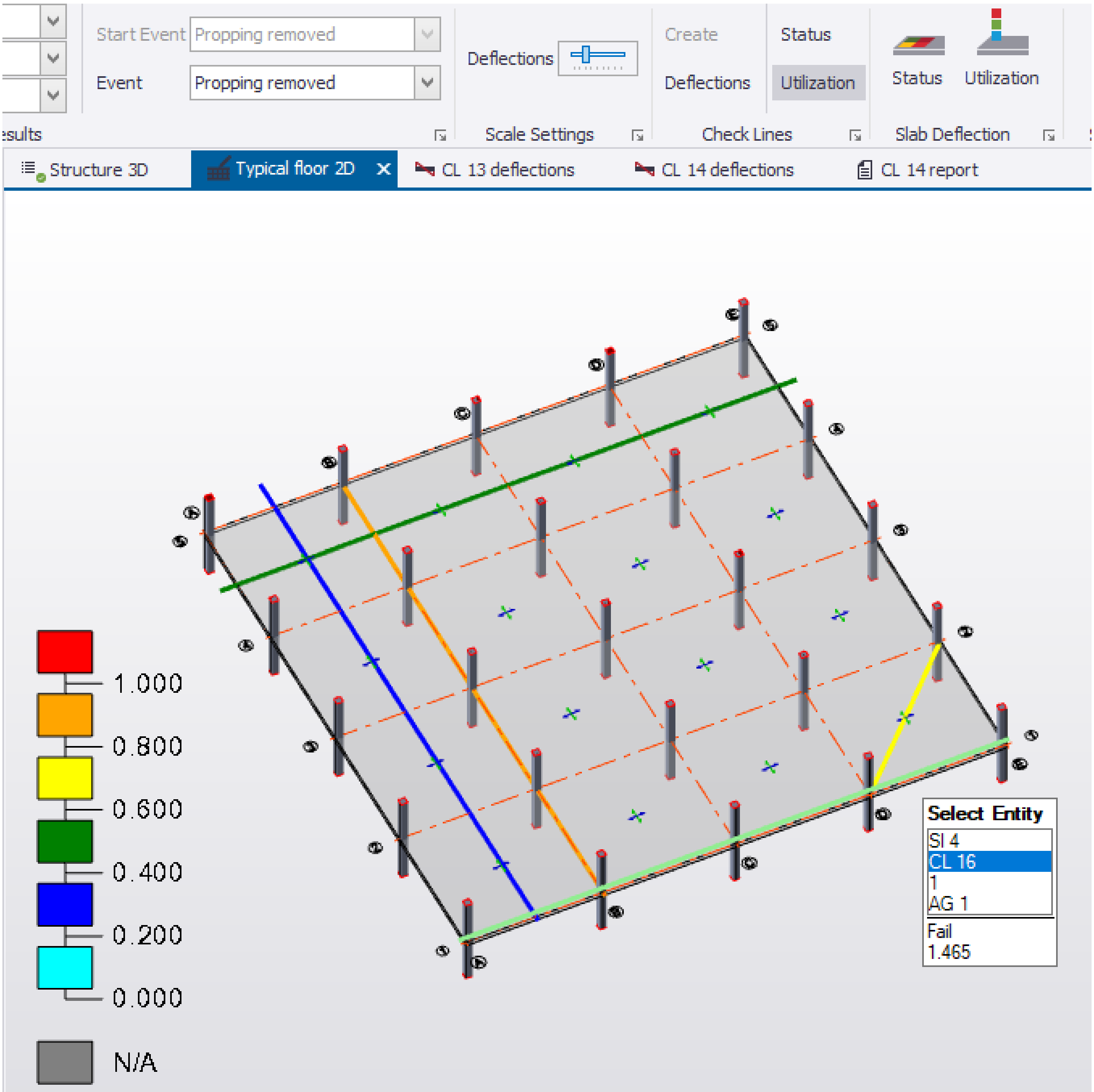
4. Click on one of the corner slab panels to select **H16 150** as the reinforcement to be applied.



5. Click on the eight slab panels currently showing H16 bars at 175 spacings to change them to the 150 spacings.



6. Change the Reinforcement Direction to **BarsX** to see the bars in that direction.
7. Edit the same 8 panels in exactly the same way, so that the bars in X match the bars in Y.
8. Click **Analyse Current** to update the results
9. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
10. Investigate the tooltip results

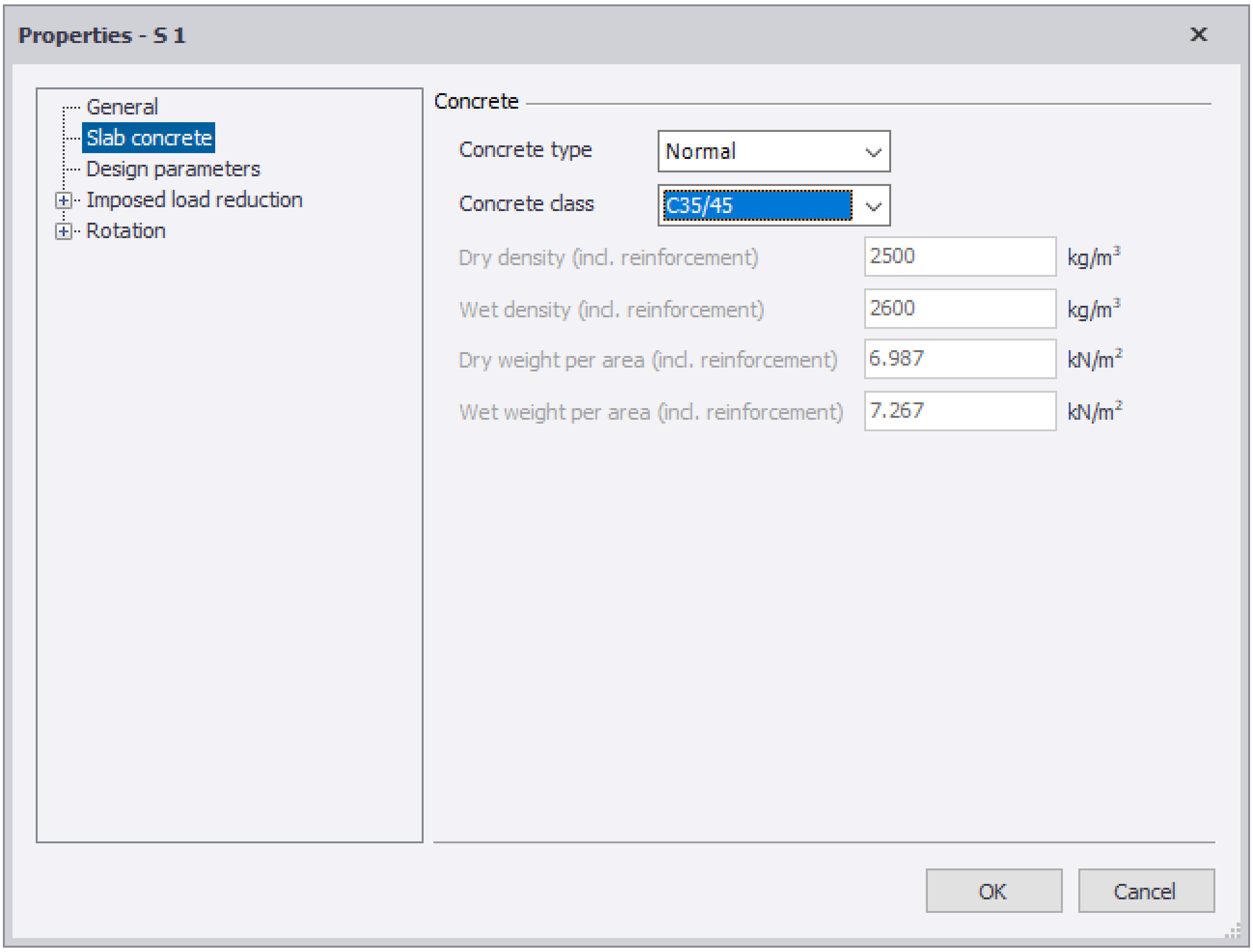


You should find that although some utilizations have reduced, the line along grid line B is still failing. At this point you could begin to look at the impact of the various other input parameters. For now, we will adjust the concrete grade of the typical floor slab group from C30/37 to C35/45.

11. Right click slab **Sl 4** between D-E/1-2 on the typical floor level and choose **Edit....** from the context menu.

NOTE Editing a slab item via the right click context menu updates the parent slab properties, not just the individual slab item. Hence any changes will be applied to all the slab items in S 1.

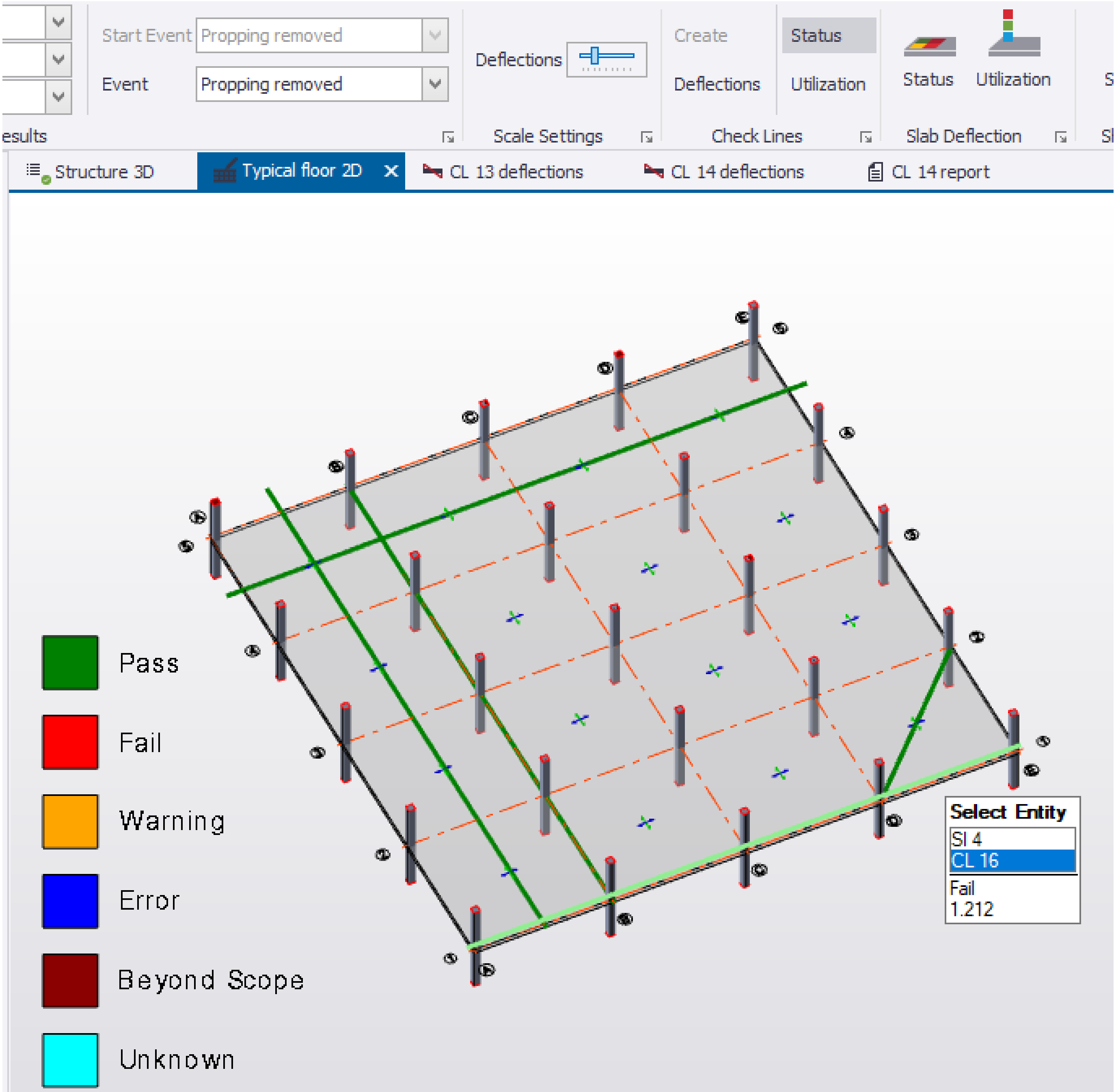
12. On the **Slab concrete** page of the dialog, change the Concrete Strength to **C35/45** and click OK



To update the check line results we need to re-run the analysis. A chasedown analysis is automatically performed as part of the slab deflection analysis, however, it should be borne in mind that some edits could affect the element design i.e. reducing the concrete slab thickness would result in an increase to the required reinforcement and hence a Design Concrete (Static), Slab or patch design may be required again.

13. Click **Analyse Current** again to update the results.

14. Review the Check Lines **Status**.



The check line with the more onerous Cladding deflection limit is still failing, but we can clearly see improvements in the results. It has reduced from a Utilization ratio of 1.538 to 1.262.

In fact, to obtain a Pass we would have needed to increase the concrete grade to C45/55.

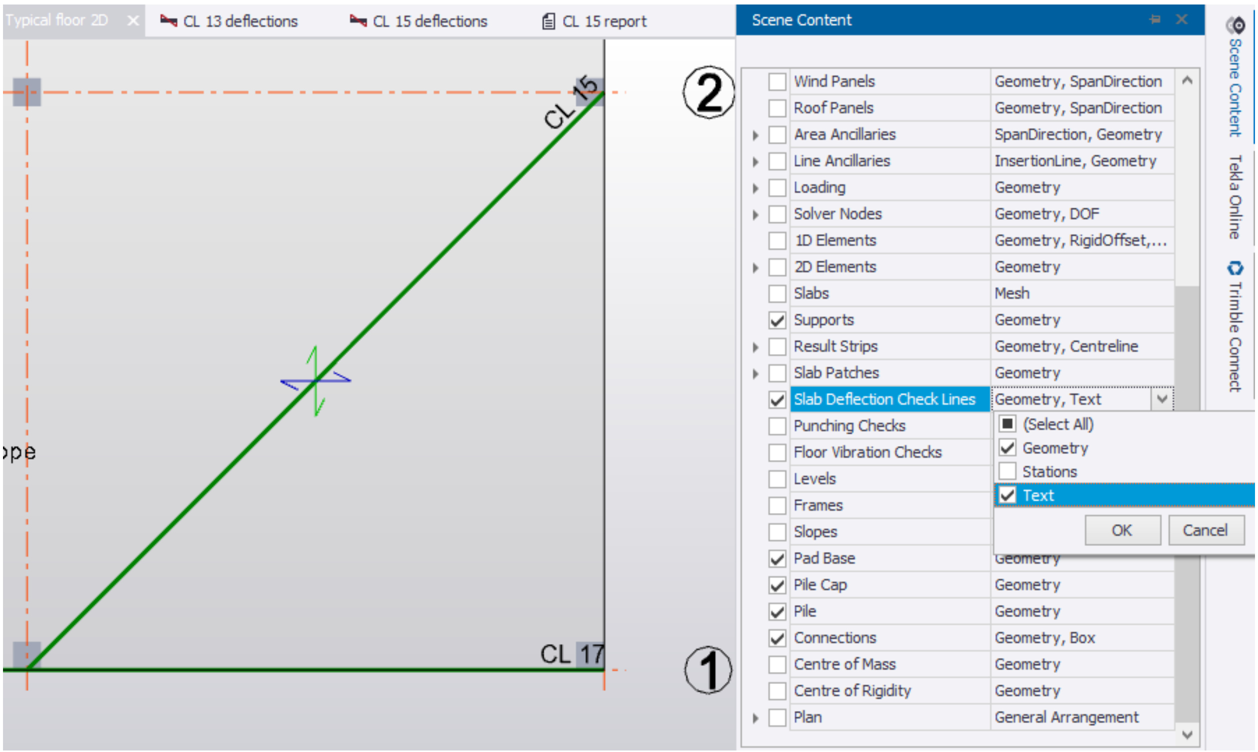
NOTE For the purpose of this exercise, as the cladding check is not a code requirement we will simply disable it before proceeding to generate the model report.

15. Select the check line along grid 1 and then in the **Properties window**, reset Check #3 to **None**.

Generate Model report

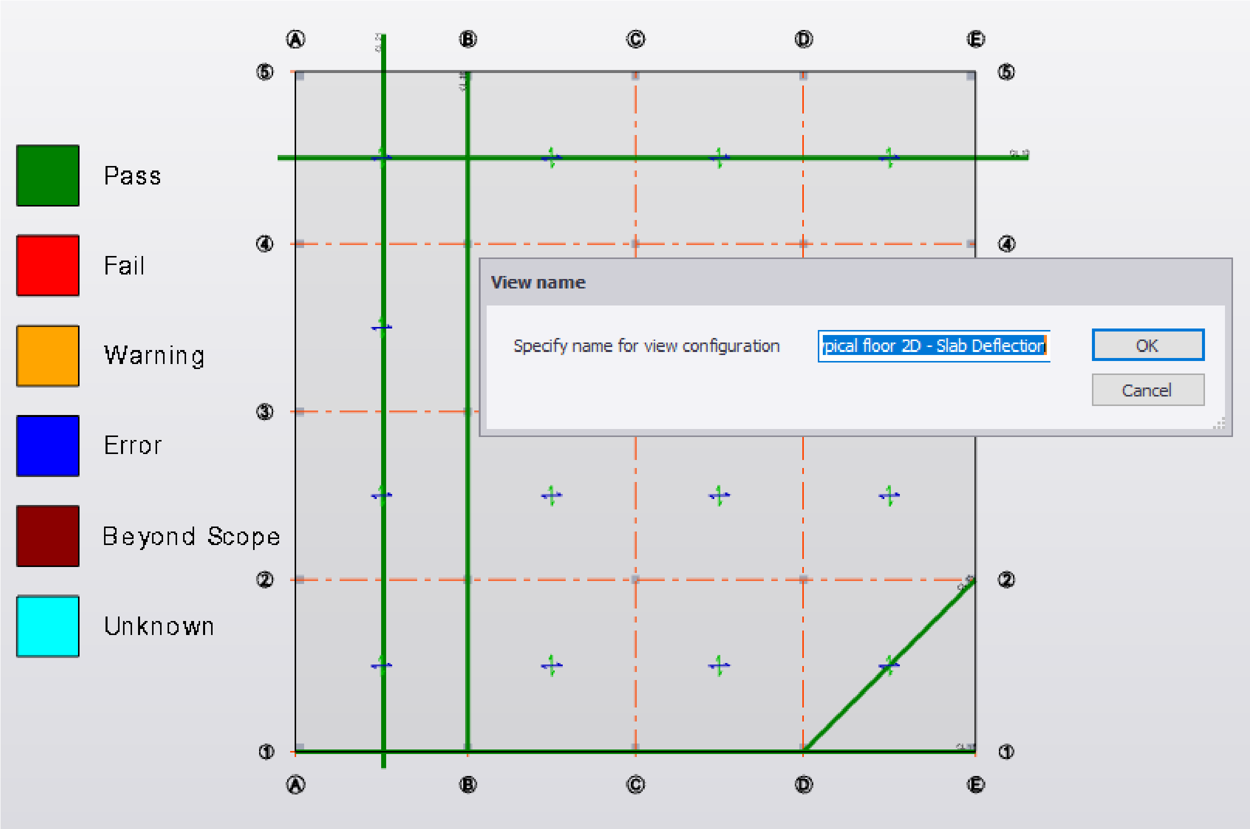
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



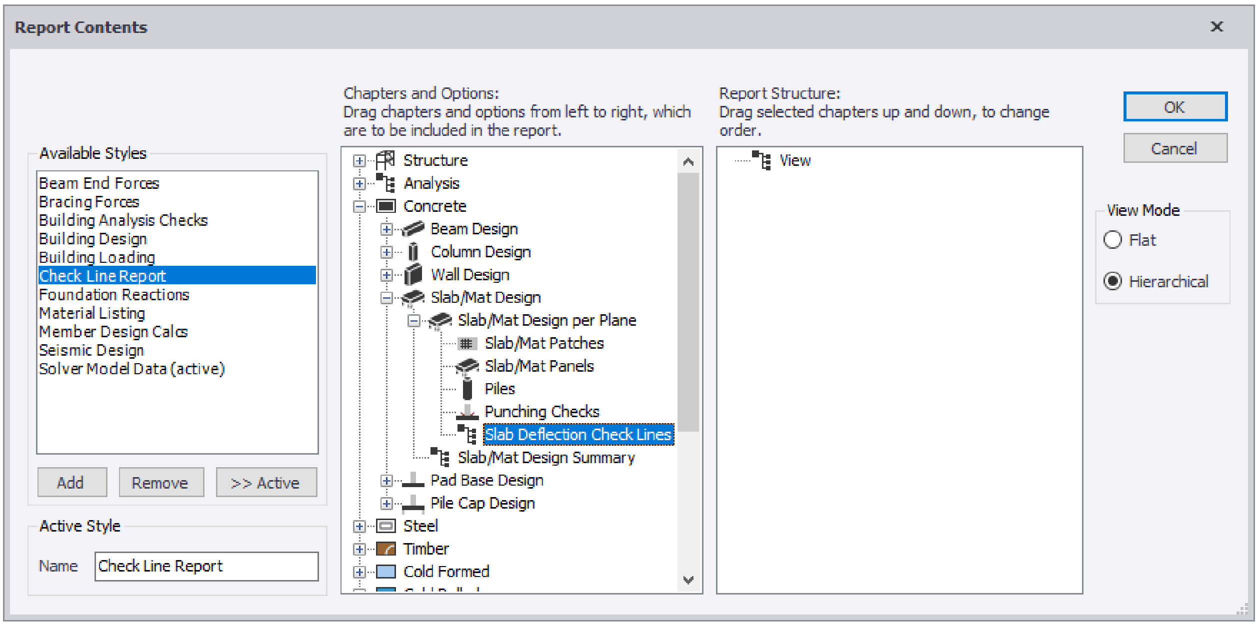
TIP The check line references can be customised using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

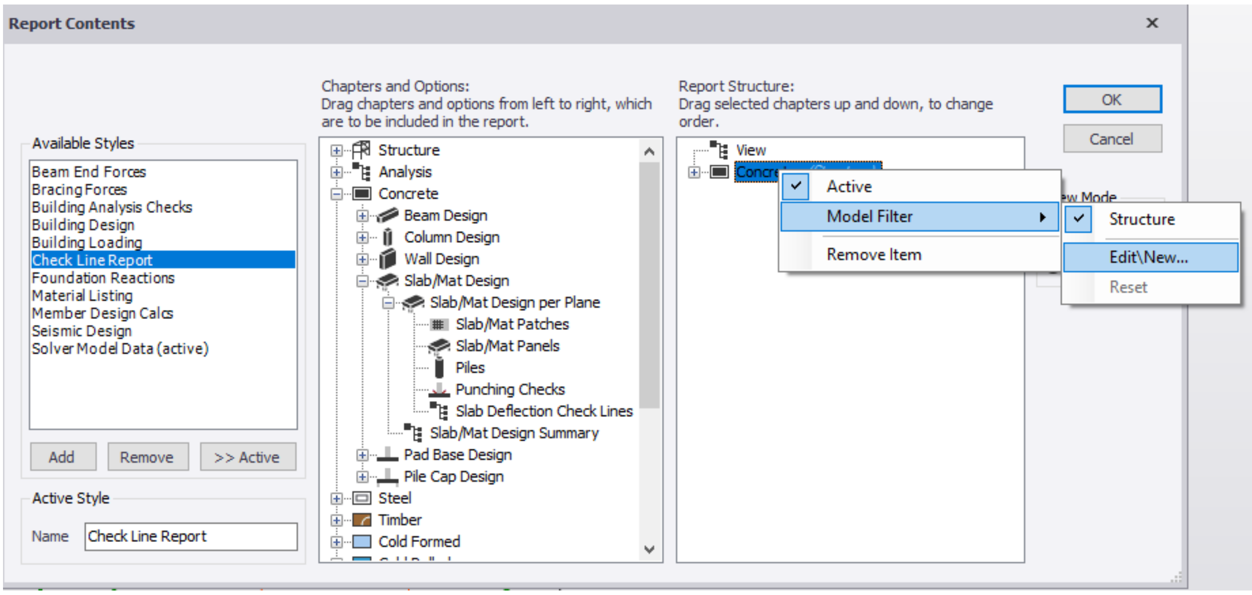


This saved view can be included in the Check line report.

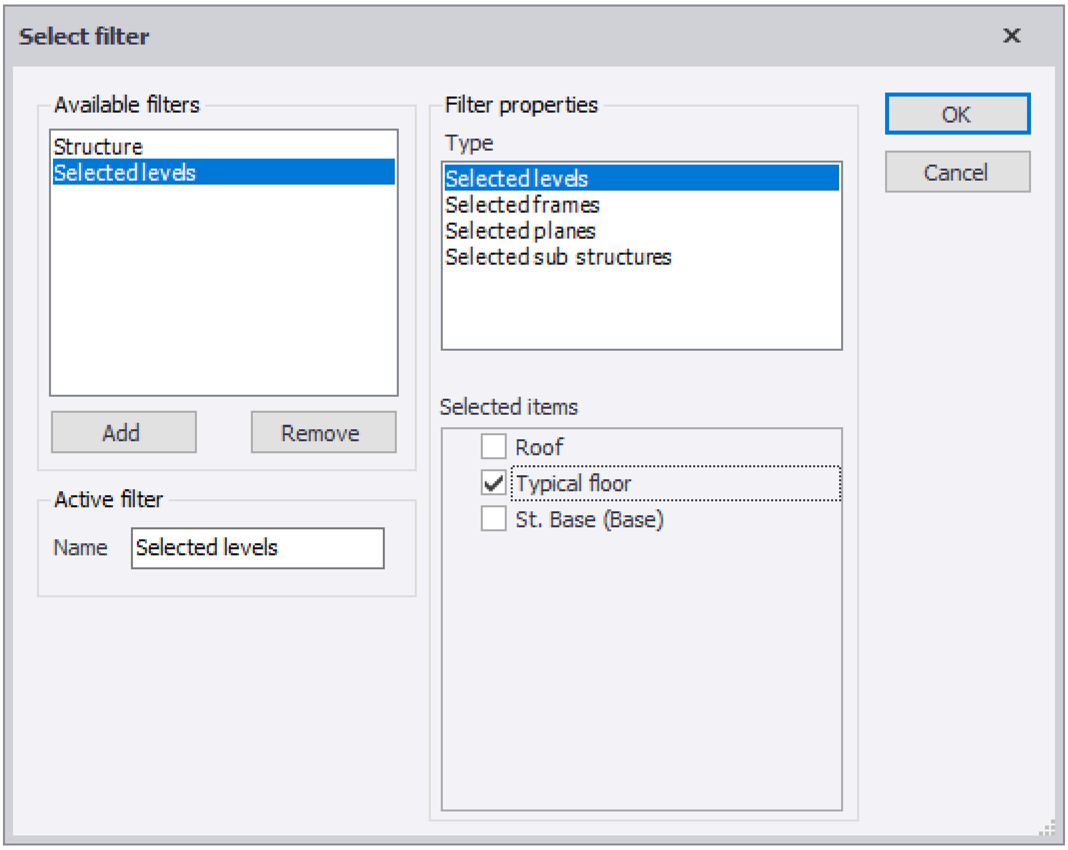
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



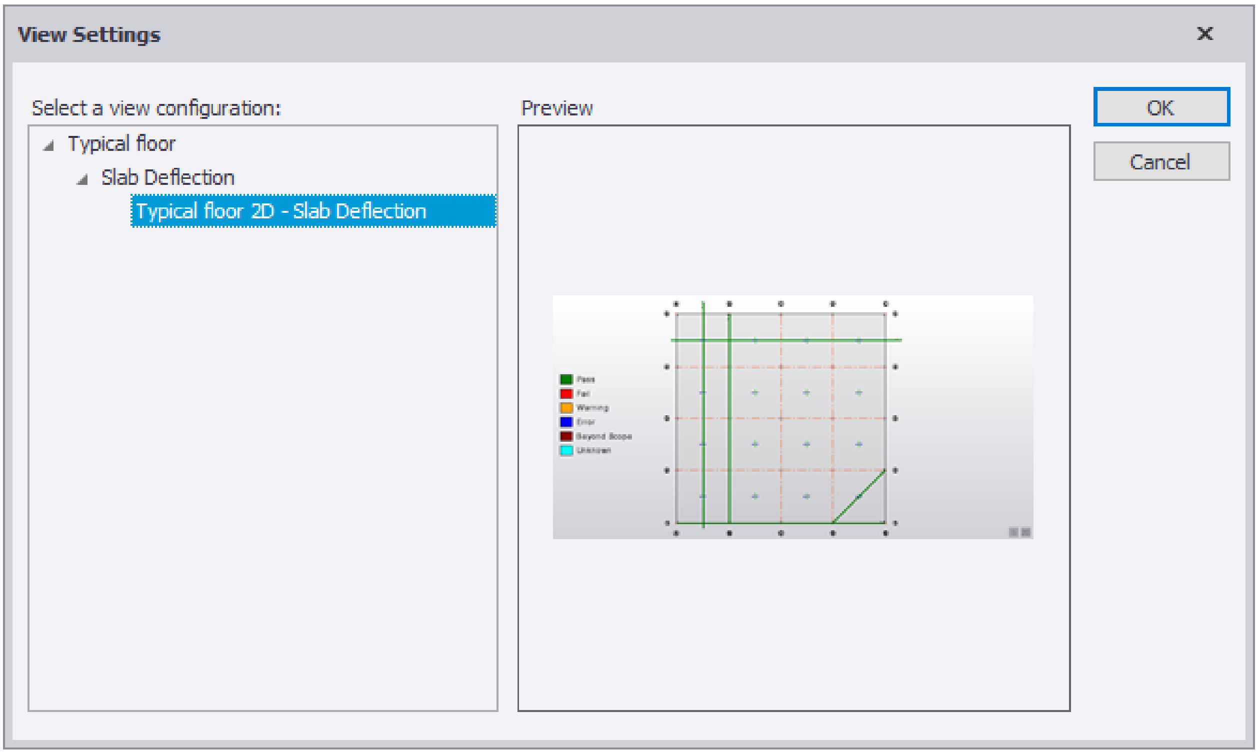
7. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**



8. In the Filter dialog, click **Add** and select Selected levels and ensure a check against Typical floor

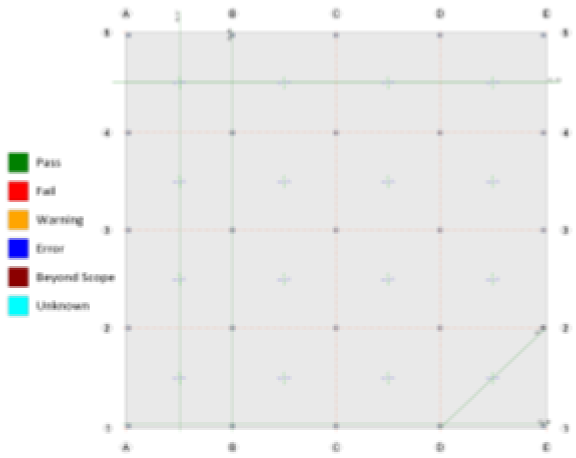


9. Click **OK** to return to the Report Contents dialog.
10. In the Report Structure, right click **View**, then choose **Settings...**
11. Select the Slab Deflection view you created earlier



12. Click **OK** to return to the Report Contents dialog
13. Click **OK** to exit and save the report.

    A report structure called Check Line Report has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select “Check Line Report”
    b. Click the Show Report command to open the report.



Typical floor 2D - Slab Deflection

Concrete

Slab/Mat Design

Slab/Mat Design per Plane

Typical floor

Slab Deflection Check Lines

CL 13

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	-3.9	3520.1	1 : 900	✓ Pass	0.278
Total	250	1 : 125	-10.7	3520.1	1 : 330	✓ Pass	0.379

CL 14

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	14.0	5515.4	1 : 394	✓ Pass	0.635
Total	250	1 : 125	-29.3	5515.4	1 : 188	✓ Pass	0.665

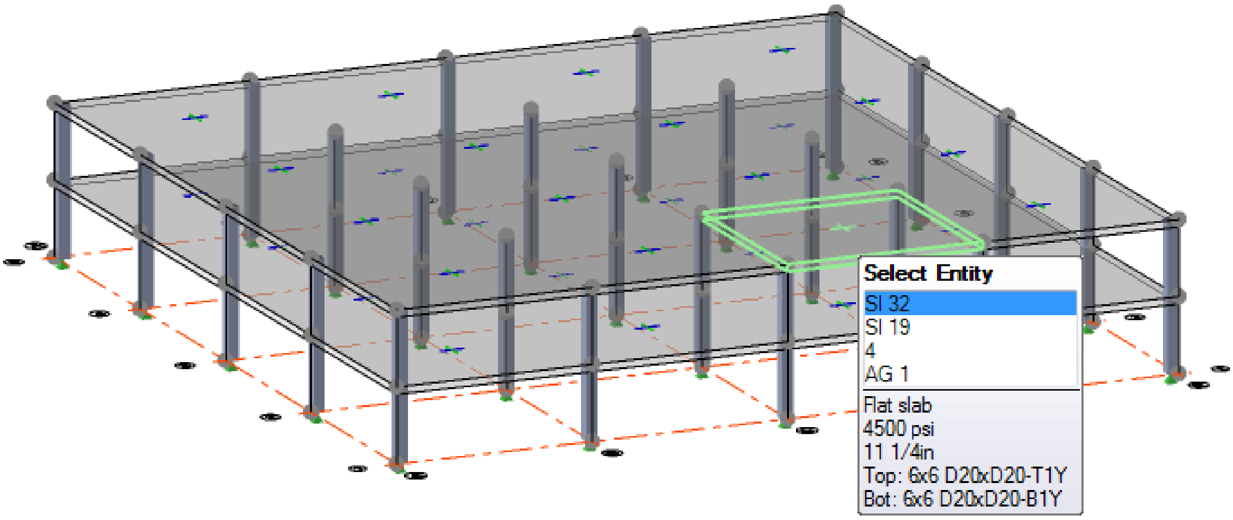
CL 15

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	8.7	3324.4	1 : 381	✓ Pass	0.656
Total	250	1 : 125	19.2	3324.4	1 : 173	✓ Pass	0.723

Slab deflection example (ACI)

In the following exercises deflections will be checked for the two story multi-bay flat slab tutorial model shown below.



Geometry:

- Bay centers 26' 3"
- Floor to Floor spacing 9' 10"
- Slab thickness 11 ¼ ", concrete grade 4500 psi
- 18" x 18" columns, concrete grade 4500 psi
- Analysis stiffness adjustment factors in accordance with ACI code guidance

Loading:

- Dead area load 30 psf
- Live area load 105 psf
- Cladding perimeter load 0.7klf

Deemed to satisfy checks (page 438) provide one method of checking, however the main focus of these exercises will be to investigate rigorous slab deflection analysis (page 440) and check lines (page 476) as the method of checking.

Deemed to satisfy slab deflection checks example (ACI)

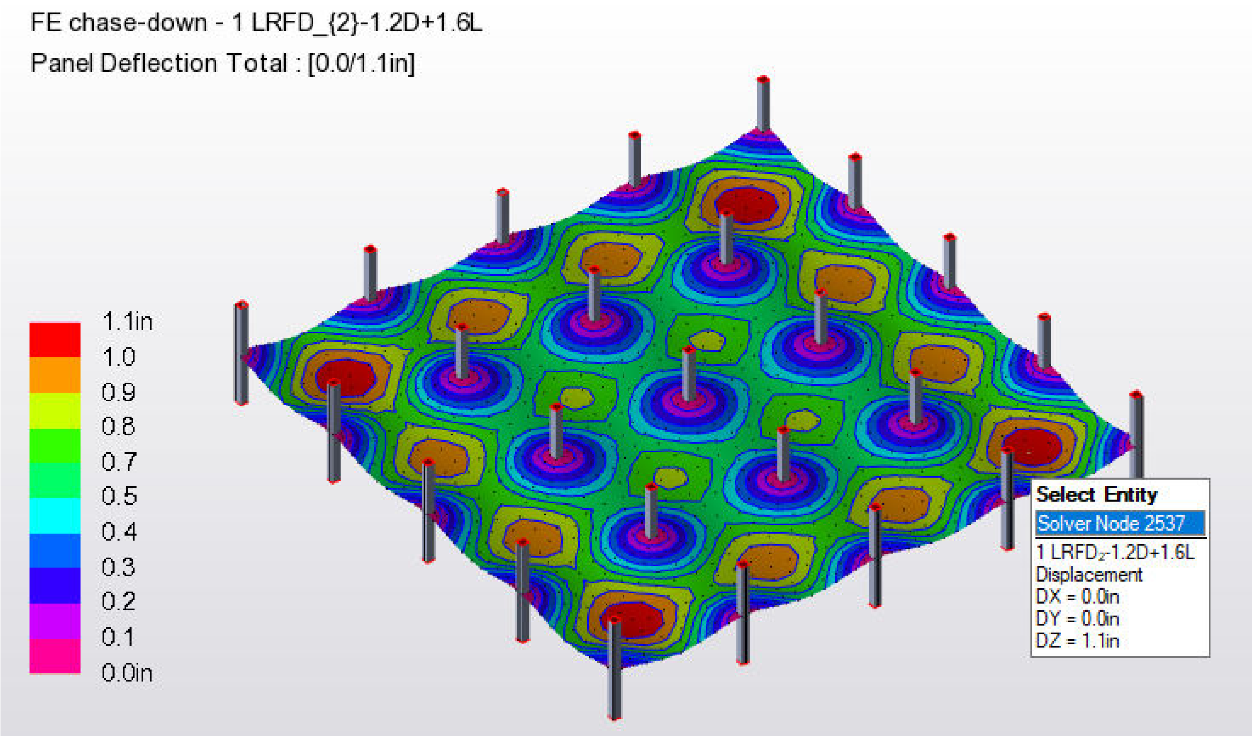
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results



Identify critical check locations

We can see that the maximum reported deflection is 1.1in, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs engineering judgment will be required to assess which deflections and span lengths require checking.

The deflection at the identified location now needs to be checked against a limiting span-to-depth ratio which we will assume for this example can be taken as span /240.

Taking the diagonal dimension across the columns, the deemed-to-satisfy span / 240 rule provides a deflection of $[\sqrt{(26' 3''^2 + 26' 3''^2)}] / 240 = 1.8\text{in}$.

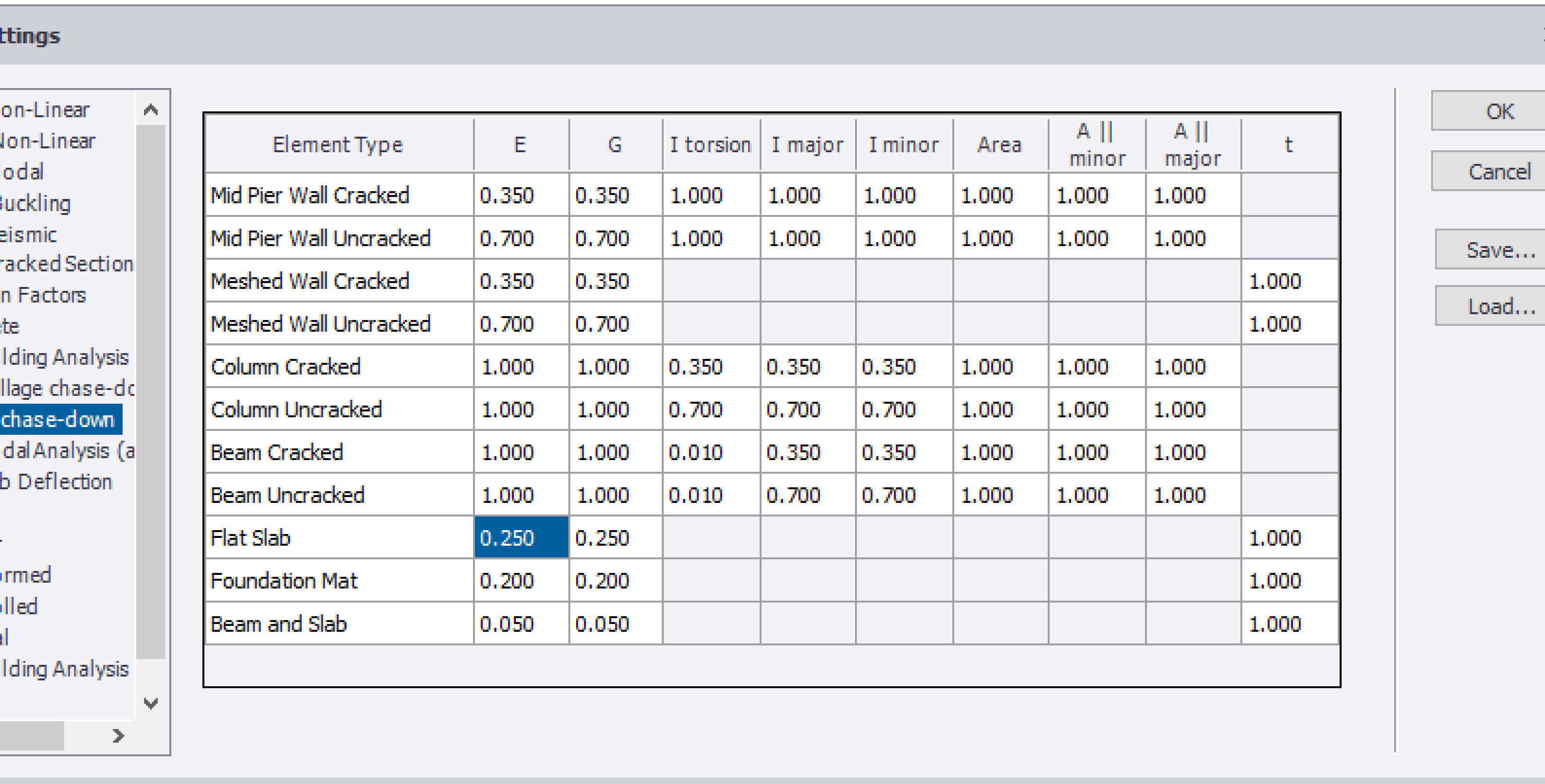
This compares unfavorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 240.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.25.



Element Type	E	G	I torsion	I major	I minor	Area	A || minor	A || major	t
Mid Pier Wall Cracked	0.350	0.350	1.000	1.000	1.000	1.000	1.000	1.000	
Mid Pier Wall Uncracked	0.700	0.700	1.000	1.000	1.000	1.000	1.000	1.000	
Meshed Wall Cracked	0.350	0.350							1.000
Meshed Wall Uncracked	0.700	0.700							1.000
Column Cracked	1.000	1.000	0.350	0.350	0.350	1.000	1.000	1.000	
Column Uncracked	1.000	1.000	0.700	0.700	0.700	1.000	1.000	1.000	
Beam Cracked	1.000	1.000	0.010	0.350	0.350	1.000	1.000	1.000	
Beam Uncracked	1.000	1.000	0.010	0.700	0.700	1.000	1.000	1.000	
Flat Slab	0.250	0.250							1.000
Foundation Mat	0.200	0.200							1.000
Beam and Slab	0.050	0.050							1.000

Rigorous slab deflection analysis examples (ACI)

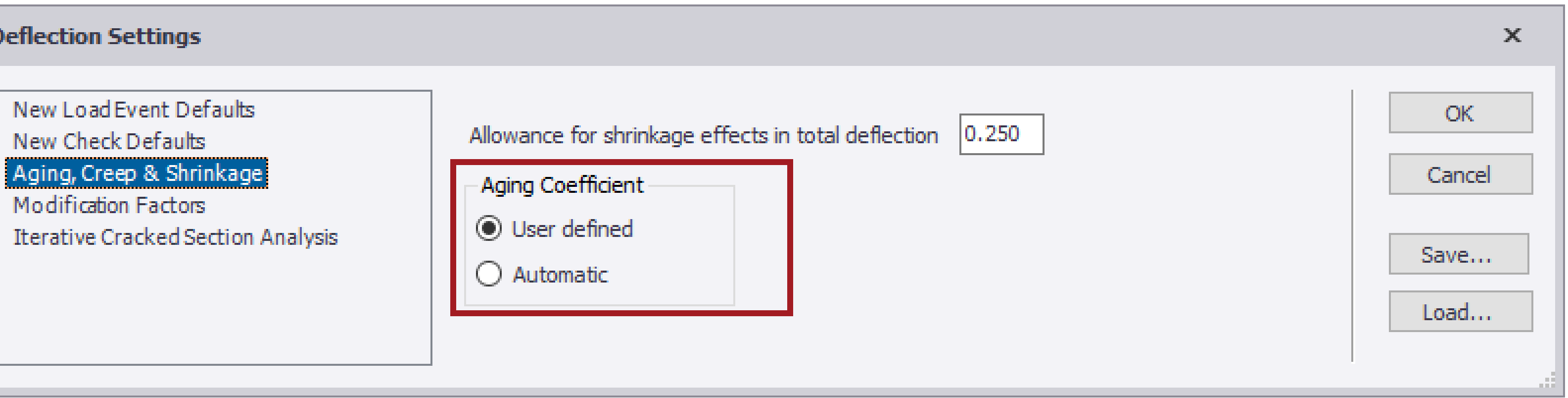
Three alternative methods of allowing for creep and shrinkage in the analysis are investigated.

Slab Deflection Property Choices

Aging Coefficient - User Defined or Automatic?

Tekla Structural Designer calculates a modulus of elasticity for each load event which accommodates both creep and aging.

The **Aging, Creep and Shrinkage** page on the Slab Deflection Settings dialog allows you to choose the method for this.



- **Automatic** - the modulus of elasticity used in the analysis (termed the composite modulus) is calculated rigorously according to the procedure defined in the Concrete Society Technical Report 58. This requires an early age event history to be defined by way of a detailed event sequence.
- **User defined** - the modulus of elasticity used in the analysis is calculated in accordance with ACI 435 Equation 3.33c:

  $\bar{E}_c(t, t_0) = E_c(t, t_0) / (1 + \chi C_t)$

  Where:

- The aging coefficient χ must be specified in the event sequence. The recommended value is 0.8. This is a user specified value.
- The creep coefficient C_t required in the above equation is automatically determined by *Tekla Structural Designer* using equation A-18 from ACI 209 taking account of shape and size effects.

The equation for C_t is $[(t_i-t_0) / (26\, e^{(0.36 \times V/S)}) + (t_i-t_0)] \times C_{ui}$

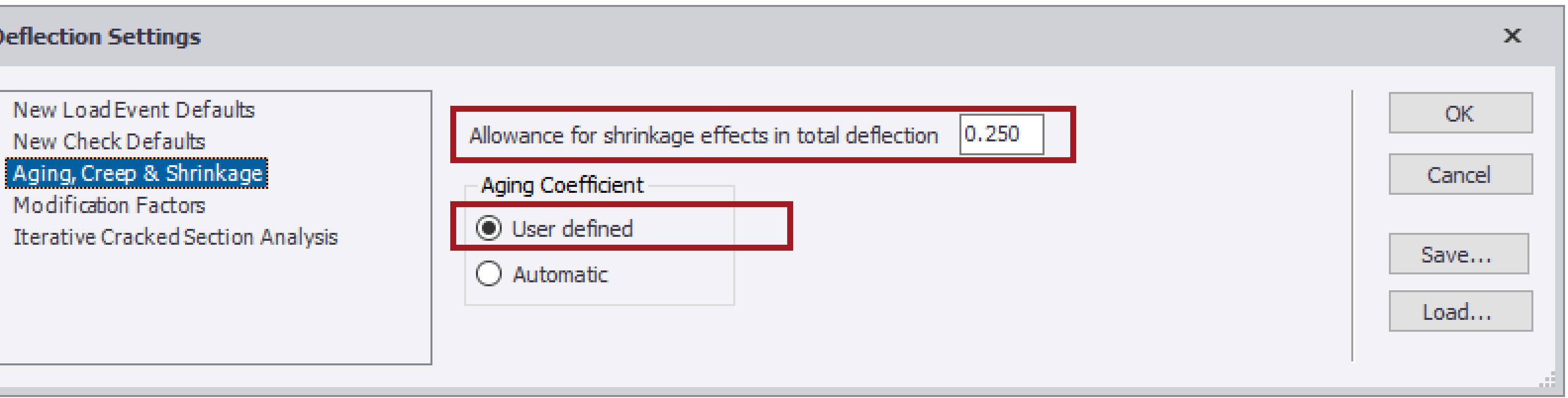
Where:

- For a slab the volume to exposed surface ratio V/S is taken as h/n where h is the slab thickness and n is the number of exposed surfaces. If n is set to zero then the creep coefficient is taken as zero thus eliminating creep from the analysis properties.
- Start of first event, t_0
- End time of event under consideration, t_i
- The Ultimate Creep coefficient, C_u must be specified in the event sequence. In normal conditions the recommended value from ACI 435R page 55 is C_u = 2.35. A different value can be specified according to guidance in ACI 209 if desired.

Shrinkage Allowance, or Combined Creep and Shrinkage Allowance?

When creep effects are included in the analysis, the Allowance for Shrinkage Effects multiplier can be set at a value which caters for shrinkage only. Alternatively you can increase the multiplier to allow for both creep and shrinkage, provided that you ensure that creep is not also accounted for in the analysis.

When using the latter approach it is important to ensure that the Aging coefficient is set to User defined.



Restraint Constant - Significant or Insignificant Restraint?

For each of the three methods considered in this example the Restraint constant (ACI) (page 364) will be varied in order to model different restraint assumptions.

Restraint constant values from ACI 435 are:

- For situations with significant restraint - **4.0**
- For insignificant restraint - **7.5**

This will allow us to compare deflections estimations based on the assumption of significant restraint against those assuming insignificant restraint.

Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance

In this approach the simplest method suggested in ACI 318 is emulated. All creep and shrinkage effects are introduced as a single amplification factor and a cracked section analysis is run on a simplified event sequence without addressing the possibility of early age loading.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
    - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the Simplified Combined Creep and Shrinkage Allowance (ignoring ACI 435)

You can increase the **Allowance for Shrinkage Effects** multiplier to allow for both creep and shrinkage based on the multipliers from ACI 435 Table 4.1.

Using the **ACI code** recommendation (highlighted in the below table), multipliers are immediate **1.0**, creep and shrinkage **2.0**. Total = **3.0**.

Thus a creep and shrinkage contribution = 2.0/3.0 = 0.66 is to be applied which equates to 66% of the total deflection.

Table 4.1 - Multipliers recommended by different authors

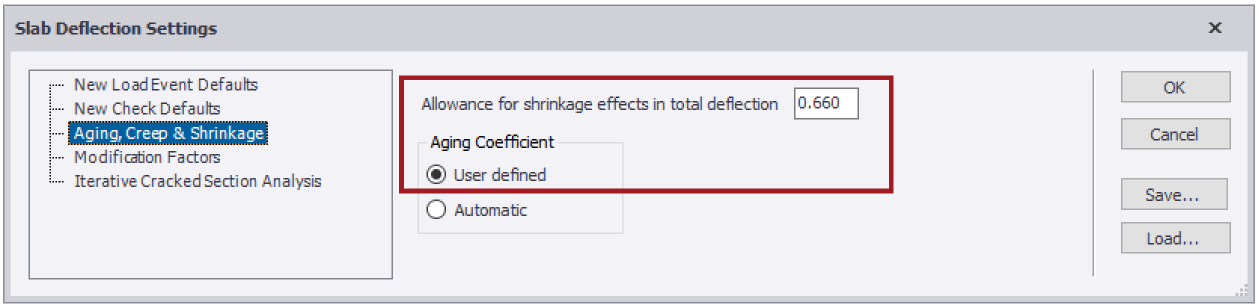
Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	$7.5 \sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	$7.5 \sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	$7.5 \sqrt{f_c'}$ $4 \sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	**$7.5 \sqrt{f_c'}$**	**1.0**	**2.0**		**3.0**

When using this approach:

- The automatic procedure using Technical Report 58 is not considered - this is ensured by setting the **Aging coefficient** to **User defined**.
- Creep is not also accounted for in the analysis - it can be excluded by setting the **Number of Exposed Faces** to **Zero** in the event sequence.

1. **To adopt the above ACI 318 Code multipliers for simplified creep and shrinkage:**

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.66**.



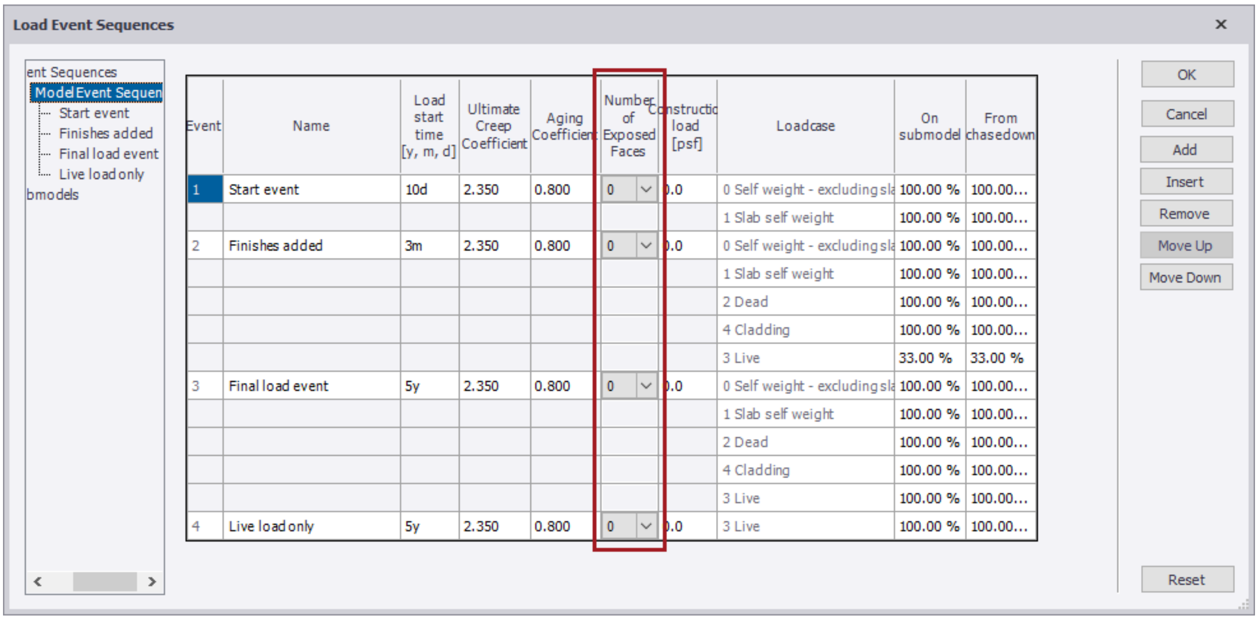
The above factor allows for both creep and shrinkage, (i.e. creep and shrinkage effects will account for 66% of the total deflection).

4. Click **OK** to close the dialog.

2. **To exclude additional creep effects from the analysis**

As creep is already being catered for by the amplification factor, we have to exclude additional creep effects from the analysis. This can be done by setting the number of exposed faces in each event to zero as follows:

5. From the **Slab Deflection** toolbar, click **Event Sequences**
6. Click **Model Event Sequence**
7. For each listed event in the model sequence, ensure the **Number of Exposed Faces** is set to **0**.



If the number of faces = 0 then the creep coefficient C_t = 0 . See: Aging Coefficient - User Defined or Automatic? (page 441)

Review the Individual Events in the Model Event Sequence

A simplified event sequence has been defined that does not include any construction stage propping events.

Various guidance documents discuss slab deflection analysis without addressing the possibility of significant early age loading events such as propping loads.

An early start event should always be created since time is measured from this start event. In this example we have assumed a start event of 10 days and allowed self weight only.

Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
Start event	10d	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
Finishes added	3m	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	33.00 %	33.00 %
Final load event	5y	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	100.00 %	100.00 %
Live load only	5y	2.350	0.800	0	0.0	3 Live	100.00 %	100.00 %

You will also note that:

- Each event has a load start time. The Final load event is set to the normal ACI requirement of 5 years.
- ACI requires that instantaneous deflection due to live loads only should be considered based upon a span/360 limit. To account for instantaneous deflection due to live load only, an end event has been included with the same load start time as the preceding event but only including live load.
- Ultimate Creep Coefficient, C_u is set with the default value of 2.350. This value can be set separately for each event.
- Aging Coefficient is set at the default value of 0.8. This value can be set separately for each event. It may be more logical to set higher values for the earlier event times, however, if your primary concern is differential deflection between later events then it will be conservative to use the same value everywhere.
- Number of exposed Faces is set to 0 - this has been done to exclude additional creep effects from the analysis, (as explained in the previous section).

Discussion:

- You need to think about the accumulation of deflection through time and hence the checks that you ultimately wish to consider.
- As an example, if you are interested in differential deflection between the final load event and the start of finishes (deflection at end of event 1), if you underestimate deflection to the end of event 1 then this check becomes more onerous.
- Is it reasonable to assume no construction load and no self weight from finishes during this period?
- How much load is reasonable to assume at this starting event is ultimately the responsibility of the engineer.

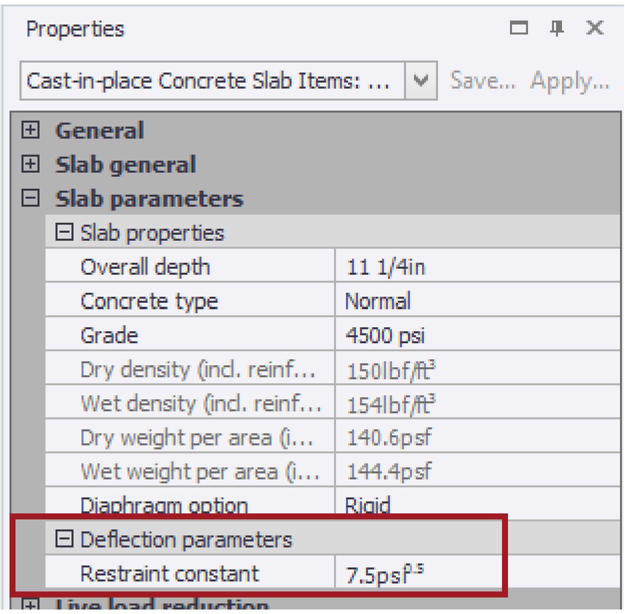
1. After reviewing the Event Sequence, click **OK** to close the dialog.

Set up the Restraint Constant

The modulus of rupture is set using the **Restraint Constant** slab deflection parameter. Since we are using the ACI code multipliers from Table 4.1 above this should be set to 7.5.

1. **To specify an appropriate restraint constant**

1. Open the **Structure** 3D view
2. Select all the slab items in the model and via the Properties Window, ensure the **Restraint Constant** is set to **7.5**



To establish some initial results:

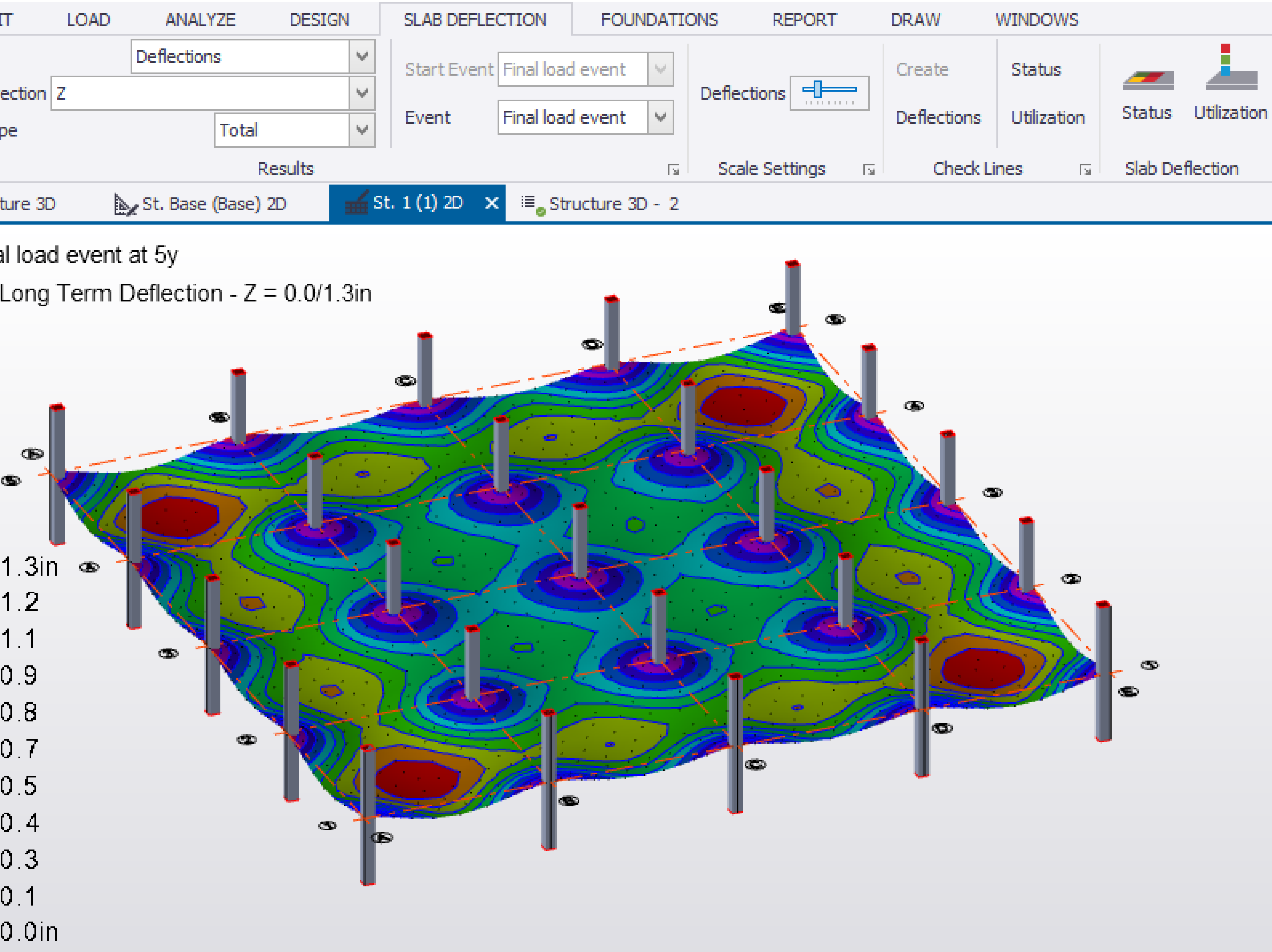
1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections for the events

   Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon. You are able to review:

   - Total deflection at the end of any event.
   - Differential deflection between any two events (Start of Event and End of Event).
   - Instantaneous deflection.

   The Total deflection at the final load event for the chosen location is 1.3”



TIP If slab patches are obscuring the above deflection contours, these can be switched in Scene Content.

Consider the Sustained Load Multiplier Effect from ACI 435

ACI 435 Clause 4.3.4.2 needs to be reviewed carefully at this stage, as this suggests that the above deflection estimation is unconservative.

- It states that if the restraint stresses are expected to be **insignificant**, (so that the restraint constant is set at 7.5) then:

- *"the multiplier for sustained-load deflection should be increased from 2 to 4, as recommended by Sbarounis (1984) and Graham and Scanlon [1986(b)]"*

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	**7.5 $\sqrt{f_c'}$**	**1.0**	**2.8**	**1.2**	**5.0**
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	**7.5 $\sqrt{f_c'}$** 4 $\sqrt{f_c'}$	**1.0** 1.0	**2.0** 1.5	**2.0** 1.0	**5.0** 3.5
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

Hence the combined creep and shrinkage contribution = 4.0/5.0 = **0.8** (i.e. 80% of the total deflection).

- Alternatively, if the restraint stresses are likely to have a **significant** effect then Clause 4.3.4.2 states that:
  - a reduced restraint constant of 4 can be used, along with a long-term sustained-load multiplier of 2.5.

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ **4 $\sqrt{f_c'}$**	1.0 1.0	2.0 **1.5**	2.0 **1.0**	5.0 **3.5**
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

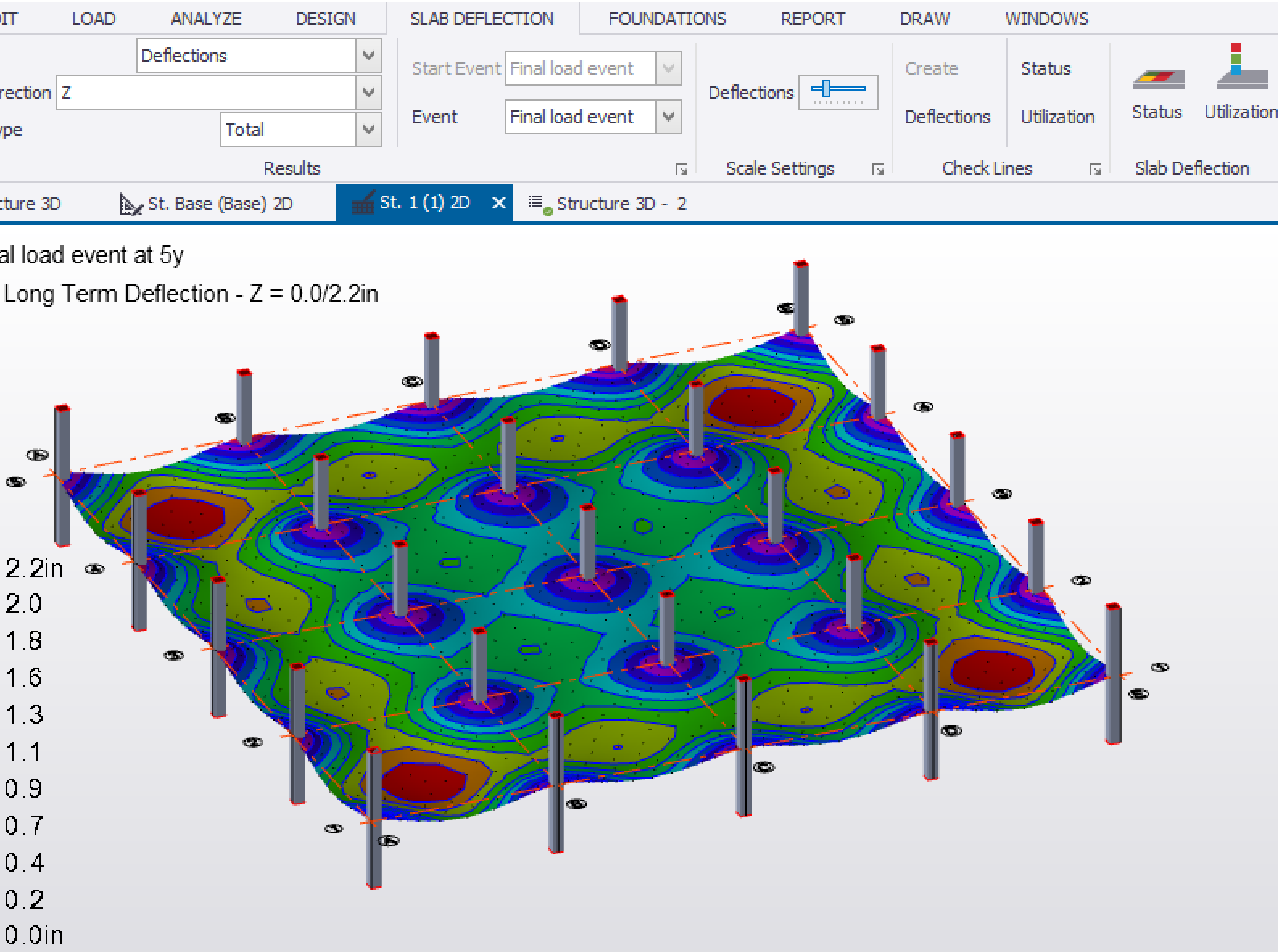
In this case the combined creep and shrinkage contribution should be reduced to 2.5/3.5 = **0.714** (i.e. 71.4% of the total deflection).

1. **To adopt the ACI 435 recommendation for insignificant restraint stresses:**

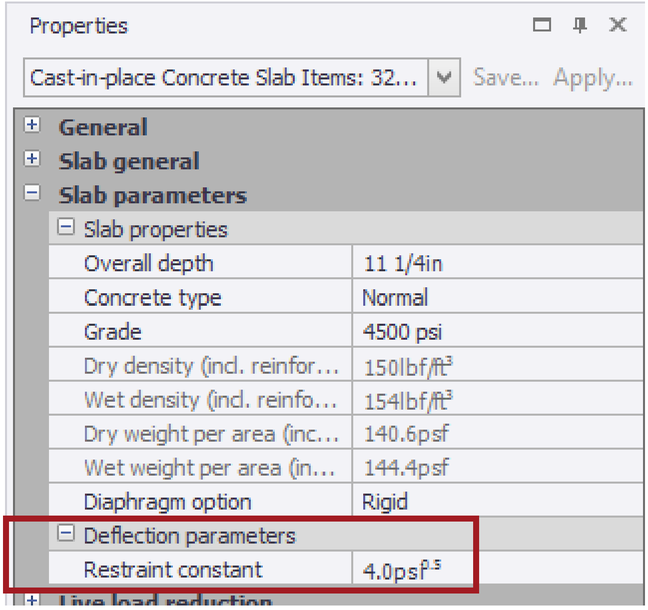
1. From the **Slab Deflection** toolbar, click **Settings**

2. In the dialog, click **Aging, Creep & Shrinkage**
3. Increase the **Allowance for shrinkage effects in total deflection** to **0.8** then click **OK**
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   Using this value, the revised deflection prediction increases to 2.2”

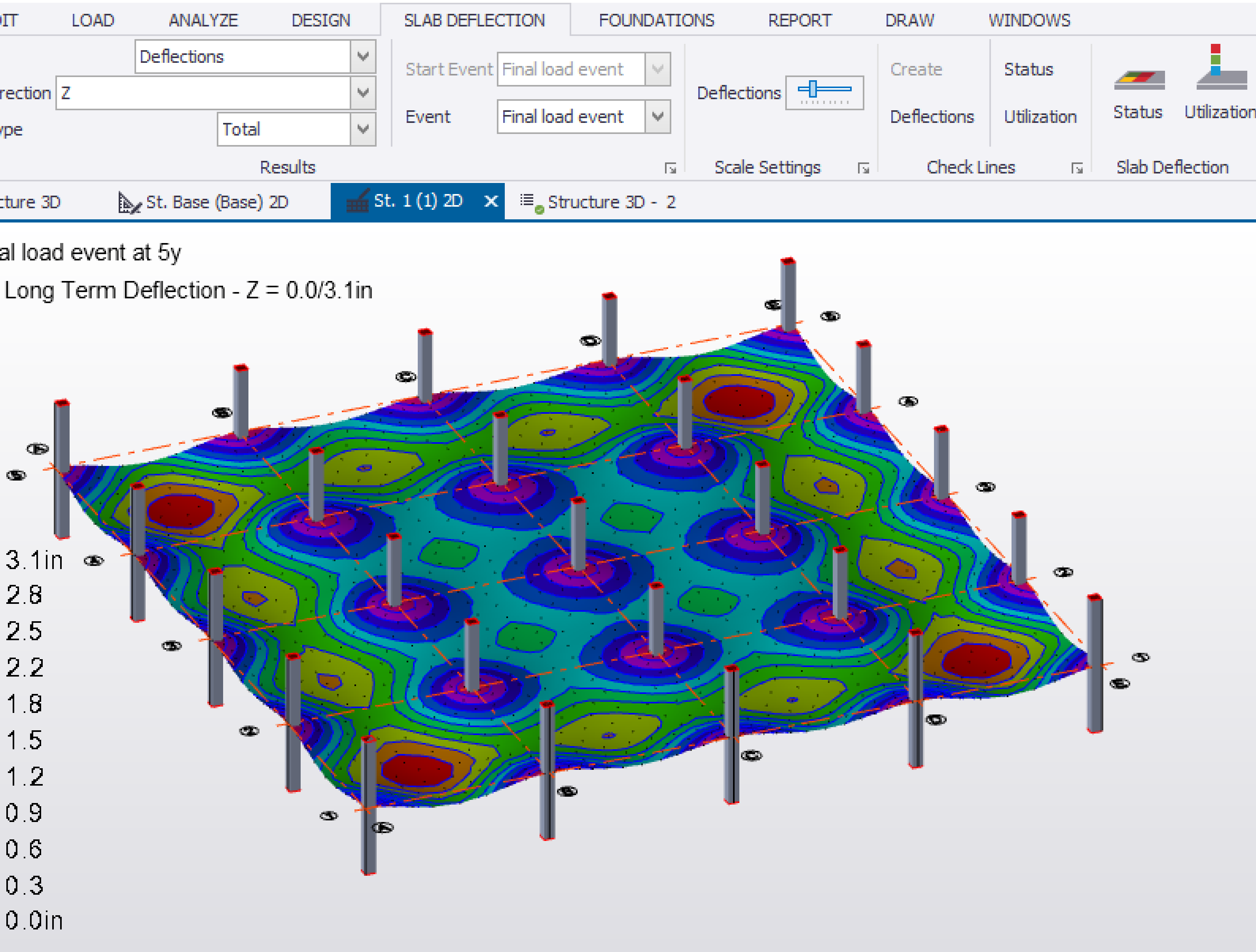


2. **To adopt the ACI 435 recommendation when restraint stresses are expected to be significant:**
5. Open the **Structure** 3D view
6. Select all the slabs in the model and via the Properties Window, adjust the **Restraint Constant** to **4.0**



7. From the **Slab Deflection** toolbar, click **Settings**
8. In the dialog, click **Aging, Creep & Shrinkage**
9. Change the **Allowance for shrinkage effects in total deflection** to **0.714** then click **OK**
10. Return to the **St.1 (1)** 2D view.
11. From the **Slab Deflection** toolbar, click **Analyze Current**

    Using this value, the revised deflection prediction increases further to 3.1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel > For the current slab item**

Composite Modulus Calculation				
Event	Start [d]	To end of Event Ultimate Creep Coefficient, c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.800	4000
2 Finishes added	91	2.350	0.800	4000
3 Final load event	1825	2.350	0.800	4000
4 Live load only	1825	2.350	0.800	4000

Slab_ S 3, Panel_ Sl 33

We can see from the report that the short term composite modulus, E_c is used.

Summary of Results

Using the Simplified event sequence + simplified combined creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Assumed Creep and Shrinkage combined allowance %	Total deflection (Final load event)
ACI 318 (ignoring ACI 435)	7.5	66%	1.3″
ACI 435 (simple approach - insignificant restraint)	7.5	80%	2.2″
ACI 435 (simple approach - significant restraint)	4.0	71.4%	3.1″

Next steps

- In Method 2 (page 455) we will re-use the same model, but edit the simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.
- In Method 3 (page 462) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 471) are discussed.

- Finally, using results from one of the methods, deflection checks are performed using check lines (page 476) and an output report is generated.

Method 2: Simplified Event Sequence + Rigorous Creep and Shrinkage Allowance

In Method 1 (page 443) we used a simplified event sequence (without addressing the possibility of early age loading), to see how a single multiplier can be applied to allow for the combined effects of creep and shrinkage.

In **Method 2** we will re-use the same model, but edit the previous simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.

Download and open the tutorial model (if required)

NOTE If you are reusing the model from the Method 1 (page 443) exercise you can skip this step.

---

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

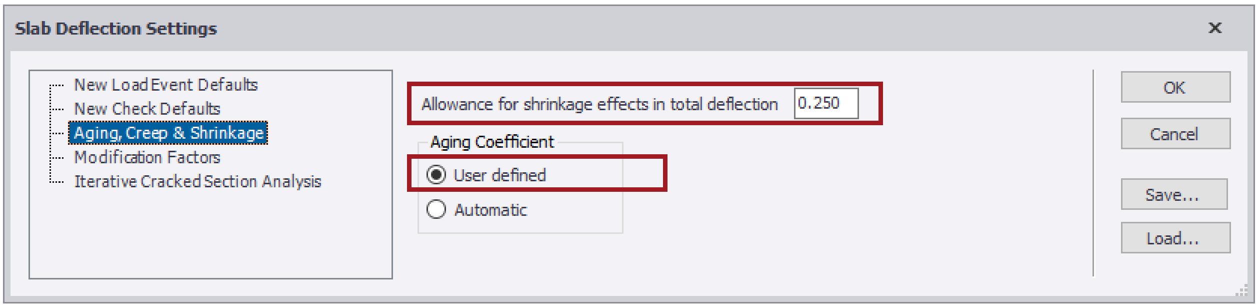
Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the rigorous creep and shrinkage allowance

In this method the **Allowance for Shrinkage Effects** multiplier is set to allow for shrinkage only.

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.25**.



The above factor allows for shrinkage only.

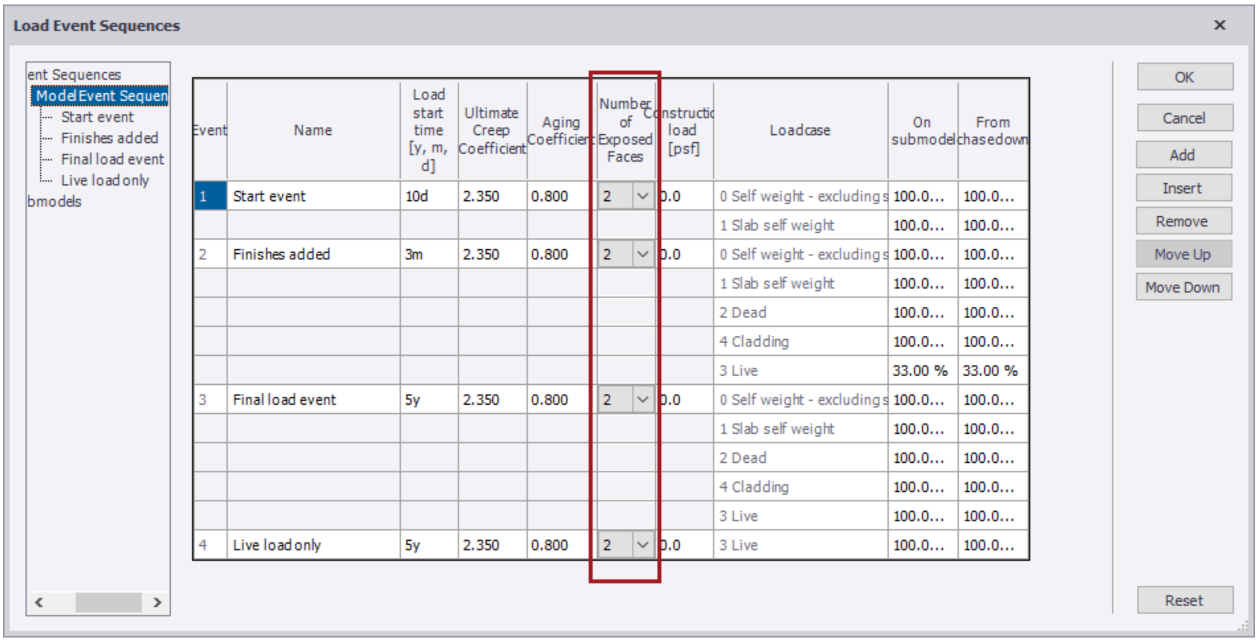
Refer to Shrinkage allowance (page 392) for an explanation of where this value comes from.

4. Click **OK** to close the dialog.

Edit the Event Sequence to ensure that creep is accounted for in the analysis

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**
3. Ensure that the **Number of Exposed Faces** is set to **2** for each event.

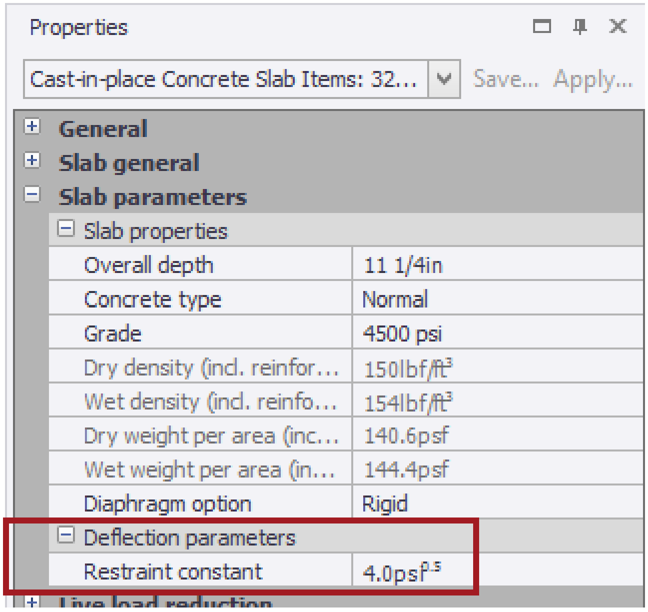
The creep coefficient C_t will be calculated based on this, thus affecting the modulus of elasticity used in the analysis. For further details of this calculation, see: Aging Coefficient - User Defined or Automatic? (page 441)



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

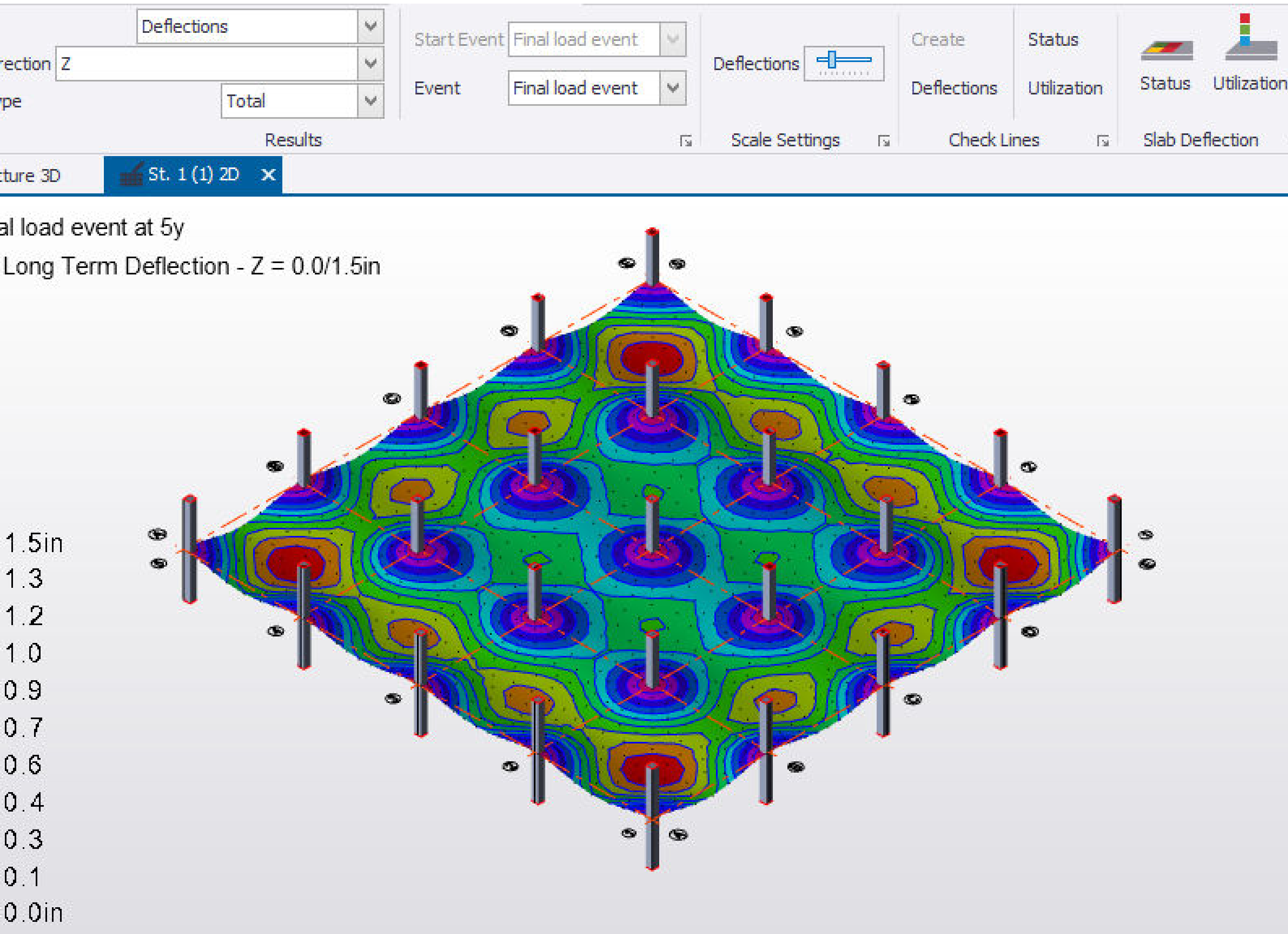


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

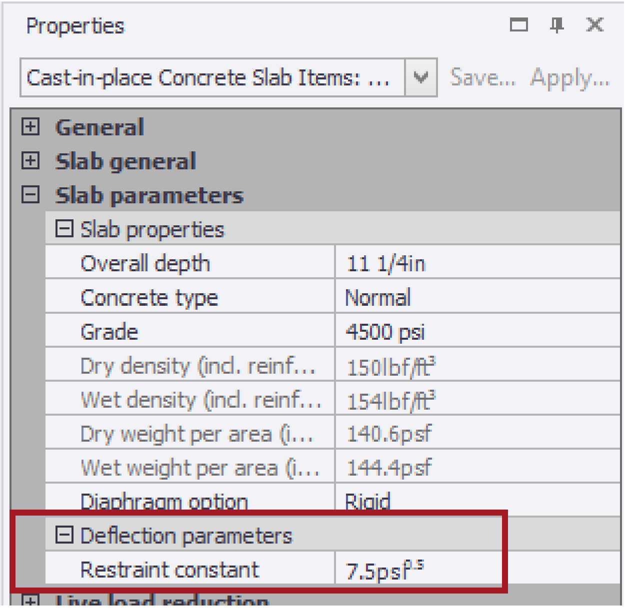
   The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



Adjust the Restraint Constant and Re-analyze

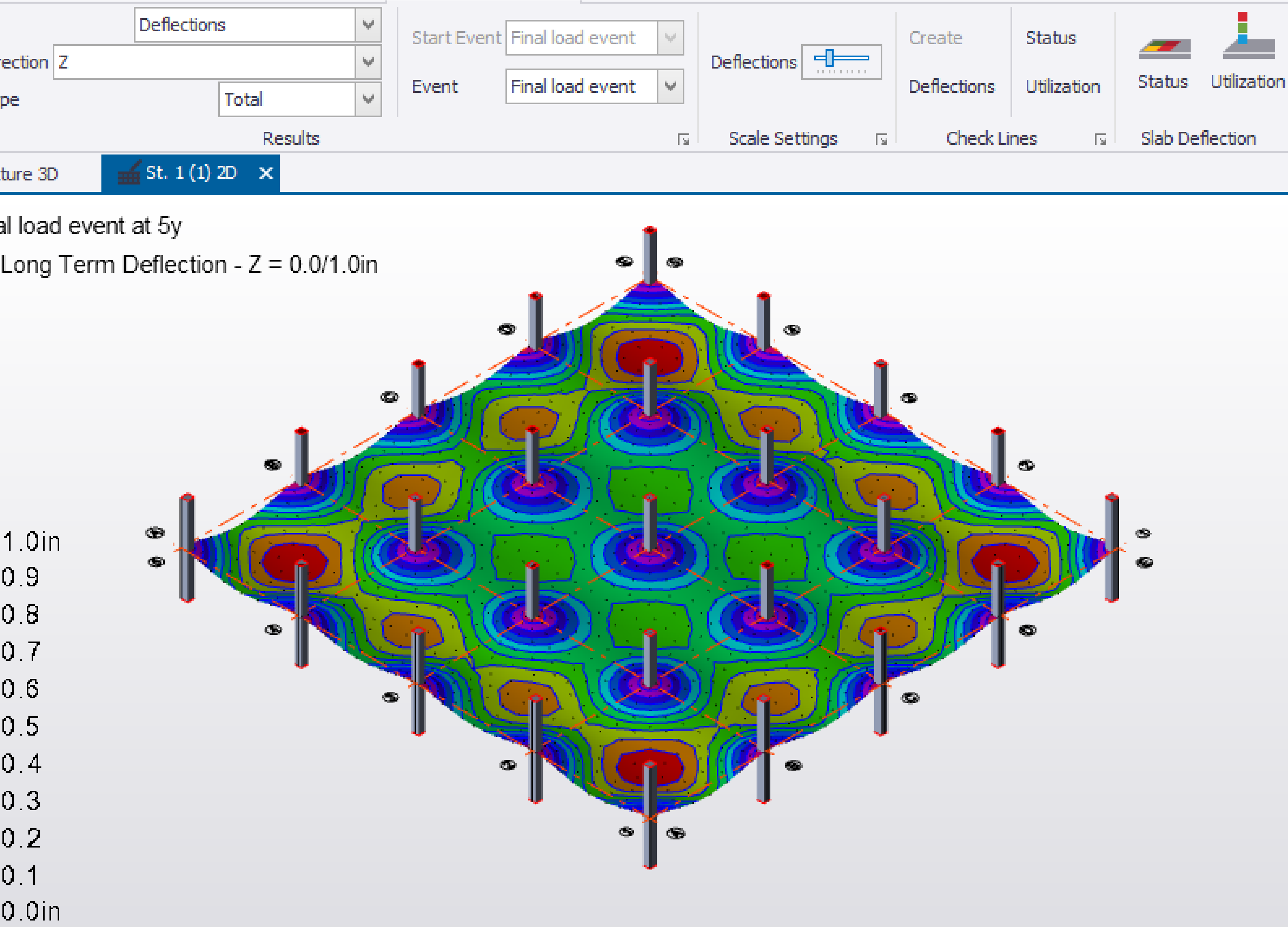
Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   With these settings the total deflection predicted at 5 years reduces to 1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**

Composite Modulus Calculation					
	Start	To end of Event			
Event	[d]	Ultimate Creep Coefficient, c_u	c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.684	0.800	2585
2 Finishes added	91	2.350	2.119	0.800	1484
3 Final load event	1825	2.350	2.119	0.800	1484
4 Live load only	1825	2.350	2.119	0.800	1484

Slab_ S 3, Panel_ Sl 33

Comparing this report to the report obtained when using the simplified creep and shrinkage allowance (page 453), we can see a difference in the value of the Composite Modulus E_c used. In this method, the age of concrete is taken into account.

To verify the result, consider Event 2, 3 and 4 where the event ends at 1825 days:

- Start event t_0 = 10 days, Event being considered, t_i = 1825 days
- Time between events (t_i-t_0) = 1825-10 = 1815 days
- Modulus of Elasticity, E for 4500 psi concrete grade = 4000 ksi (from Material database)
- Assumed Ultimate Creep Coefficient, C_u = 2.35
- Assumed Aging coefficient χ = 0.8. This is typically in the range 0.7 to 0.9
- Slab depth, d = 11 ¼ in
- Number of exposed faces, h = 2.

  $C_t = [(t_i-t_0) / (26\ e^{(0.36 \times d/h)}) + (t_i-t_0)] \times C_{ui} = [1815 / (26\ e^{(0.36 \times 11\ ¼ / 2)} + 1815] \times 2.35 = 2.1199$

  Thus

  $\bar{E}_c = (t, t_0) = E_c\ (t, t_0) / (1 + \chi\ Ct)$

  For event 2, 3 and 4, Composite Modulus, E_c = 4000 / (1 + 0.8 x 2.1199) = 1484 ksi

Summary of Results

Using the Simplified event sequence plus rigorous creep plus shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = 0.8	25%	1.5"
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = 0.8	25%	1.0"

Next steps

- In Method 3 (page 462) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 471) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 476) and an output report is generated.

Method 3: Detailed Event Sequence + Rigorous Creep and Shrinkage Allowance

In this approach, a detailed event sequence has been defined which includes propping loads at early events to create an overall load history. Creep is considered rigorously alongside this load history by setting the Aging coefficient to Automatic. This means that an effective composite modulus value to the end of each event is calculated in accordance with the Concrete Society Technical Report 58.

NOTE The aging coefficient method should only be set to Automatic when defining a detailed event sequence that includes a realistic assessment of early propping load events.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
    - Slab deflection ACI - Detailed Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Detailed Event Sequence

1. **To display the Event Sequence**

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

   A detailed event sequence has already been created:

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Number of Exposed Faces	Constructio load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	160.00 %	0.00 %
3	Propping slab 2 above	30d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	140.00 %	100.00 %
4	Propping removed	1m 9d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	50.00 %	50.00 %
5	Sensitive Finishes added	4m	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	33.00 %	33.00 %
6	Final load event	5y	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	100.00 %	100.00 %
						4 Cladding	100.00 %	100.00 %
						3 Live	100.00 %	100.00 %
7	Live load only	5y	2.350	2	0.0	3 Live	100.00 %	100.00 %

☑ Update custom event sequences

You will note that:

- Ultimate Creep Coefficient, C_u set at **2.35** for this example
- Number of exposed faces, h set at **2** for this example
- Construction load assumed set at **10.4 psf** in this example
- Propping load % assumed during early age propping events **160%** and **140%** assumed.

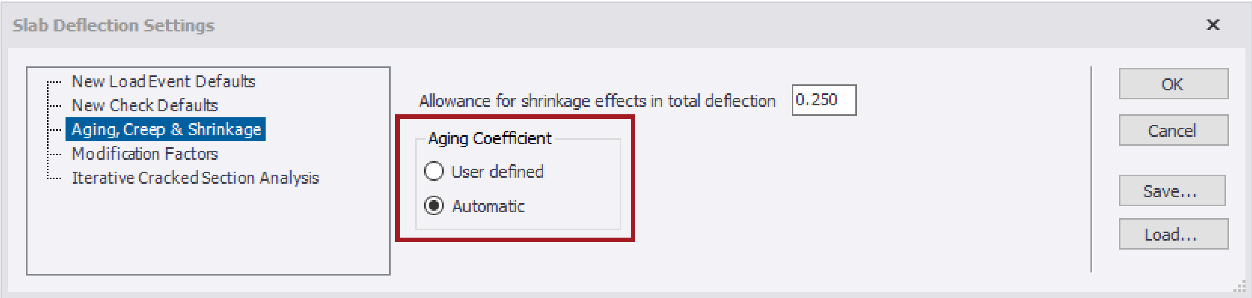
Set up Rigorous Creep

This method requires that the **Aging coefficient** is set to Automatic. Creep is then considered automatically as part of the analysis for each event sequence.

A user defined allowance for Shrinkage effects in the total deflection is also specified.

1. **To specify the multiplier for shrinkage effects only but also consider creep in the analysis**

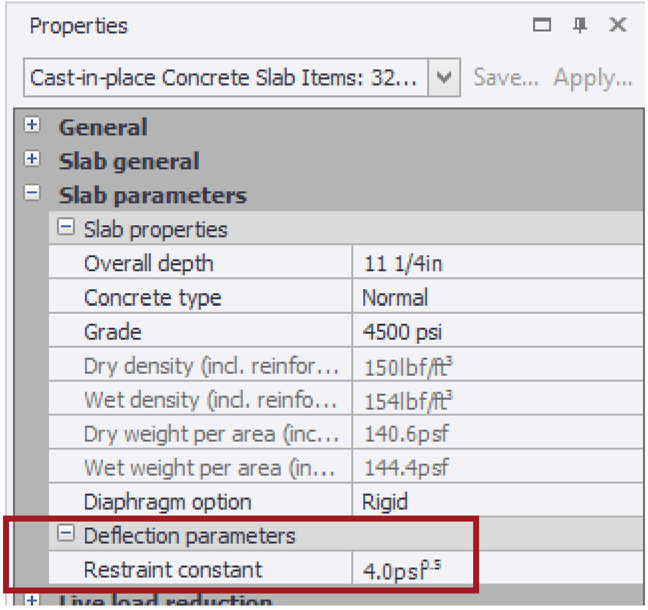
1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Allowance for shrinkage effects in total deflection** is set to **0.25**, and the **Aging Coefficient** is set to **Automatic**



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

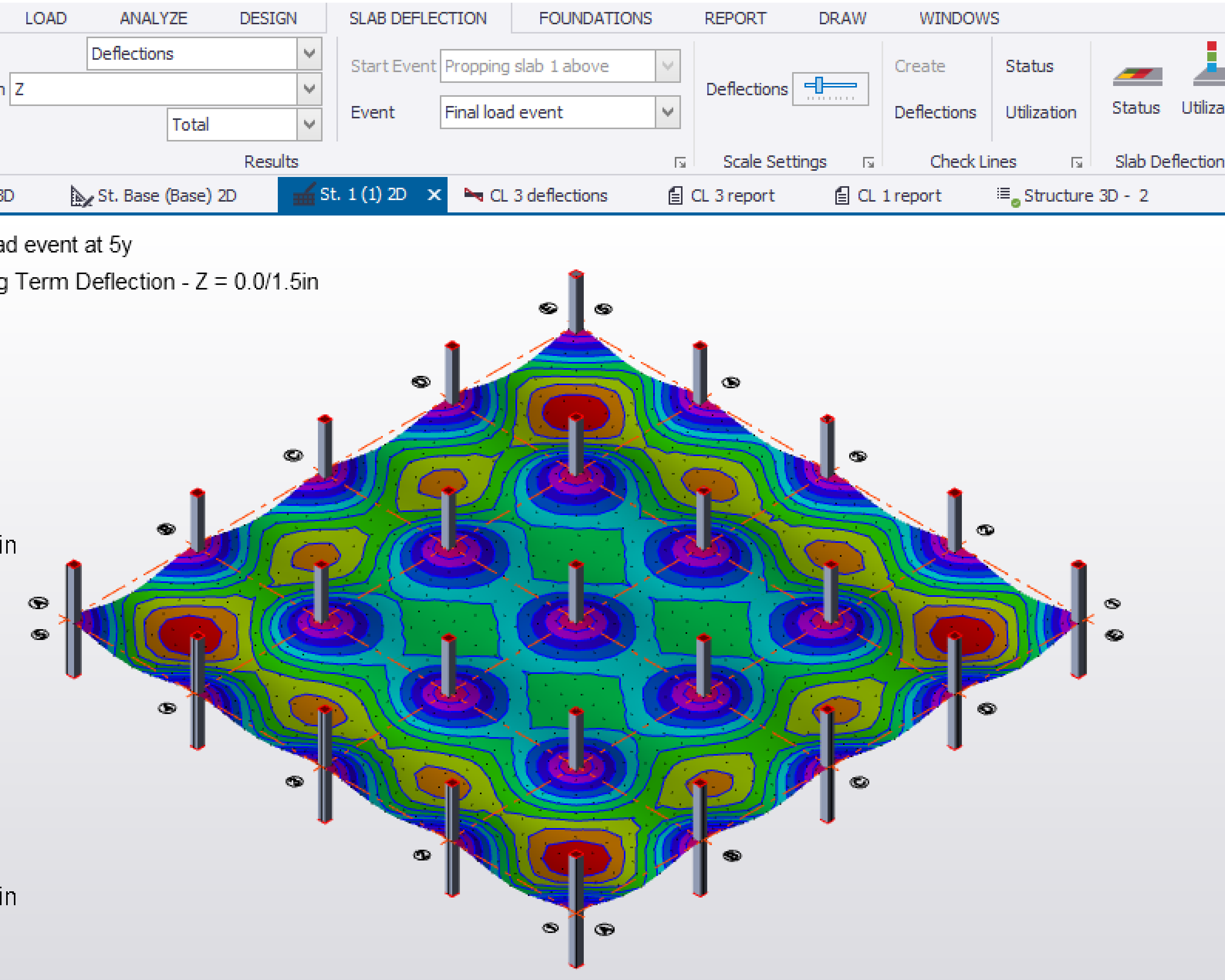


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

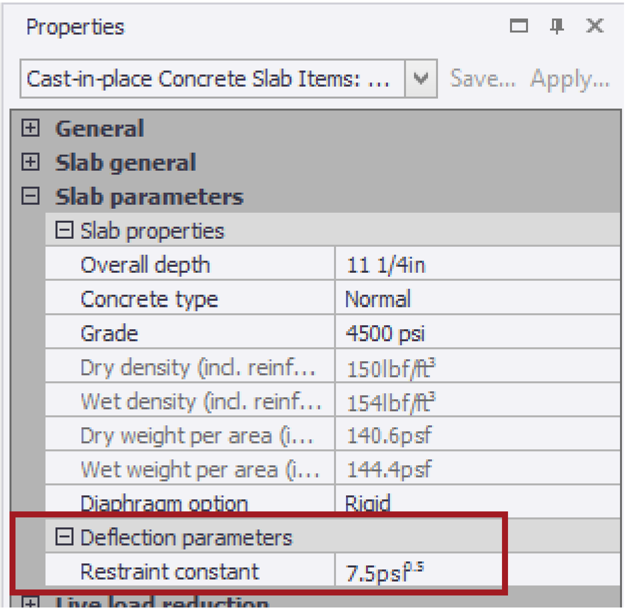
   The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



Adjust the Restraint Constant and Re-analyze

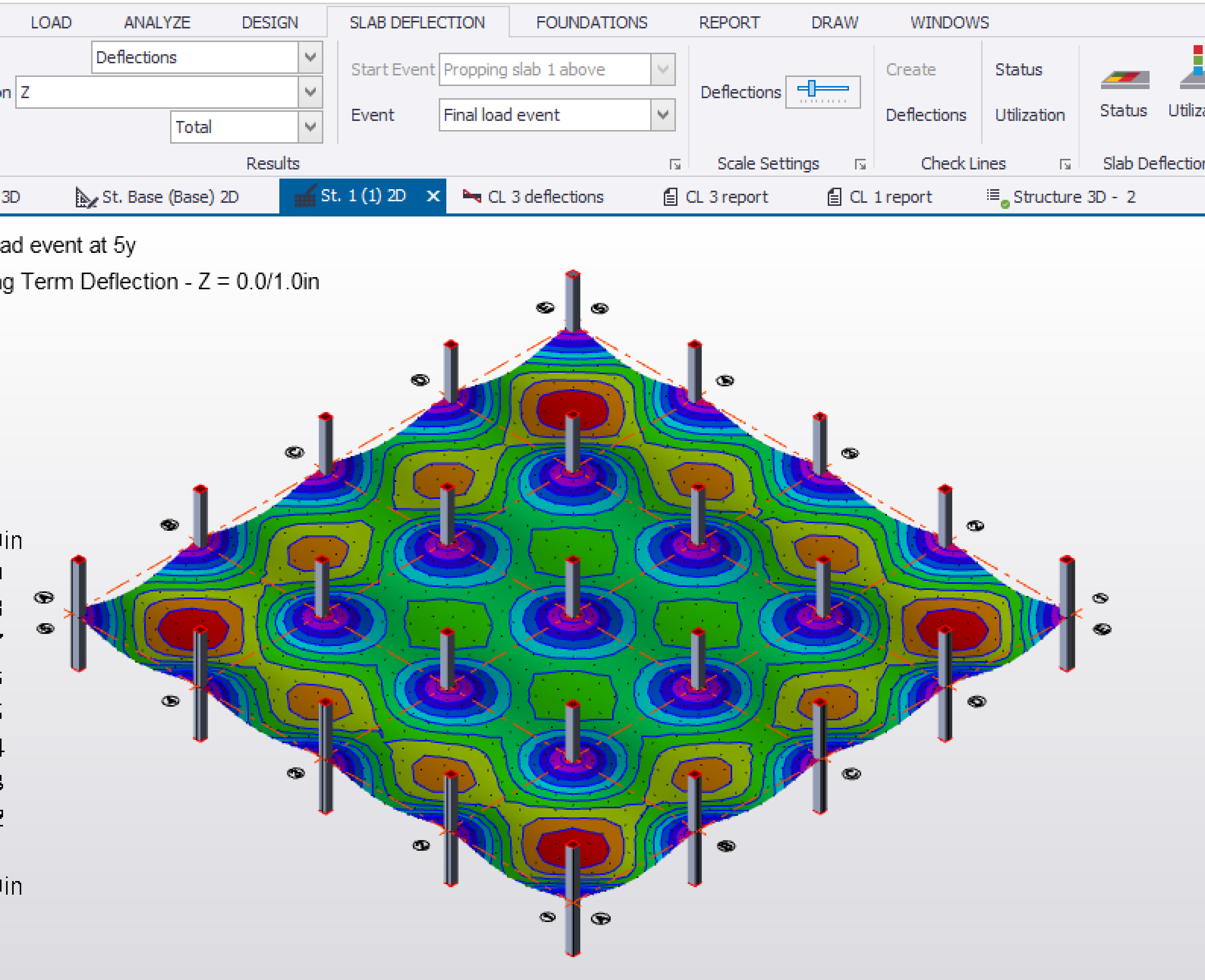
Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   With these settings the total deflection predicted at 5 years reduces to 1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.

2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**

Composite Modulus Calculation

Event	Start [d]	Incremental load factor, λ	To end of Event 1			To end of Event 2			To end of Event 3			To end of Event 4			To end of Event 5			To end of Event 6			To end of Event 7		
			φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	φ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]
1 Strike and backprop slab	10	6.735	0.113	3593	0.00187	0.216	3290	0.00205	0.304	3067	0.00220	0.848	2164	0.00311	2.119	1282	0.00525	2.119	1282	0.00525	2.119	1282	0.00525
2 Propping slab 1 above	20	4.539	-	-	-	0.113	3593	0.00126	0.210	3305	0.00137	0.798	2225	0.00204	2.118	1283	0.00354	2.118	1283	0.00354	2.118	1283	0.00354
3 Propping slab 2 above	30	-1.349	-	-	-	-	-	-	0.107	3614	-0.00037	0.744	2293	-0.00059	2.117	1283	-0.00105	2.117	1283	-0.00105	2.117	1283	-0.00105
4 Propping removed	39	-1.976	-	-	-	-	-	-	-	-	-	0.690	2366	-0.00083	2.116	1284	-0.00154	2.116	1284	-0.00154	2.116	1284	-0.00154
5 Sensitive Finishes added	122	2.831	-	-	-	-	-	-	-	-	-	-	-	-	2.106	1288	0.00220	2.106	1288	0.00220	2.106	1288	0.00220
6 Final load event	1825	3.367	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	0.000	4000	0.00084	0.000	4000	0.00084
7 Live load only	1825	-9.120	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	0.000	4000	-0.00228
Total of λ / $E_{c,eff}$ [1 / ksi]					0.00187			0.00331			0.00320			0.00373			0.00840			0.00924			0.00696
E_c to end of Event [ksi]					3593			3406			3105			2132			1284			1531			722

Slab_ S 3, Panel_ SI 33

Comparing this report to the report for the more simplified ACI "rigorous" approach to creep (page 460) undertaken earlier.

- ACI composite modulus calculation for the Final Load Event = 1484 ksi (also verified by hand calculation earlier)
- TR58 composite modulus calculation for the Final Load Event = 1531 ksi
- These values compare favorably.

Note, however, that if we compared event 5 above where the composite modulus, E_C = 1284 ksi using the ACI method, we can establish the aging coefficient as:

$E_C = E / (1 + X * Ct)$

$1284 = 4000 / (1 + \chi \times 2.1199)$

$\chi = [(4000 / 1284) - 1] / 2.1199 = 0.9978$

This increased aging coefficient occurs due to the significant propping loads at early events, which creates an overall load history that is almost equivalent to constant loading.

Summary of Results

Using the Detailed event sequence plus rigorous creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic	25%	1.5"

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
		based on TR58		
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58	25%	1.0”

Next steps

- Having obtained results for all three methods, observations on the different methods (page 471) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 476) and an output report is generated.

Observations on the Different Methods

Combined Table of Results

Approach		Restraint Constant for Modulus of Rupture	Creep approach	Assumed Creep and Shrinkage combined allowance %	Assumed Shrinkage %	Total deflection (Final load event)
Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance	ACI 318 (ignoring ACI 435)	7.5		66%		1.3”
	ACI 435 (simple approach - insignificant restraint)	7.5		80%		2.2”
	ACI 435 (simple approach -	4.0		71.4%		3.1”

Approach		**Restraint Constant for Modulus of Rupture**	**Creep approach**	**Assumed Creep and Shrinkage combined allowance %**	**Assumed Shrinkage %**	**Total deflection (Final load event)**
	significant restraint)					
Method 2: Simplified Event Sequence + Rigorous Creep + Shrinkage Allowance	**ACI 435 (creep included in analysis)**	**4.0**	**C_u = 2.35, h = 2 and χ = 0.8**		**25%**	**1.5”**
		7.5	C_u = 2.35, h = 2 and χ = 0.8		25%	1.0”
Method 3: Detailed Event Sequence + Rigorous Creep + Shrinkage Allowance	ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.5”
		7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.0”

Discussion

When the guidance in ACI 435 about the use of a combined allowance for creep and shrinkage is taken into account, the simplified approach using a combined creep and shrinkage multiplier seems to determine very conservative deflection estimations.

Looking at table 4.1 in ACI 435, it is clear that the creep contribution is the most significant part of the overall deflection estimate. It is also the area where there seems to be greatest variation in opinion on the contribution level.

The creep contribution can be dealt with more rigorously by including it in the analysis. So rather than using a short term E value and then amplifying the result to account for creep, the analysis at the end of each event uses an effective E value which includes for creep up to that point. This impacts on the analysis properties of every shell in the FE analysis and even on the extent of cracking.

It is suggested that when the creep is included in the analysis the use of a modulus of rupture which assumes low restraint (restraint constant = 7.5) be used cautiously.

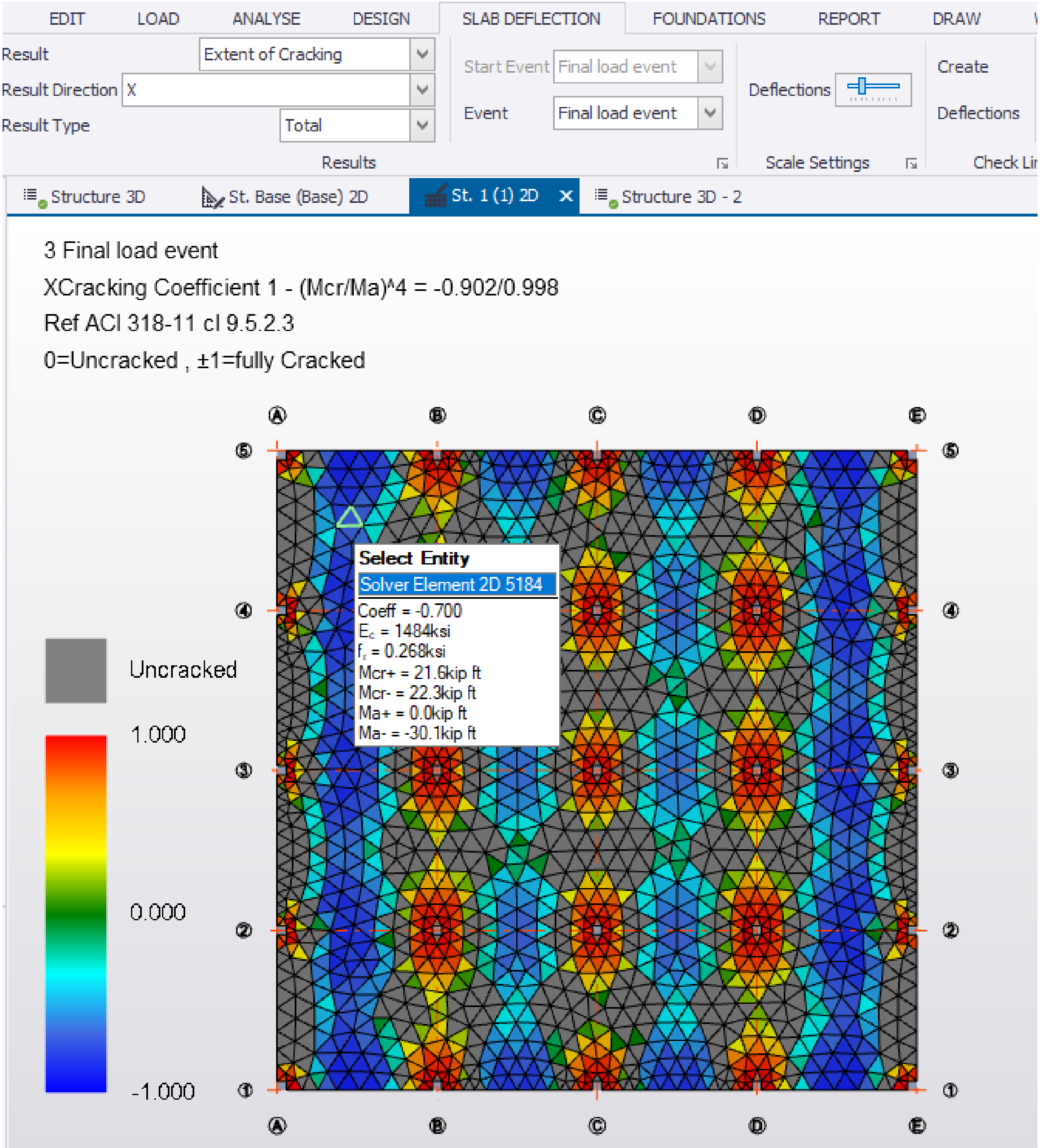
The option to calculate the effective E value based on some UK guidance in TR58 makes little difference in this example. It should be borne in mind that the aging coefficient χ is user defined and 0.8 is just a default for "normal" situations. In situations such as a transfer slab where load will accrue more slowly over time 0.8 is likely to be conservative and it may be of interest to consider the TR58 alternative.

Overall, it is expected that most engineers would use the settings highlighted in bold in the table above as a default starting point and then make refinements if necessary.

Extent of Cracking

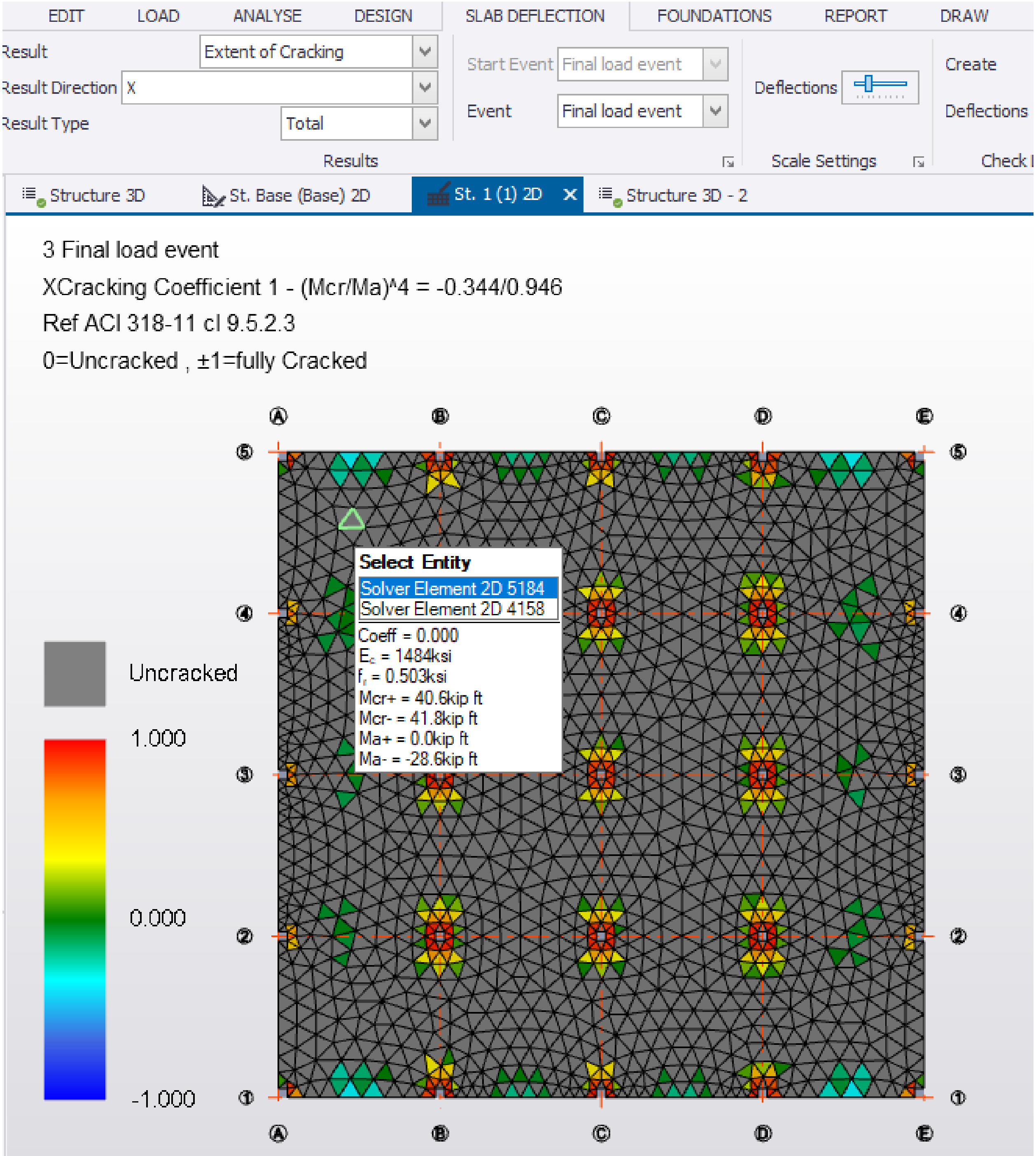
It is clear from the **Combined Table of Results**, that the Restraint Constant and hence the Modulus of Rupture has a huge impact on the predicted deflections. This is due to the impact the modulus of rupture has on the extent of cracking that develops. This is illustrated below where the extent of cracking for the final load event is displayed for the Method 2 model.

Extent of cracking resulting from a restraint constant of 4.0



You can clearly see the majority of the slabs have cracked where the restraint constant is 4.0.

Extent of cracking resulting from a restraint constant of 7.5



The majority of the slabs remain uncracked where the restraint constant is 7.5.

Next steps

- Using results from one of the methods, deflection checks are performed using check lines (page 476) and an output report is generated.

Use of check lines to check deflections (ACI)

Once the input parameters have been set as required to allow for creep and shrinkage, and the analysis has been performed, check lines can then be placed in order to check the deflections.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Check Lines.tsmd

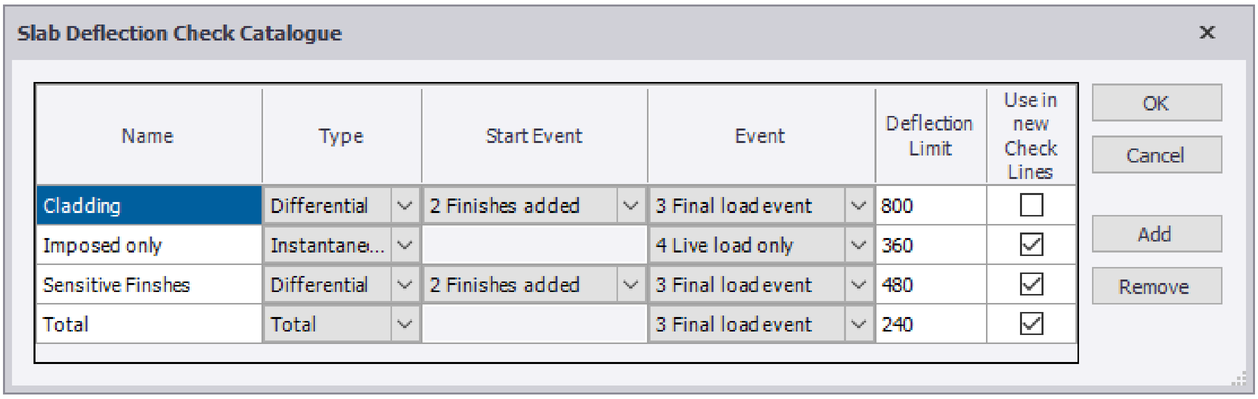
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the Slab Deflection ribbon, click **Deflection Checks**



Whilst four checks have been defined above, only three of these have been set to be used in new Check Lines. The more onerous cladding check which does not come from code requirements is not applied by default:

- **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/480
- **Total** will check the total deflections to the final load event against a deflection limit of 1/240

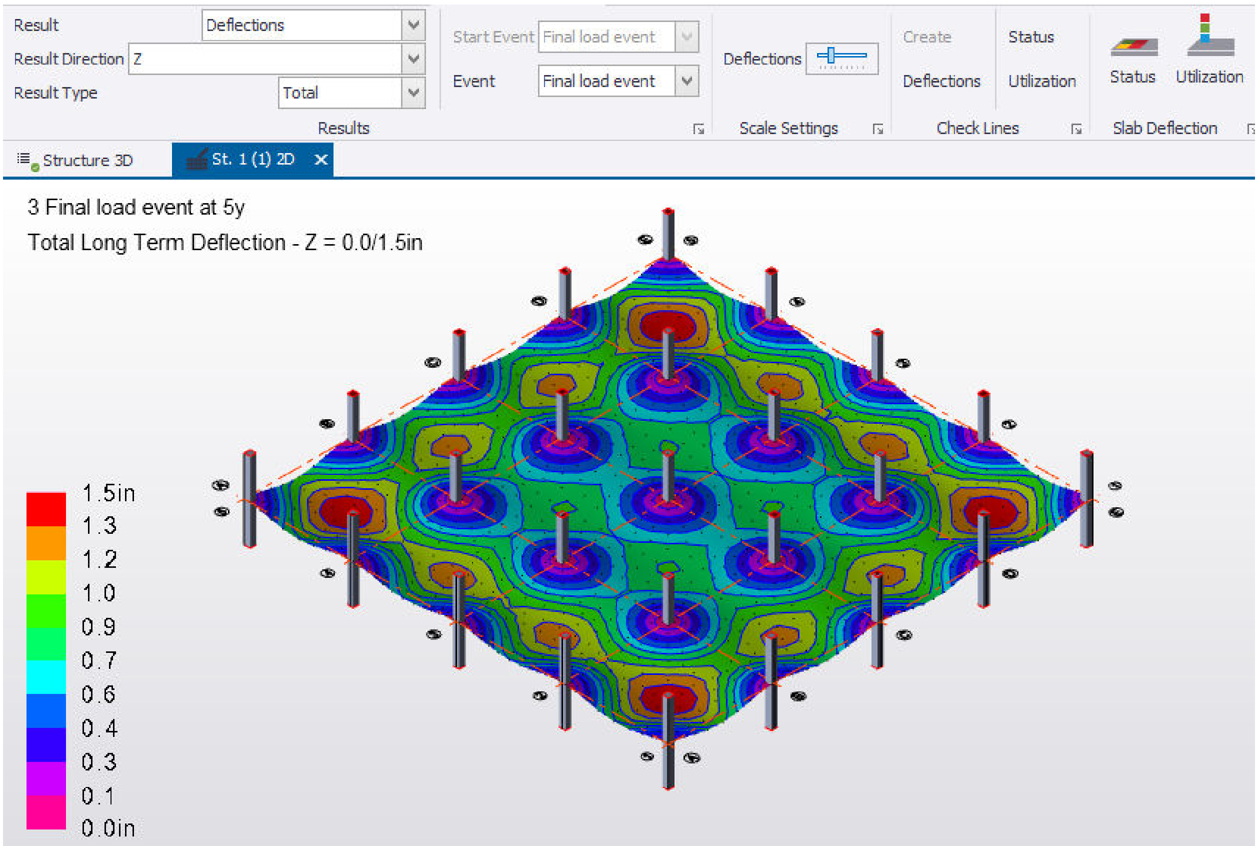
- **Live load only** will check the instantaneous deflections for the live load only event against a deflection limit of 1/360

2. Click **OK** to close the dialog.

Place Check Lines

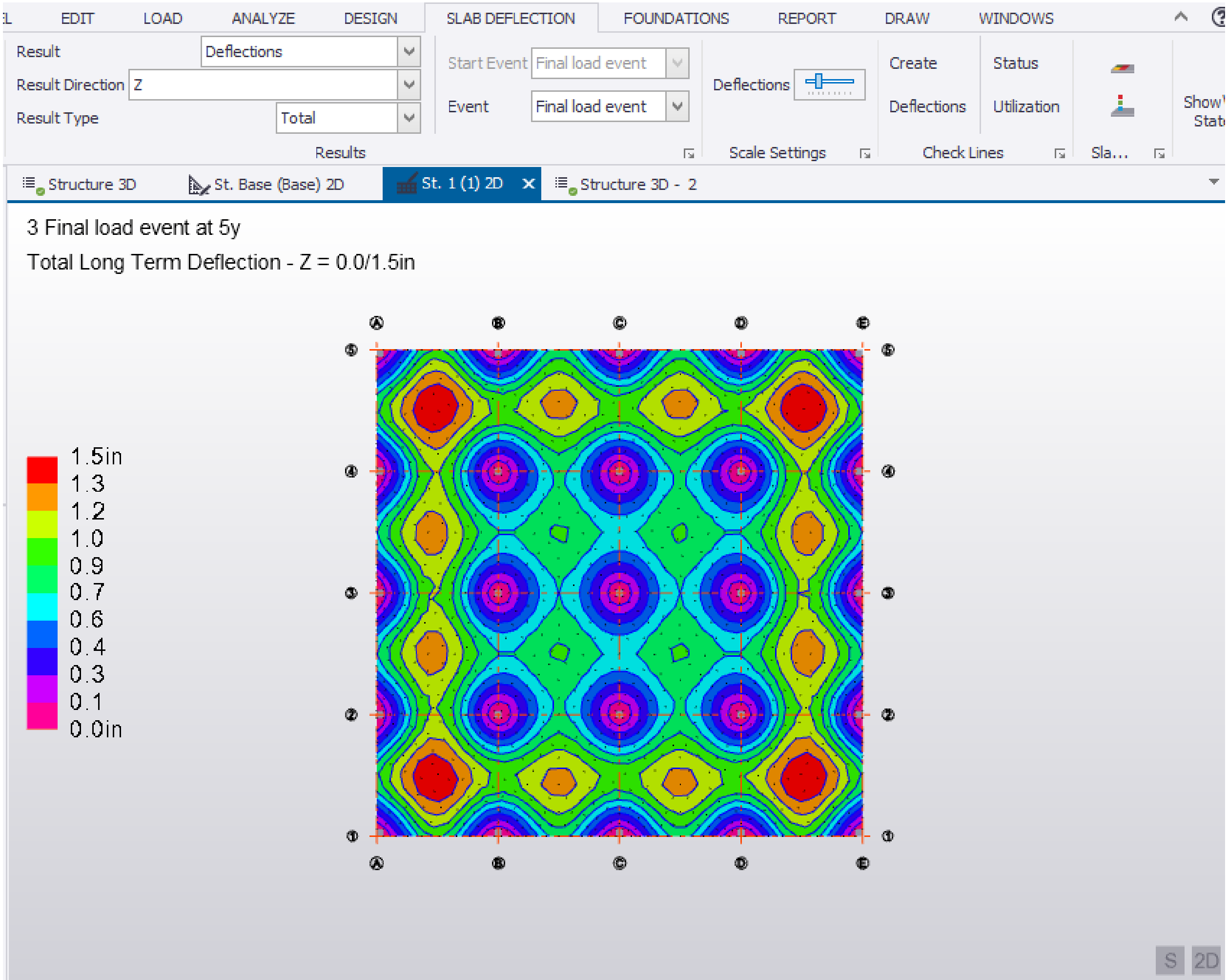
We will define two check lines in this example:

1. Open the **St.1 (1)** 2D plan view



NOTE When the 2D plan view is being viewed in 3D as above, the **Create** button on the **Slab Deflection** toolbar is grayed out.

2. Click 3D in the bottom right corner of the view to toggle the view until it is displayed in 2D.

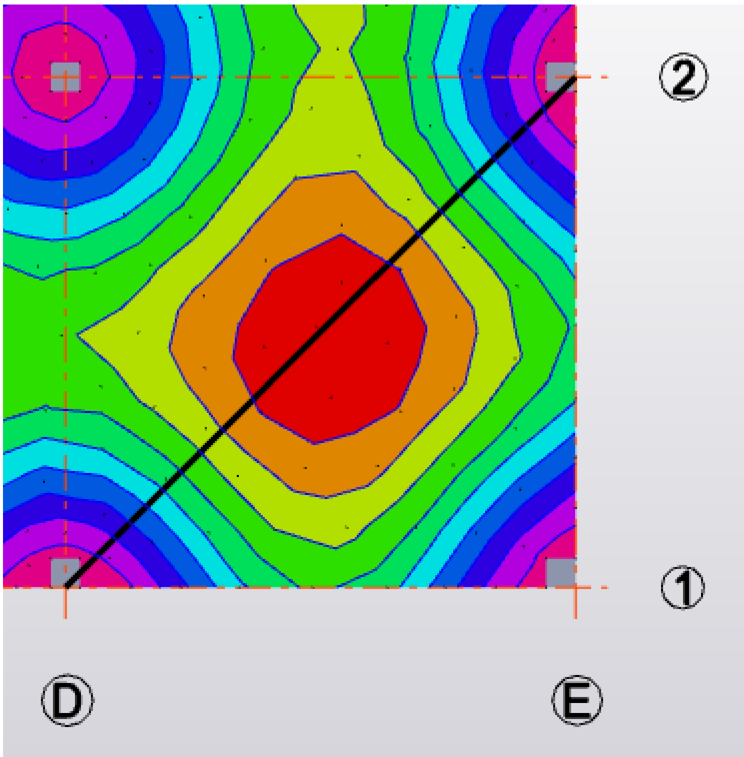


3. From the **Slab Deflection** toolbar, ensure the Results are set as above:
   a. Result - **Deflections**,
   b. Result Direction - **Z**,
   c. Result Type - **Total**,
   d. Event - **Final load event**
4. Click **Create**

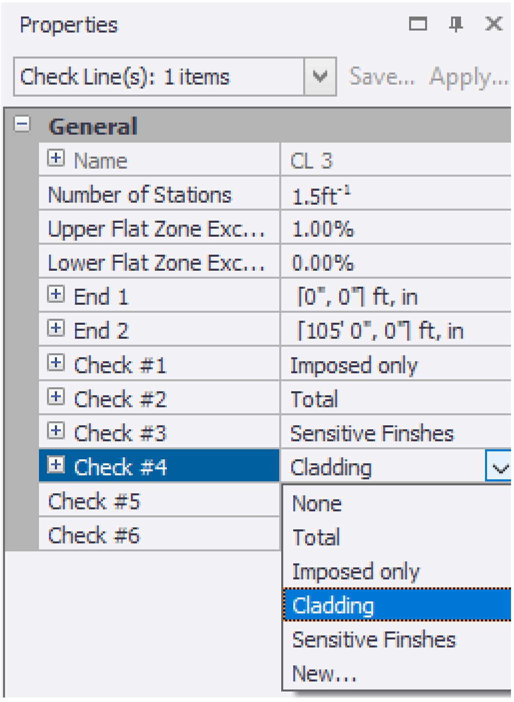
NOTE Check lines can only be created in a 2D view.

NOTE When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which “Use in new Check Lines” was checked.

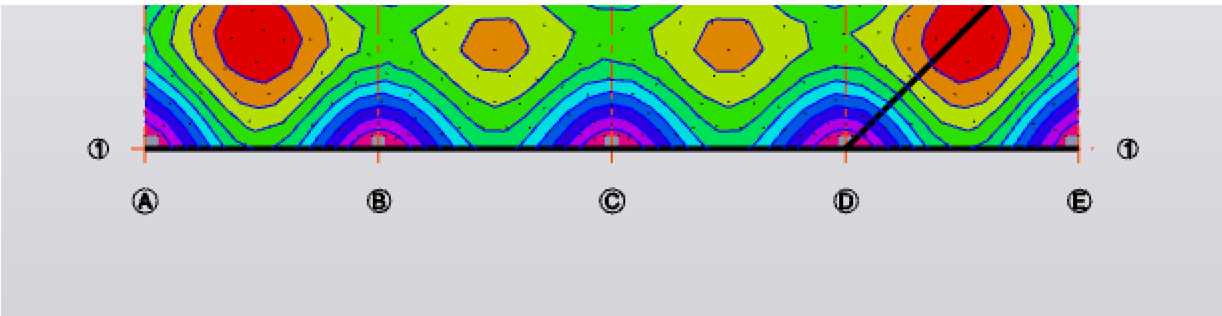
5. To place a check line running diagonally between columns in bottom right corner panel where the peak deflection occurs:
    a. Pick the start point as grid line intersection D/1.
    b. Pick the end point as grid line intersection E/2.



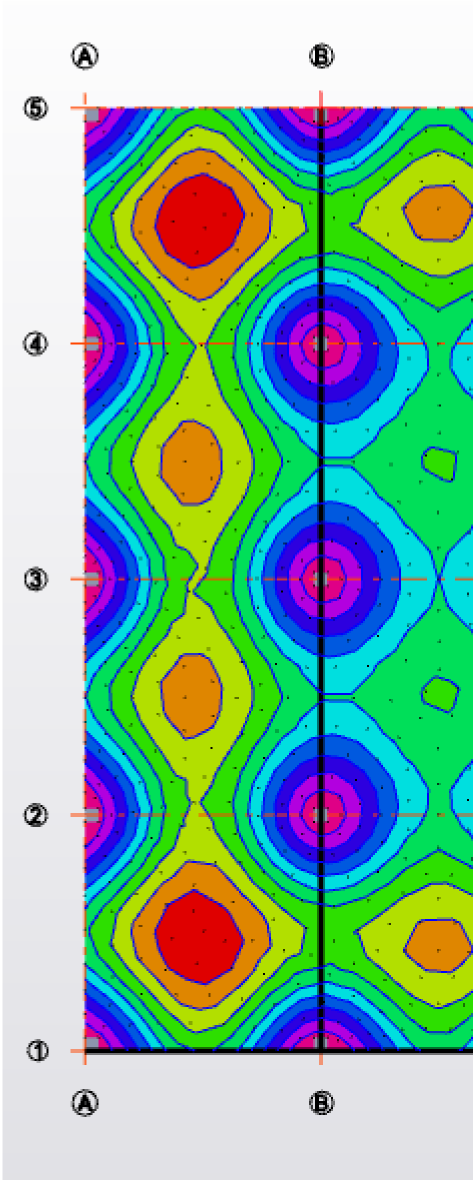
Before creating a second check line, in the Properties Window add a check #3, by selecting Cladding from the droplist.



6. Create the second check line along grid line 1.
   a. Pick the start point as grid line intersection A/1.
   b. Pick the end point as grid line intersection E/1.

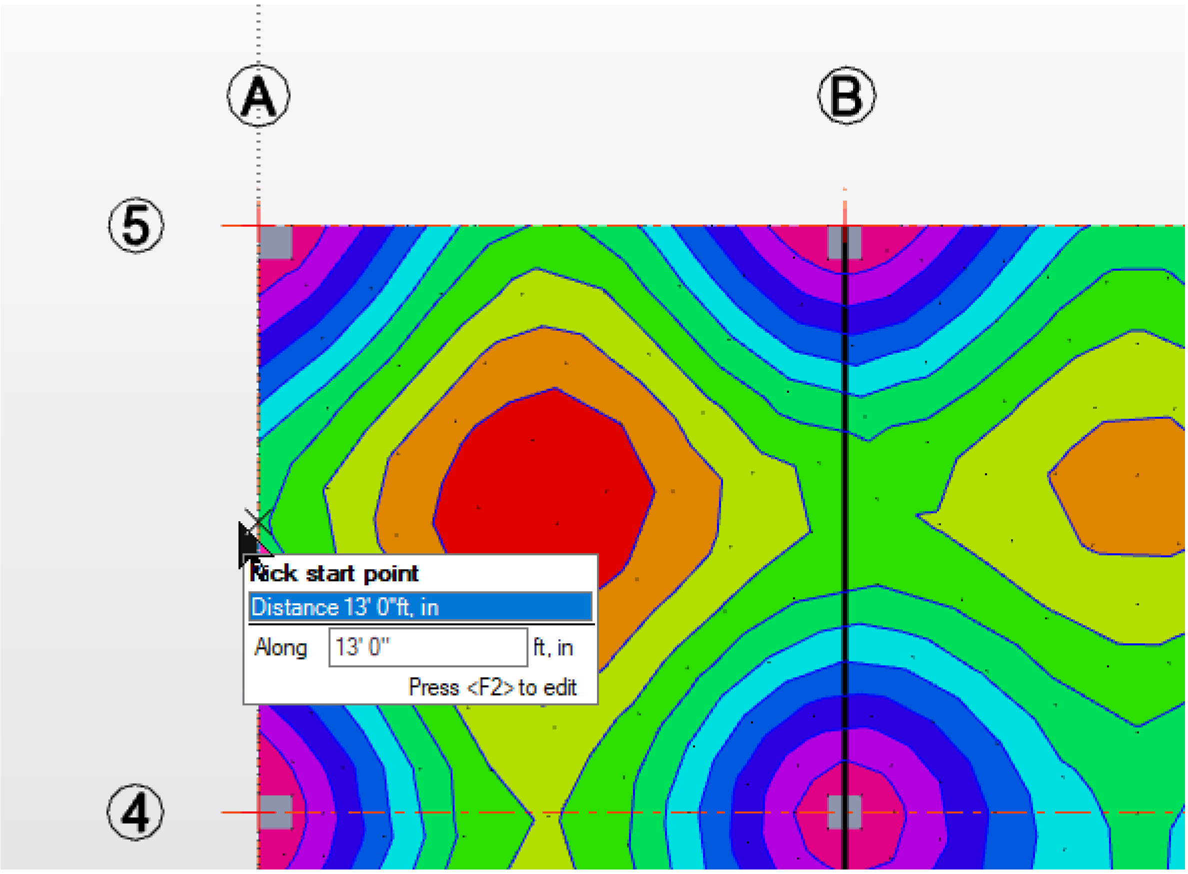


7. Create the third check line along grid line B.
   a. Pick the start point as grid line intersection B/1.
   b. Pick the end point as grid line intersection B/5.

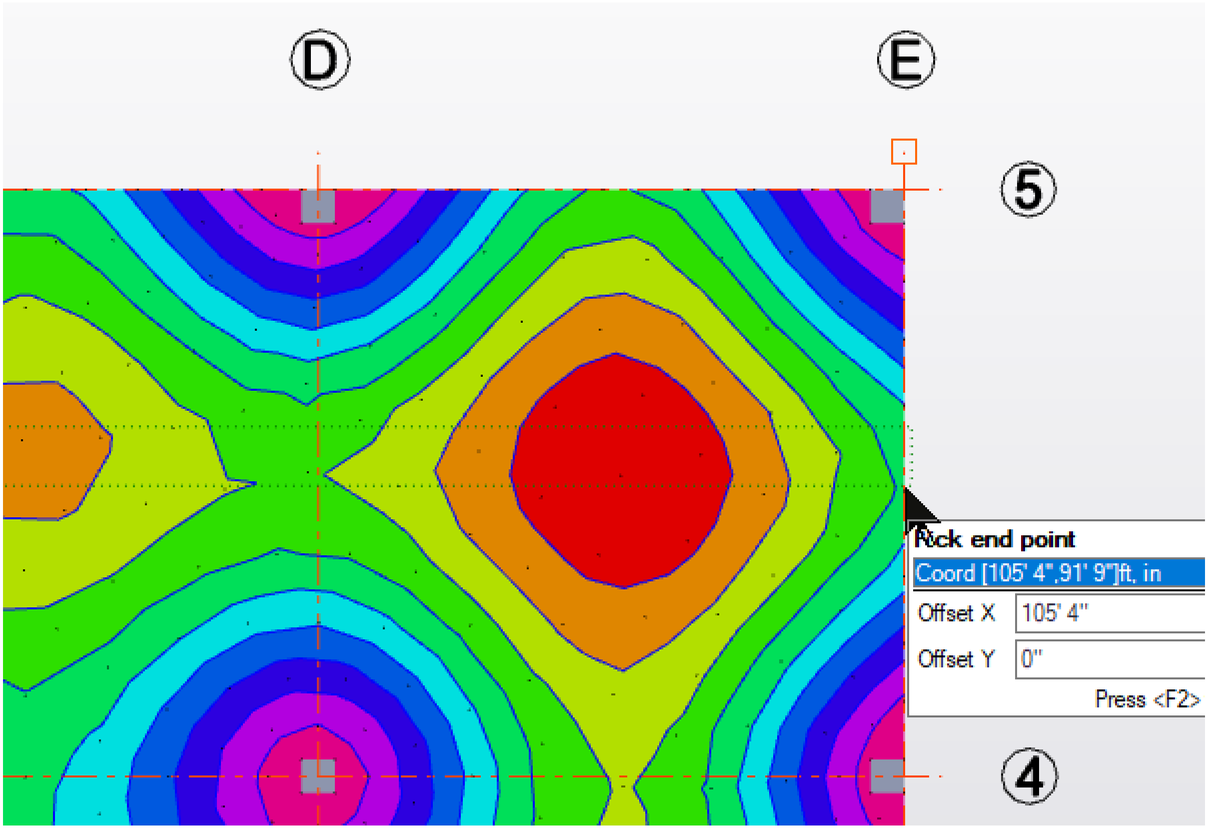


8. Create the final check line mid-way between grid lines 4-5 from grid line A to E.

    a. Pick the start point approximately as shown:



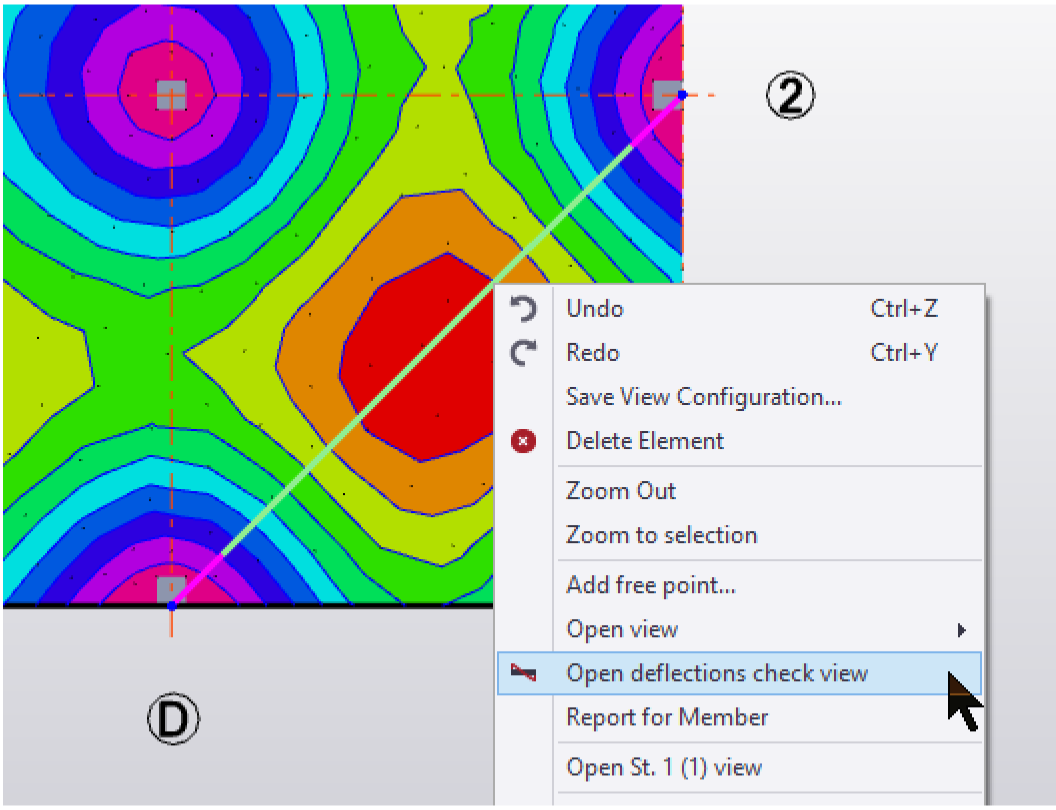
b. Pick the end point approximately as shown:

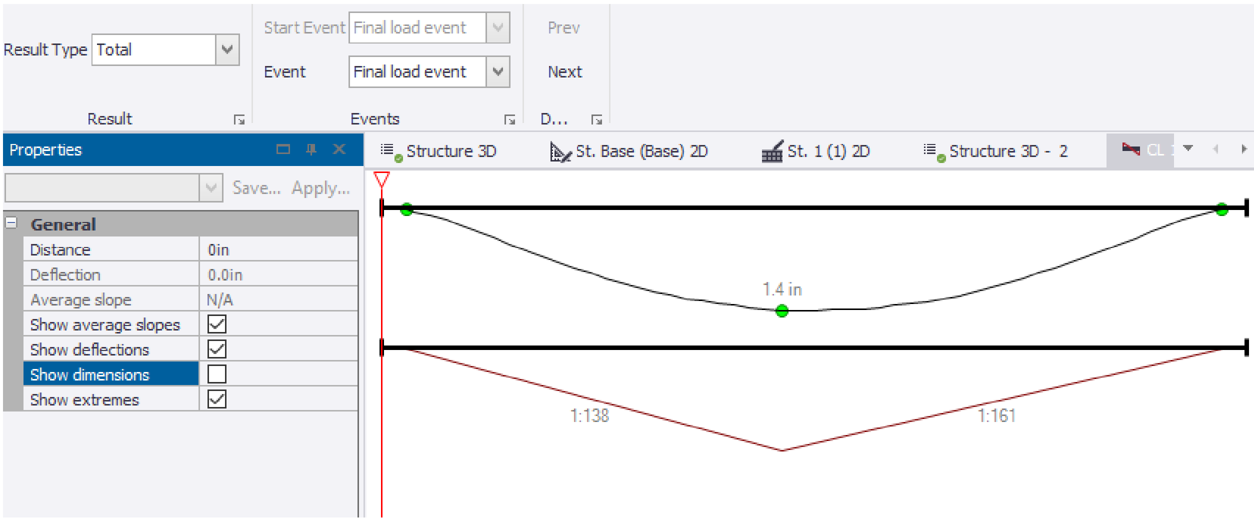


9. Press **Esc** to end the command.

NOTE In a real model you can add as many check lines to the model as you consider appropriate.

10. Right click on the diagonal check line and from the context menu select **Open deflections check view**.



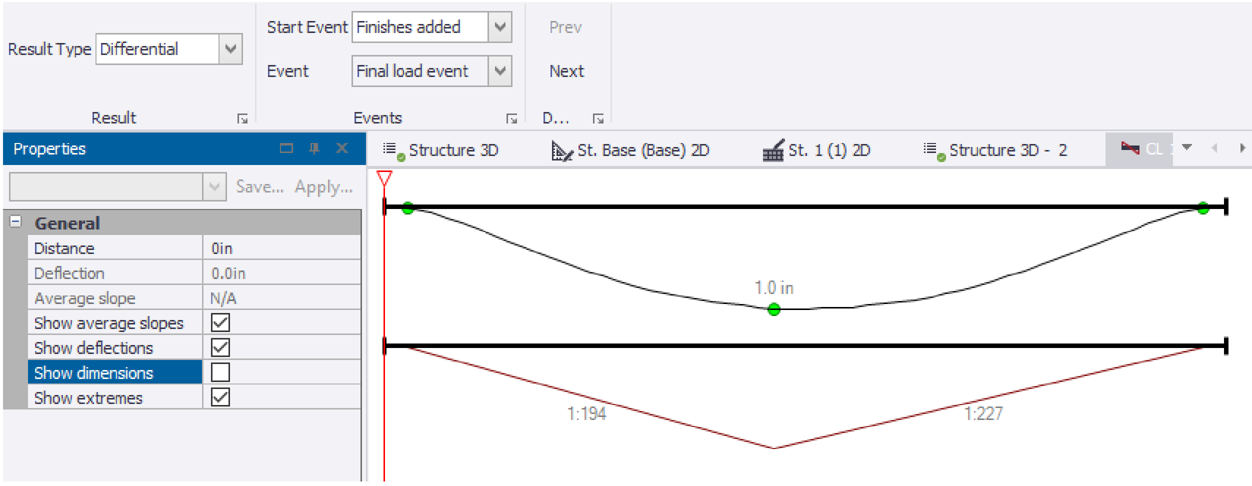


The ribbon allows you to specify the total (as shown above), or differential or instantaneous results for the selected events. Tekla Structural Designer then draws average slopes between maximum and minimum points.

A total deflection limit of Span / 240 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 120. In the view above the average slope between these points is 1 / 138, so the check passes.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

11. Click within the Deflection load analysis view and change the Result type to **Differential** and check deflection and slopes between the **Finishes added** and **Final load event**. The resulting differential deflection is as shown:



For the Sensitive Finishes check of differential deflection between finishes added and the final load limit had a limit of Span / 480, which is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 240. In the view above the average slope between these points is 1 / 194, so this check fails.

Generate Check Line Reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **St.1 (1) 2D** view, right click on the diagonal check line and select **Report for Member**

Deflection Checks Summary

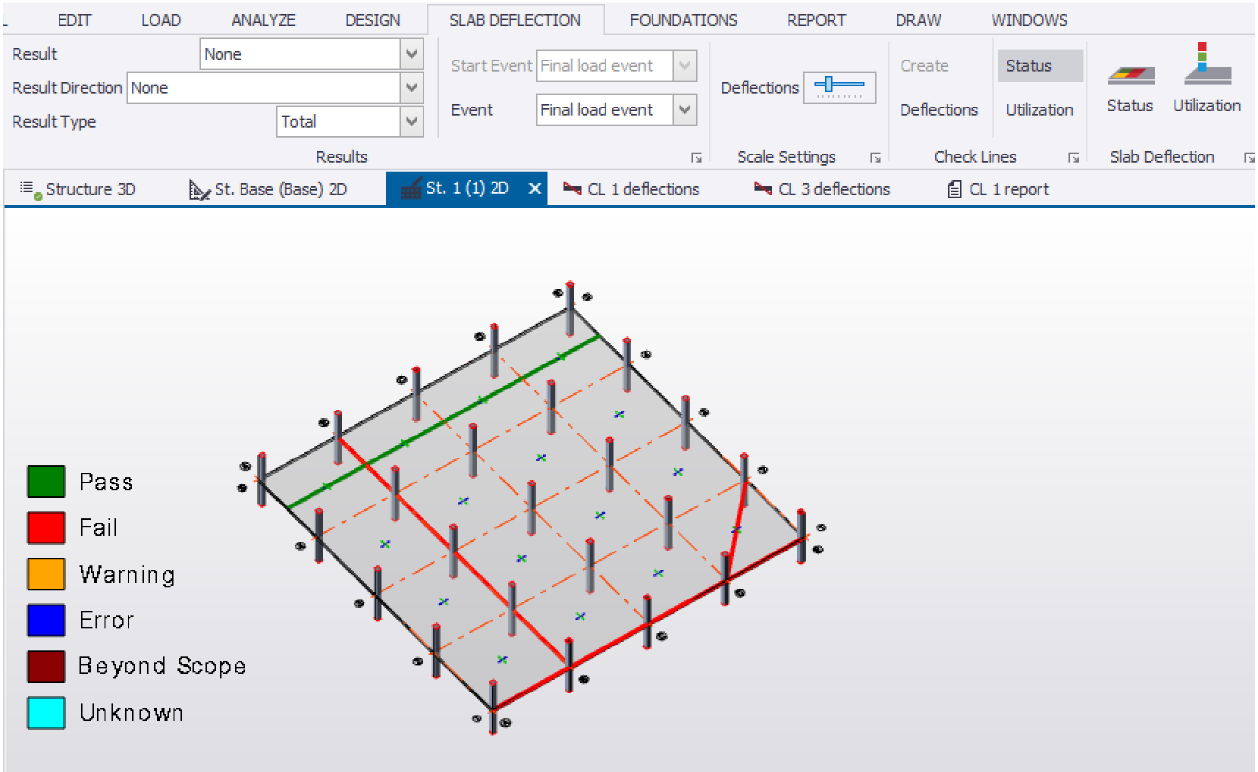
Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	193 33/64	1 : 811	✓ Pass	0.222
Sensitive Finshes	480	1 : 240	1.0	193 33/64	1 : 194	✗ Fail	1.237
Total	240	1 : 120	1.4	193 33/64	1 : 138	✓ Pass	0.870

As previously stated, the Sensitive Finishes check fails since the slope for differential deflection is reported as 1:206 which is greater than the allowable slope limit of 1:240.

Review Check Line Status and Utilization

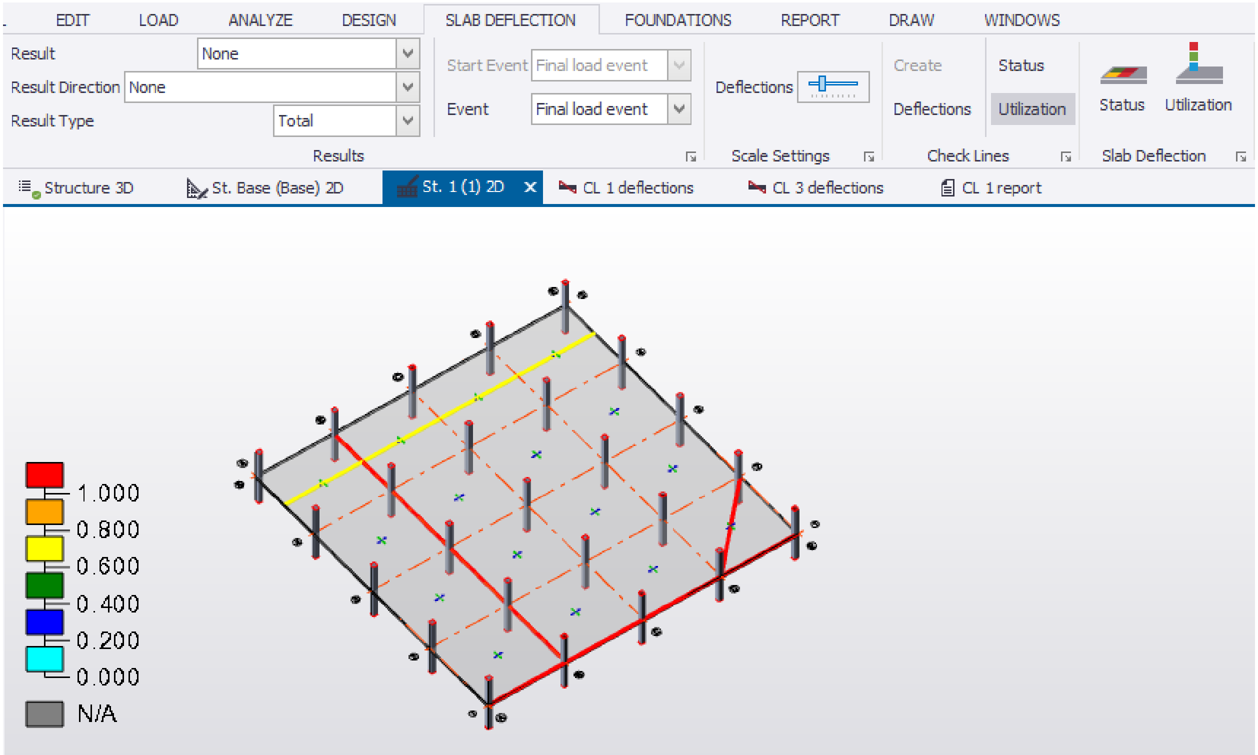
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the **St.1 (1) 2D** 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



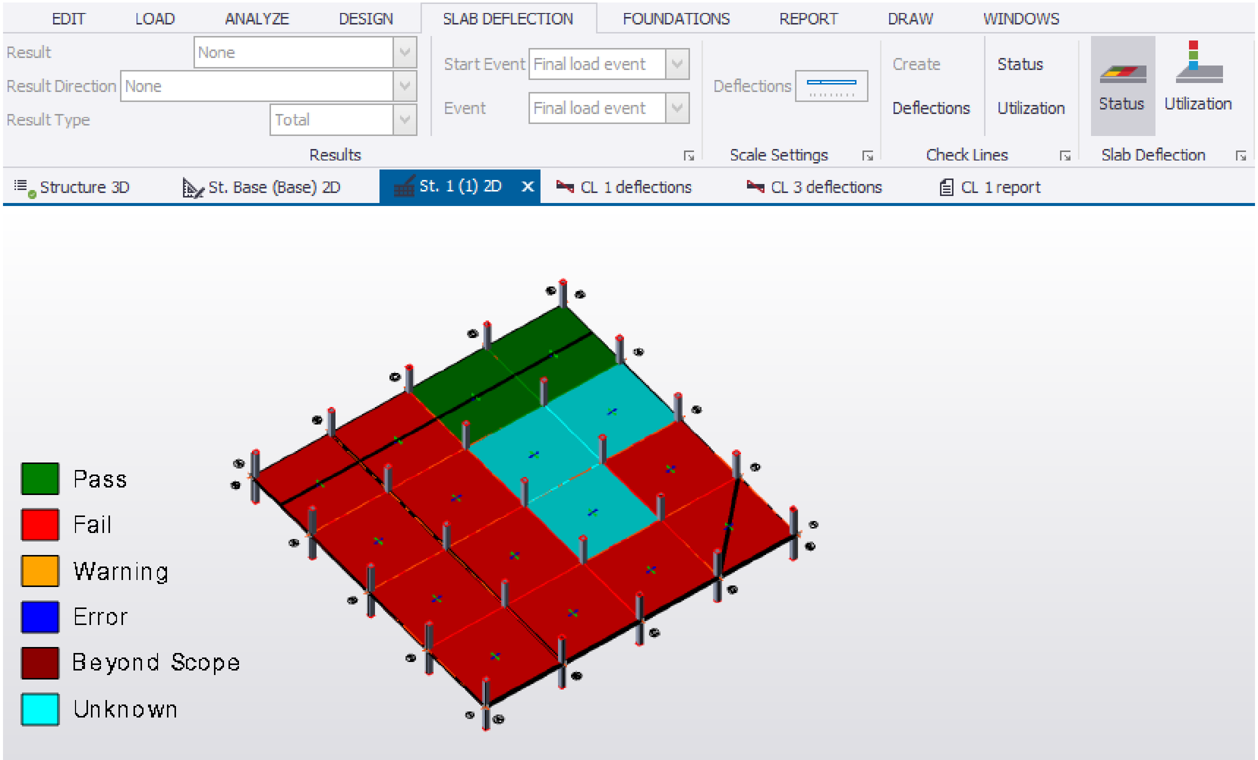
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

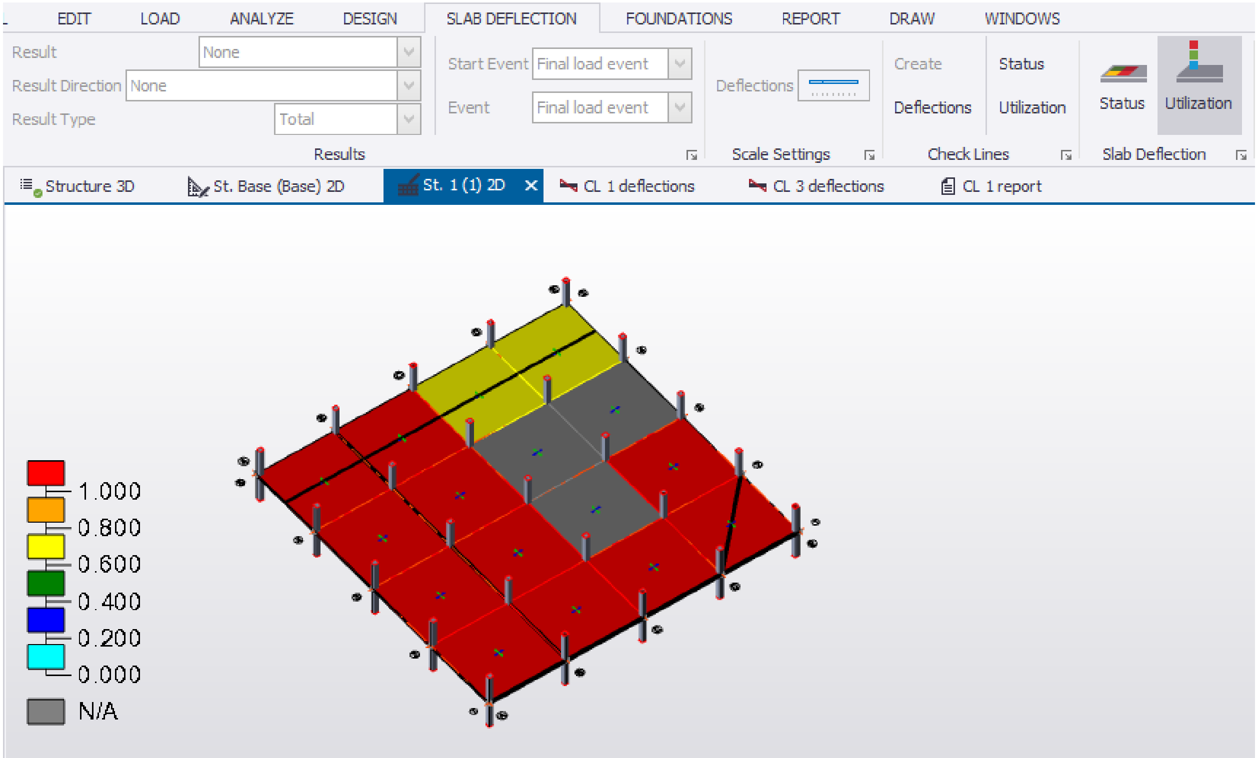
1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/ fail status of each slab.



- No check lines cross the majority of the slab items so they currently have a status of Unknown.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the 4 slab items crossed by the horizontal check line Fail.
- One slab item is being crossed (only just, in one corner) by the passing diagonal check line, this is the only slab to have a pass status.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

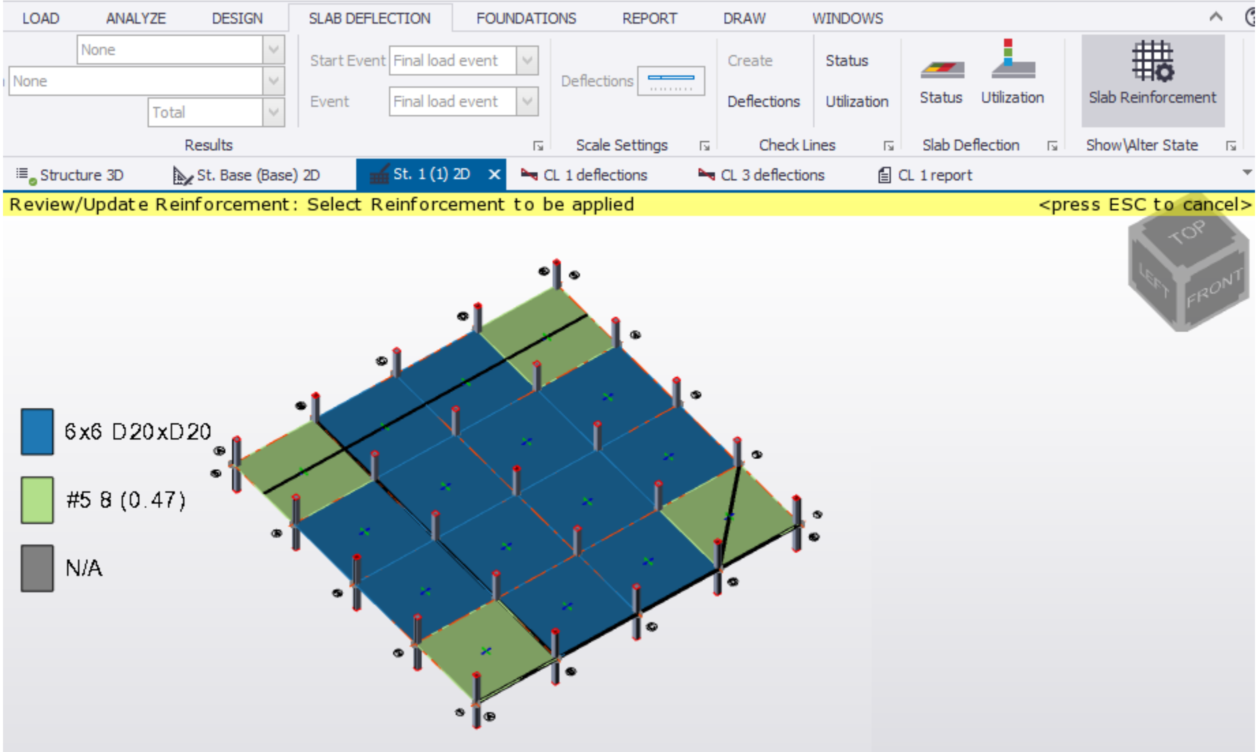
Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

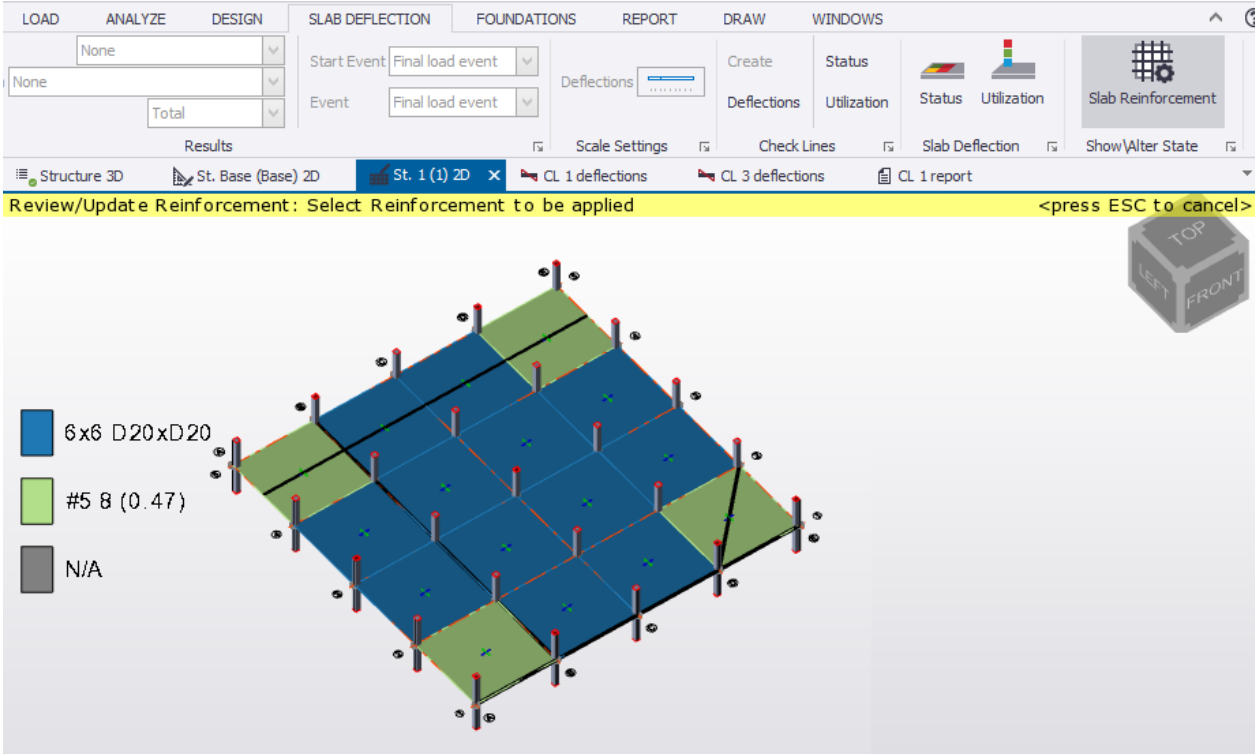
The analysis is extremely quick and since everything is contained within one model file, it allows "What If" scenarios to be considered to find the optimum solution.

In this exercise we will start by looking at the impact of adjusting the reinforcement.

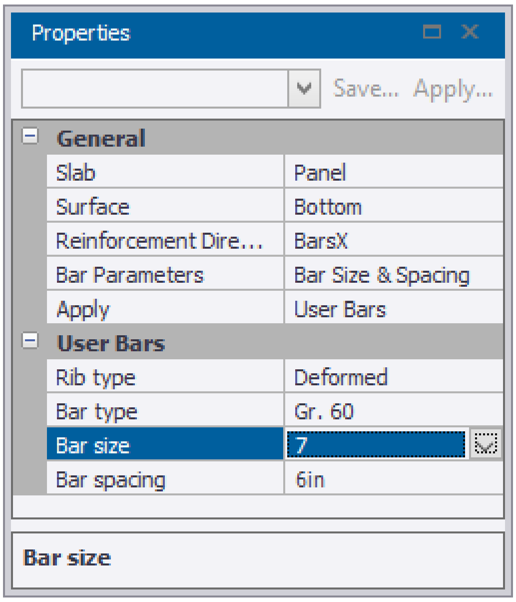
1. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.



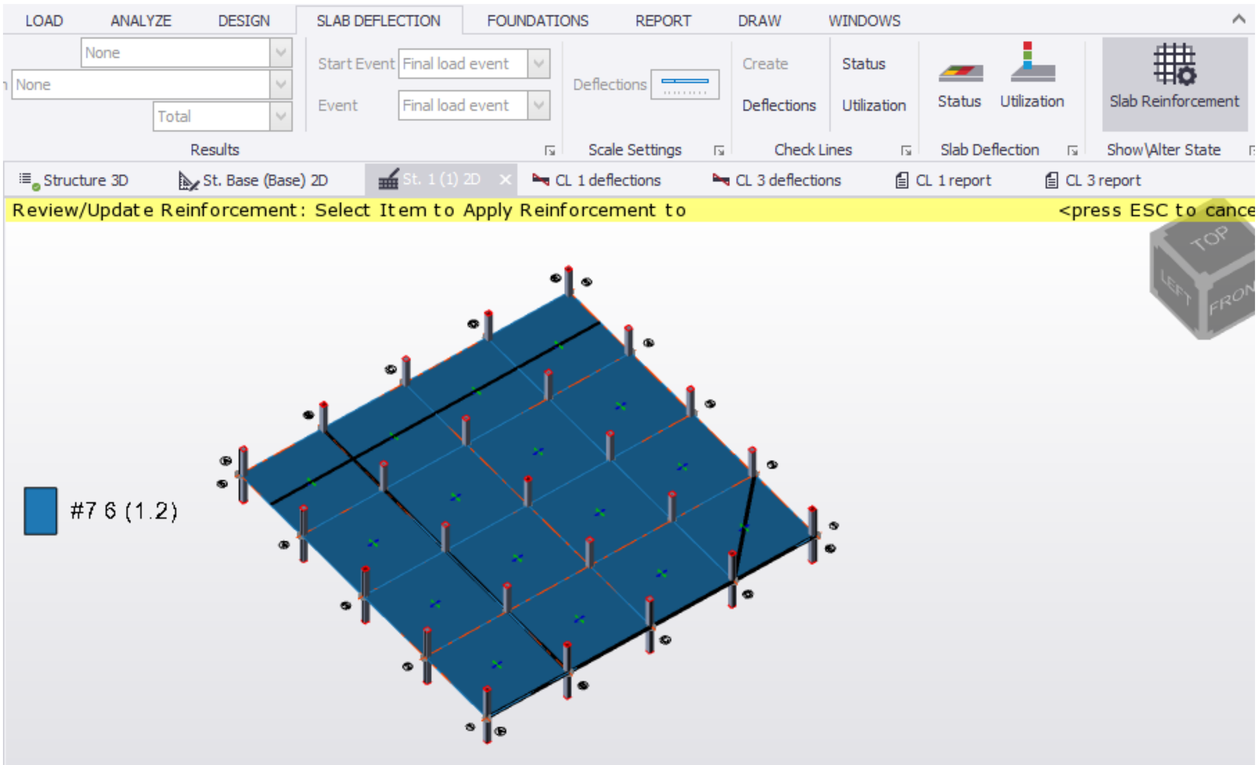
2. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



3. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change Apply to **User Bars**
   d. Change Bar size to **7**
   e. Change Bar spacing to **6in**

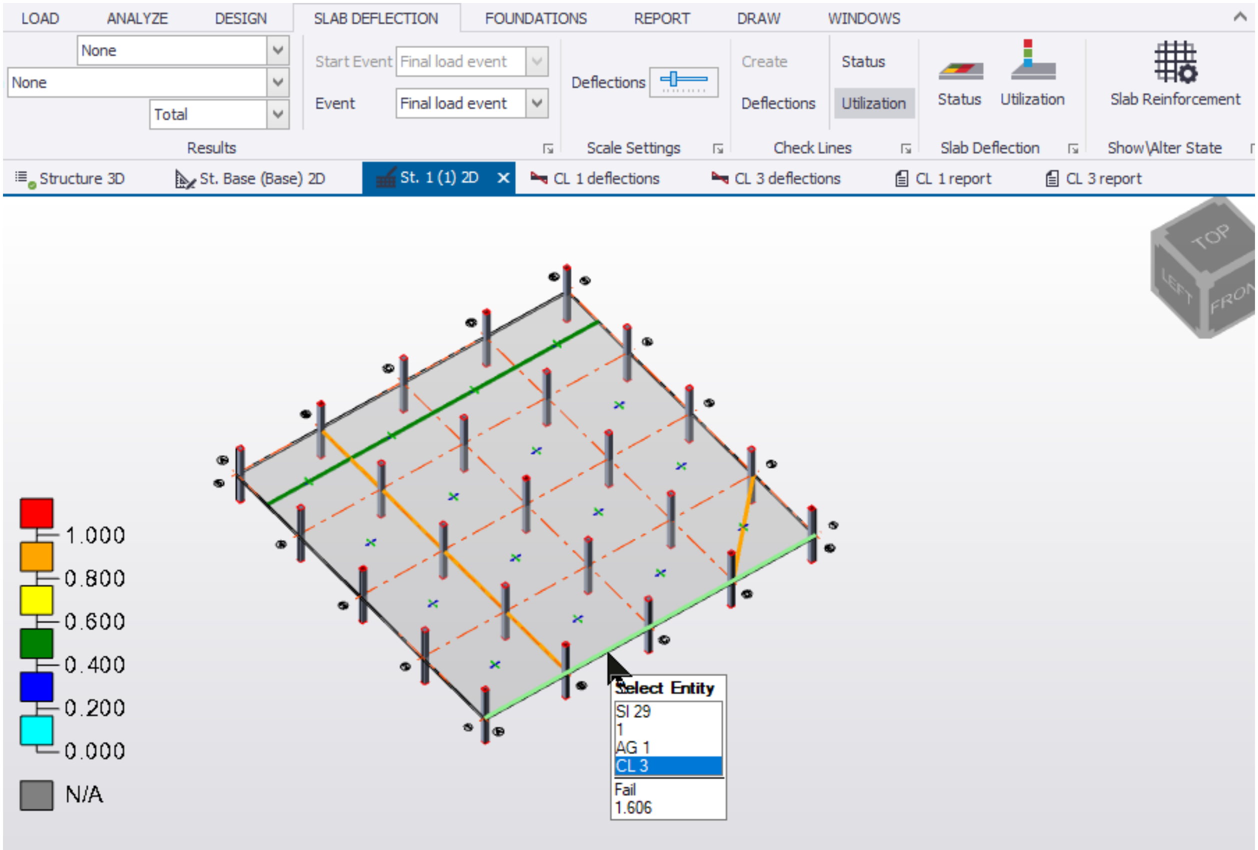


4. Click on each panel to apply the new reinforcement.



5. Repeat the above process to apply the same bars in the X reinforcement direction also.
6. Click **Analyze Current** to update the results

7. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
8. Investigate the tooltip results

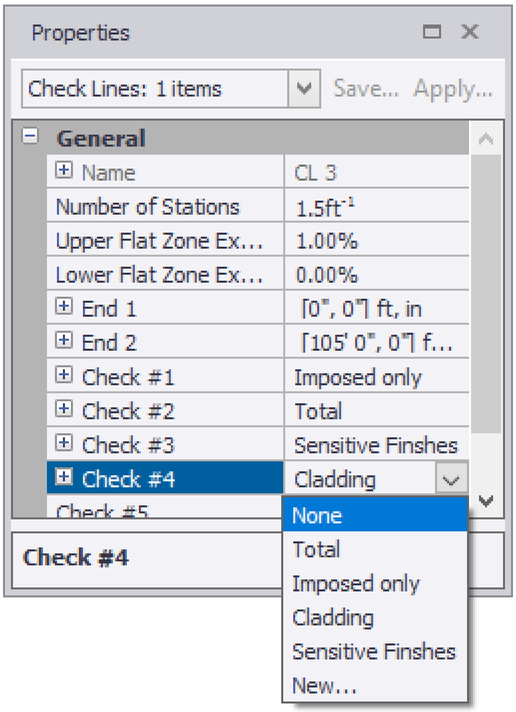


We can clearly see improvements in the results, however the check line along grid 1 with the more onerous Cladding deflection limit is still failing.

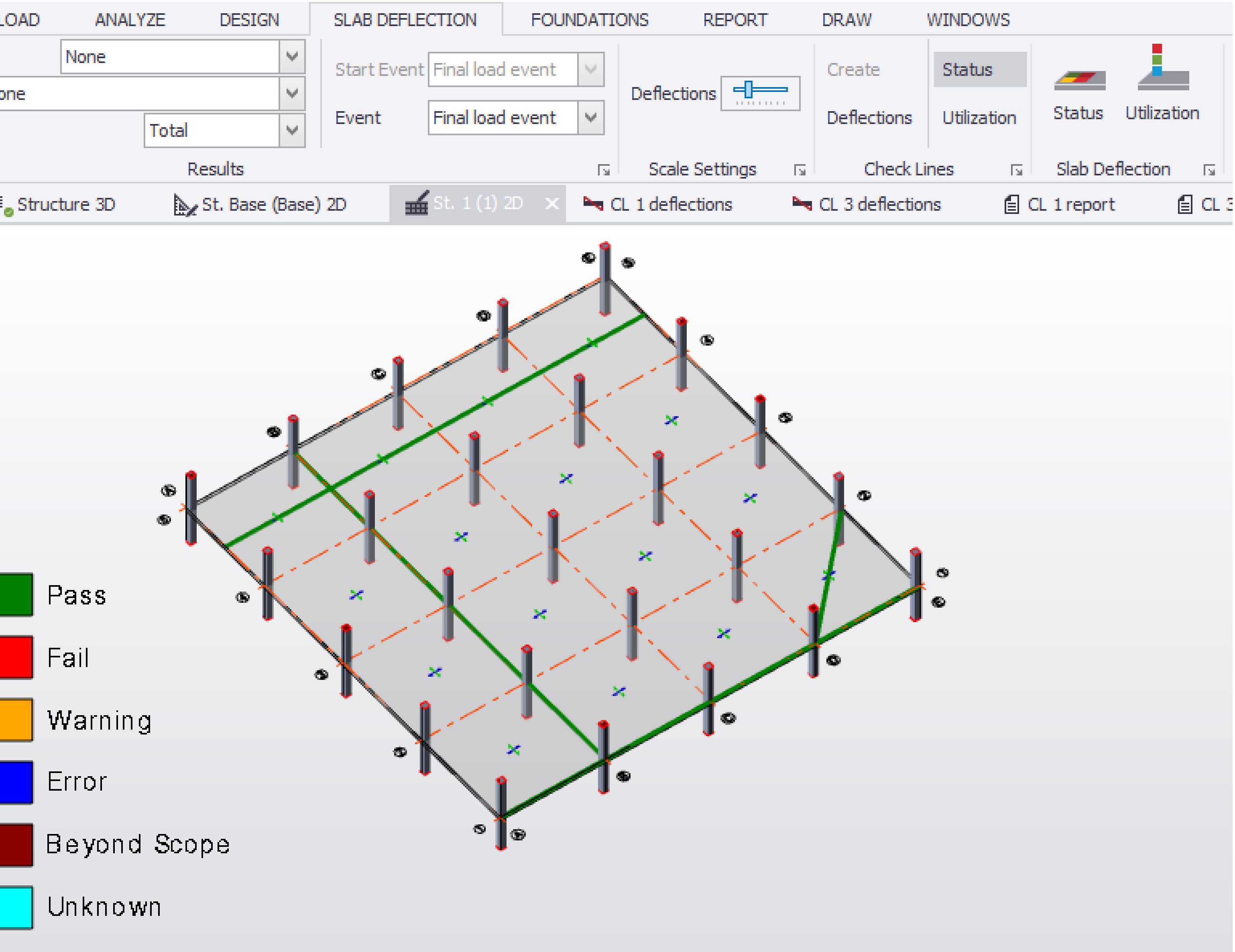
At this point in a real model you could begin to look at the impact of the various other input parameters, such as changing the concrete grade.

For the purpose of this exercise, and as the cladding check is not a code requirement, we will simply disable it, as follows:

9. Select the check line along grid 1
10. In the Properties window, reset Check #4 to **None**.



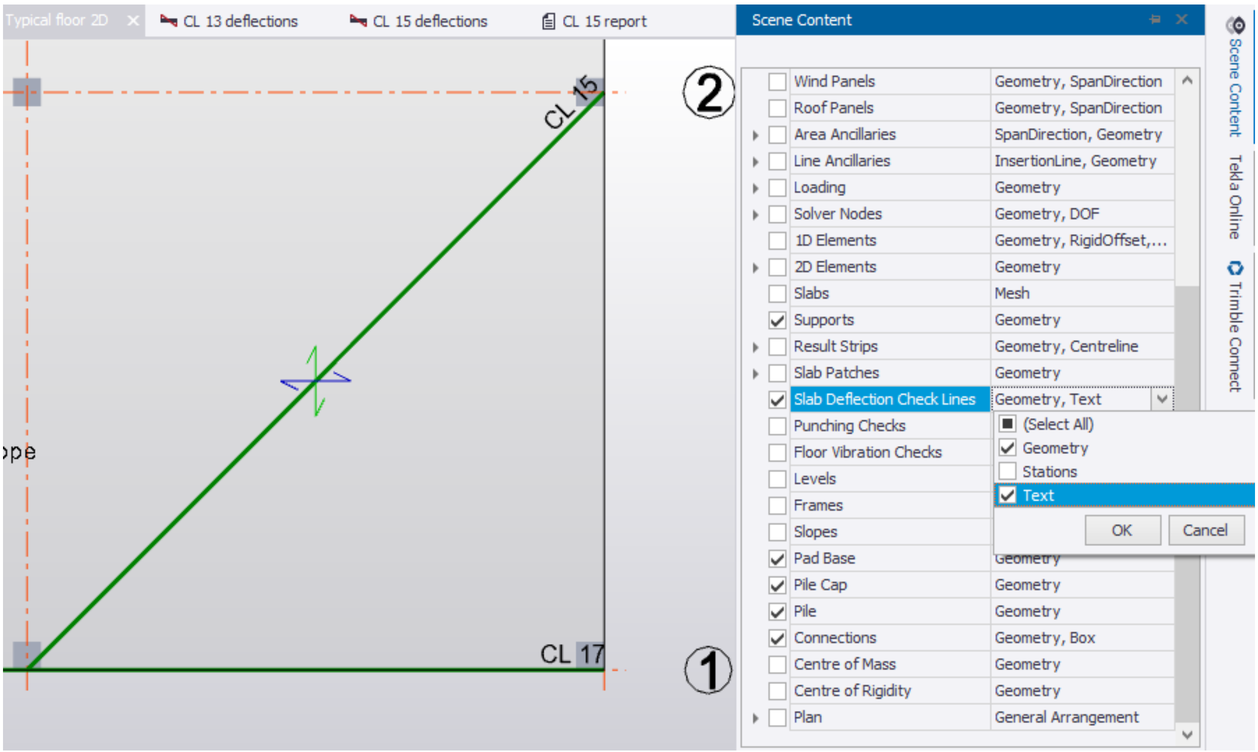
11. Click **Analyze Current** to update the results
12. Click **Status** in the Check Lines group. All of the check lines should now pass.



Generate Model report

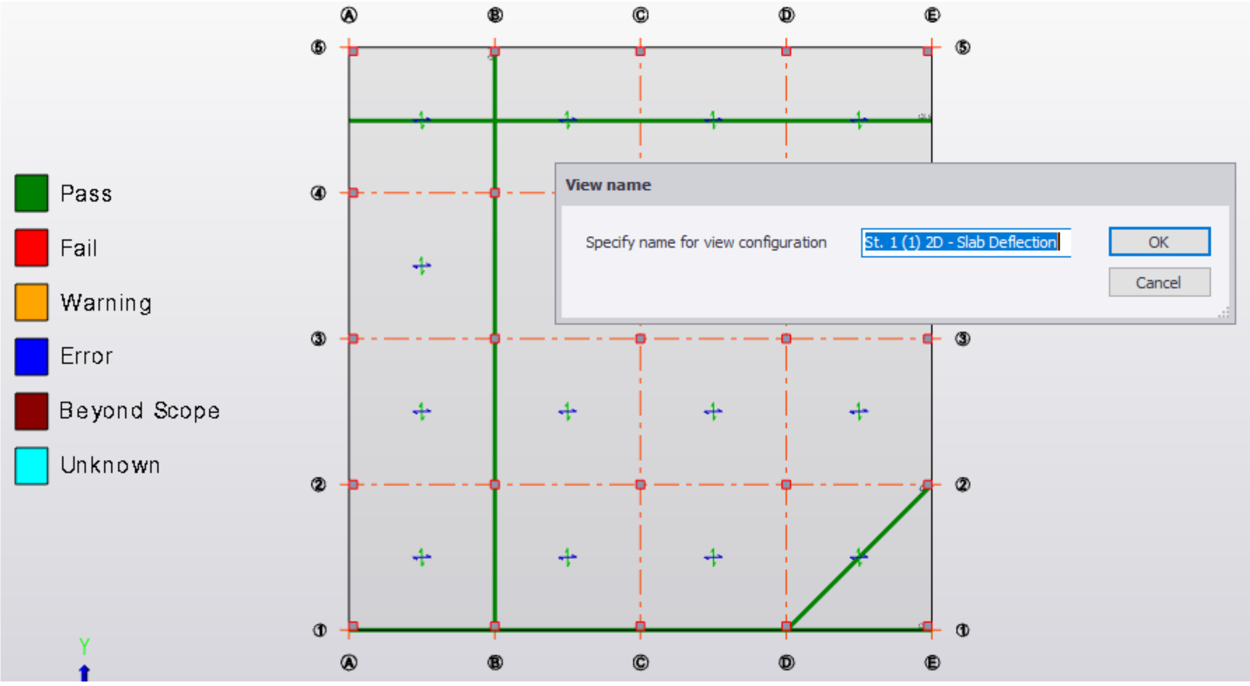
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



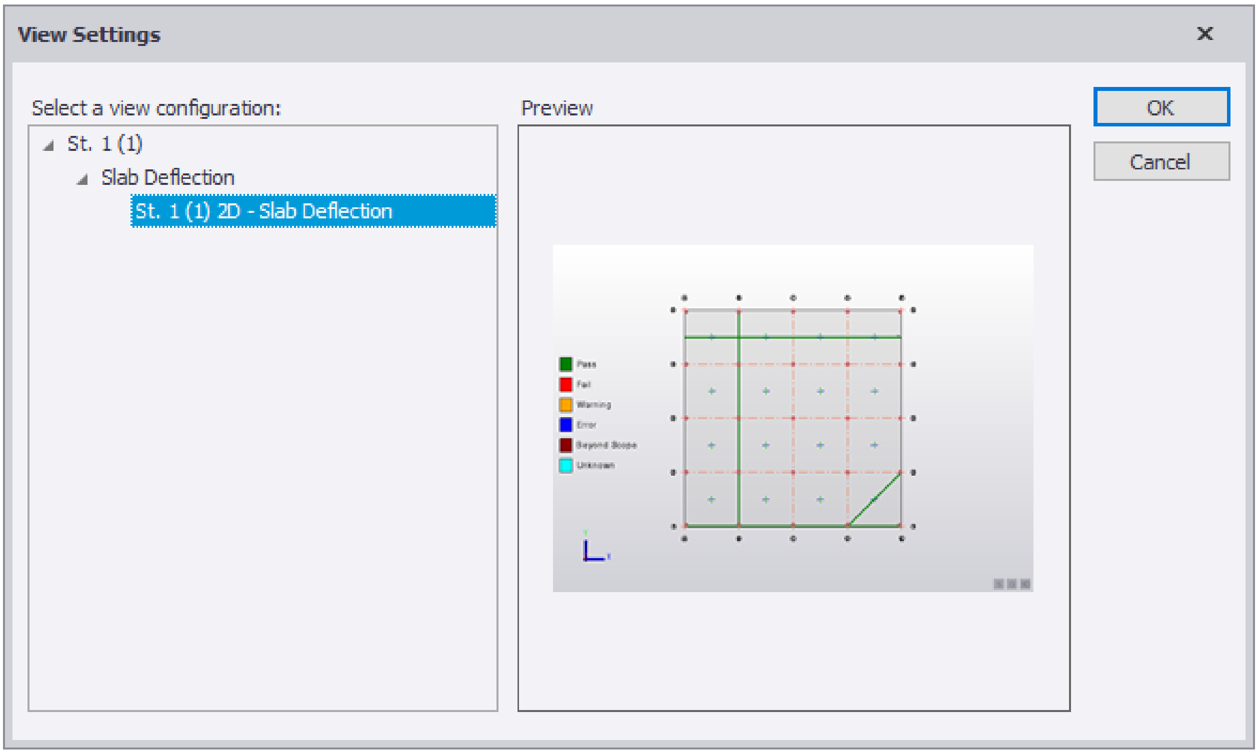
TIP The check line references can be customized using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

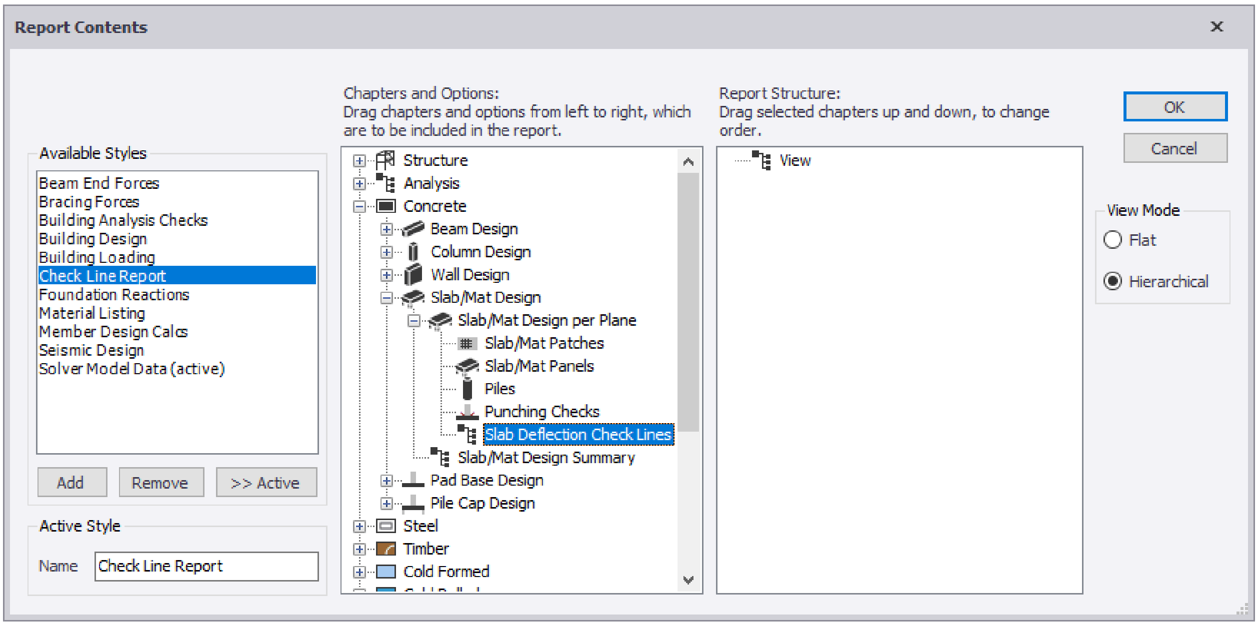


This saved view can be included in the Check line report.

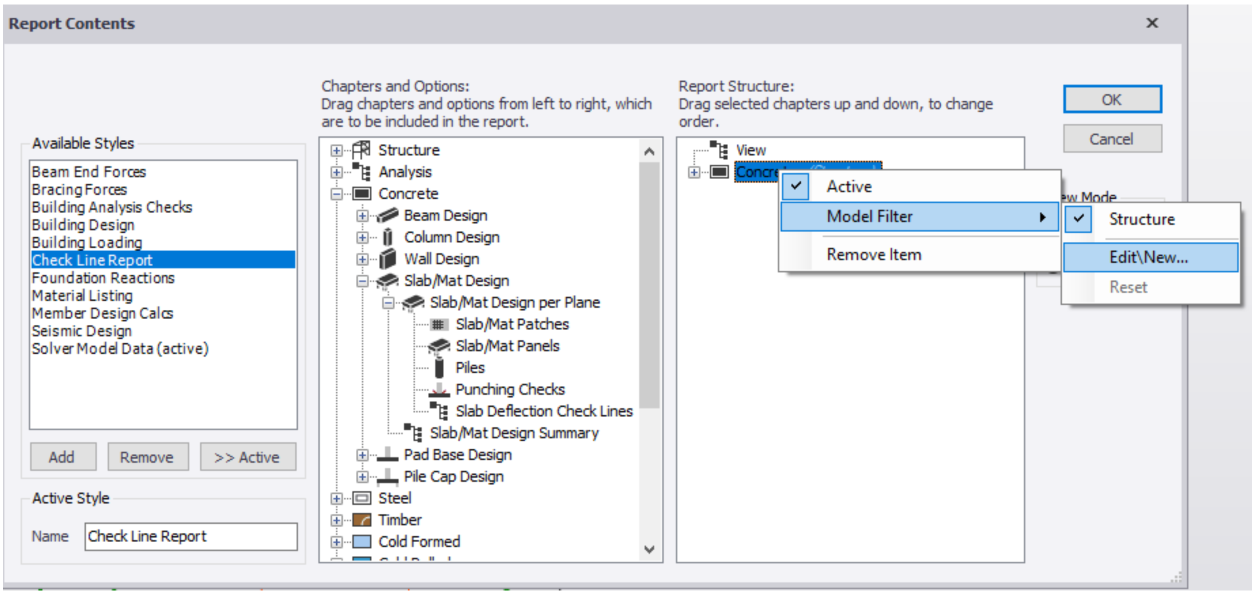
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In the Report Structure, right click **View**, then choose **Settings...**
7. Select the Slab Deflection view you created earlier



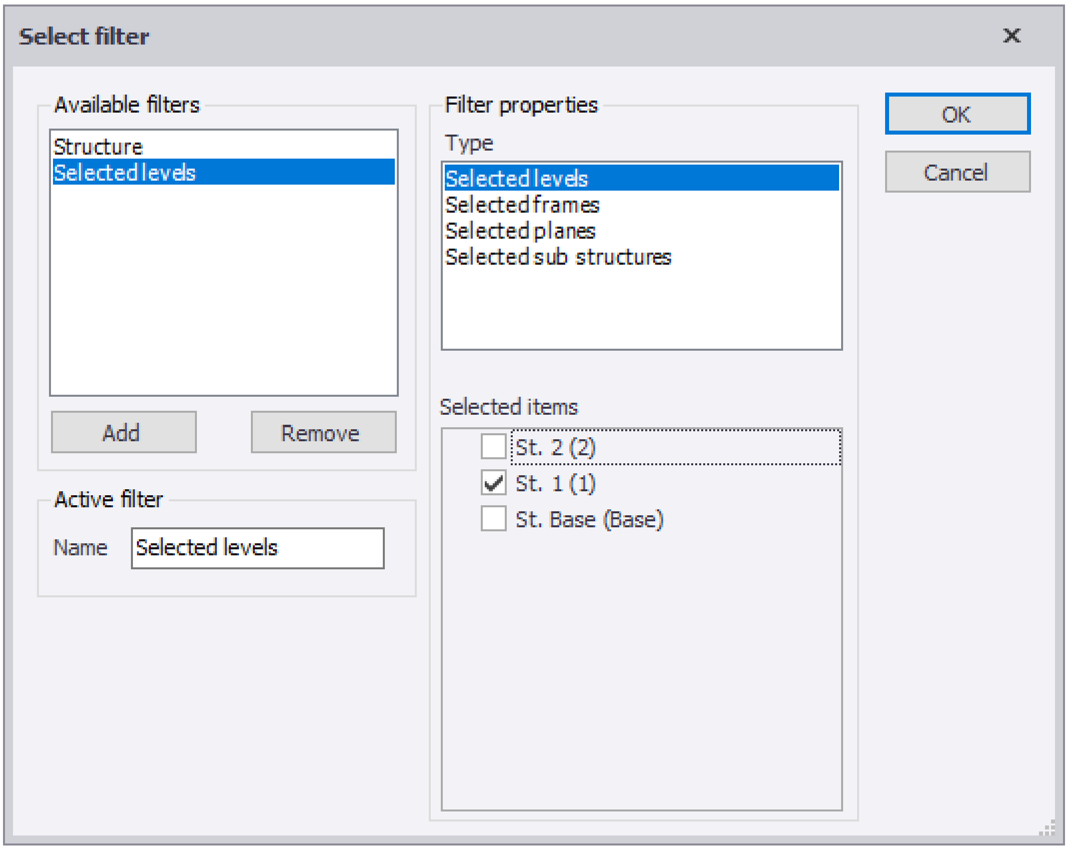
8. Click **OK** to return to the Report Contents dialog
9. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



10. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**

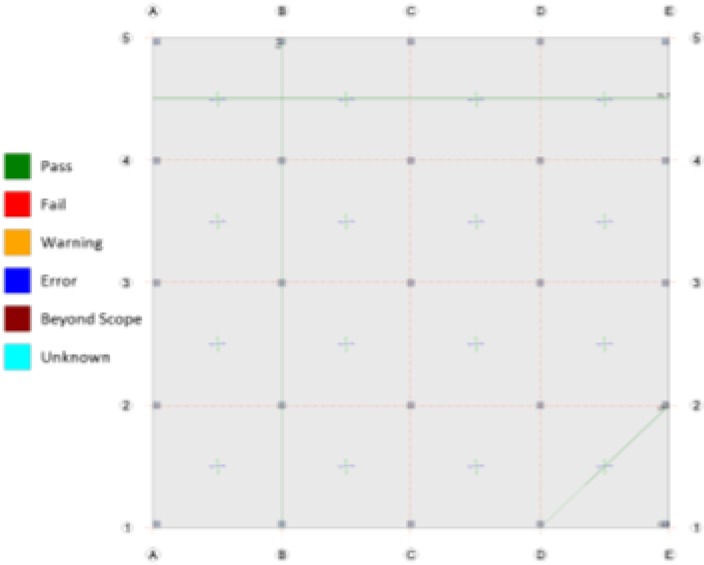


11. In the Filter dialog, click **Add** and select Selected levels and ensure a check against **St. 1 (1)**



12. Click **OK** to return to the Report Contents dialog.
13. Click **OK** to exit and save the report.

    A report structure called **Check Line Report** has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select **Check Line Report**
    b. Click **Show Report** to open the report.



St. 1 (1) 2D - Slab Deflection

Concrete

Slab/Mat Design per Plane

St. 1 (1)

Slab Deflection Check Lines

CL 1

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	201 49/64	1 : 1087	✓ Pass	0.166
Sensitive Finshes	480	1 : 240	0.7	201 49/64	1 : 273	✓ Pass	0.879
Total	240	1 : 120	1.1	201 49/64	1 : 185	✓ Pass	0.649

CL 3

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	-0.1	144 15/32	1 : 1232	✓ Pass	0.146
Total	240	1 : 120	-0.8	144 15/32	1 : 183	✓ Pass	0.656
Sensitive Finshes	480	1 : 240	-0.6	144 15/32	1 : 254	✓ Pass	0.945

CL 5

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.1	135 15/32	1 : 1059	✓ Pass	0.170
Sensitive Finshes	480	1 : 240	0.5	135 15/32	1 : 284	✓ Pass	0.845

1.8 Precast member design handbook

Precast structures can be modeled and analyzed in Tekla Structural Designer. Precast beams and columns can then be designed, provided that you have access to a license of Tekla Tedds.

This handbook describes the workflow required in Tekla Structural Designer for running the Tekla Tedds precast calculations.

NOTE The following limitations apply:

- Only design of normal weight precast concrete beams and columns to Eurocodes is supported.
- Design of precast concrete slabs and walls is beyond scope for all head codes.
- The process is only applicable to precast structures that don't make use of in-situ structural toppings.
  - Precast construction which adopts a more hybrid construction involving the use of in-situ toppings exceeds the limitations of the Tekla Tedds calculations and should therefore be considered beyond scope.

The following topics are covered:

Related video

Precast modeling, analysis and design using Tekla Tedds

Precast member design workflow

The basic workflow for precast design in Tekla Structural Designer is described in the sections below:

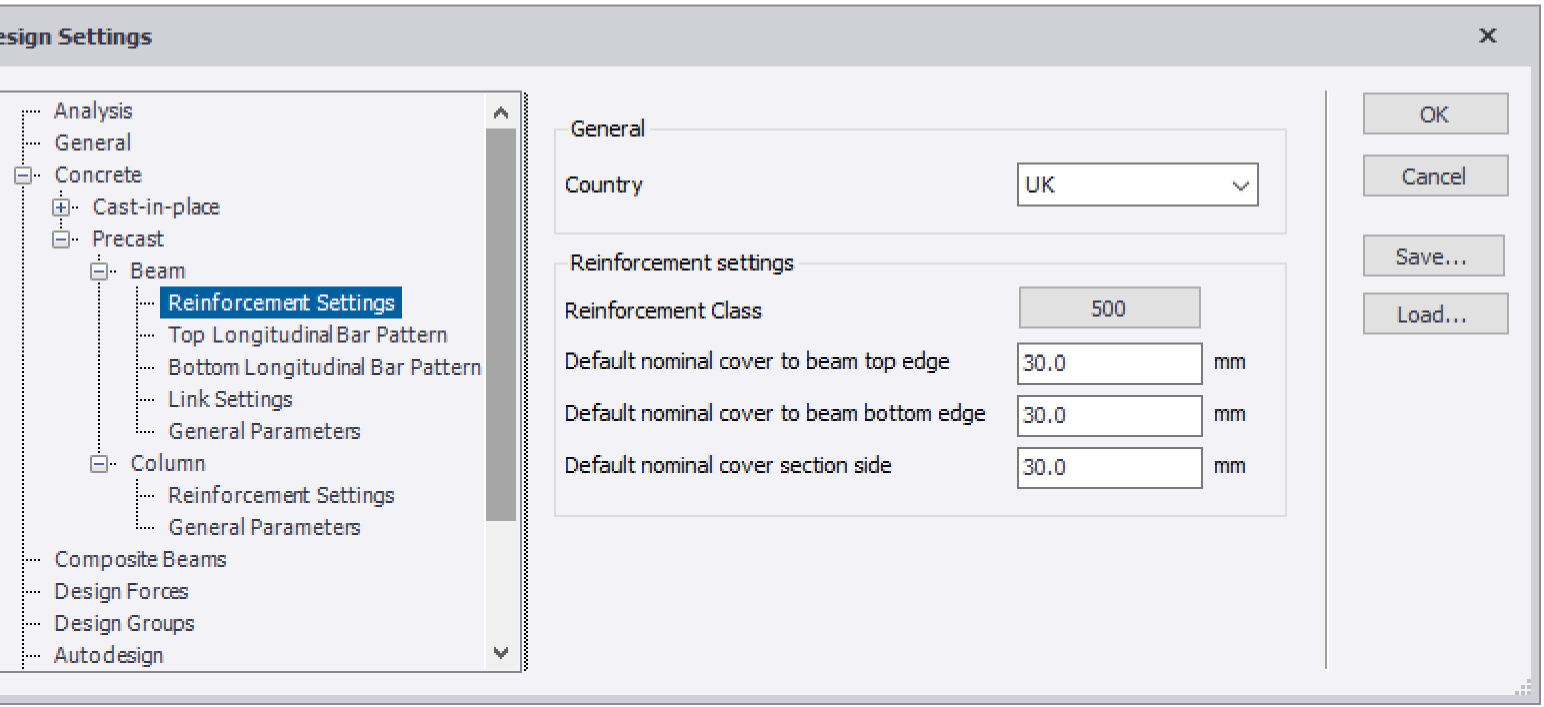
Related video

Precast modelling, analysis and design using Tekla Tedds

Configure precast beam and column design settings

Specific precast beam design settings and precast column design settings that apply to all the beams and columns in the model should be specified prior to running the designs.

These are set on the repective **Concrete>Precast** pages of the **Design Settings** dialog.

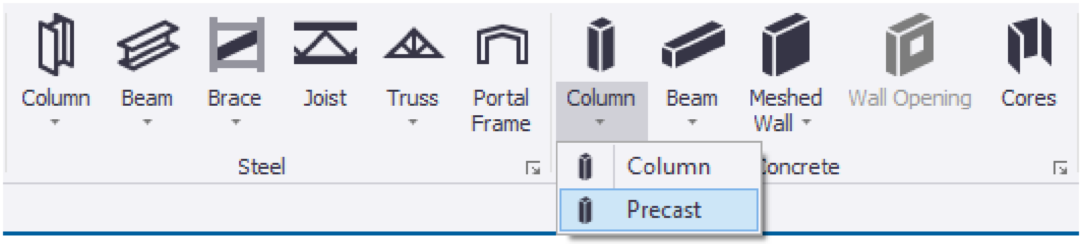


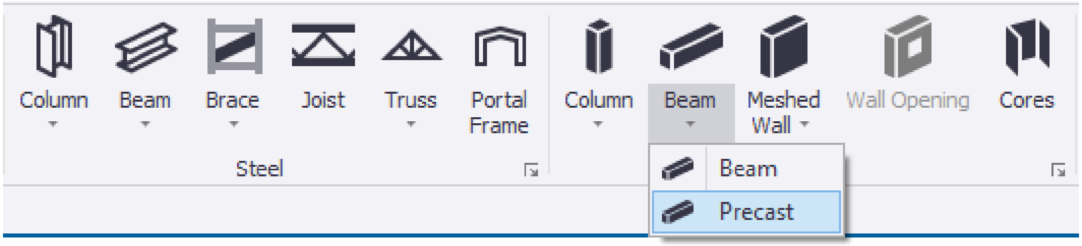
By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds precast calculation as it is run.

You still have control within individual calculations to adjust these settings on a member by member or group basis.

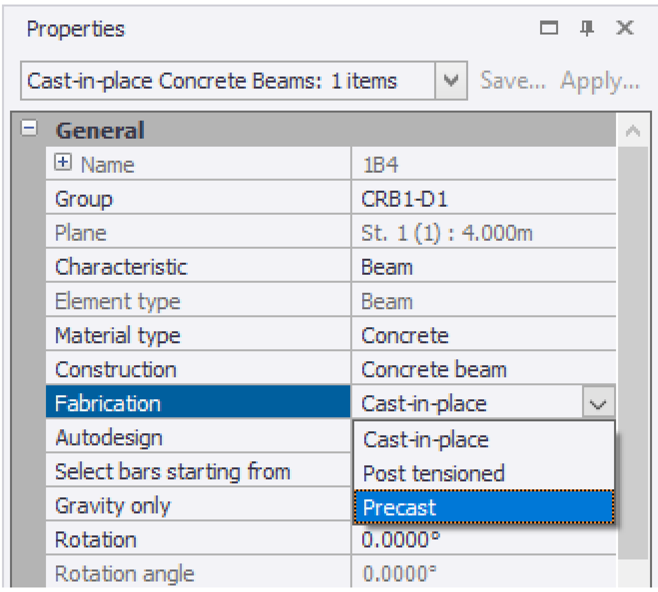
Define and place precast members

To place precast members, simply use the **Concrete Column > Precast**, or **Concrete Beam > Precast** commands on the Model tab.



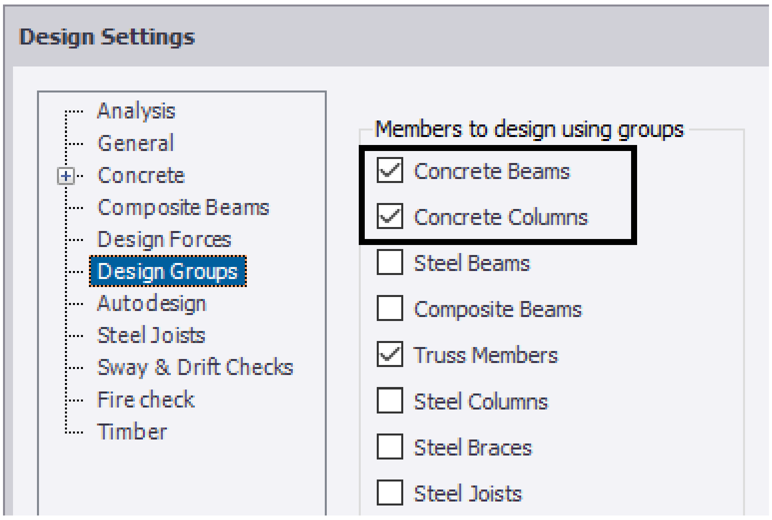


An alternative to this would be to use the **Concrete Column > Column**, or **Concrete Beam > Beam** commands and to subsequently change the Fabrication parameter to be Precast.



Configure precast groups

Customizable member groups are created automatically. You can choose whether to utilize them for design purposes via the **Design Groups** page of the **Design Settings** dialog.



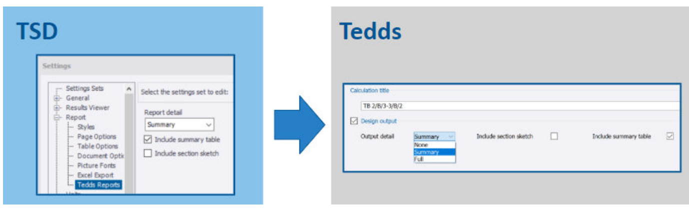
Using precast member design groups (page 513) can help speed the overall design process, particularly so in the case of medium to large size models.

- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

The initial precast member design groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can move members into new groups as required.

Set the Tedds results output level

You can choose the output level for the Tedds precast calculations in advance by clicking **Home > Settings > Report > Tedds Reports**



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds precast calculation as it is run.

The setting applies to all Tedds linked calculations (precast and timber).

Establish design forces by running the analysis

Precast members can only be designed provided a set of analysis results exist. These can be generated from the **Analyze** ribbon by running .

A design force envelope is established, and all critical load combinations are considered in one Tedds calculation.

Provided load combinations have been created, once analysis has been performed the **Design using Tekla Tedds** options become available.

Design using Tekla Tedds

NOTE We recommend that the Tekla Structural Designer model is developed as much as possible prior to considering the design of the precast members. This should ensure that the correct distribution of forces is carried through into the Tekla Tedds calculations.

Design a Selection

To design several precast members or groups in one go:

1. Select the precast members you want to design.
2. Right click and select Design using Tekla Tedds> Selection
3. Design the selected members.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whther ot not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

4. When changes are made to the model, you can check if the existing sections are still sufficient by by running
5. Check In Tedds from the Design tab.

   NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Design a group

If you have activated concrete member design groups, each group can be designed as follows:

1. Highlight any member in the group, right click and select Design using Tekla Tedds> Group

   This design gathers the analysis results for all members in the group and collates them in to one design. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

2. Design the selected member for these loads.

3. Click **Finish**.

   If the section size was changed during the design, all members of the group will be updated to the new section. If the reinforcement data was changed this is also copied to all group members.

   Irrespective of whether the section or reinforcement has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

4. In the Review View, review the utilization ratios for all members in the group - if these all indicate an efficient design, the process can stop at this point.

   NOTE For columns in particular, the envelope of design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

5. If a group member has a lower than desired utilization, right click on it and select Design using Tekla Tedds> Member

6. In the Review View, review the resulting utilizations for the group members - note that some of might now fail.

7. Using the new utilizations to better inform you choice, add extra groups as necessary and re-allocate the members between the groups.

8. Run Design using Tekla Tedds> Group for one member in each of the new groups.

9. Iterate the process, starting from step 5 above.

10. When changes are made to the model, you can check if the existing designs are still sufficient by by running Check In Tedds from the Design tab. Alternatively you can check groups individually by running Check using Tekla Tedds> Group for one member in each group.

    NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Ungrouped Design

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member
2. Design the selected member.
3. Continue to design additional members in the same way as required.
4. When changes are made to the model, you can check if the existing sections are still sufficient by by running Check In Tedds from the Design tab.

> **NOTE** Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Design model

If you want to design every precast (and timber) member in the model in one go:

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Design using Tekla Tedds > Model**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check the design after changes

If changes are made to the model you can run a 'check' design to determine if the existing sections are still sufficient. A check is quicker to perform than a design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

The updated utilizations can then be reviewed in a Review View.

Check the whole model

To check all the existing Tedds member designs,

1. Click Check In Tedds from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.
2. Right click and select **Check using Tekla Tedds> Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds> Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

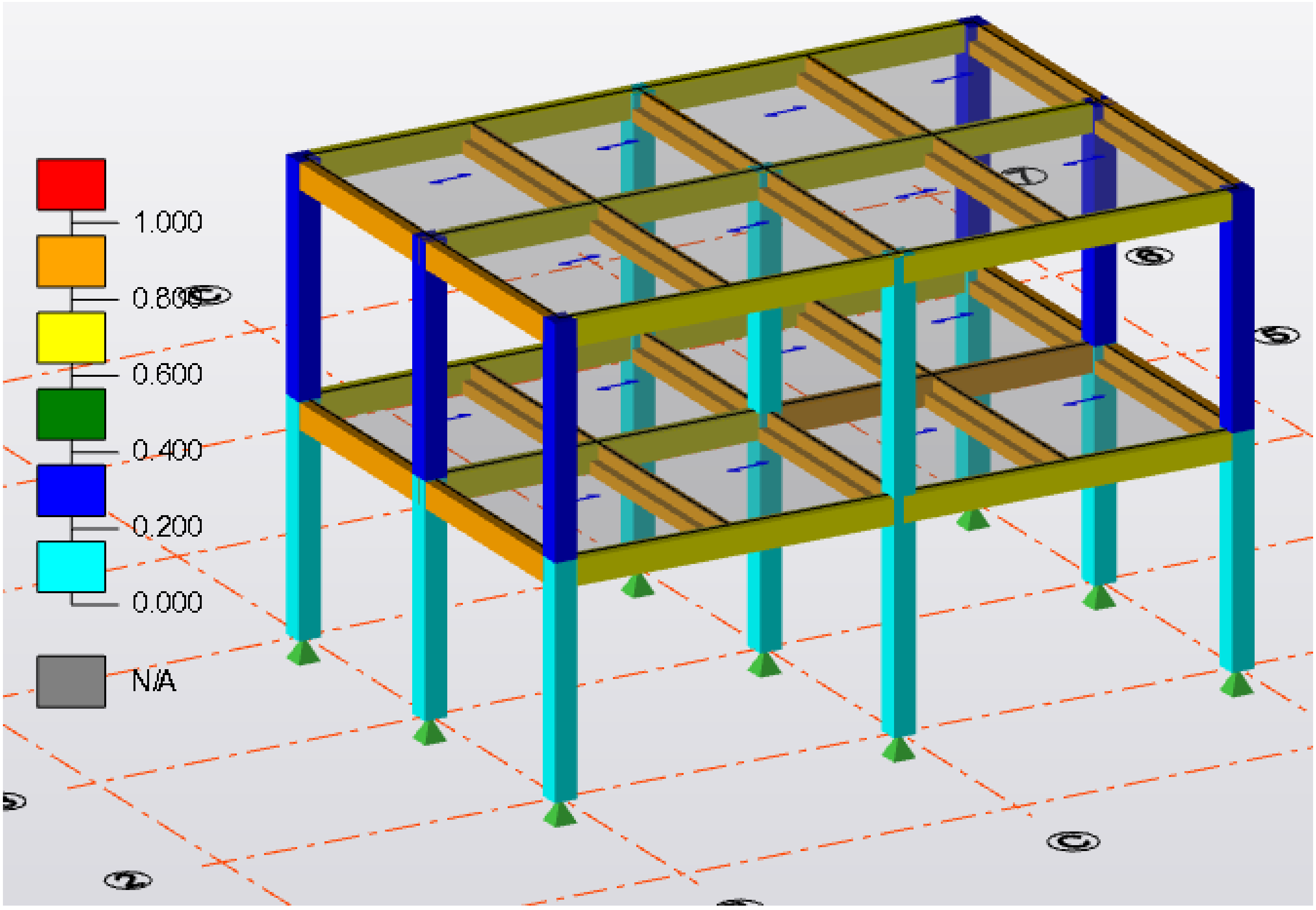
To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds> Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

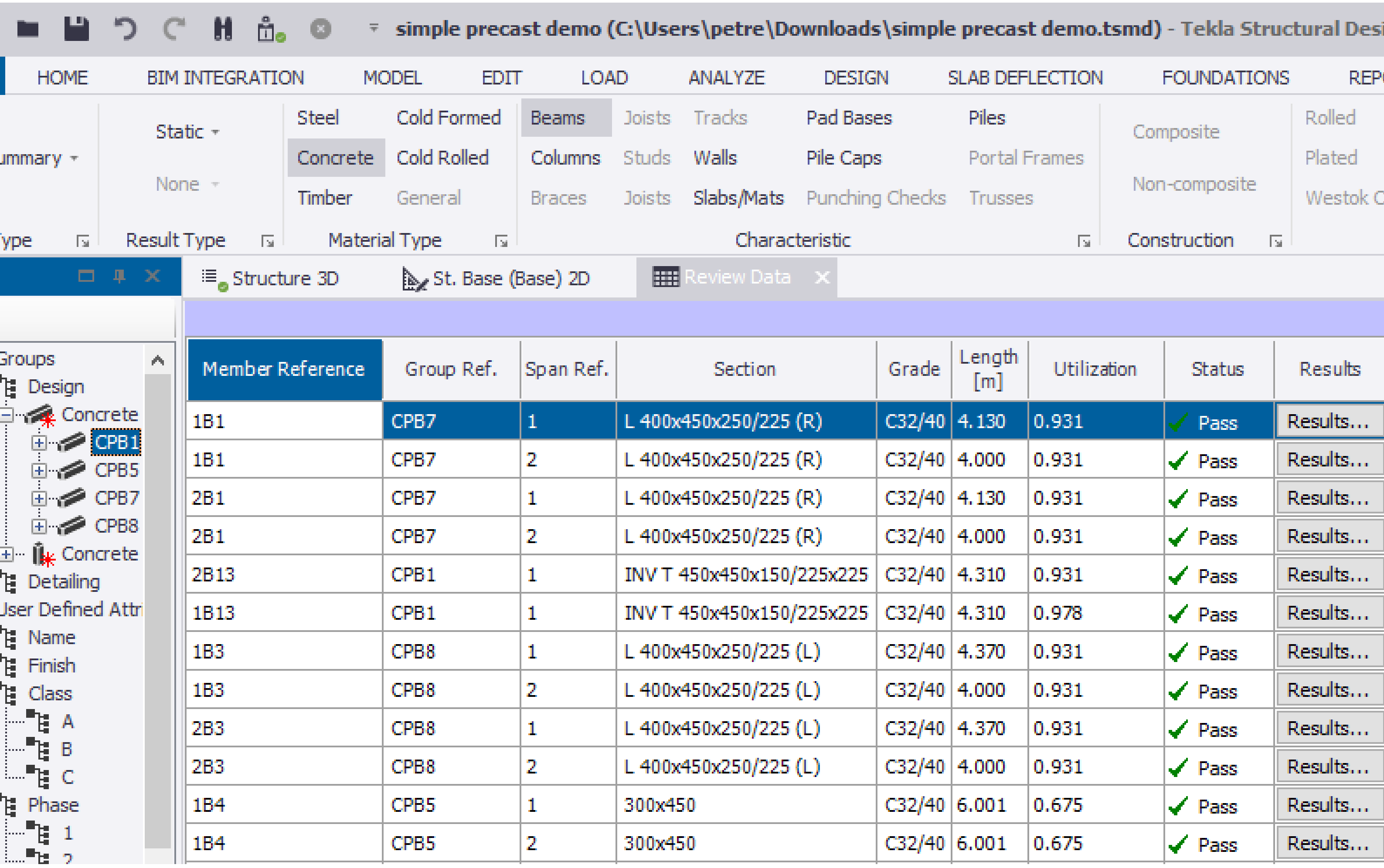
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

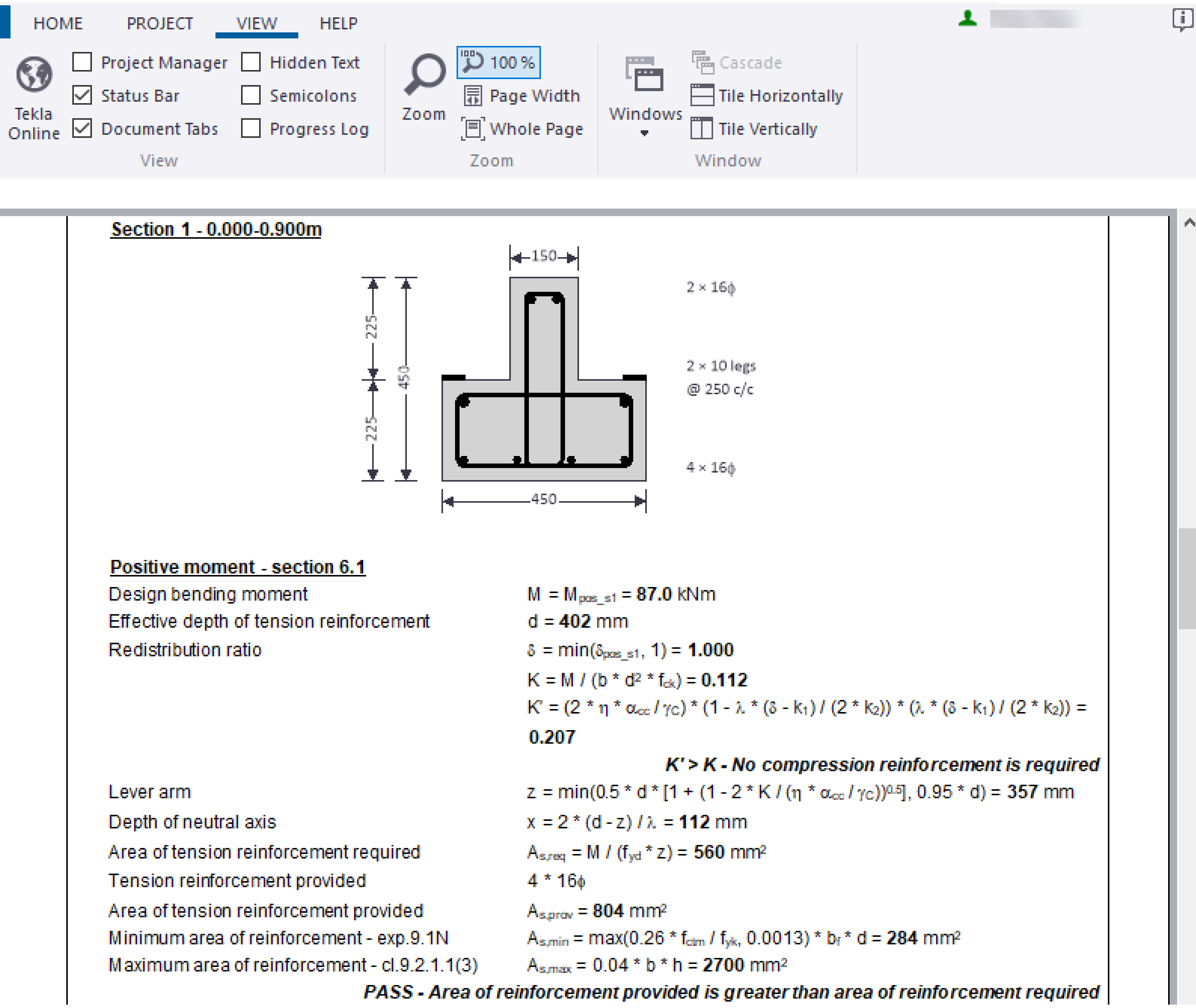
At this stage you can display member design status and utilization ratios from a Review View.



You can also display the design status tabular results.



Detailed Tekla Tedds calculations for precast members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds.



Precast member design groups

NOTE The use of suitable design groups for precast members is recommended. Using design groups will help speed the overall design process and allow for identical reinforcement amounts to be provided for each element within the group.

Why use precast design groups?

Precast beams and columns are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design - to standardise designs, reduce processing time, and reduce the volume of output created.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same number of stacks. Members in all member groups must be of similar lengths and cross-section.

NOTE Although precast members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect concrete beams and/or concrete columns as required.

Activating precast member design groups

Precast beam and column design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups
2. Select concrete beams and/or concrete columns to activate precast beam and/or precast column design groups.

Group management

Automatic Grouping

Precast beams and columns are grouped automatically according to a fixed set of requirements.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See:

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Precast design group requirements

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

Member type	Design group rules
Precast beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties.
Precast column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties.

Precast beam design

Specific aspects of the precast beam design workflow in Tekla Structural Designer are described below:

Section shapes

The following section shapes can be used within the Tekla Structural Designer - Tekla Tedds workflow.

- Rectangular
- L-Section
- Inverted L-Section
- T-Section
- Inverted T-Section

Beam arrangement

The Tekla Tedds calculations have no concept of the model geometry within Tekla Structural Designer and so there are no limitations on the design of curved beams in both major and minor axes. Continuous beams, cantilevered beams etc. are similarly supported by the workflow.

Concrete type

While you can apply both normal and lightweight concrete in the beam properties, precast beam design using lightweight concrete is currently beyond scope.

Nominal cover

The default nominal top, bottom and side cover values set in **Design Settings> Precast> Beam> Reinforcement Settings** are automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

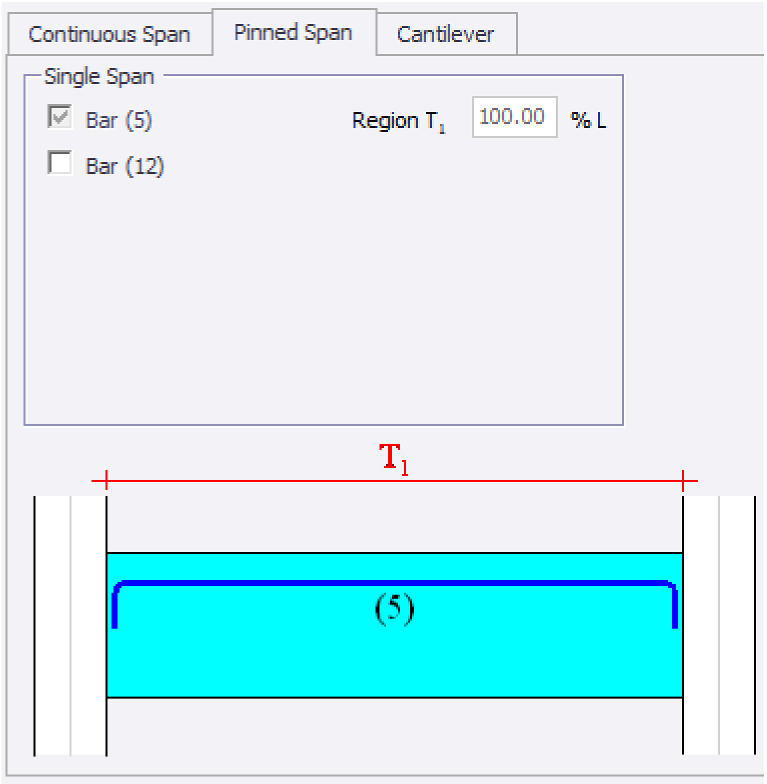
Reinforcement - longitudinal bar patterns

NOTE The reinforcement regions are considered to be the extent of the effective reinforcement. Anchorage and anchorage lengths are not considered in precast beam design within Tekla Tedds.

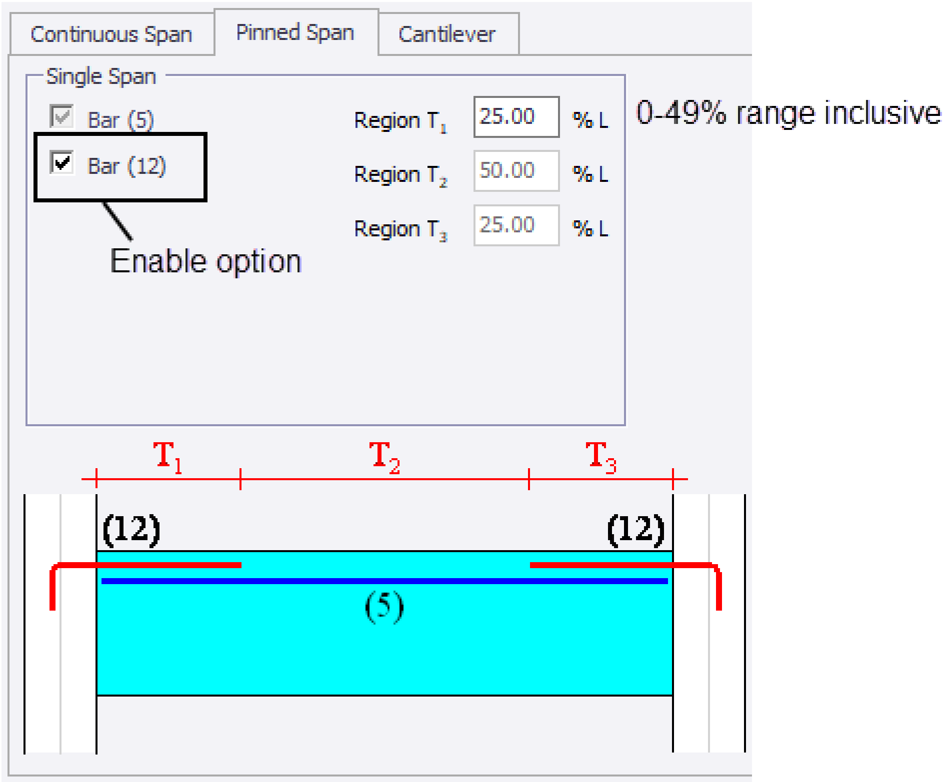
In **Design Settings> Precast> Beam** there is a one Standard Pattern available for top longitudunal reinforcement, Precast Top 1, and one Standard Pattern available for bottom longitudunal reinforcement, Precast Bottom 1 as illustrated in the figures below.

Standard Pattern of Top Reinforcement - Precast Top 1

Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.

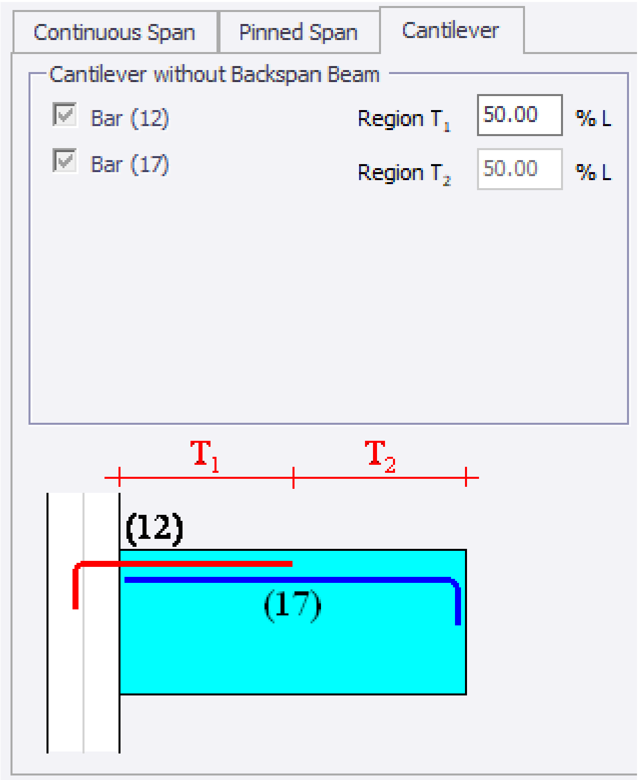


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (5)
Bar (12)
Region T1
100.00
% L
T1
(5)

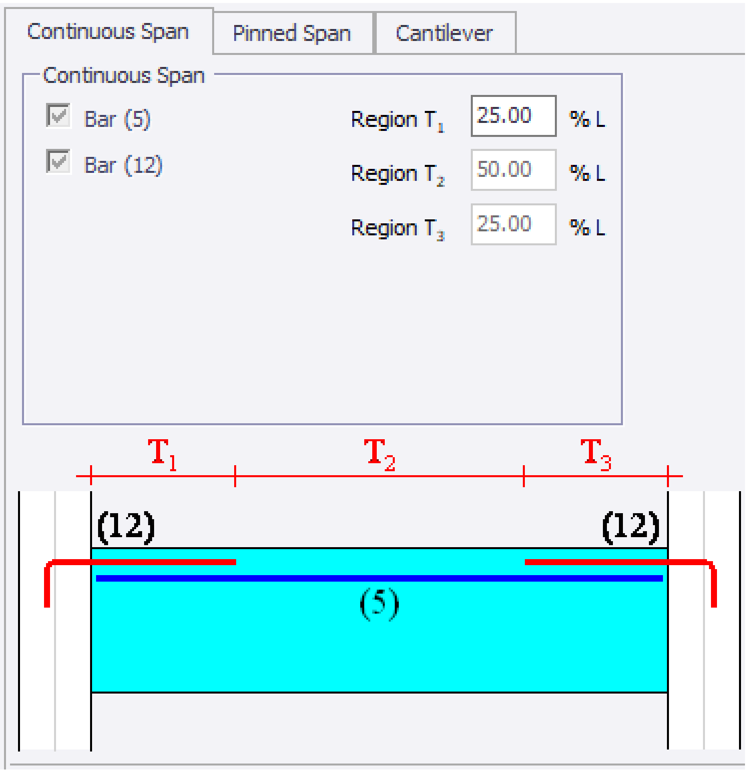


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (12). When enabled, the end region length is allowed to be 0%.

Cantilever span settings are applied to all spans with cantilever selected.

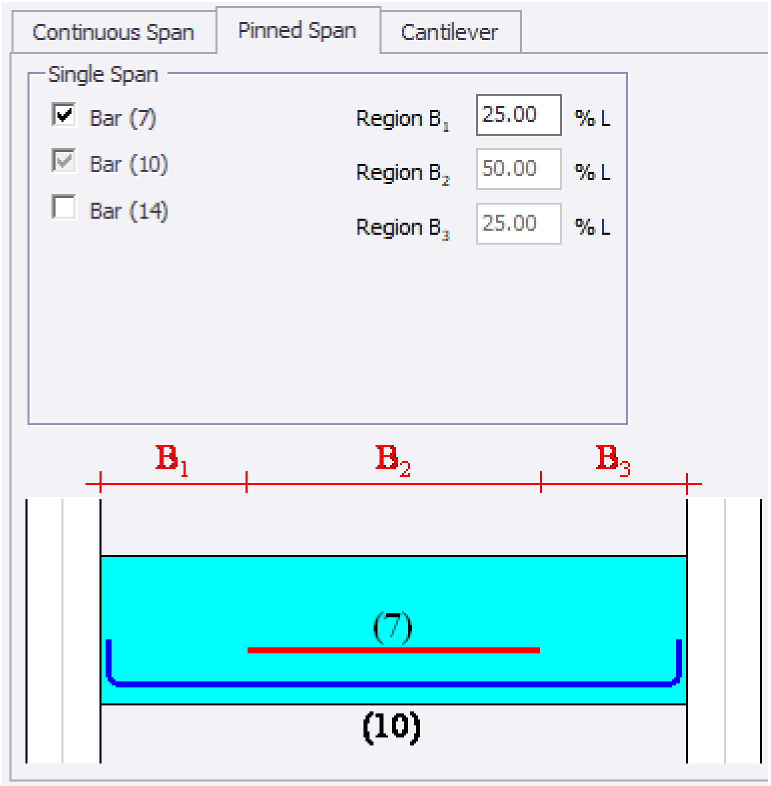


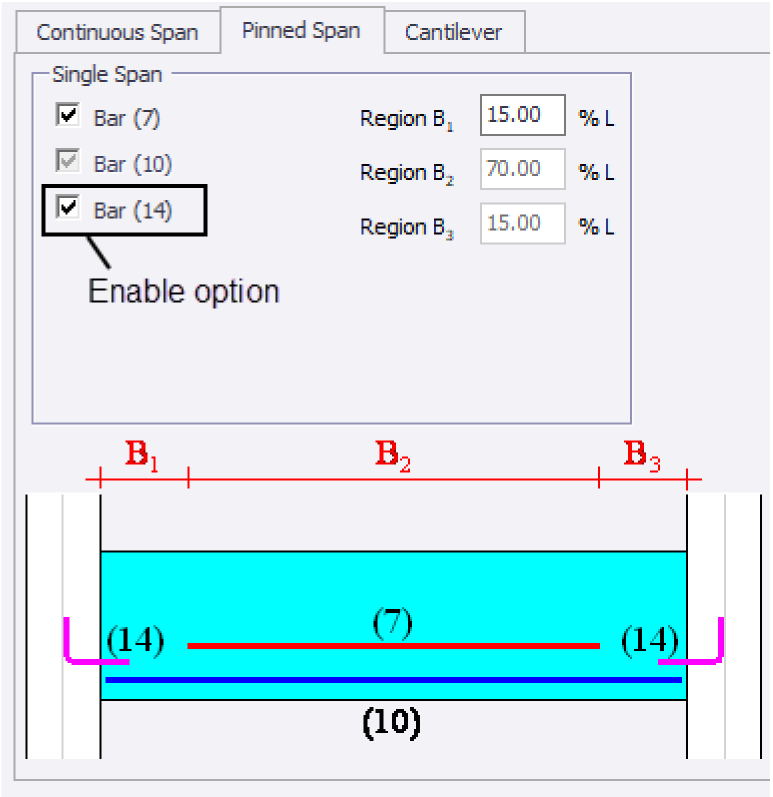
Continuous span settings apply to all other spans.



Standard Pattern of Bottom Reinforcement - Precast Bottom 1

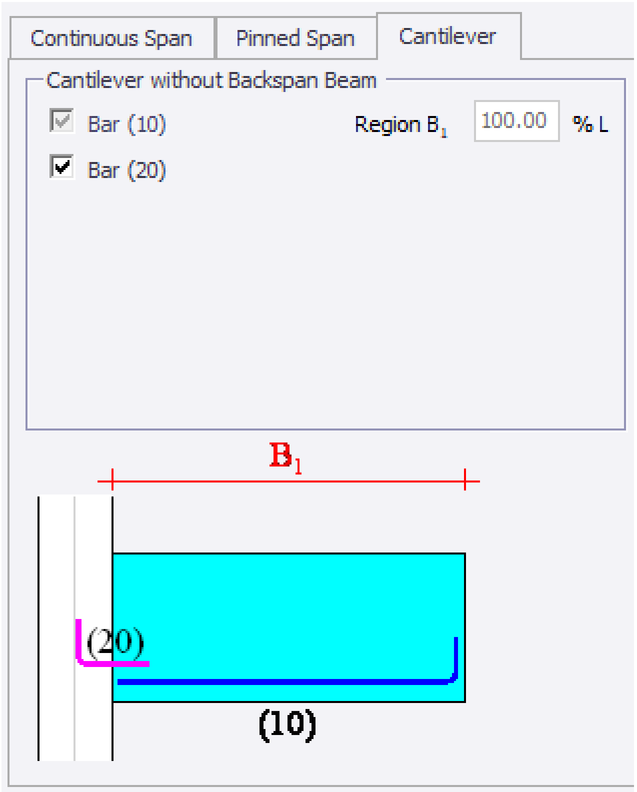
Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (7)
Bar (10)
Bar (14)
Region B_1 25.00 % L
Region B_2 50.00 % L
Region B_3 25.00 % L
B_1
B_2
B_3
(7)
(10)

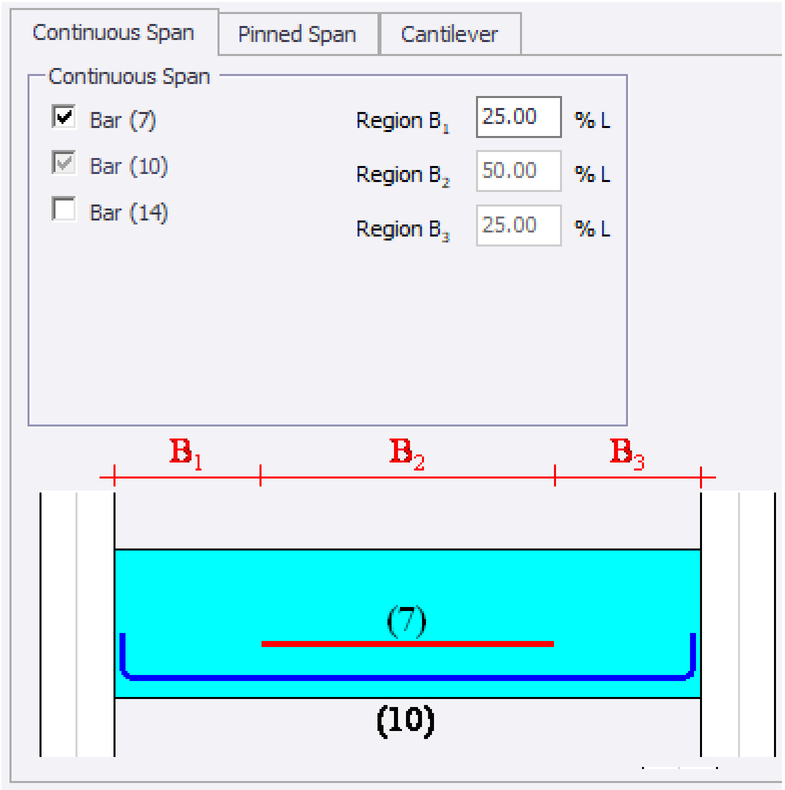


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (14).

Cantilever span settings are applied to all spans with cantilever selected.



Continuous span settings apply to all other spans.



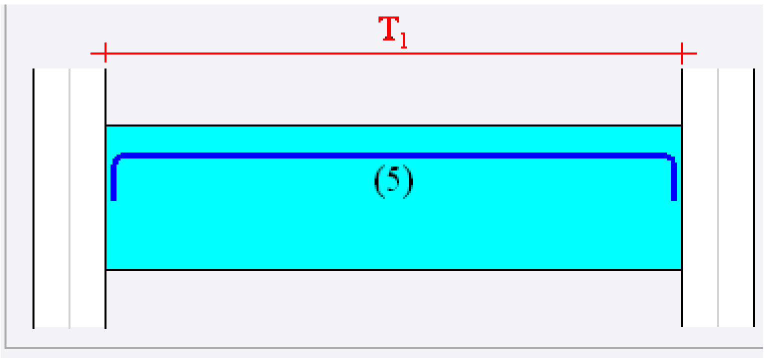
Design sections

An individual design section is created for each length of beam with similar reinforcement definitions. Each design section is then designed for the maximum forces in that region of similar reinforcement.

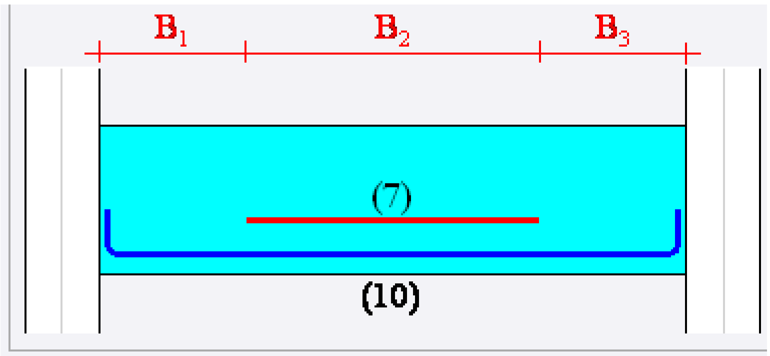
The numbers of design sections and their locations are determined by aggregating the different regions specified in the **Design Settings> Precast> Beam**: **Top Longitudinal Bar Pattern**, **Bottom Longitudinal Bar Pattern** and **Link Settings**.

Example: The default **Precast Top 1** and **Precast Bottom 1** longitudinal bar patterns for a Pinned Span are adopted without any continuity bars added as follows:

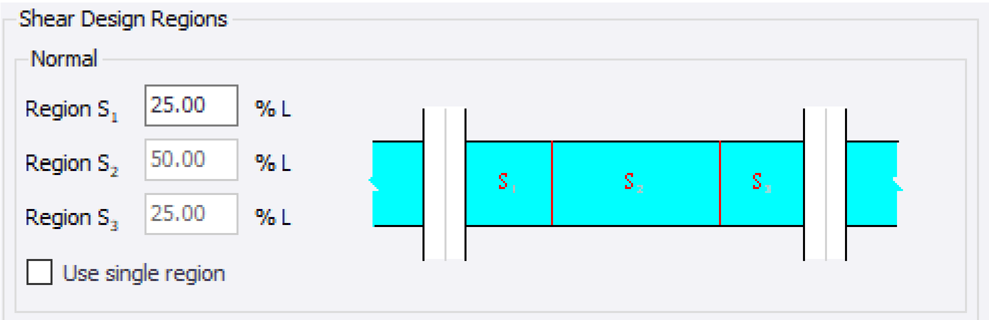
- **Precast Top 1**: single region 100% of the beam span.



- **Precast Bottom 1**: B1 and B3 regions set as 15% of the beam span.

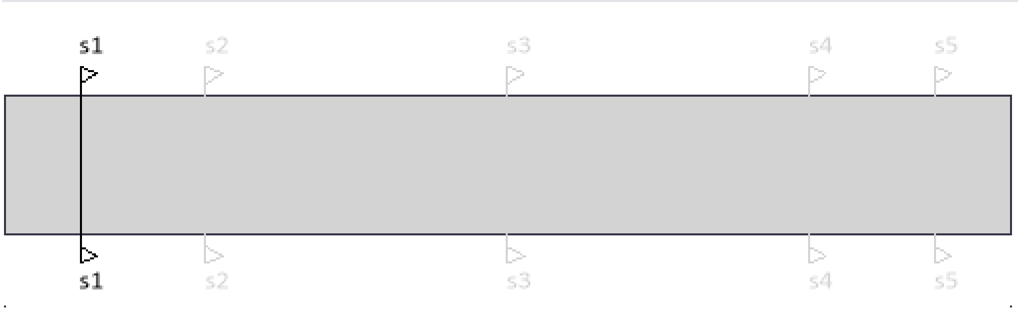


- Default **Link Settings** are also adopted: S1 and S3 regions set as 25% of the beam span.



For a pinned single span precast beam, different region boundaries would therefore exist at 15%, 25%, 75% and 85% of the span.

Consequently, when the span is designed in Tekla Tedds, 5 design sections would be created:



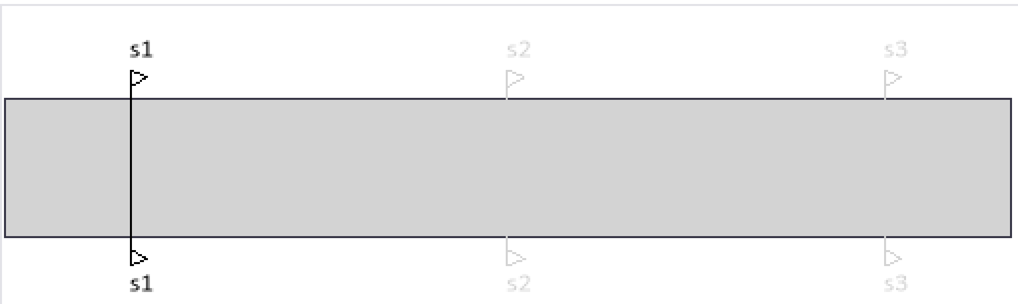
- s1 - positioned half way along the first bottom span region (7.5% of the span)
- s2 - positioned half way between the first and second region boundaries (20% of the span)
- s3 - positioned half way along the second and third region boundaries (50% of the span)
- s4 - positioned half way between the third and forth region boundaries (80% of the span)
- s5 - positioned half way along the last bottom span region (92.5% of the span)

NOTE In this example the first design section lies in the B1 bottom span region, so the positive design moment at s1 would be the maximum that occurs within B1; the second, third and forth design sections all lie in B2, so the positive design moment at s2, s3 and s4 would be the maximum that occurs within B2.

The engineer can of course adjust the standard bar patterns in the Tekla Structural Designer design settings to suit their requirements.

For instance, if in this example the **Precast Bottom 1** pinned span pattern were to be amended so that B1 and B3 regions were set as 25% of the span; because the bottom and link regions would then coincide, region boundaries would only exist at 25% and 75% of the span.

Consequently, when the span is designed in Tekla Tedds, only 3 design sections would be required:



- s1 - positioned half way along the first region (12.5% of the span)

- s2 - positioned half way between the first and second region boundaries (50% of the span)
- s3 - positioned half way along the last region (75% of the span)

NOTE In the above elevation diagrams:

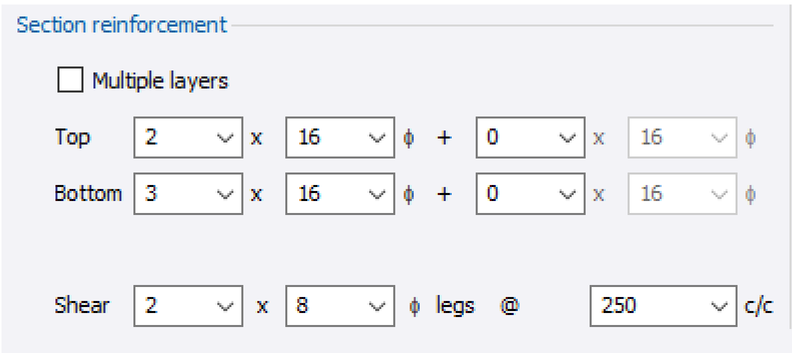
- A black section mark indicates the selected design section is passing the design criteria.
- Grey section marks indicate unselected design sections which are passing.
- A red section mark would indicate a design section which is failing.

Default reinforcement in the Tekla Tedds calculation

Regardless of beam shape and size imported from Tekla Structural Designer, the reinforcement in each beam member is always defaulted to the same values Tekla Tedds calculation.

This default reinforcement is:

- Top: 2x 16 dia bars
- Bottom: 3x 16 dia bars
- Shear: 2 legs of 8 dia bars at 250 cross centres



NOTE The Tekla Tedds calculation contains some limitations to the error trapping when it comes to maximum reinforcement values. Although the main bars in both the top and bottom of the beam have a check allocated to prevent unrealistic distances being entered, the shear links have no such limitation.

The engineer is expected to use their own knowledge to place in shear link values and distances which are both realistic and constructible.

NOTE The Tekla Tedds calculation contains some limitations to the section diagram. It may be noted that the diagram is never altered if the number of shear legs is increased from the default value of 2 legs. It should be noted that shear link legs are indicative only within Tekla Tedds and in reality there are a variety of ways that reinforcement can be detailed. The Tekla Tedds calculation therefore chooses to omit additional shear reinforcement in the section diagram and to flag it instead in the notes and the calculations.

Lifting checks

Lifting checks are disabled by default. This has been done to ensure a rapid design of grouped beams can take place. If lifting checks are required then simply enable the switch within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check for beam members makes allowance for reinforcement for both bending and shear checks.

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Positive moment
  - Positive design moment
  - Positive quasi-permanent moment
  - Redistribution ratio
- Negative moment
  - Positive design moment
  - Negative quasi-permanent moment
  - Redistribution ratio
- Shear
  - Maximum Design shear force
  - Design shear force
- Torsion
  - Design torsional moment

NOTE Minor axis forces (as defined in Design > Settings > Design Forces) are not used in the design of the member. Where minor axis forces are present within a beam, a warning is displayed within the process dialog of Tekla Structural Designer.

Other precast beam properties

The following properties which are common to both cast-in-place and precast beams, are not transferred to the Tekla Tedds calculation, but still require consideration as they will impact the analysis in Tekla Structural Designer.

- Cracked, partially cracked, and uncracked concrete members (page 255)
- Design parameters (page 269)

Precast column design

Specific aspects of the precast column design workflow in Tekla Structural Designer are described below:

NOTE The current Tekla Tedds calculation makes no allowance for any shear checks.

Section shapes

Whist a variety of different section shapes can be defined in Tekla Structural Designer, the Tekla Tedds calculation only supports the design of rectangular and circular column sections.

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Nominal cover

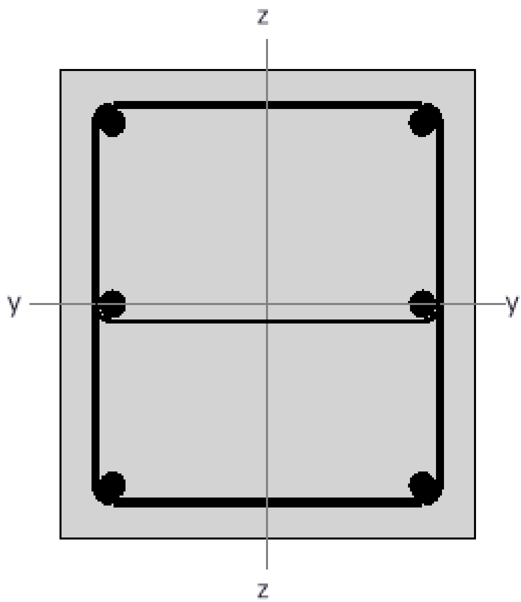
The default nominal cover value set in Design Settings> Precast> Column> Reinforcement Settings is automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

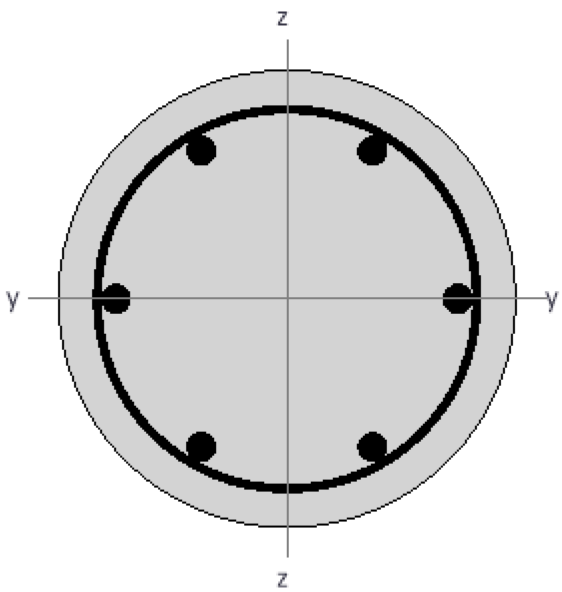
Default reinforcement in the Tekla Tedds calculation

Regardless of the column size imported from Tekla Structural Designer, the initial default reinforcement for all stacks in the Tekla Tedds calculation is as follows:

- Rectangular columns: a 2x3 formation of 25mm dia rebars with 8mm shear links.



- Circular columns: 6x 25mm dia rebars with 8mm shear links.



NOTE Tekla Tedds has no concept of multiple stack columns and Tekla Structural Designer has to associate separate Tekla Tedds calculations with each stack. It is the responsibility of the engineer to ensure that rebar on adjacent stacks are of appropriate quantities and sizes to ensure correct detailing can take place at the construction stage. It should be noted that Tekla Structural Designer does not hold any reinforcement information relating to precast members. Reinforcement is completely managed by the Tekla Tedds calculations.

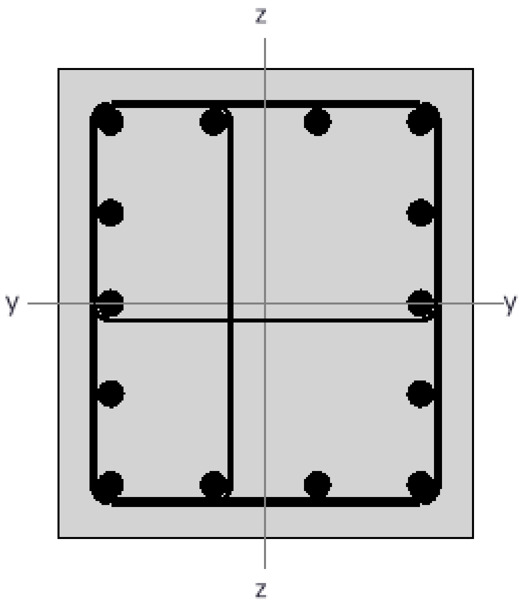
The maximum number of bars which can be entered in either direction for rectangular columns is 10. For circular columns, the maximum total number of bars is 20. Please note that these restrictions will have an effect on the maximum allowable column size for the design workflow.

There is a basic minimum distance calculation within the Tekla Tedds document relating to bar amounts to avoid unrealistic values of rebar from being selected. This amount is to be the minimum of:

- K1 x bar dia
- 20mm
- k2 x max aggregate size

NOTE For values of coefficients k1 and k2, refer to clause 8.2 of the Eurocode.

The number of link legs shown in the Tekla Tedds calculation will increase or decrease depending on the number of reinforcing bars assigned within the document.



NOTE The link legs are displayed to indicate required confinement reinforcement for vertical bars in compression, but have no effect on the overall design process.

Lifting Checks and Splice Design

Lifting checks and splice checks are disabled by default. This has been done to ensure a rapid design of grouped columns can take place. If these checks are required then simply enable the switches within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check is performed against the unreinforced area of the column, hence there is no input for the shear link centres within the calculation. (Bending checks (for splice design) do include reinforcement allowances within the design process.)

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Axial load, 1st case (Max value)
- Axial load, 2nd case (Min value)
    - Where tension exists in the column, the lowest compression value is populated into this parameter and the tension value ignored. A warning will be displayed in the process log within Tekla Structural Designer.

- Moment about y axis at top
- Moment about z axis at top
- Moment about y axis at bottom
- Moment about z axis at bottom

NOTE Tekla Structural Designer populates values from all design combinations. For columns subjected to low axial loading, the moment capacity will not be as high as a column subjected to larger axial loading. Low axial load and high moment can easily be a critical combination for concrete column design.

Other precast column properties

The following properties which are common to both cast-in-place and precast columns, are not transferred to the Tekla Tedds calculation, but require consideration as they impact the overall performance of the model.

- Cracked, partially cracked, and uncracked concrete members (page 255)
- Design parameters (page 275)
- Stiffness (page 277)
- Sway/Drift Checks (page 278)

Precast column connection eccentricity moments

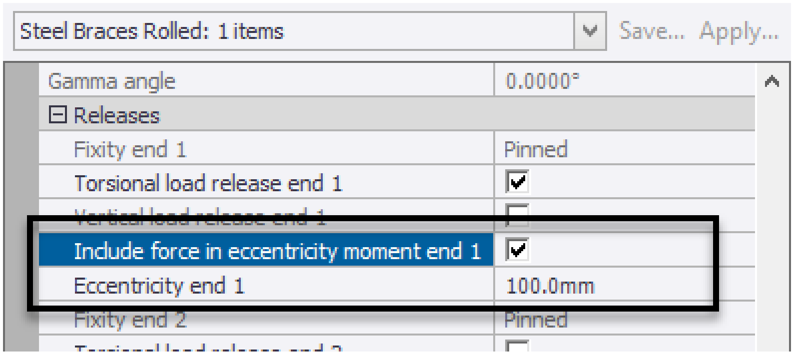
Overview

Nominal eccentricity moments arise in precast columns due to the reactions from pin ended beams being applied at eccentricities to the column centerline. The eccentricity moments are calculated for all headcodes, and can be interrogated in the Load Analysis View.

If working to Eurocodes, these eccentricity moments are also taken into account when precast columns are transferred to Tedds for design.

These moments do not come directly from the global analysis but instead are calculated at the ‘load analysis’ post-processing stage as follows:

- At each level the eccentricity of each connection is specified as a user defined offset from the column face. If rigid zones are not being used this is increased by half the depth of the supporting column.
- At each level the pinned beam end reactions connecting to the column at each face are determined.
- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

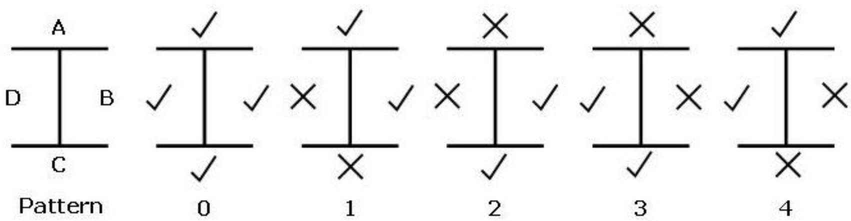
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are 'true' patterns switching live load 'on' and 'off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design to Eurocodes

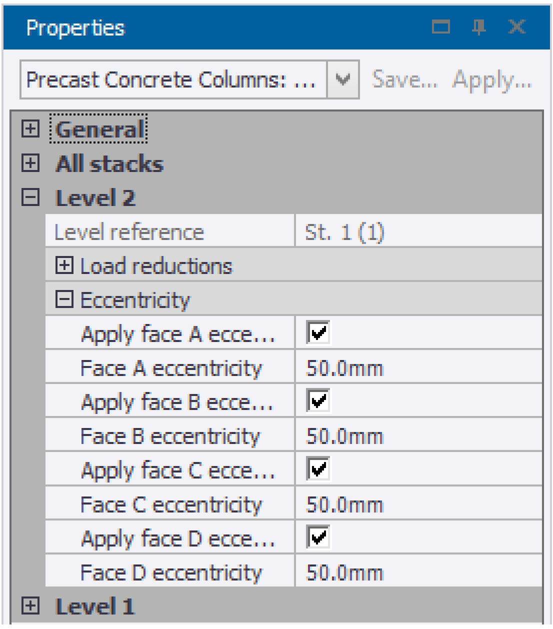
In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.

Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



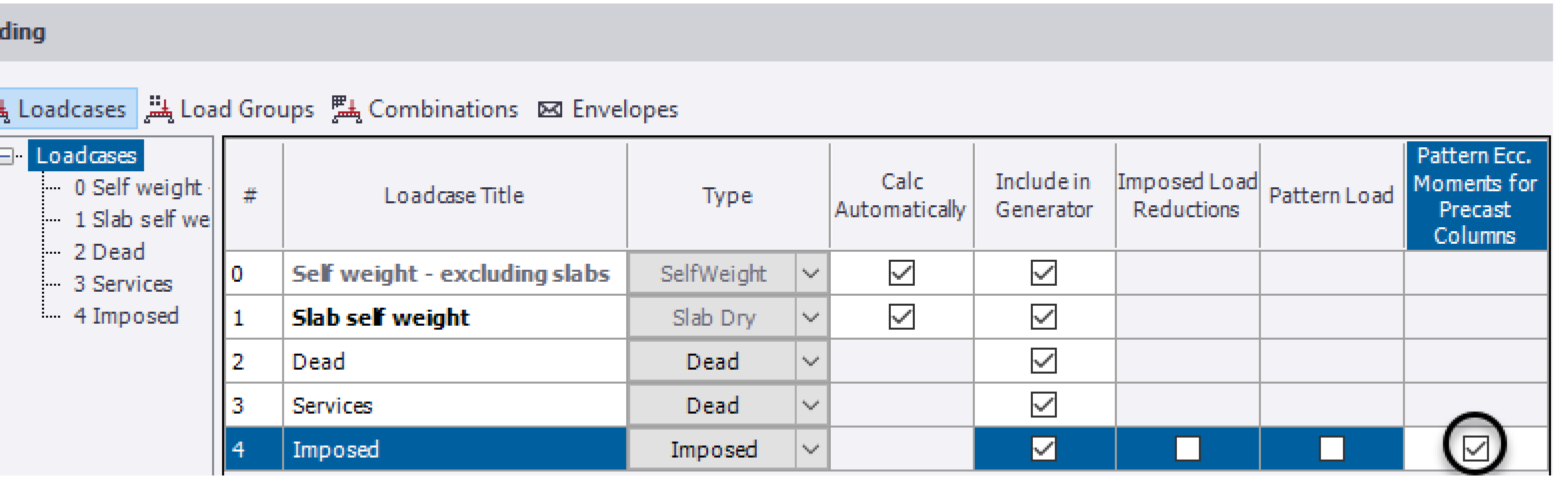
If you uncheck the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 219)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
    a. Click **Model Settings > Loading > General**
    b. Select **Use patterning of eccentricity moments for precast columns**
    c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
    a. Select a live loadcase that you want to be patterned
    b. Select **Pattern Ecc. Moments for Precast Columns**
    c. When patterning has been selected for each of the required loadcases, click **OK**

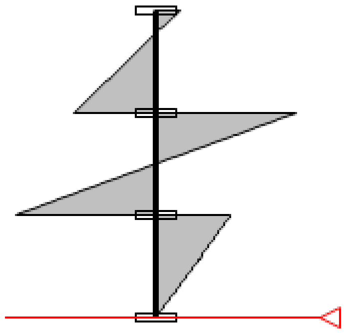


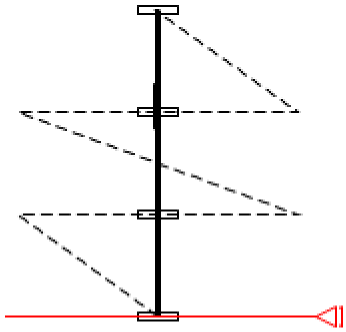
Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View.

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:



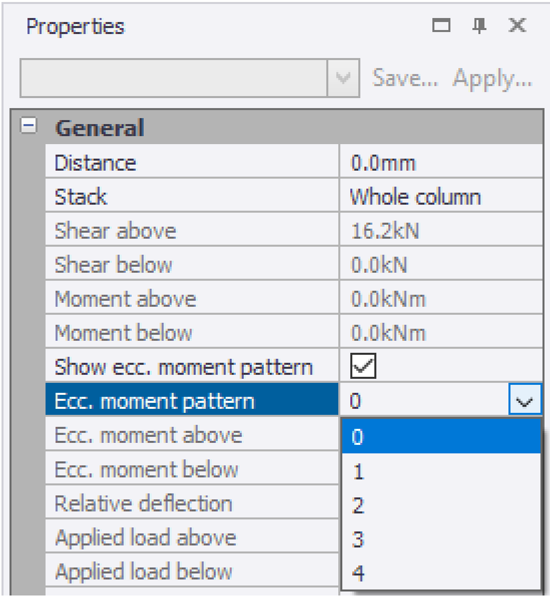


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.



Precast member design commands

By right clicking over the required member in a view or appropriate branch of the **Project Workspace**, the following interoperative design commands can be accessed:

- Design using Tekla Tedds
- Check using Tekla Tedds
- Export to Tekla Tedds
- Clear Tekla Tedds Data

NOTE Initially only Design using Tekla Tedds command is shown, but once this has been run the other commands then become available.

Each of the above offers a sub-menu of choices, depending on context:

- > Model
- > Member
- > Group
- > Selection
- > *<Substructure name>*

NOTE Check using Tekla Tedds > Model is repeated on the Design ribbon tab as  Check in Tedds

1.9 Timber member design handbook

Timber structures can be modelled and analyzed in Tekla Structural Designer. If a licence of Tekla Tedds is available, timber beams, columns and braces can then also be designed.

NOTE The following limitations and assumptions apply:

- The user should ensure the locale is set correctly in Tekla Tedds before attempting to run the design from Tekla Structural Designer.
- Design is supported for NDS LRFD, NDS ASD, base Eurocode, and the following Eurocode National Annexes: UK; Ireland; Norway; Finland; Sweden.
  - If the Singapore or Malaysia Eurocode National Annex is selected, the calculation will adopt base Eurocode recommended values.
- All timber sections should be consistent with the appropriate to the selected head code.
- If working to the NDS code, size classifications taken from the NDS supplement 'Design Values for Wood Construction' are used in the Tedds calculation to distinguish which grades are valid for particular timber species. The same classification rules are not implemented in Tekla Structural Designer itself. Therefore, after a member has been designed in Tedds, if the member size or grade is subsequently changed in Tekla Structural Designer the associated Tedds calculation is deleted, so that the member cannot be checked and a redesign is required.
- Column design: If designing a multi-stack column the user must indicate a splice in the stack properties **before** running the design if a change in cross section is required between stacks. If a splice has not been specified the change in section size will not be correctly returned to the Tekla Structural Designer model.

The following topics are covered in this handbook:

Timber member design workflow

The essentials of design of timber members using Tekla Tedds can be thought of as being very similar to Interactive concrete member design (page 285) within the program:

- Just like the concrete beam/ column/ wall interactive design dialog, the Tedds Timber member design calculation interface lists the design forces and settings for a selected member/ group, populated from the model and analysis, the design pass/fail status and individual check results and utilization ratios. You can make changes in the interface - such as to section size and/ or grade - and immediately see the design results for these. All the data you input and changes you make are updated to and stored in the model. You can then re-analyze and run a check on all designed members to ensure the design results are up to date. You can perform design and analysis loops as necessary so that all members are passing, and to cater for any other changes to the model or loading.

- Design Groups (page 552) work in a similar manner also - these are automatically created for Timber members and listed in the Project Workspace Groups Tree. The resulting groups can be reviewed and customized as necessary. Group design is enabled by default (in **Design Settings > Design Groups > Timber Beams/ Columns/ Braces**) and can be run both via the model and from the Groups Tree.

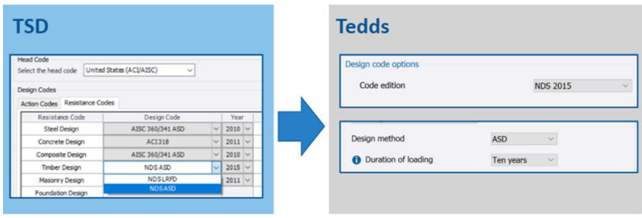
Key aspects of the Tedds integrated timber design process and workflow in Tekla Structural Designer are:

Set the timber design code

The timber design code is set from the **Home** ribbon by clicking **Model Settings > Design Codes > Resistance Codes**.

- If working to **US codes**, there are two different methods, ASD or LRFD to choose from and also a choice of code year to apply. The design forces consider only the selected design method.
- If working to **Eurocodes**, the timber design code will be applied with a national annex that is appropriate to the head code that has been selected.

The selected design code is automatically applied in all Tedds timber member designs.



Define and place timber members

To place timber members, select the appropriate member type from the **Timber** group on the **Model** ribbon, ensure the values in the **Properties** window are as required and then pick the points to locate the members in position.

Note the following with regard to the properties:

- **Section shapes** - When defining a timber model in Tekla Structural Designer it may appear that the range of timber sections is somewhat limited. This is because Tekla Structural Designer doesn't currently have a facility to specify timber sections by simply inputting a breadth and depth. Instead you have to select the required size from Tekla Structural Designer's database of timber sections.

  While this might seem to be a significant limitation, it is actually not too great an issue, because the Tedds timber calculation allows you to override the original size to input the breadth and depth you require. Then when the member data is transferred back from Tedds to Tekla Structural Designer the new size is automatically added to the timber section database.

  Once the new section is in the section database it becomes available for future Tekla Structural Designer models also.

- **Timber fabrication and grade** - You can specify timber, glulam, or structural composite lumber fabrication types.

  Initially the grade must be selected from the existing grades in the Tekla Structural Designer materials database, although if the grade is changed in the the Tedds timber calculation, when the member data is transferred back from Tedds to Tekla Structural Designer the new grade is automatically added to the materials database.

  NOTE If a situation arises such that the material properties in the programs don’t align, Tedds ignores unmatched data and selects calculation defaults.

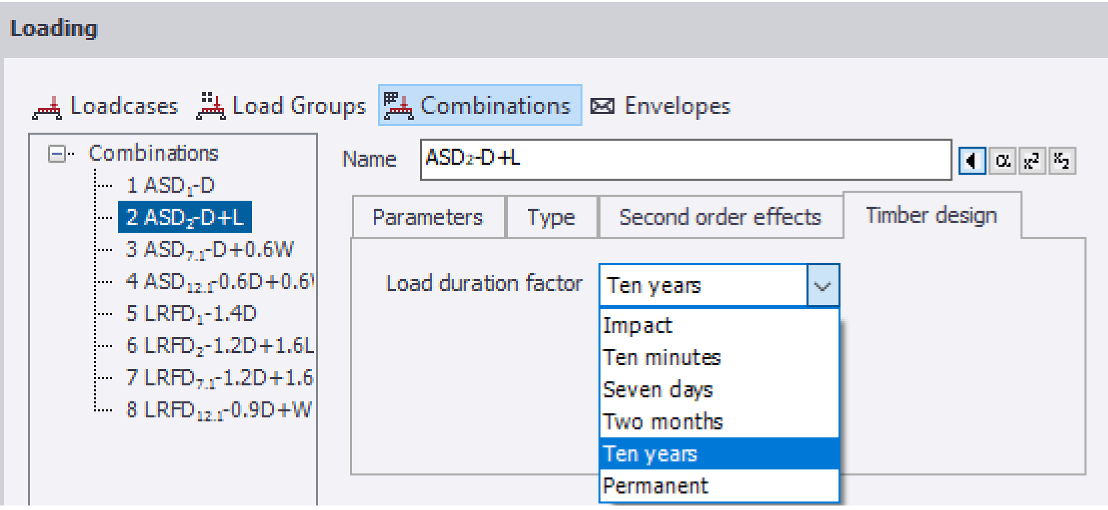
- **Multi-stack columns** - splices should be specified in the stack properties when the column is being defined if a change in cross section is required between stacks. If a splice has not been specified before the column is designed in Tedds the change in section size will not be correctly returned to the Tekla Structural Designer model.

Create load combinations and set load duration/time effect factors

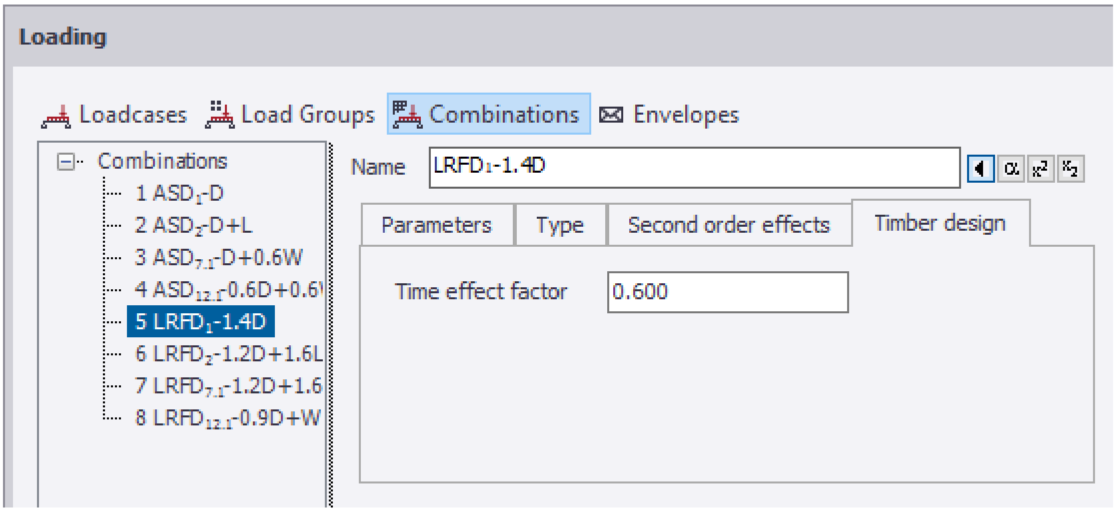
Loads are applied in the standard manner. Once load combinations have been created you should review their load duration/ time effect factors.

These factors are set on the **Timber design** tab on the **Combinations** page of the **Loading** dialog once you have selected an individual combination from the list in the dialog. The value is set automatically according to the constituent loadcases.

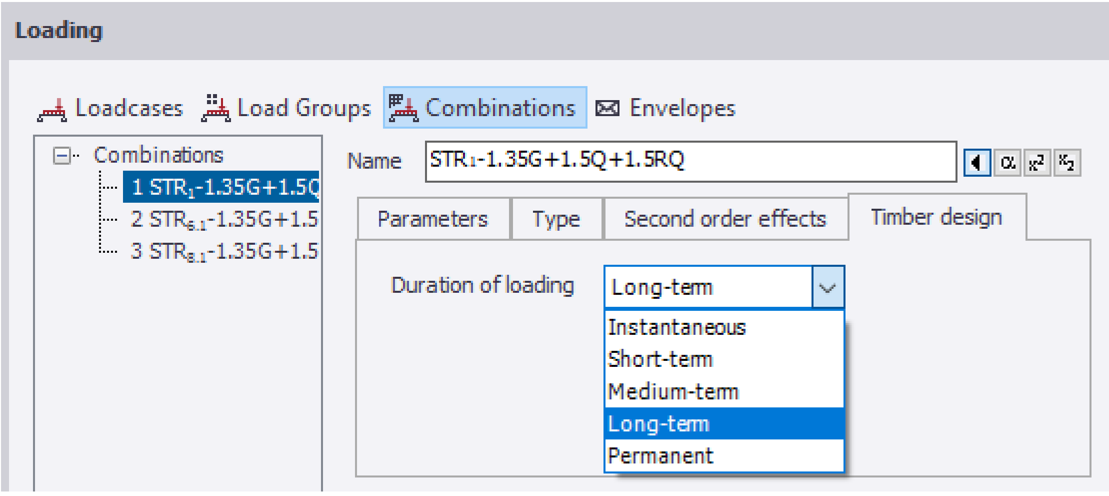
- **ASD combination:**



- **LRFD combination:**



- **Eurocode combination:**



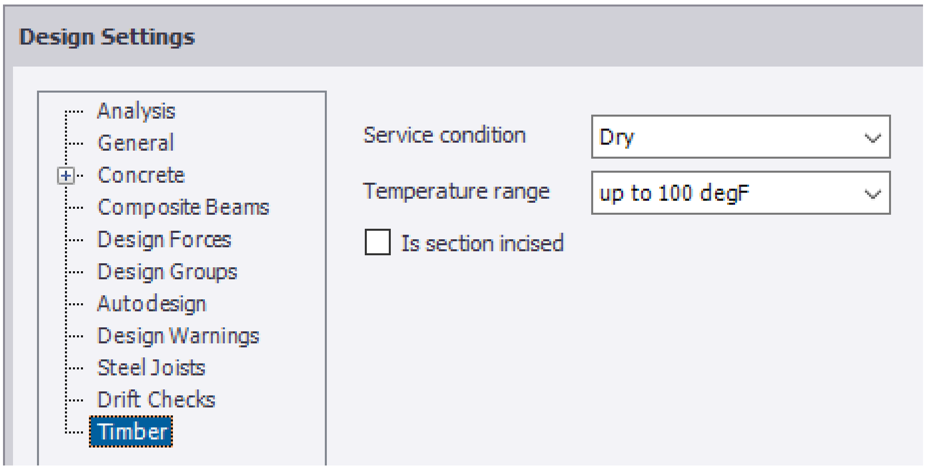
You have control to adjust the default value on a combination by combination basis.

NOTE The actual factor that gets applied in the Tedds calculation will be the one that is associated with the worst case load combination.

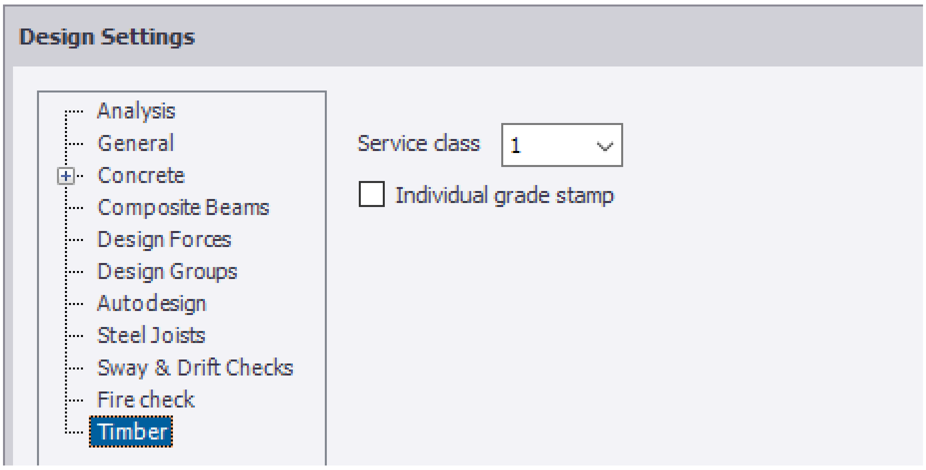
Configure timber design settings

Certain timber design settings which apply to the entire building can be specified on the .

- If working to **US codes**:



- If working to **Eurocodes**:



By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds timber calculation as it is run.

You still have control within individual calculations to adjust these settings on a member by member or group basis.

NOTE Tedds data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established.

If you change timber design settings after a member has already been designed you would need to manually clear the Tedds data from the design in order to have it use the revised design settings. To do this highlight the member, right-click and select Clear Tekla Tedds Data from the context menu.

Configure timber groups

When timber members are placed in the model, to make editing easier they are automatically grouped according to a set of rules (page 554).

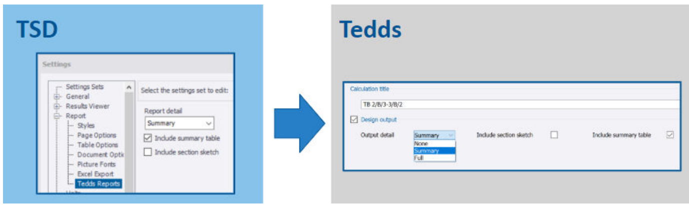
These initial groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can add new groups and move members between groups as required.

Groups are not only useful for editing, they can (optionally) be activated for design purposes also. There are a number of reasons why you would choose to do this.

- Why use timber design groups? (page 552)
- Activating timber member design groups (page 552)

Set the Tedds results output level

You can choose the output level for the Tedds timber calculations in advance by clicking **Home > Settings > Report > Tedds Reports**



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds timber calculation as it is run.

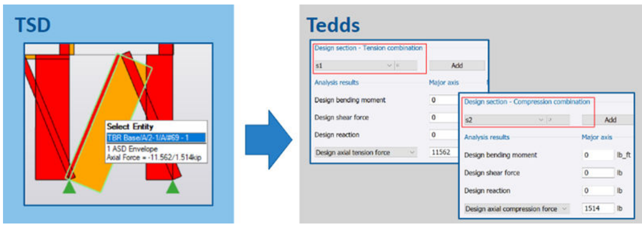
The setting applies to all Tedds linked calculations (precast and timber).

Establish design forces by running the analysis

Timber members can only be designed provided a set of analysis results exist. These can be generated from the **Analyze** ribbon by running .

A design force envelope is established, which if designing to the US head code considers the selected design method (ASD or LRFD) only.

All critical load combinations are considered in one Tedds calculation.



Provided load combinations have been created, once analysis has been performed the **Design using Tekla Tedds** options become available.

Design using Tekla Tedds

After Analysis, Design or Check using Tekla Tedds can be initiated from both the Project Workspace and any graphical view of the model:

You can right-click over any individual member in the Project Workspace, or make a graphical selection, then right-click to open the context menu listing the Tekla Tedds design options. You can then select Design of the Member, Group or Selection as appropriate. Design of a Selection can also be used for example to design all members of a truss - which can be selected with one click - in one operation.

- Design a selection (page 547)
- Design a group (page 548)
- Design a grouped member (page 550)

- Design an ungrouped member (page 550)
- Design model (page 551)

Check the design after changes

If changes are made to the model you can run a 'check' design to determine if the existing sections are still sufficient. A check is quicker to perform than a design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

You can check the whole model from the **Design** tab, or right-click over any individual member in the Project Workspace, or you can make a graphical selection, then right-click to open the context menu listing the Tekla Tedds design options as shown below. You can then select Design of the Member, Group or Selection as appropriate.

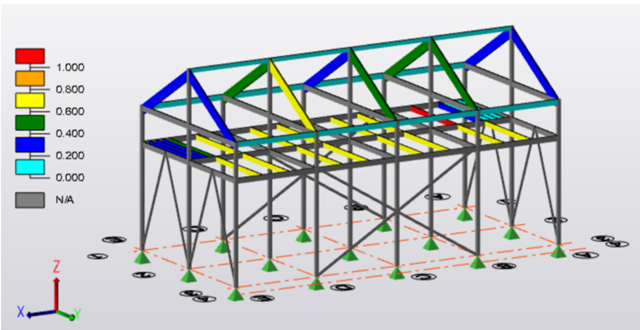
- Check model (page 551)
- Check a selection (page 551)
- Check a member (page 551)
- Check a group (page 551)

The updated utilizations can then be reviewed in a Review View.

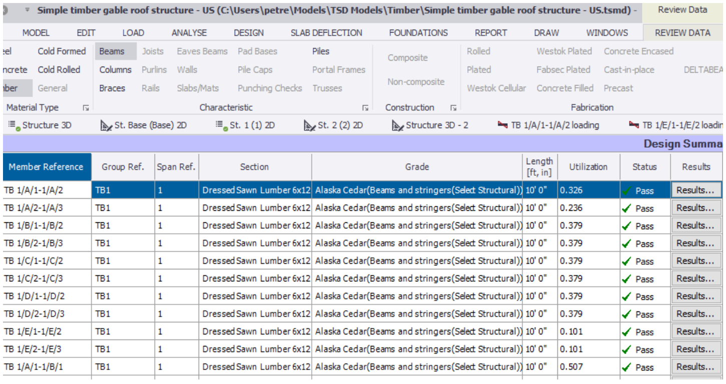
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

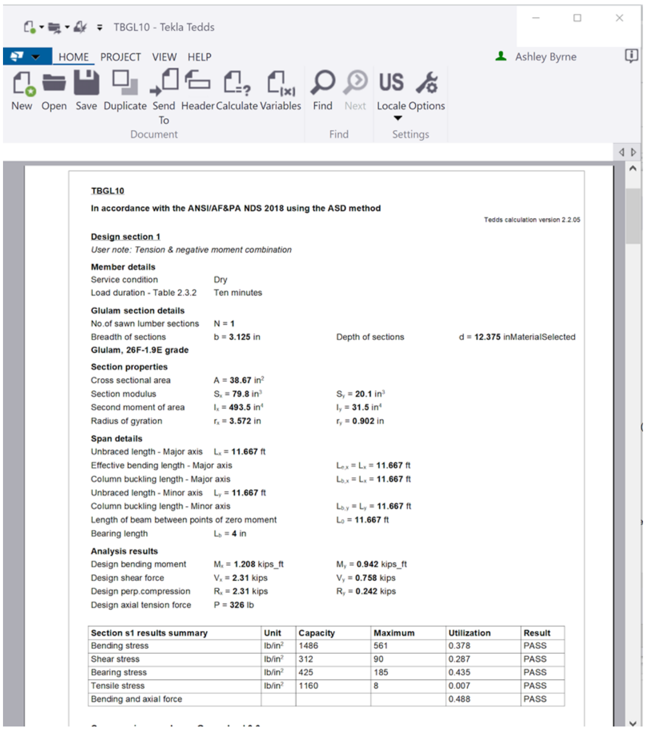
At this stage you can display member design status and utilization ratios from a Review View:



You can also display the design status tabular results:



Detailed Tekla Tedds calculations for timber members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds.



Design timber members using Tekla Tedds

Design a selection

To design several timber members or groups in one go:

1. Select all the timber members you want to design.

2. Right click and select Design using Tekla Tedds> Selection

   Tekla Structural Designer highlights the first member or group in the selection and a Tedds dialog opens displaying the design forces at the member's first design section.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whether or not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.
4. When all sections are satisfactory, click **Finish**.

   Tekla Structural Designer highlights the next member or group in the selection and the Tedds dialog re-opens to allow it to be designed.

   Continue in the same way until all the selected members/groups have been designed.

   NOTE If a section is changed during the design to a size that does not exist in Tekla Structural Designer's Timber Section database, it will be automatically added to the local section database that exists for the current model only.

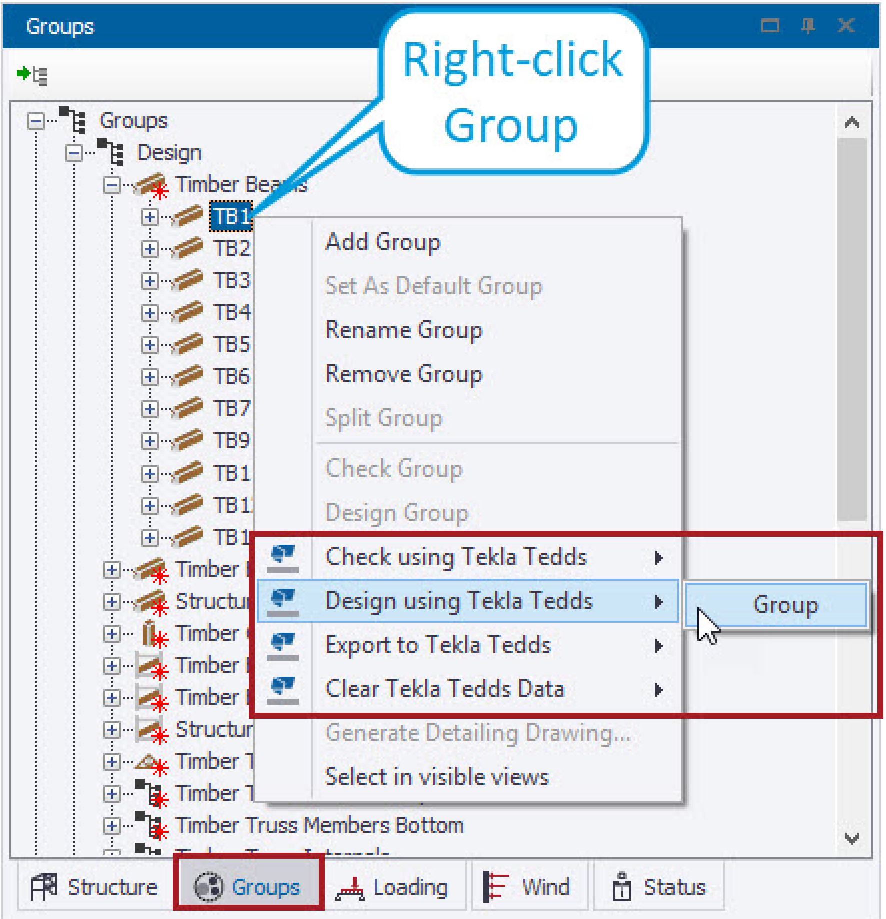
Design a group

If you have activated timber member design groups, each group can be designed as follows:

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, expand **Timber Beams**, **Timber Columns**, or **Timber Braces** as required.

   NOTE When designing timber brace pairs, the groups for these are always located under **Timber Braces** and not **Timber X Braces, A Braces, V Braces, K Braces.** These latter groups are used for editing purposes only, but not design.

3. In the **Design** tree, right-click the timber member group you want to design.
   a. It can be useful at this point to click **Select in visible views** to confirm all the group members.
4. In the context menu, select Design using Tekla Tedds> Group.



This design gathers the analysis results for all members in the group and collates them into one set of critical design forces. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

5. Using the results preview, design the first design section in the group for its critical design forces, then if there is a second section design that also.

6. When all sections are satisfactory, click **Finish**.

   If the section size was changed during the design, all members of the group are updated to the new section.

   Irrespective of whether the section has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

7. In the Review View, review the utilization ratios for all members in the group - if these indicate an efficient design, the design can stop at this point.

NOTE For columns in particular, the envelope of critical design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

If the utilizations indicate the existing grouping is not very efficient, you can investigate alternative grouping arrangements by adding extra groups as necessary and re-allocating members between the groups. You might also consider individually designing grouped members (as described below) when optimizing the groups.

Design a grouped member

If after running a grouped design some group members have a lower than desired utilization, you can re-design them individually to investigate the effect optimizing them would have on the rest of the group.

1. Highlight a group member which has a lower than desired utilization, right click and select Design using Tekla Tedds> Member

   The analysis results of the selected member are used to establish the set of critical design forces.

2. Optimize the design of the member for its forces.
3. Click **Finish**.
4. If the section size was changed during the design, all members of the group are updated to the new section and checked against only the loads that they see individually.
5. In the Review View, review the new utilizations for the group members - note that some of the group might now be failing.
6. Using the new utilizations to better inform you choice, add extra groups as necessary and re-allocating members between the groups.
7. Run Design using Tekla Tedds> Group for each of the new groups.
8. Iterate the process as necessary until the groups are fully optimized.

Design an ungrouped member

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member
2. Tekla Structural Designer highlights the member and a Tedds dialog opens displaying the design forces at the member's first design section.
3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.

4. When all sections are satisfactory, click **Finish**.
5. If the section size was changed during the design, it is updated to the new section in the Tekla Structural Designer model.
6. The member status and utilization are displayed in the member tooltip.

Design model

If you want to design every timber (and precast) member in the model in one go:

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Design using Tekla Tedds > Model**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check timber members using Tekla Tedds

Check model

To check all the existing Tedds member designs,

1. Click Check In Tedds from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.
2. Right click and select **Check using Tekla Tedds> Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds> Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds> Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

Timber member design groups

Why use timber design groups?

Timber beams, columns and braces are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design -
   a. to standardize designs,
   b. to reduce the volume of output created,
   c. to speed up the design process, particularly so in the case of medium to large size models.

When groups are used for design:

- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

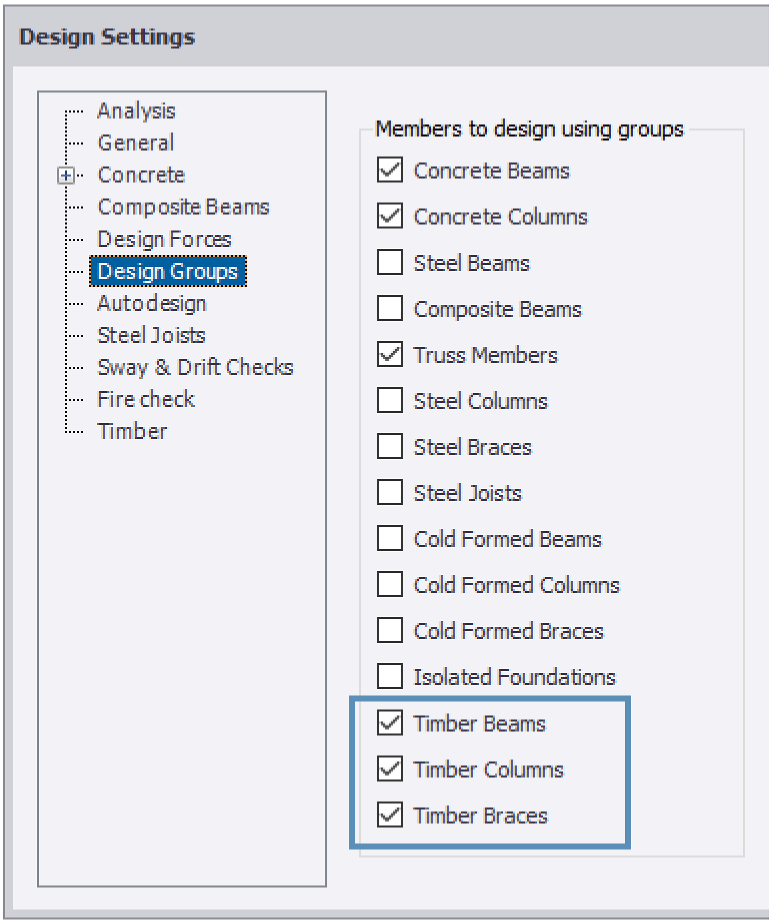
NOTE Although timber members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

Activating timber member design groups

Timber beam, column and brace design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups

2. Select **Timber Beams**, **Timber Columns**, **Timber Braces** as required.



Group management

Automatic Grouping

Timber beams, columns and braces are grouped automatically according to a fixed set of requirements, including section size, grade, and member length.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are of sufficiently similar length to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See:

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup Members

If you have made changes in the model that impact on grouping of a specific member type, you can update the affected groups from the Groups tab of the Project Workspace by right-clicking **Timber Beams**, **Timber Columns**, or **Timber Braces** as required and selecting **Regroup Members**.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Regroup ALL Model Members

If you have made changes in the model that impact on grouping, you can update all affected groups accordingly from the Groups tab of the Project Workspace, by clicking Re-group ALL Model Members. (Located at the top of the Groups tab.)

NOTE Any manually applied grouping will be lost if you elect to re-group!

Timber design group requirements

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

Member type	Design group rules
Timber beam	• A beam may be in only **one** design group. • All beams in the group must have an identical cross section. • All beams in the group must have an identical grade. • All beams in the group must have an identical number of spans. • Individual span lengths must be the same across the group.
Timber column	• A column may be in only **one** design group. • All columns in the group must have an identical cross section. • All columns in the group must have an identical grade. • All columns in the group must have an identical number of stacks. • Any splices must be located in the same stacks for all columns of the group. • Individual stack lengths must be the same across the group.

Member type	Design group rules
Timber brace	• A brace may be in only **one** design group. • All braces in the group must have an identical cross section. • All braces in the group must have an identical grade. • All braces in the group must have an identical span length.

Timber member design commands

By right clicking over the required member in a view or appropriate branch of the **Project Workspace**, the following interoperative commands can be accessed:

- Design using Tekla Tedds
- Check using Tekla Tedds
- Export to Tekla Tedds
- Clear Tekla Tedds Data

NOTE Initially only Design using Tekla Tedds command is shown, but once this has been run the other commands then become available.

Each of the above offers a sub-menu of choices, depending on context:

- > Model
- > Member
- > Group
- > Selection
- > *<Substructure name>*

NOTE Check using Tekla Tedds > Model is repeated on the Design ribbon tab as  Check in Tedds

1.10 Foundation design handbook

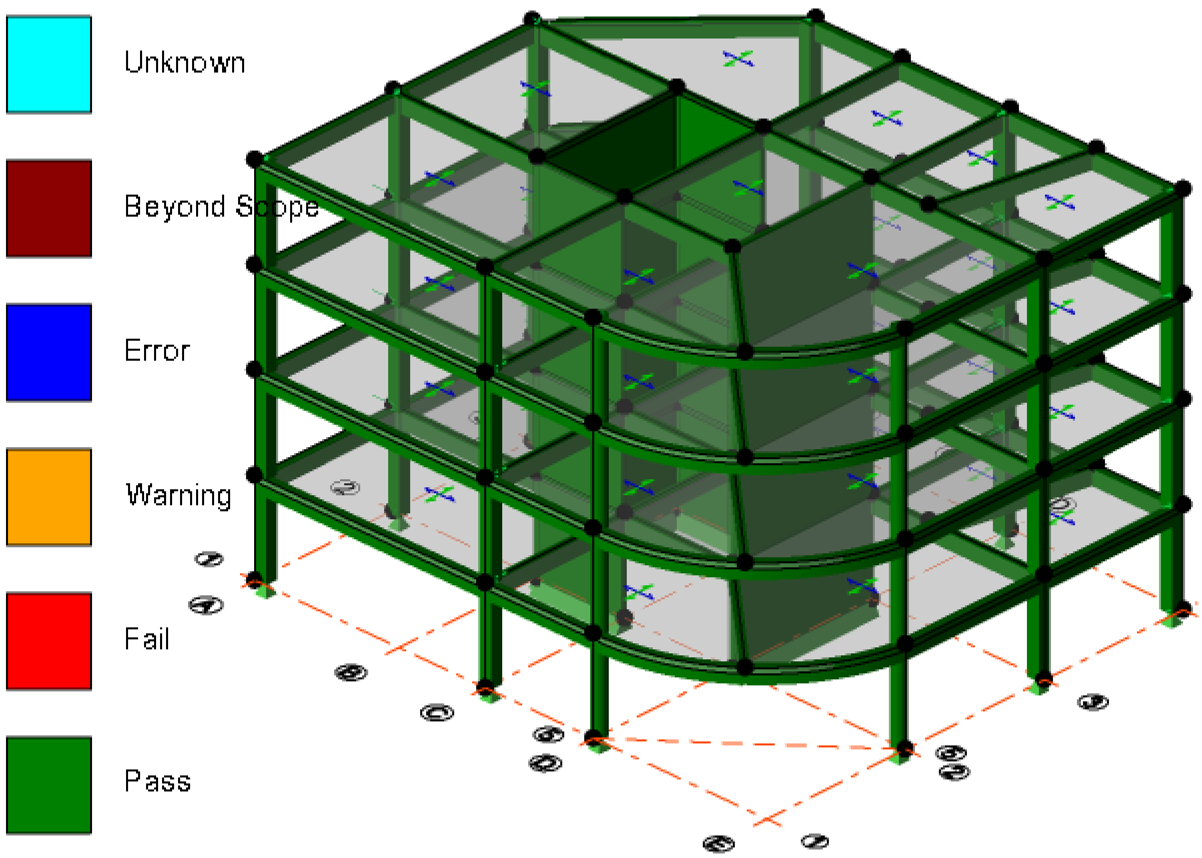
This handbook contains information relevant to the design of foundations in Tekla Structural Designer.

You can find the following information in this handbook:

- Pad base design workflow (page 556)

Pad base design workflow

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



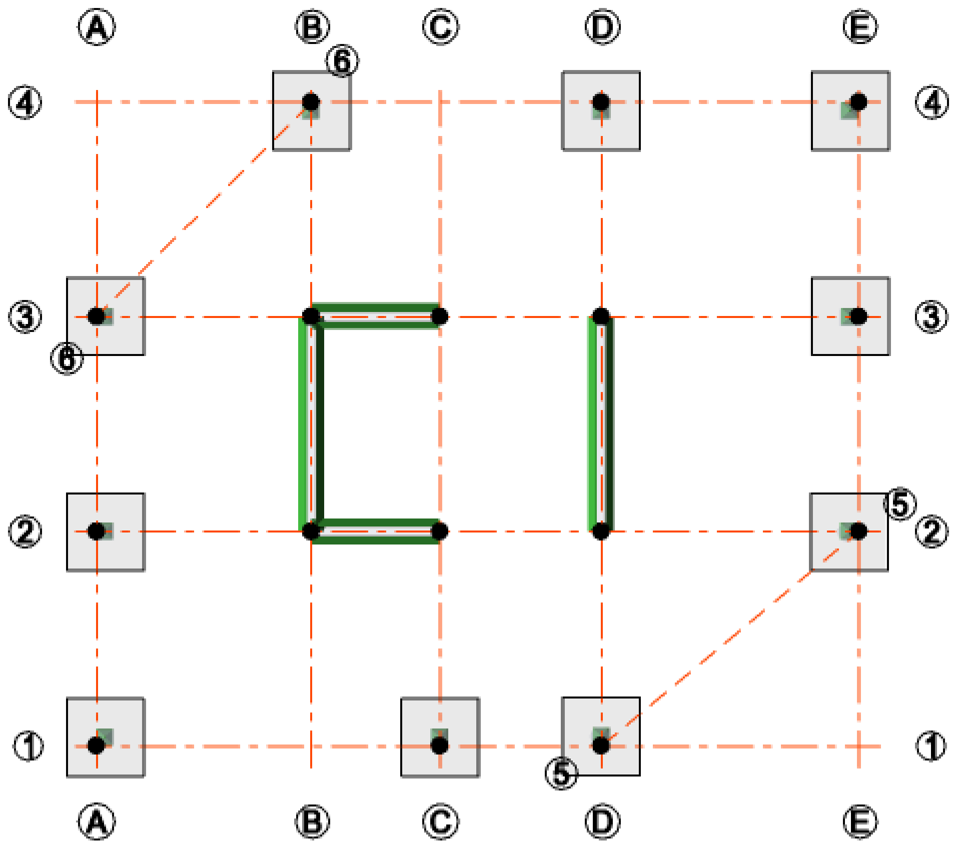
The model has already been designed prior to placing the bases.

Apply pad bases under supported columns

Pad bases can only be placed under, and be loaded by supported columns; strip bases can only be placed under and be loaded by supported walls.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the base design.

At this stage, as you are not aware of the individual base size and depth requirements; you can simply choose to place the bases where required, accepting the default size/depth offered.



Auto-size pad bases individually for loads carried

To obtain an idea of the range of potential sizes, bases should initially be designed individually for their respective loads, as follows:

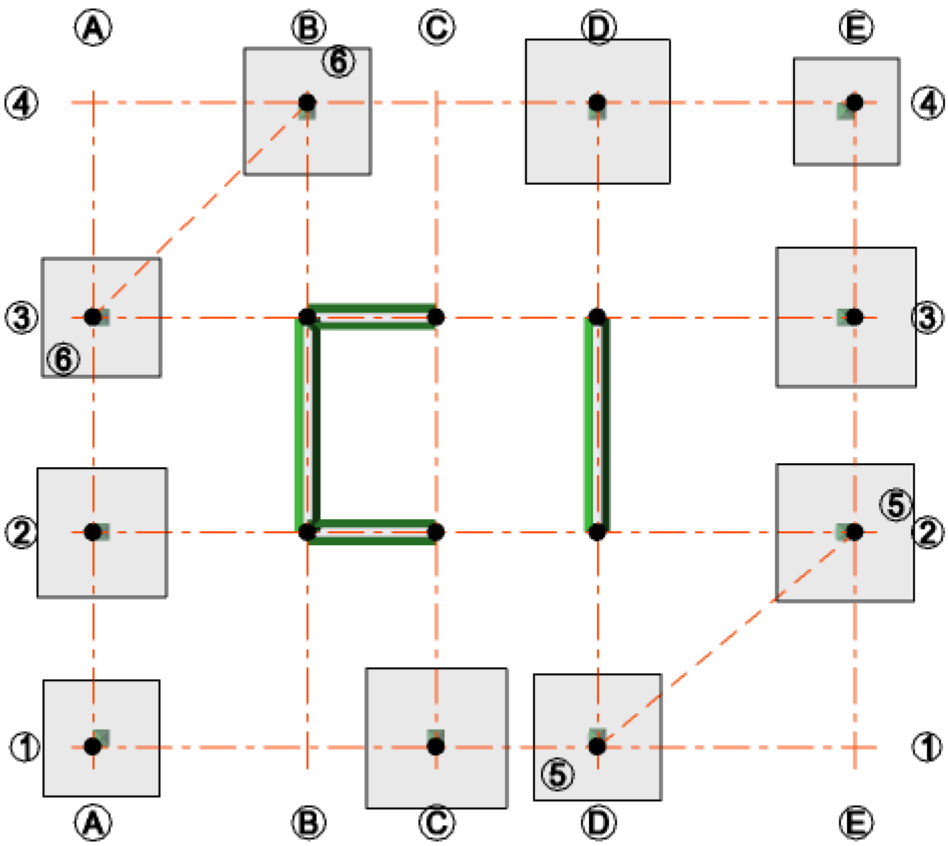
1. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.

2. Select the bases to be auto-sized and in the Properties Window and choose to auto-design both the size and depth; In this way the program establishes suitable base dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

3. From the **Foundations** tab click **Design Pad Bases**.

   Each base will be sized accordingly (any that are not in auto-design mode will simply be checked).



4. With the auto-design options cleared, you can then adjust individual base dimensions and re-check if required (by right-clicking the base that has been edited and choosing Check Member).

   The site boundary may impose restrictions on the positioning of an isolated foundation relative to the column/wall it supports. This restriction may result in a requirement for an offset base, this can be achieved by specifying the eccentricity required in the base properties.

NOTE The overall auto-design procedure is summarized as follows:

1. Bearing Design: - increase size

2. Bending Design: - increase reinforcement - If max allowable reinforcement is reached then increase depth, set reinforcement back to start point, and go back to step 1.
3. Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
4. Punching Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
5. Sliding Checks - increase depth, set reinforcement back to start point, and go back to step 1
6. Uplift Checks - increase size, set reinforcement back to start point, and go back to step 1

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

Apply grouping to rationalize pad base sizes

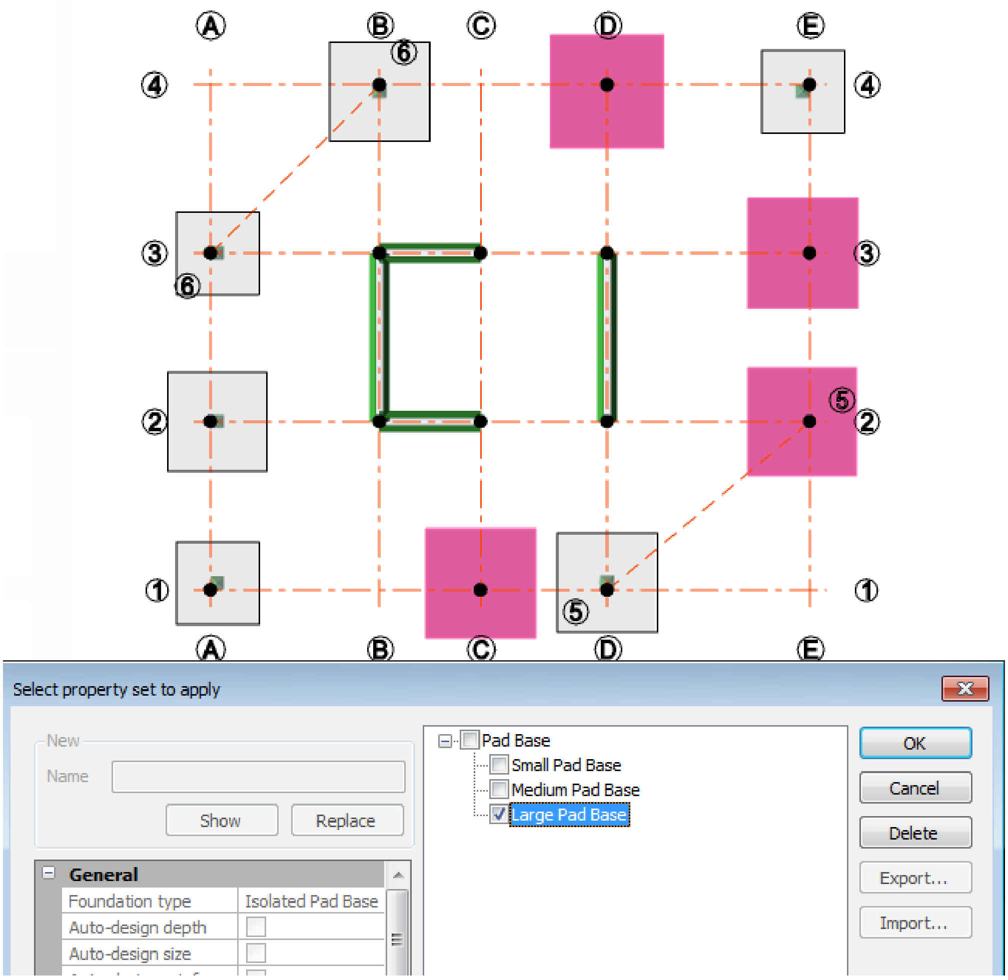
NOTE Grouping can only be applied to pad bases - not to strip bases.

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

This is done as follows:

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

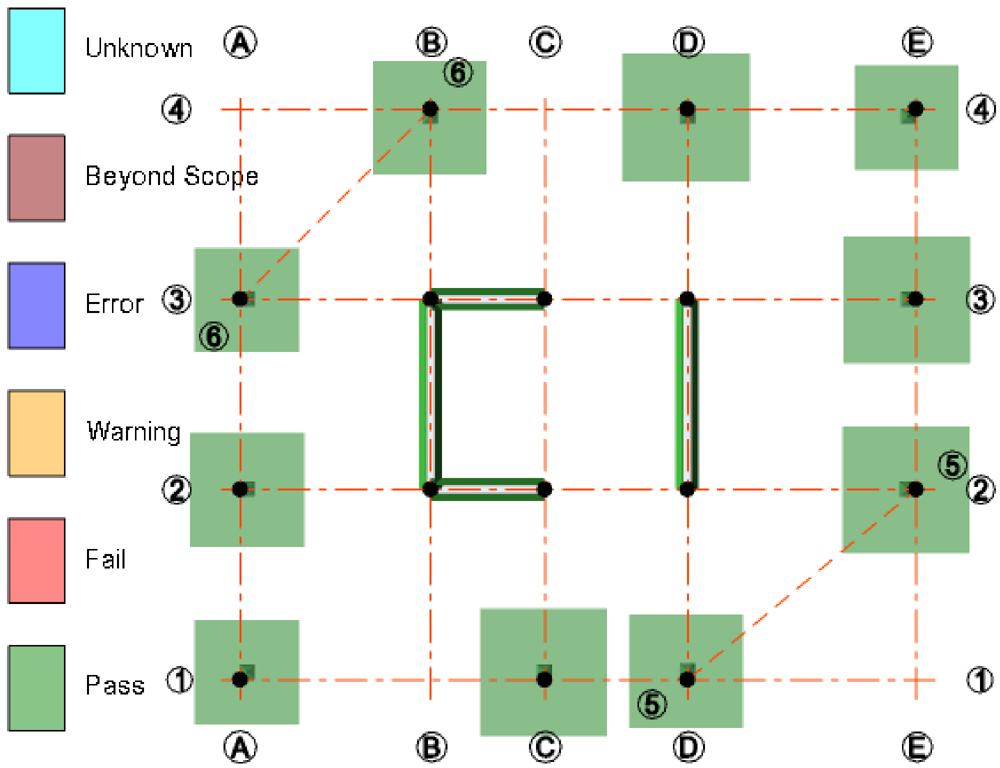
5. In the Properties Window, click **Apply...** to apply the property set you have just created to the selected bases.



NOTE Ensure you click **Apply...** from the Properties Window and not from the right-click menu, otherwise the properties will only be applied to the last base selected.

6. From the Groups page of the Project Workspace, right-click Pad Bases (under the Design branch) and choose Regroup Members - this will put those bases that share similar properties into the same group.
7. Open **Design Settings** dialog box, and from the Design Groups page select the option to design isolated foundations using groups.

8. Click **Design Pad Bases** - the results obtained will reflect the grouping that has been applied.



Review/optimize base design

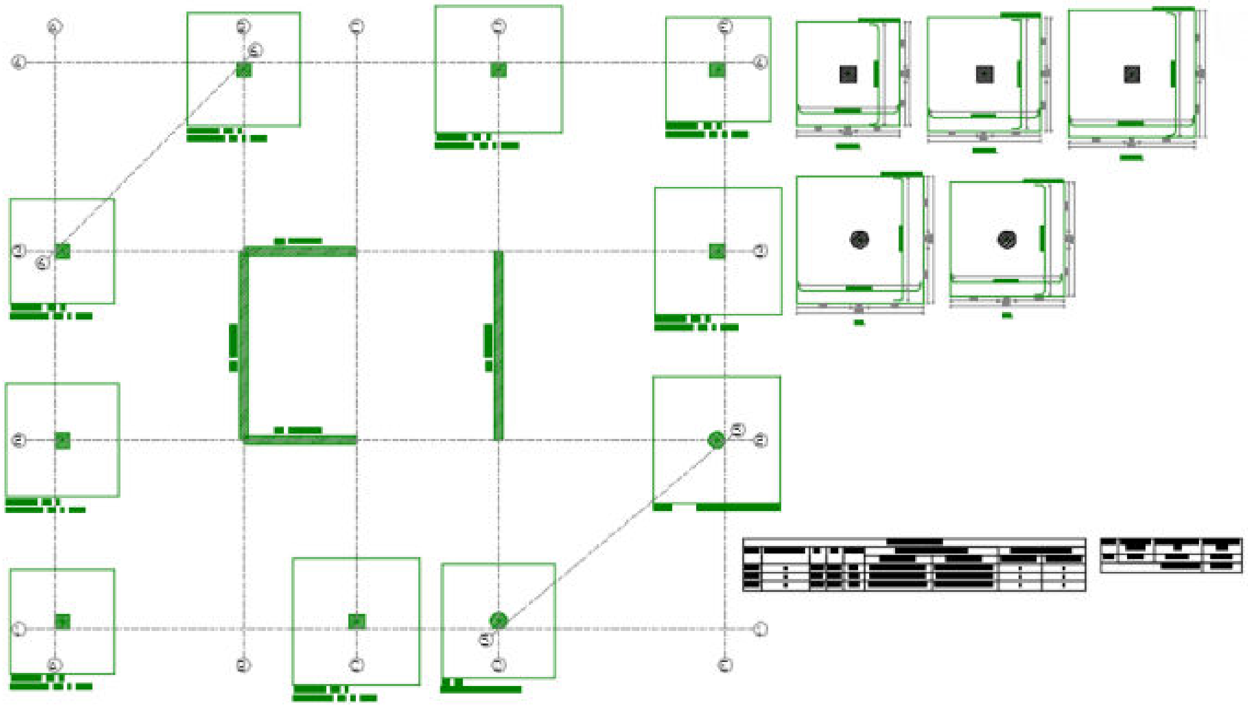
In the Review View you can examine the design efficiency by switching from Foundations Status to Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in **Design Settings> 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

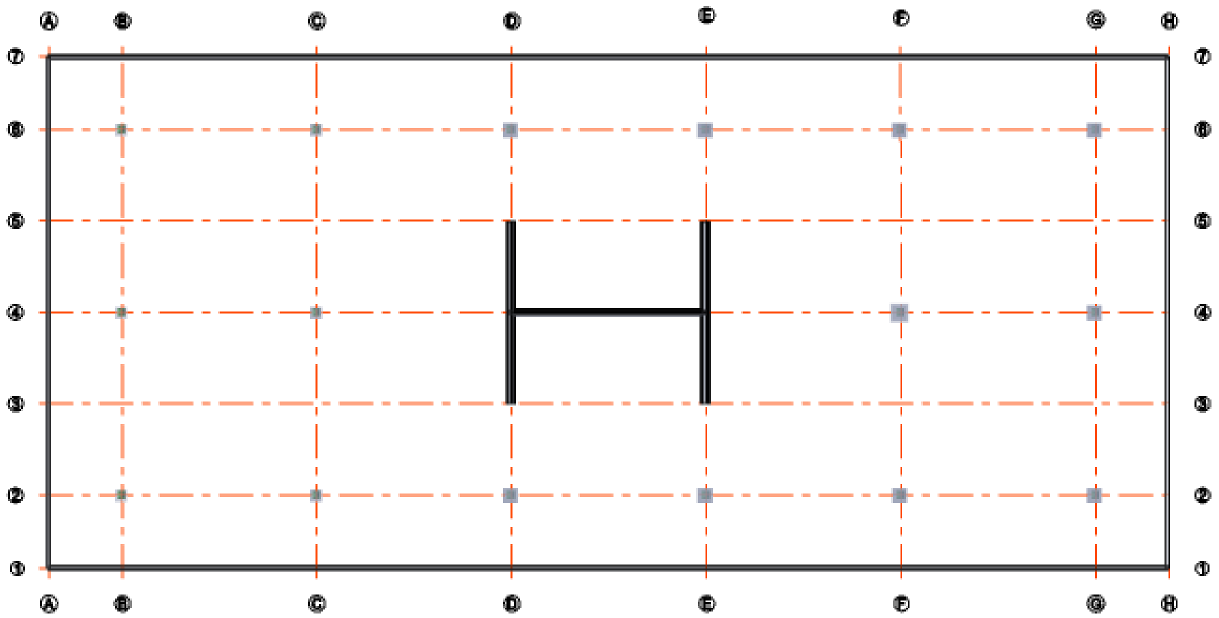


Print calculations

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).

Pile cap design workflow

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



The model has already been designed prior to placing the pile caps.

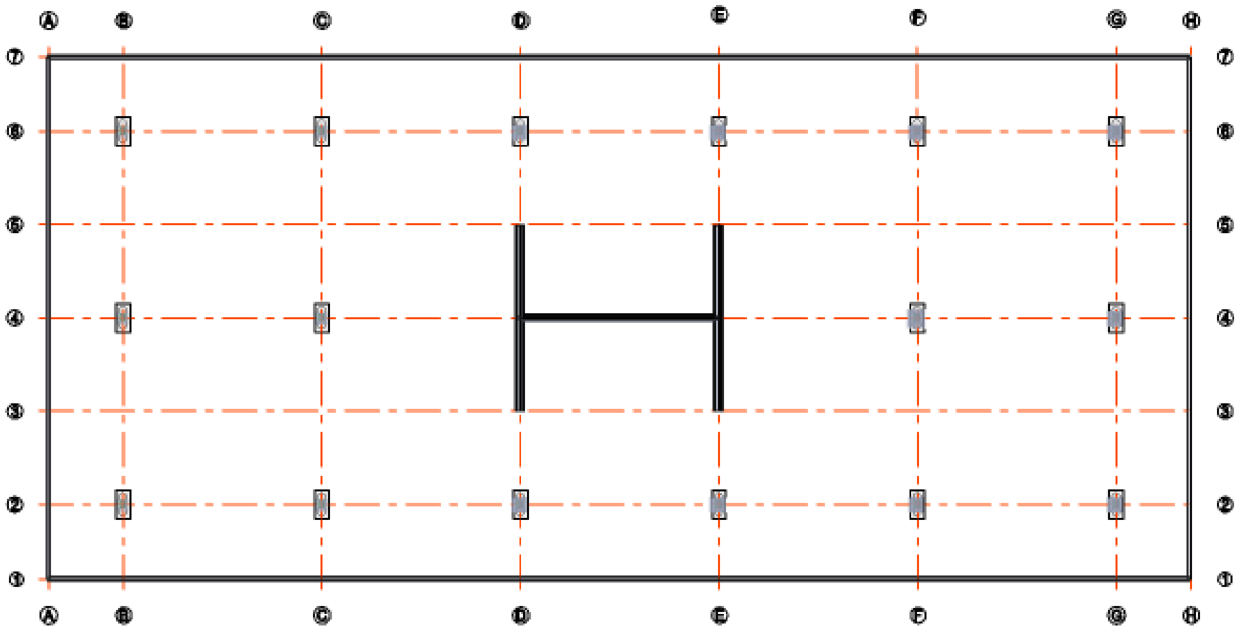
Apply pile caps under supported columns

Before a pile cap can be placed the **Pile Type Catalogue** must contain at least one pile type.

Pile caps can only be placed under and be loaded by supported columns.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the pile cap design.

At this stage, as you are not aware of the individual pile cap size and depth requirements; you can simply choose to place pile caps where required, accepting the default size/depth offered.



Auto-size pile caps individually for loads carried

To obtain an idea of the range of potential sizes, pile caps should initially be designed individually for their respective loads

Note that when piles are auto-designed the outcome will depend on the auto-design method selected; the pile cap size will either be based on the minimum number of piles required, or on the minimum pile capacity.

To individually size the pile caps:

1. In the **Design Settings** dialog box got to **Concrete> Foundations> Isolated Foundations > Piles** to choose the pile auto-design method required: (minimize pile capacity, or minimize number of piles).

2. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.

3. Select the pile caps to be auto-sized and then in the Properties Window choose to auto-design both the piles and depth; In this way the program will establish suitable pile cap dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

4. From the **Foundations** tab, click **Design Pile Caps** and all the pile caps set in auto-design mode will be sized accordingly. (Those not in auto-design mode will simply be checked). Similarly the piles beneath the pile caps will either be designed (if pile auto-design mode is enabled) or checked.

At any point you can switch to a user defined arrangement, modify the pile cap configuration and have the design re-checked.

One example where you might choose a user defined arrangement is where the site boundary imposes restrictions on the positioning of the pile cap relative to the column/wall it supports. Switching to a user defined arrangement allows you to specify an eccentricity and create an offset pile cap.

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

Apply grouping to rationalize pile cap sizes

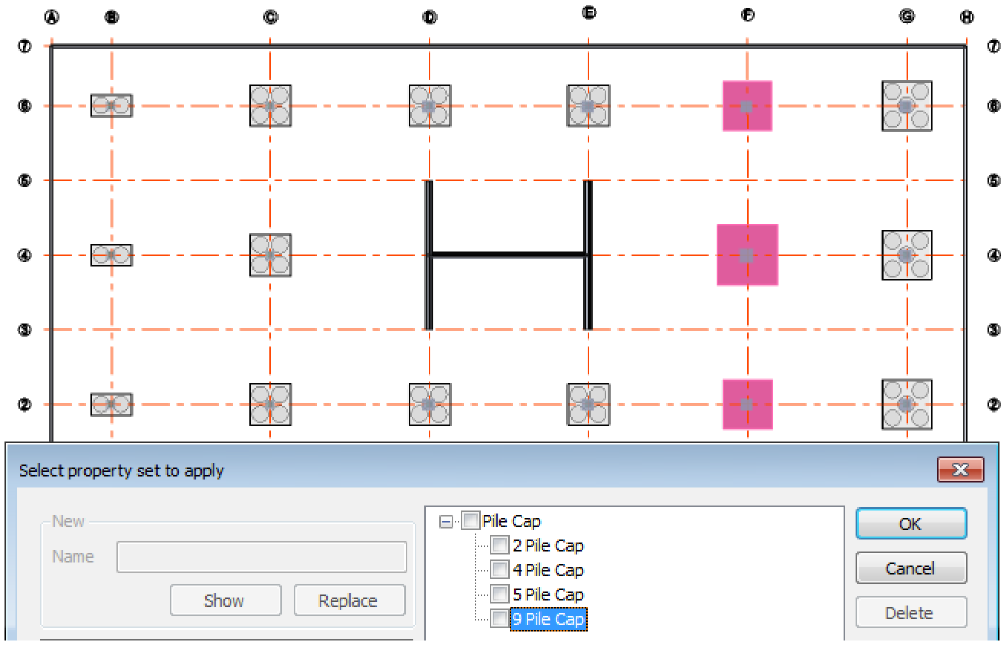
Once pile caps have been sized individually, the designs can be rationalized by activating grouping, in order to obtain one design per group sufficient for all pile caps within the group.

This is done as follows:

1. Select a pile cap that you want to be in a particular group.
2. In the Properties Window, ensure it is set to auto-design.
3. Right-click the same pile cap and from the context menu choose Create Property Set...
4. Select all the other pile caps that you want to be in the same group.

   NOTE When applied moments are significant, be cautious when grouping pile caps where auto-design has initially determined different principal directions.

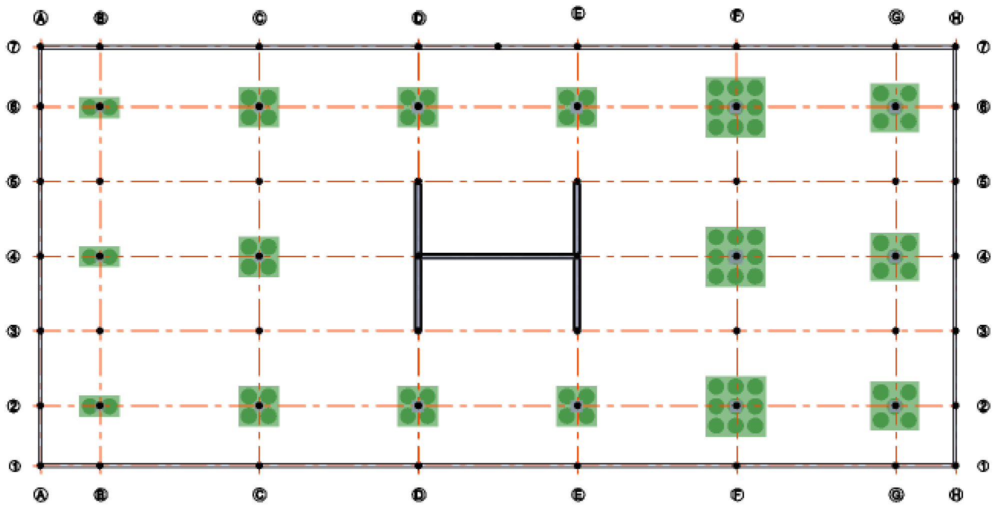
5. In the Properties Window, click **Apply...** to apply the property set you have just created to the selected pile caps.



NOTE Ensure you click **Apply...** from the Properties Window and not from the right-click menu, otherwise the properties will only be applied to the last pile cap selected.

6. From the Groups page of the Project Workspace, right-click **Pile Caps** (under the Design branch) and choose Regroup Members - this will put those pile caps that share similar properties into the same group.
7. Open the **Design Settings** dialog box, and from the Design Groups page select the option to design isolated foundations using groups.

8. Click **Design Pile Caps** - the results obtained will reflect the grouping that has been applied.



Review/optimize pile cap design

In the Review View you can graphically examine the pile cap design status & efficiency by switching between Foundations Status and Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

Similarly you can graphically examine the pile design status & efficiency by switching between Piles Status and Piles Ratio.

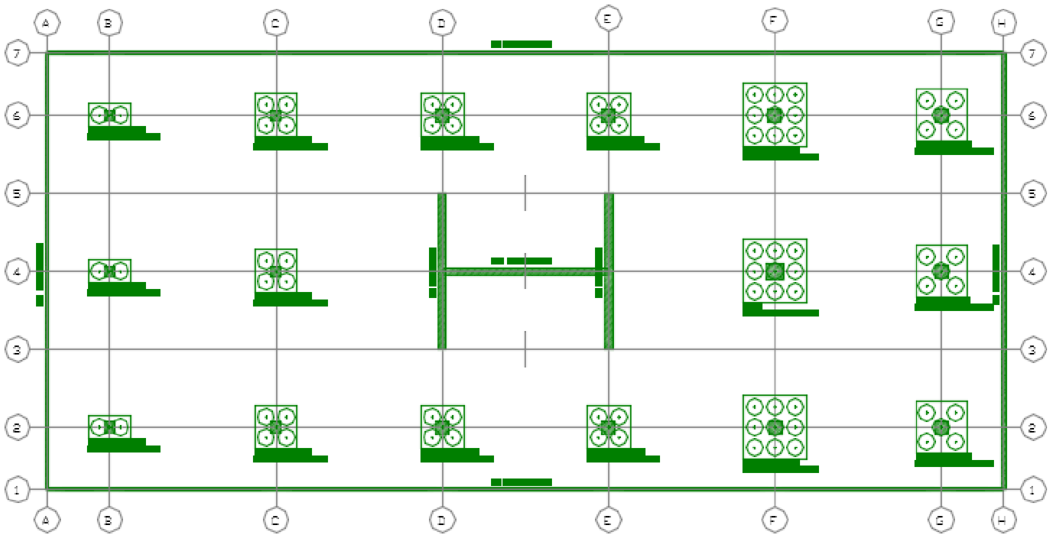
If the selections are unacceptable you may need to review the settings in **Design Settings> Concrete> Foundations**

Design results can be examined in more detail by selecting Tabular Data and setting the View Type to Design Summary. By selecting the Pile Caps or Piles as the Characteristic (and the appropriate material) the full results for each pile cap or pile can be examined.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to

eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.



Print calculations

Create a model report that includes the concrete pile cap design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Pad base, strip base and pile cap design forces

Forces acting on supports

The following forces and moments on the supports are determined from the analysis of the active load combinations:

- Vertical force in direction Z
- Horizontal forces in directions Y and X
- Moments around X and Y axis

NOTE A torsional moment around the Z axis is also determined, but the base/pile cap is not designed for this in the current release.

Foundation self weight

The foundation self-weight is automatically calculated and applied as an added load, F_{swt}

Soil self weight

The surcharge depth and soil unit weight that have been specified in the base/pile cap properties are used to determine the soil self weight. This is applied as an added load, F_{soil}

NOTE In the current release, horizontal pressure caused by soil is not considered.

Additional surcharge loads

For isolated foundations user can apply additional surcharge loads: acting in the global Z direction.

- Permanent (dead) surcharge load
- Variable (live) surcharge load

Design Forces

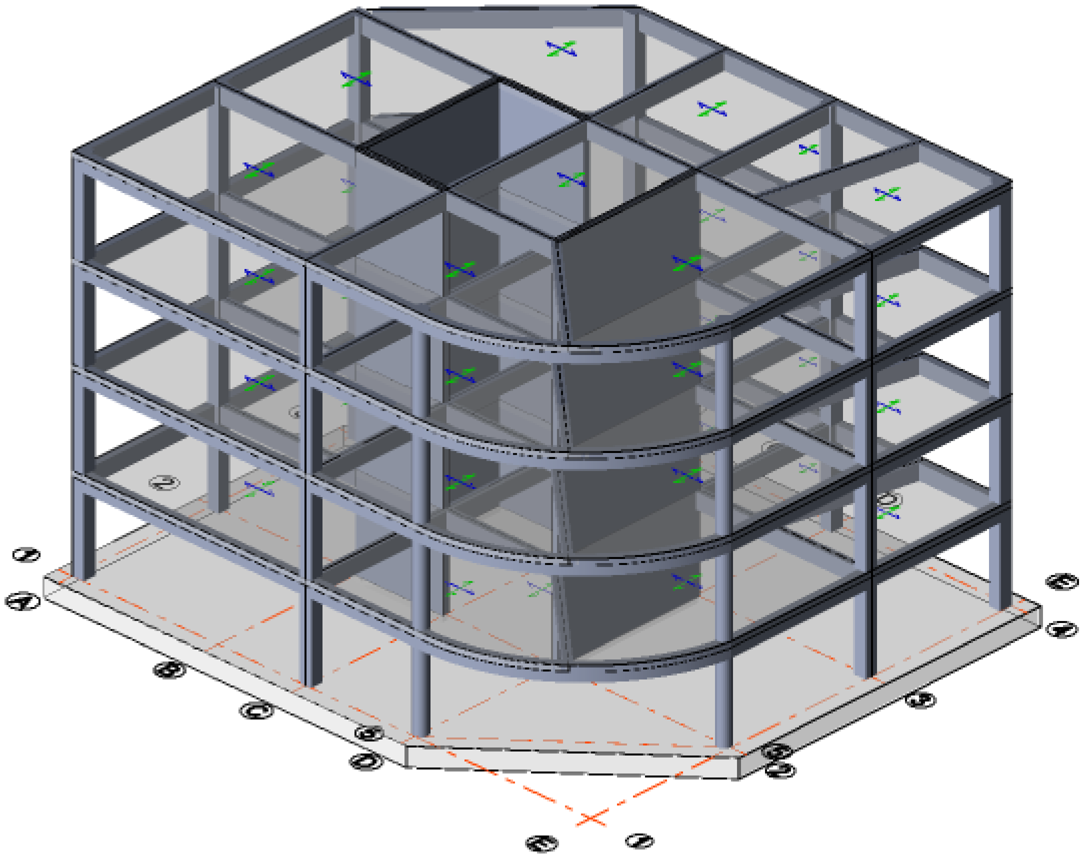
To obtain the design forces, the loads due to foundation self weight, soil self weight and additional surcharge are added to the forces acting on the supports.

These design forces (axial load and bi-axial shear and moment) are then applied to the base/pile cap at the foundation level.

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

Mat foundation design workflow (metric units)

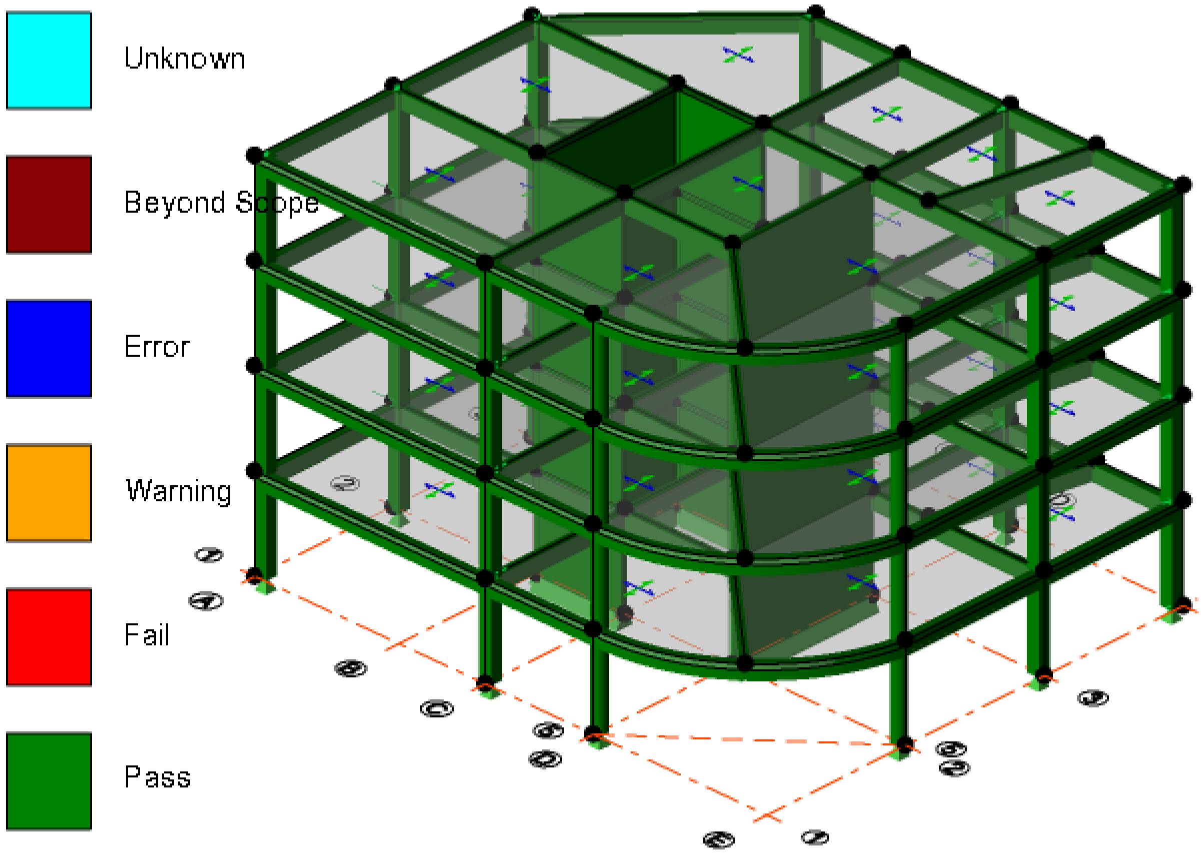
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



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

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
    - Linear ground stiffness, or,
    - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear spring are applied:

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

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit (kN/m3)	Upper Limit (kN/m3)
Loose Sand	4,800	16,000
Medium Dense Sand	9,600	80,000
Dense Sand	64,000	128,000
Clayey Medium Dense Sand	32,000	80,000
Silty Medium Dense Sand	24,000	48,000
Clayey Soil (qa<200kPa)	12,000	24,000

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

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

Horizontal support

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

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

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

Determine the remaining mat properties

You are required to manually specify the Reduce live loads by mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

In this example the minimum area method is used to create a mat with the following properties:

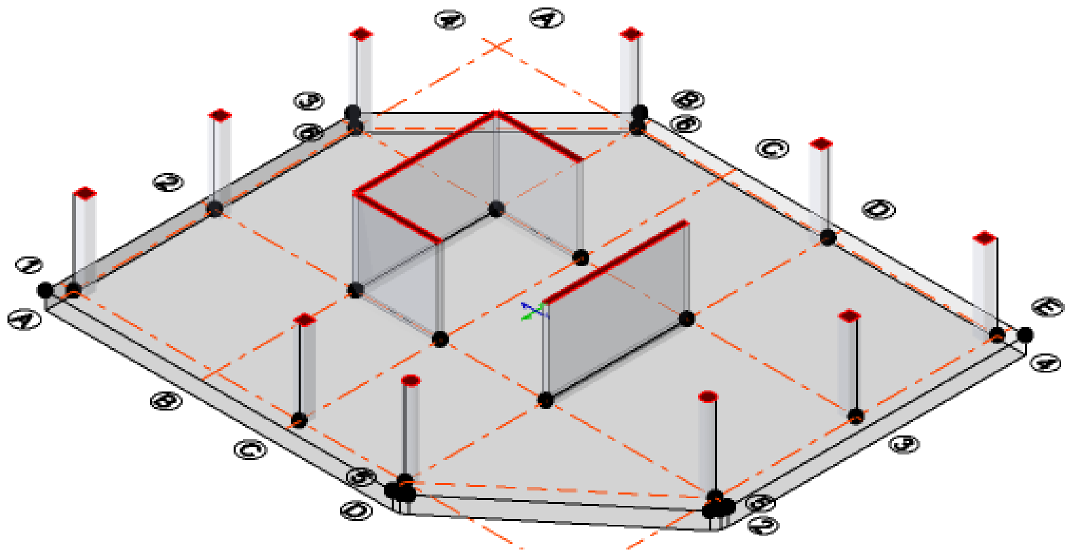
- Imposed loads reduced by 30%
- Default overhang
- Mat thickness 600mm
- Ground bearing springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)

3. Click on those columns (or walls) that define its perimeter,
4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

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

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

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

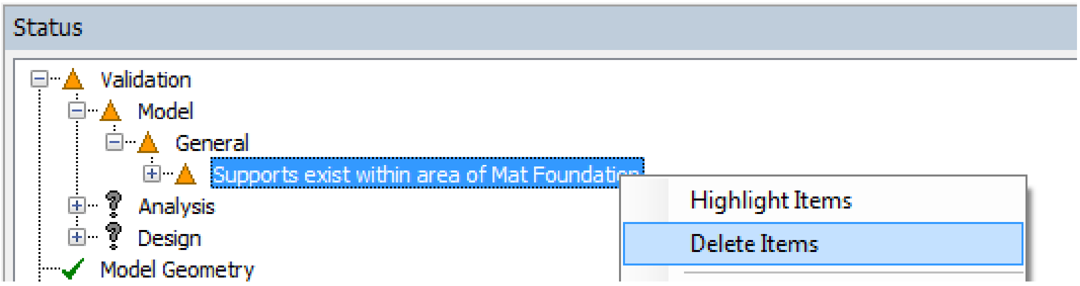
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Design All (Static), and any seismic RSA combinations by running Design All (RSA). These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

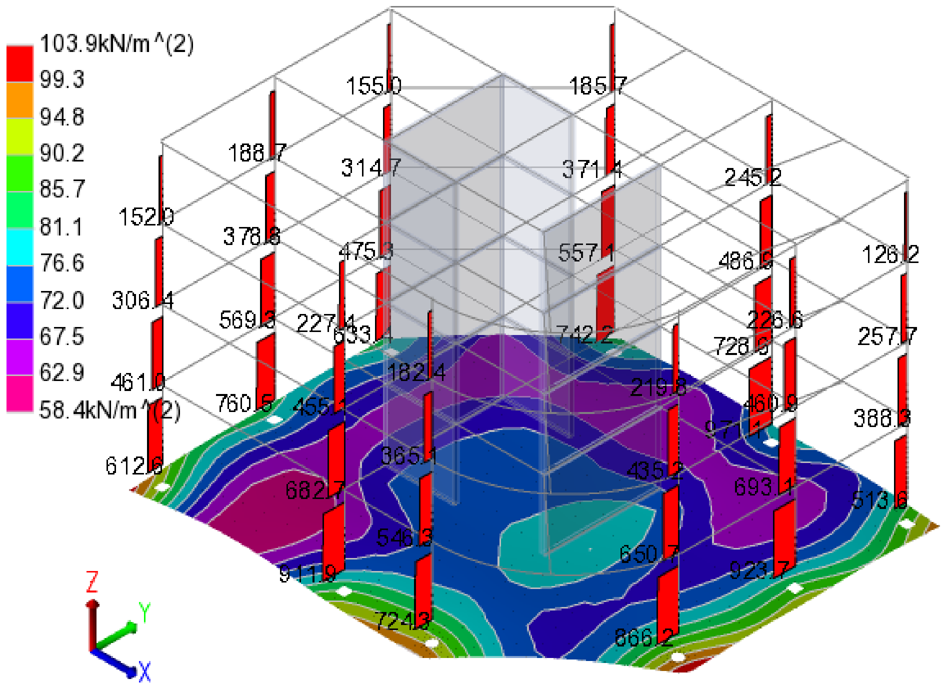
In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

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

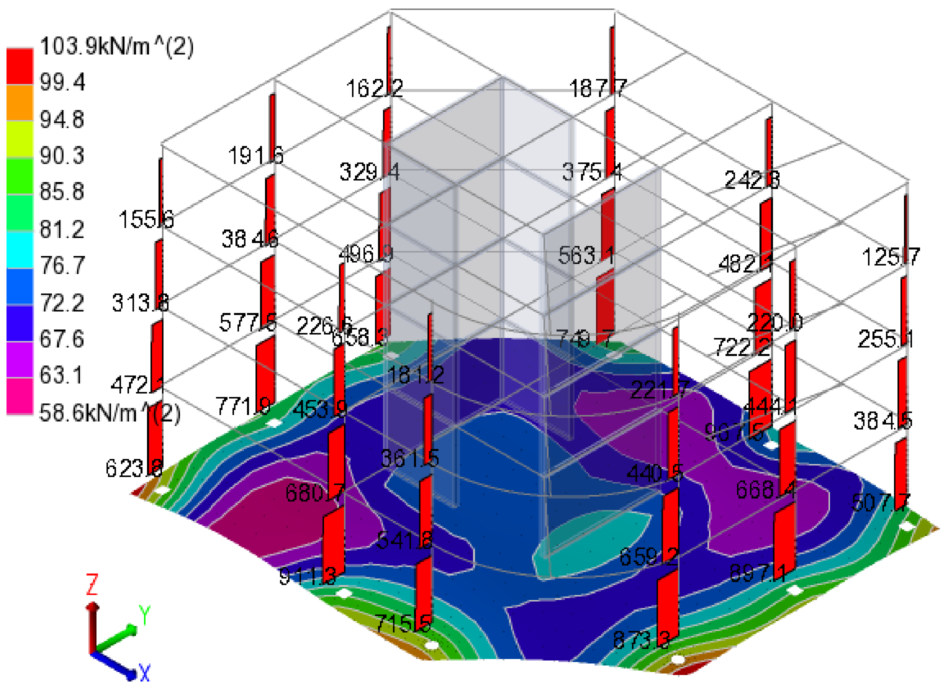
NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

The default stiffness adjustments are dependent on the design code. For design to EC2 the default adjustment factor applied is 0.2. For design to ACI the default adjustment factor applied is 0.25.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking Design All (Static) from the Design toolbar.

NOTE Similarly if an RSA design has previously been performed, but is now out of date Design All (RSA) should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

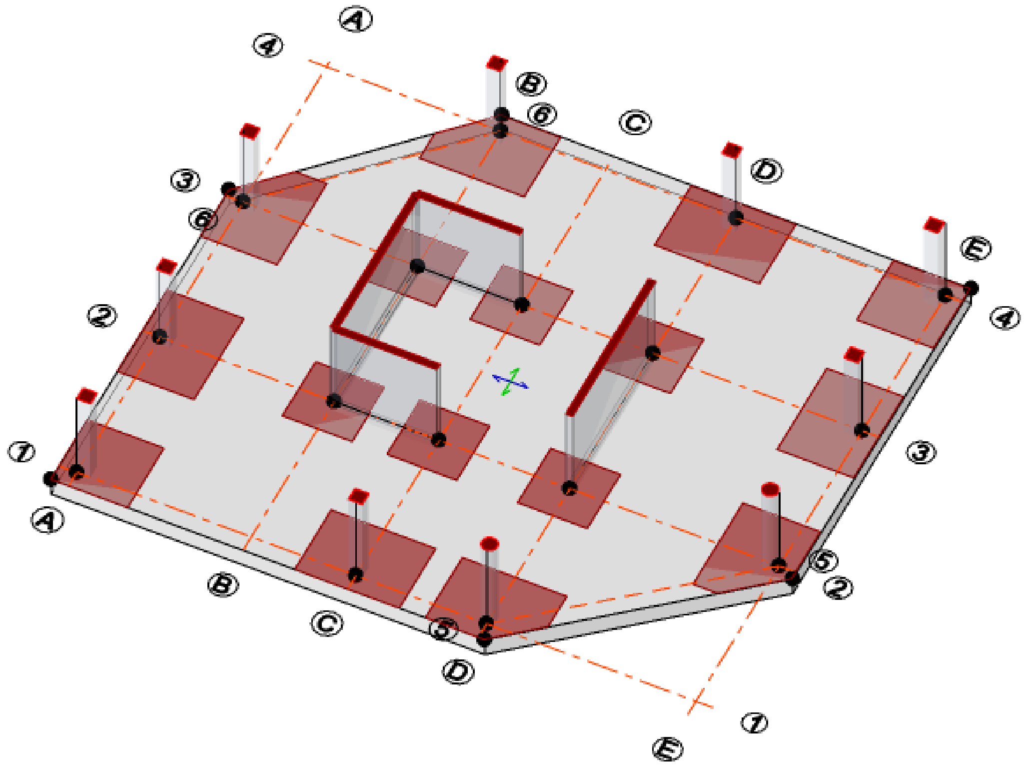
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgment.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run Design Mats in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either Design Slabs or Check Slabs.

NOTE When accessed from the right-click menu, these commands operate as follows:

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

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.

- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run Design Patches in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

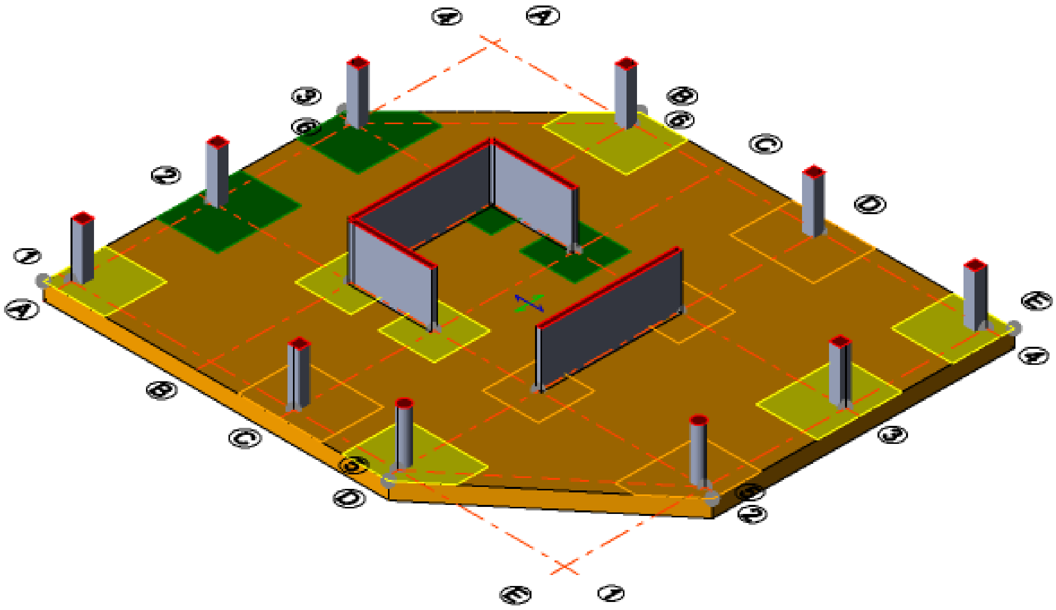
- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has

H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

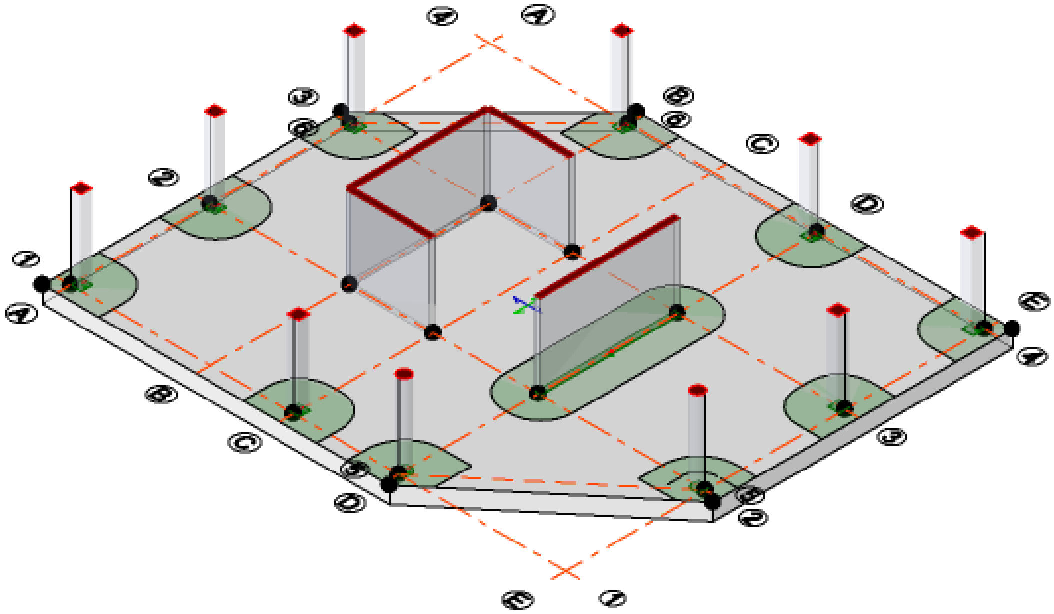


Add and run punching checks

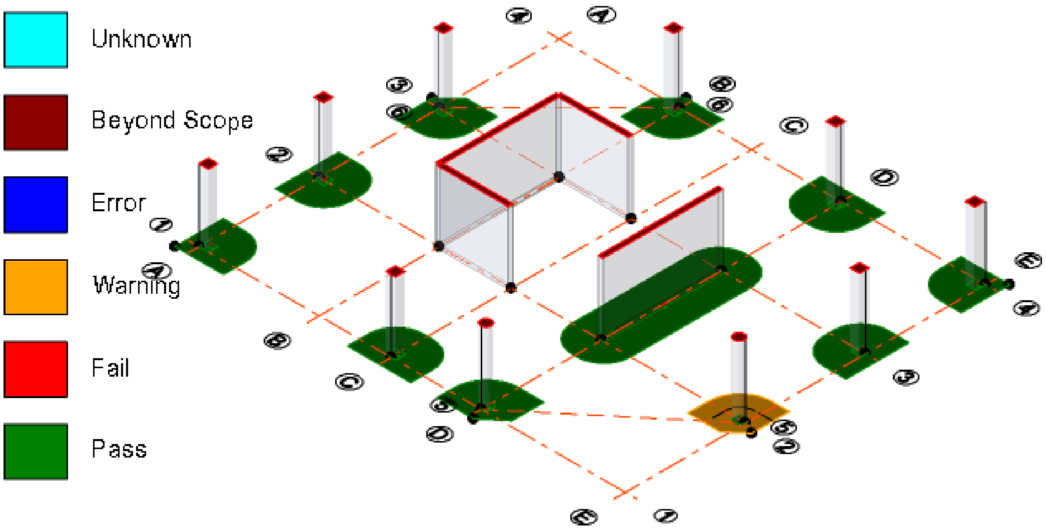
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.



Once added click Design Punching Shear. See:



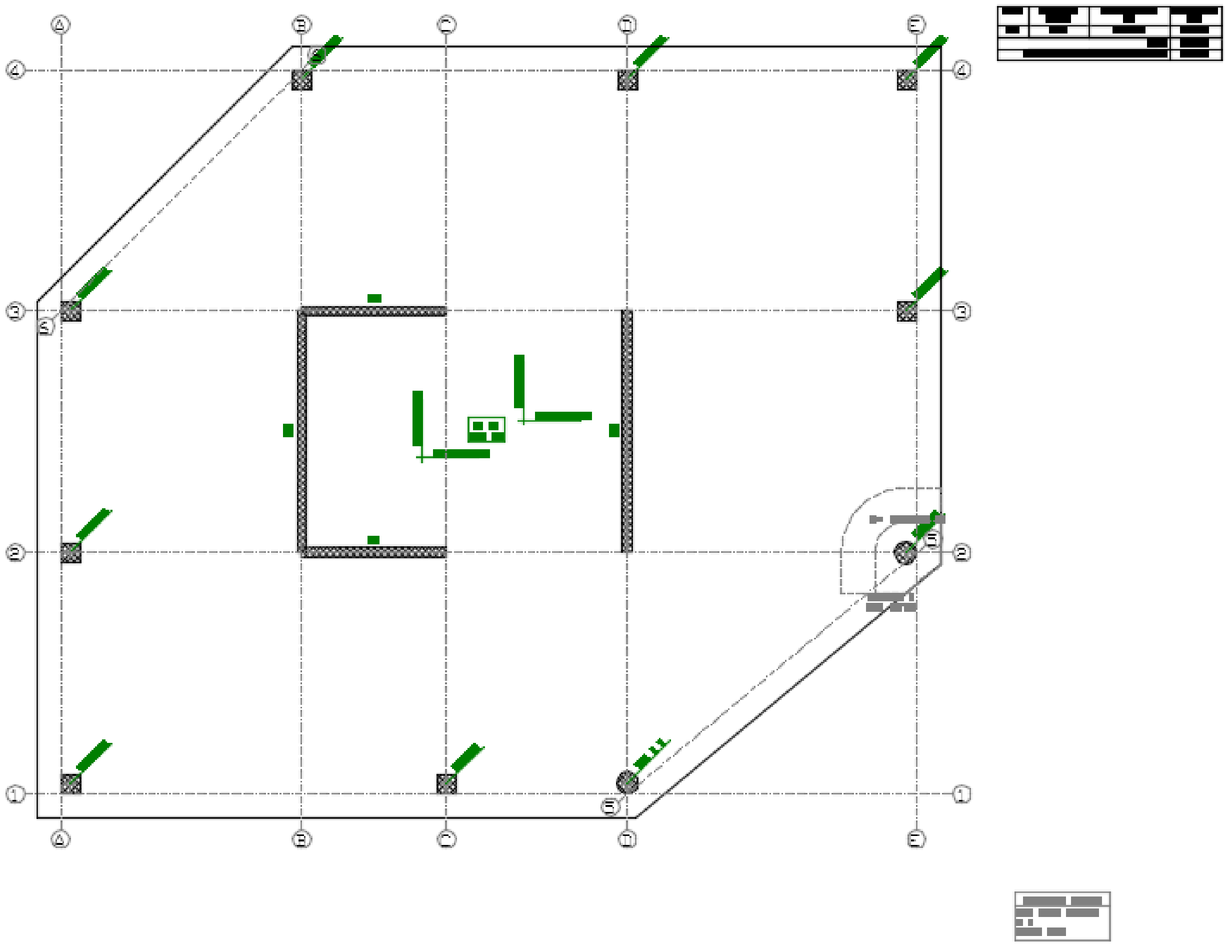
The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to

eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

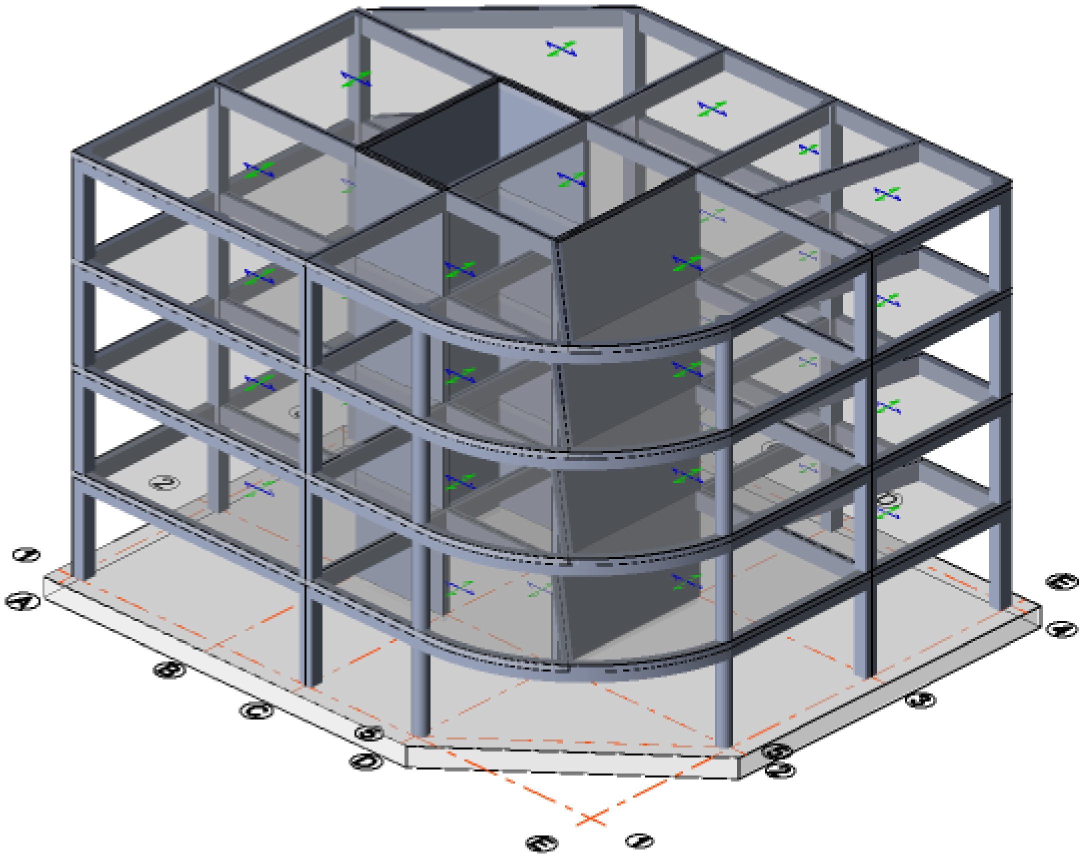


Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Mat foundation design workflow (US customary units)

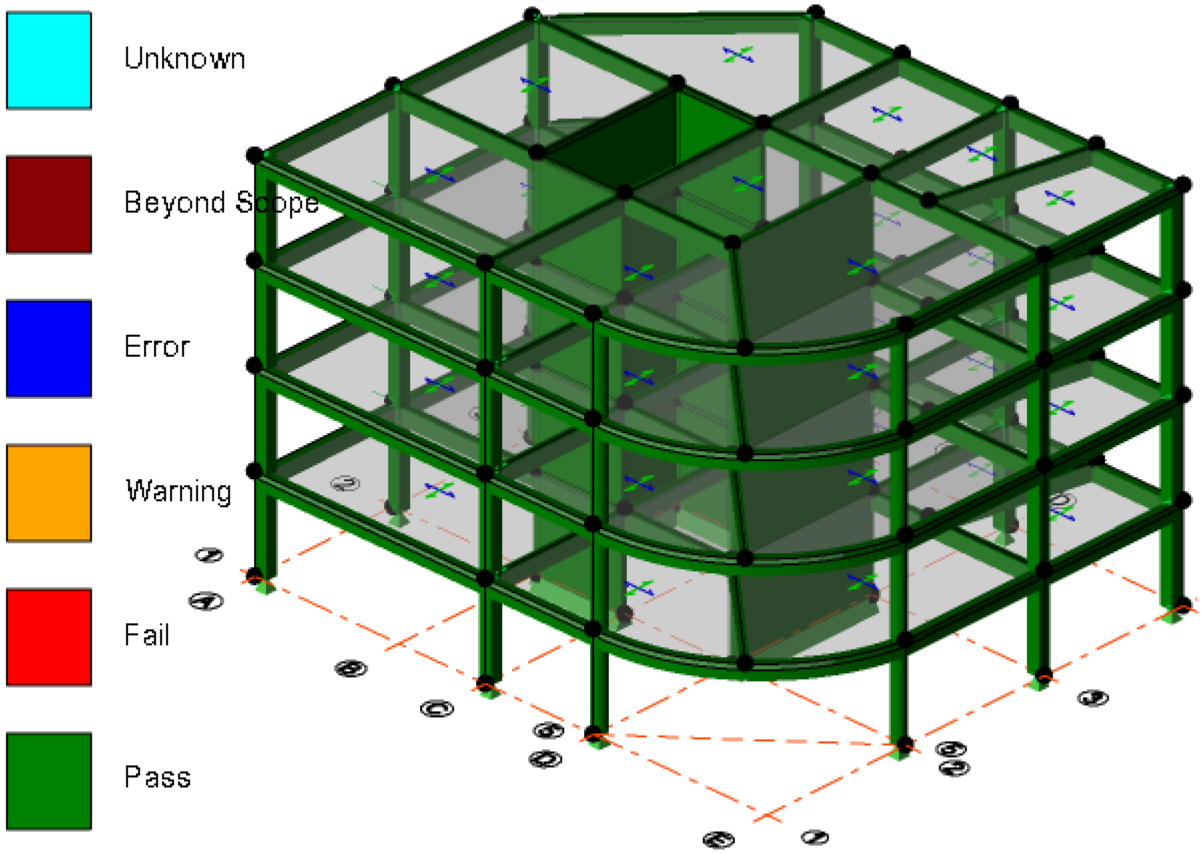
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



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

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
    - Linear ground stiffness, or,
    - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear springs are applied:

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

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit		Upper Limit	
	(kN/m3)	(kip/ft2/ft) approx.	(kN/m3)	(kip/ft2/ft) approx.
Loose Sand	4,800	31	16,000	102
Medium Dense Sand	9,600	61	80,000	509
Dense Sand	64,000	407	128,000	815
Clayey Medium Dense Sand	32,000	204	80,000	509
Silty Medium Dense Sand	24,000	153	48,000	306

Material	Lower Limit		Upper Limit	
	(kN/m3)	(kip/ft2/ft) approx.	(kN/m3)	(kip/ft2/ft) approx.
Clayey Soil (qa<200kPa)	12,000	76	24,000	153
Clayey Soil (200<qa<800kPa)	24,000	153	48,000	306
Clayey Soil (qa>800kPa)	48,000	306	200,000	1273

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

Horizontal support

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

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

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

Determine the remaining mat properties

You are required to manually specify the Reduce live loads by mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

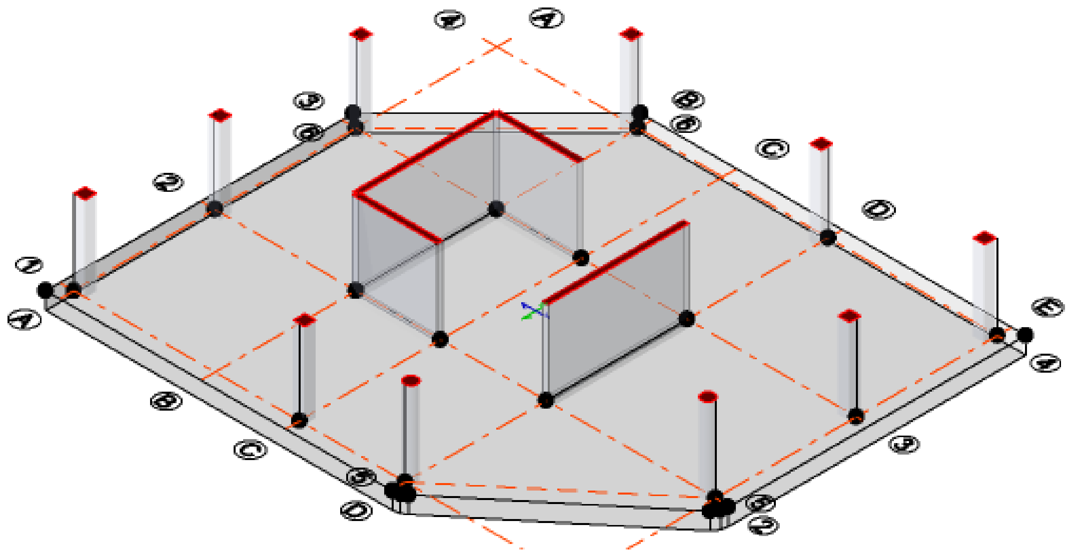
In this example the minimum area method is used to create a mat with the following properties:

- Live loads reduced by 30%
- Overhang, 20in.
- Mat thickness, 24in.
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)
3. Click on those columns (or walls) that define its perimeter,
4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

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

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

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

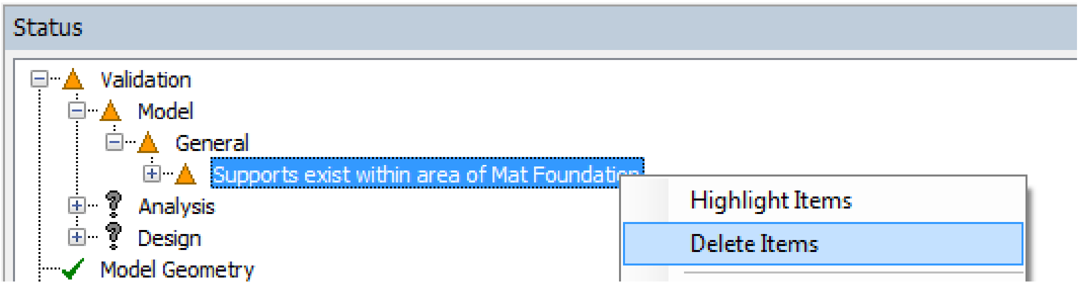
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Design All (Static), and any seismic RSA combinations by running Design All (RSA). These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

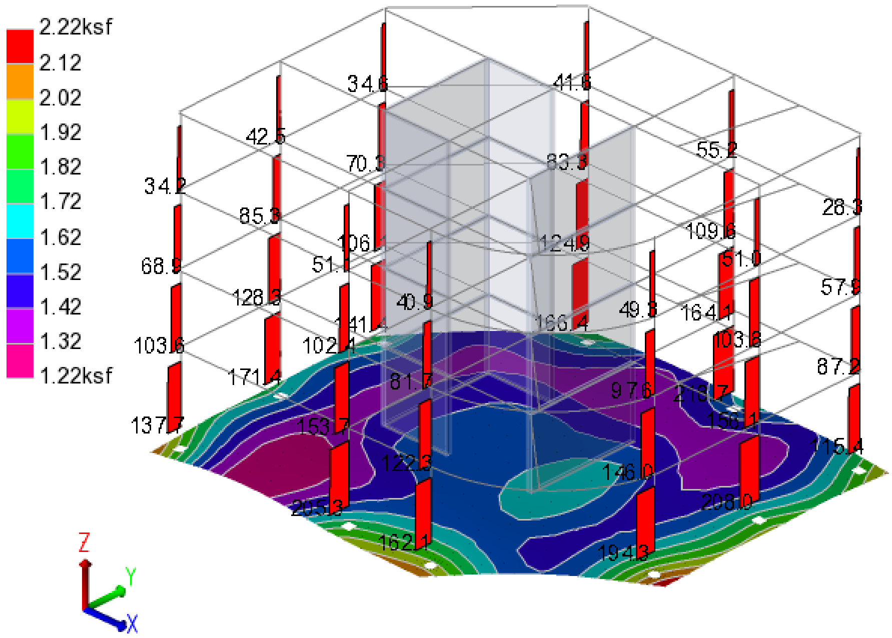
In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

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

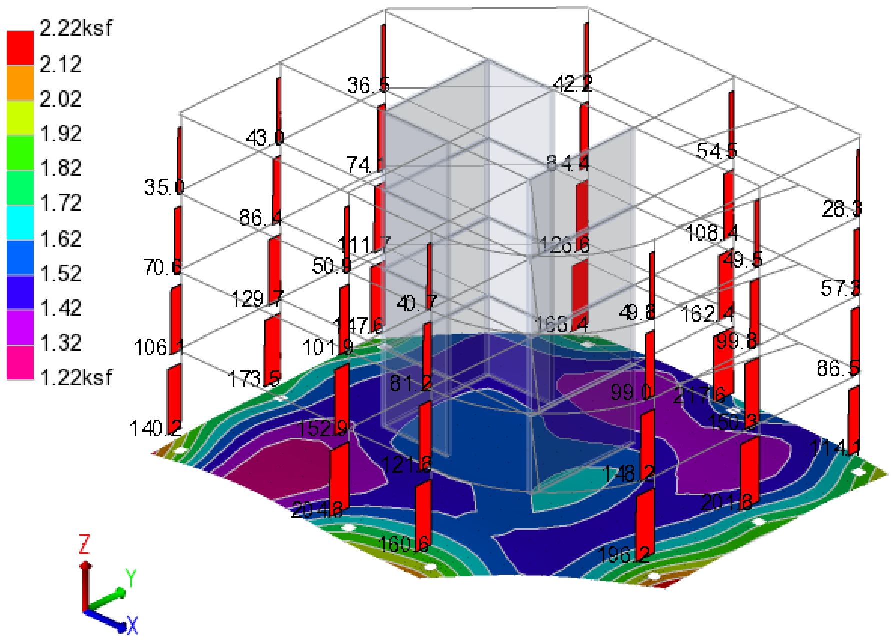
NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

For the above analysis Tekla Structural Designer default stiffness adjustment factors were used, the default factor for foundation mats being 0.2 for both ACI and EC2 design codes.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking Design All (Static) from the Design toolbar.

NOTE Similarly if an RSA design has previously been performed, but is now out of date Design All (RSA) should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

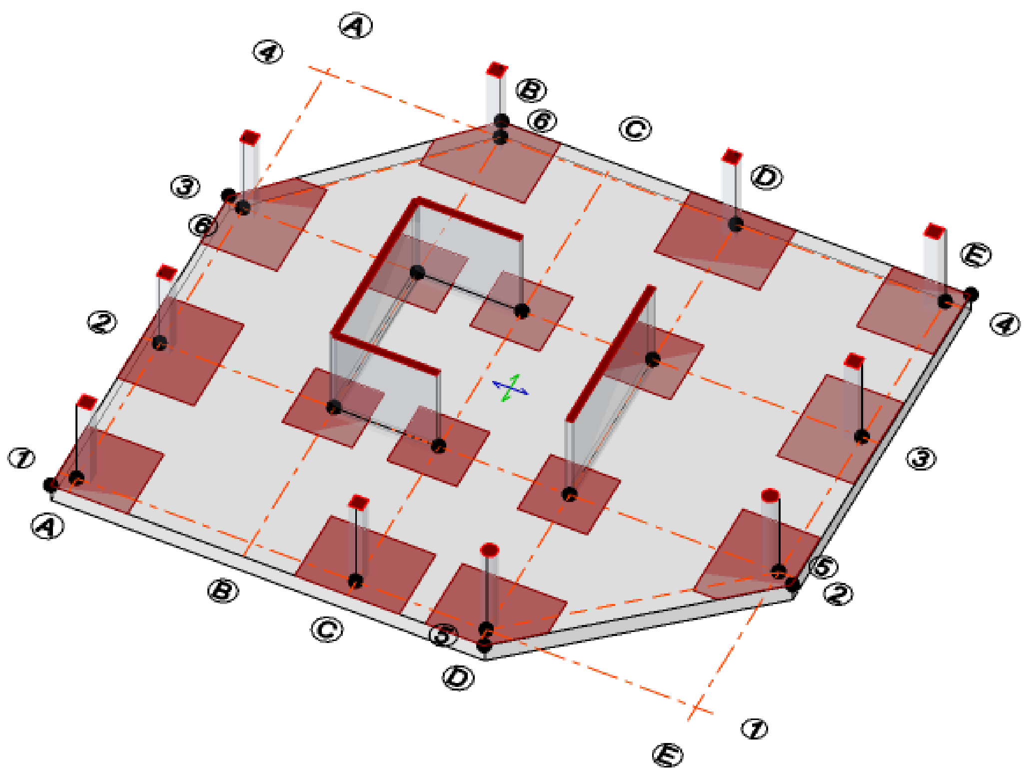
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgment.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run Design Mats in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either Design Slabs or Check Slabs.

  NOTE When accessed from the right-click menu, these commands operate as follows:

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

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 3 size bars are selected and you would just never use anything less than a 4 size bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 6".

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run Design Patches in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

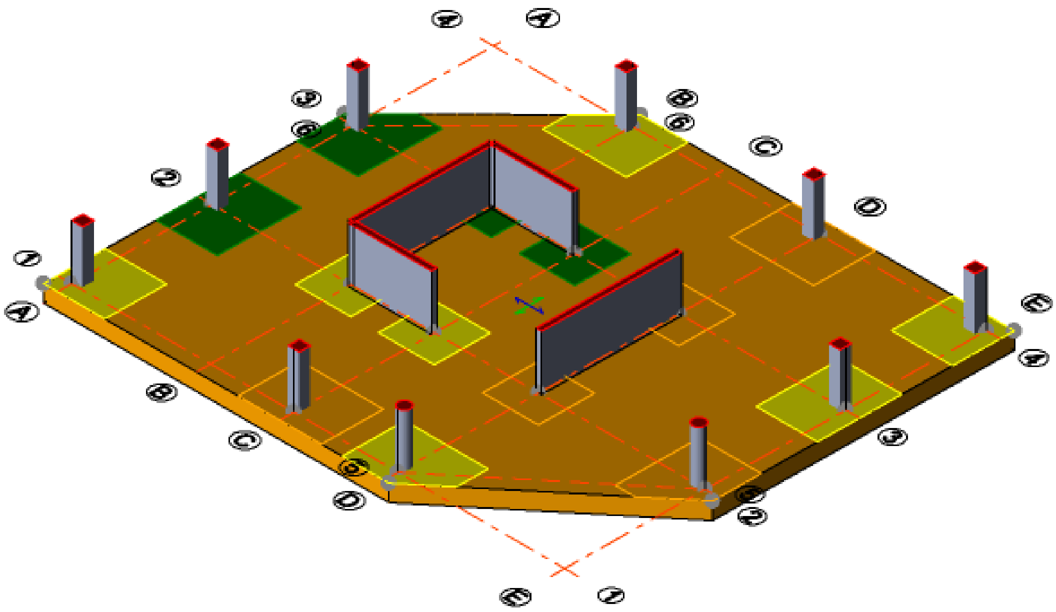
NOTE When accessed from the right-click menu, these commands operate as follows:

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

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has size 4 at 8in, then in the patch don't add size 4 at 5in, swap to size 5 at 8in - then there will be alternate bars at 4in crs in this strip of the patch.

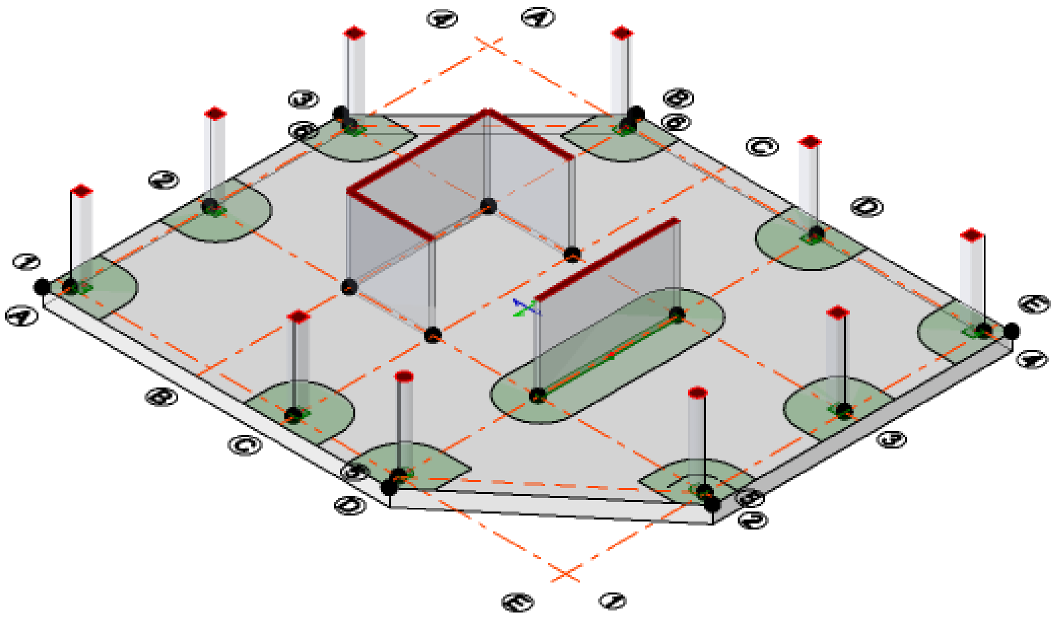


Add and run punching checks

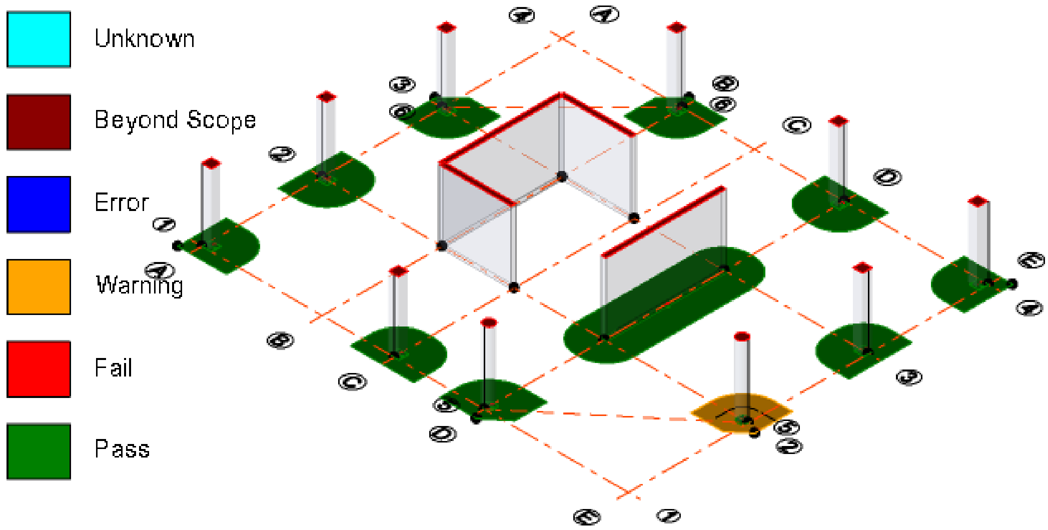
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.



Once added click Design Punching Shear. See:

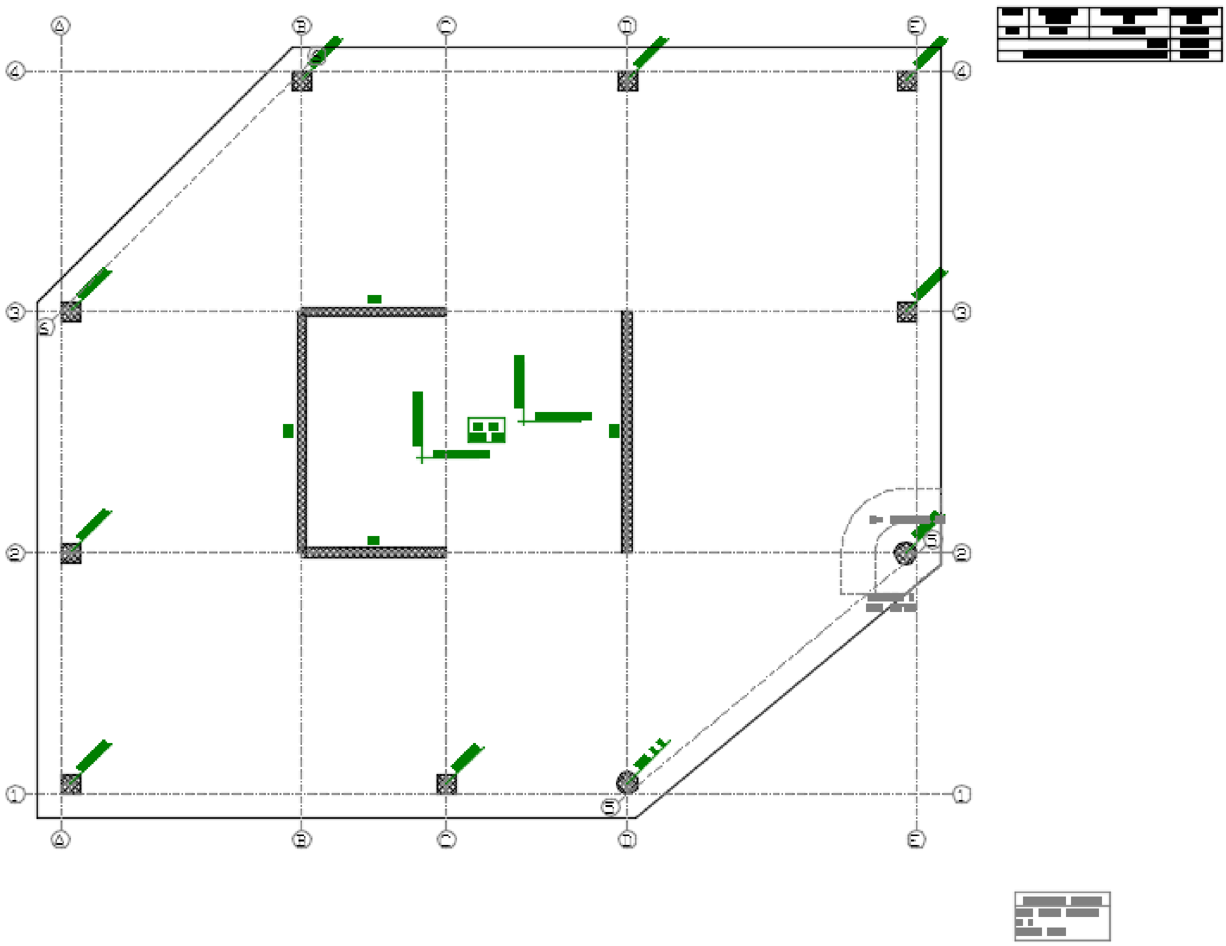


The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

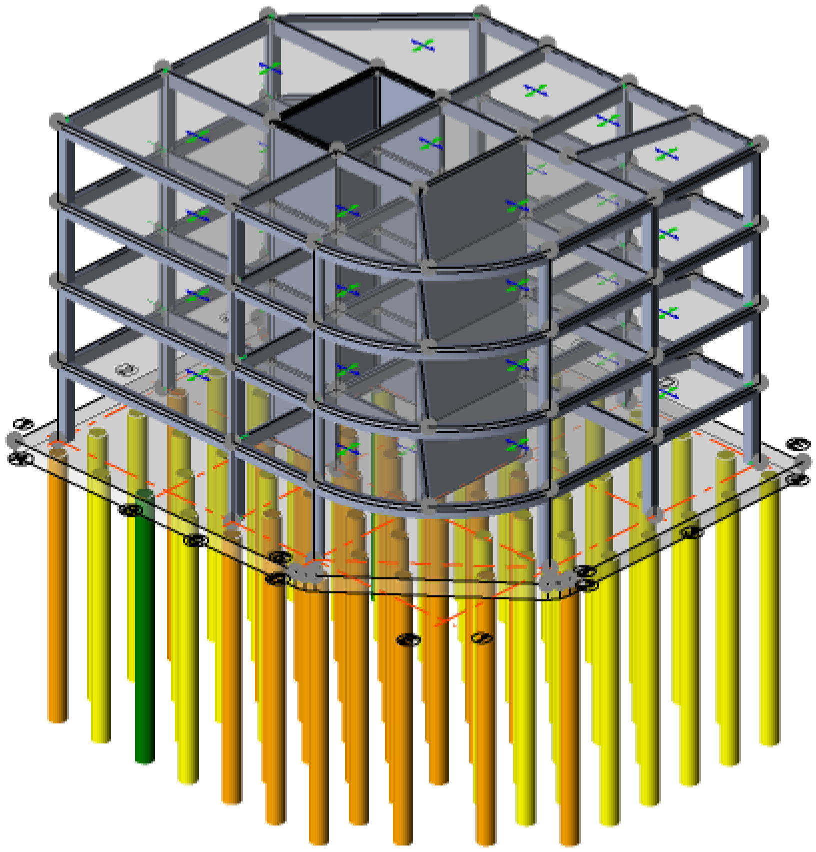


Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

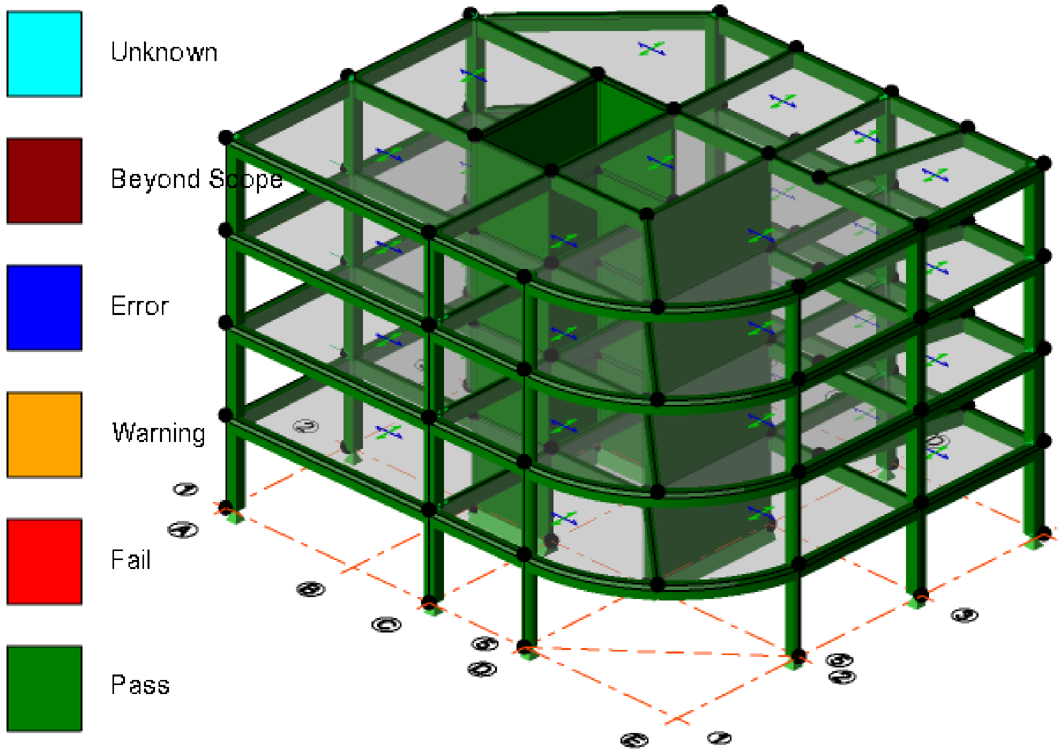
Piled mat foundation design workflow (US customary units)

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



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

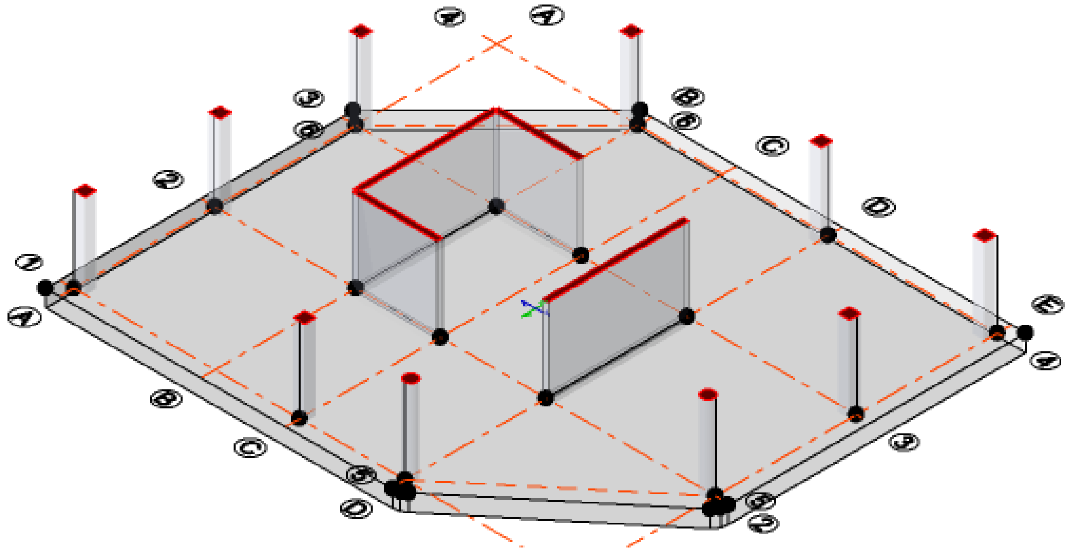
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 40in.
- Mat thickness 24in.
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

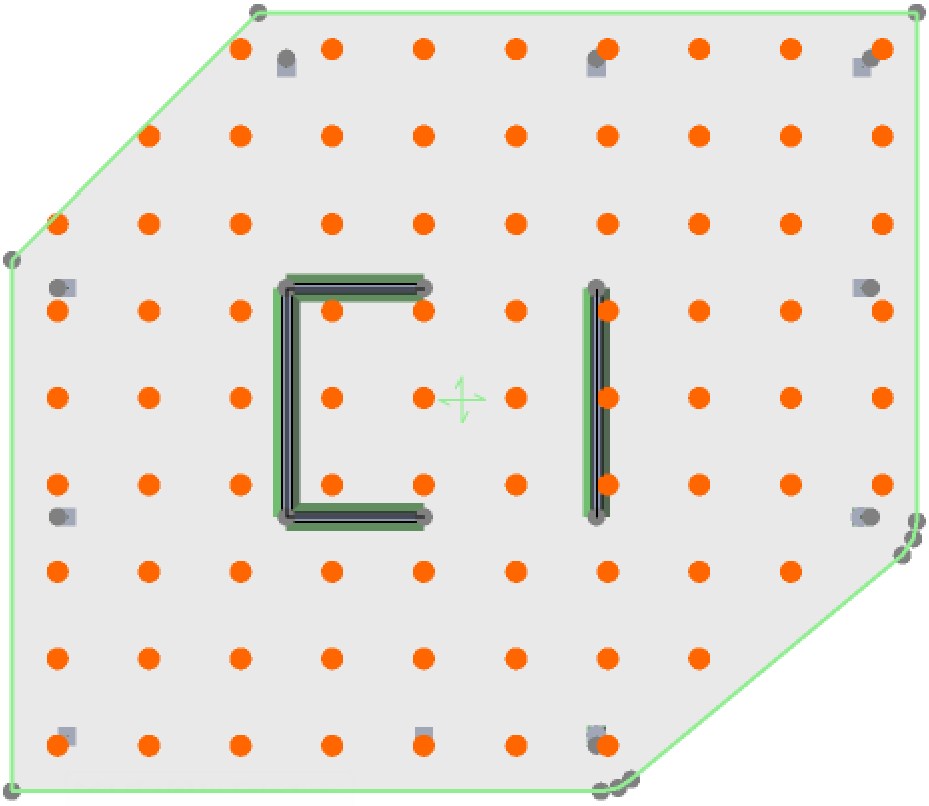
NOTE In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

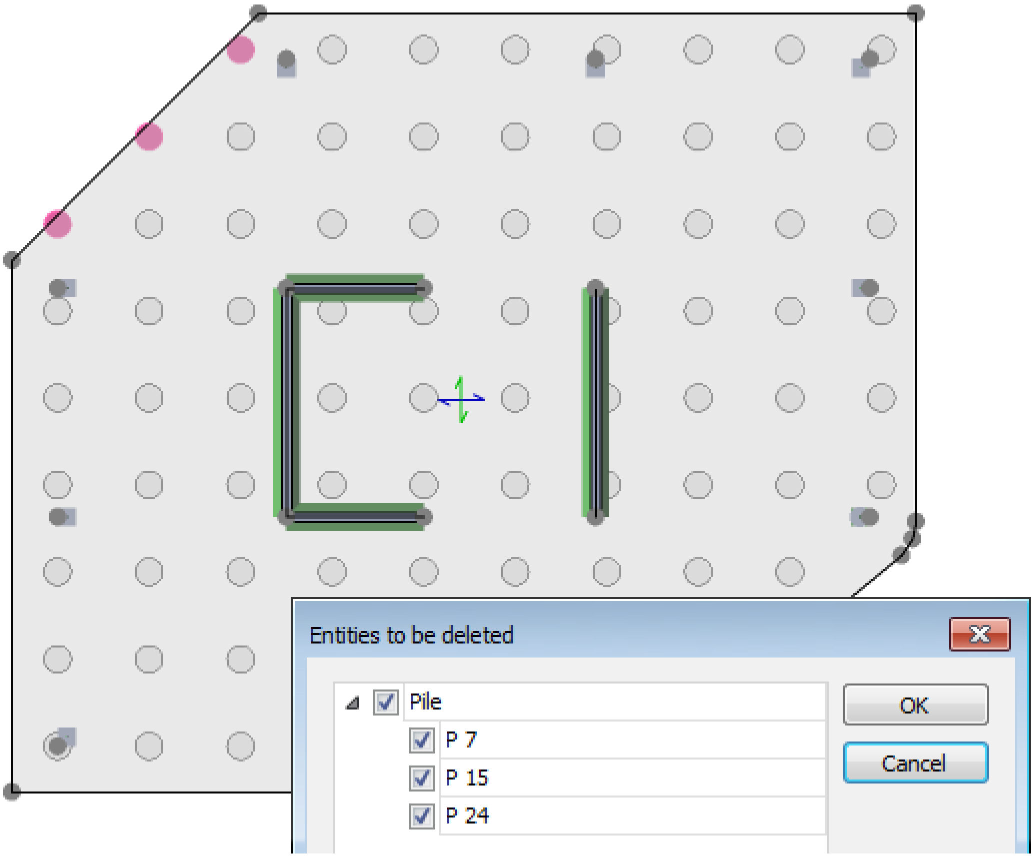
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

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

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

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

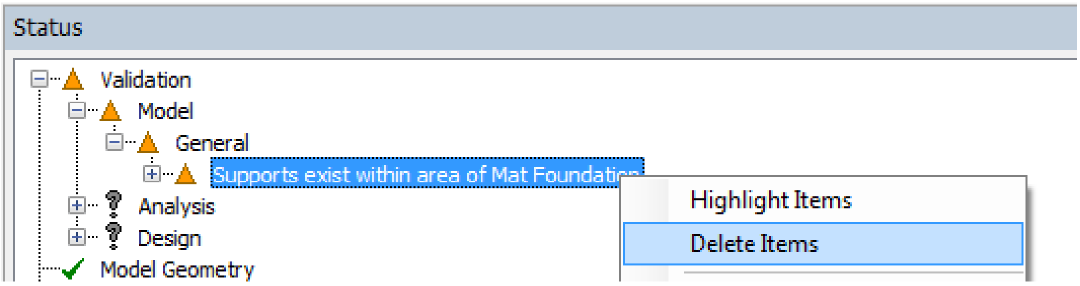
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

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

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

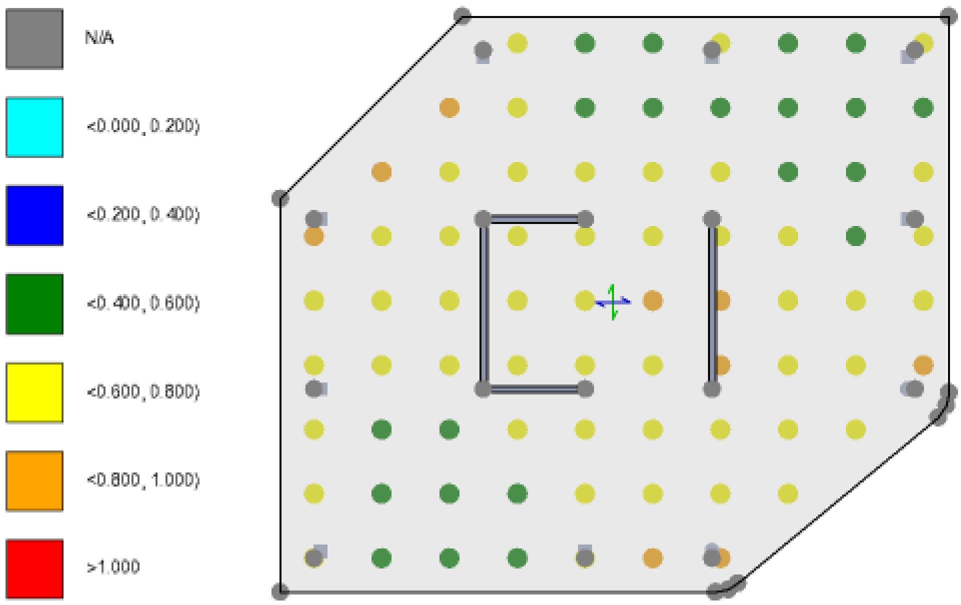
Perform the pile design

The piles are checked (and the mat is designed) by running Design Mats from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process

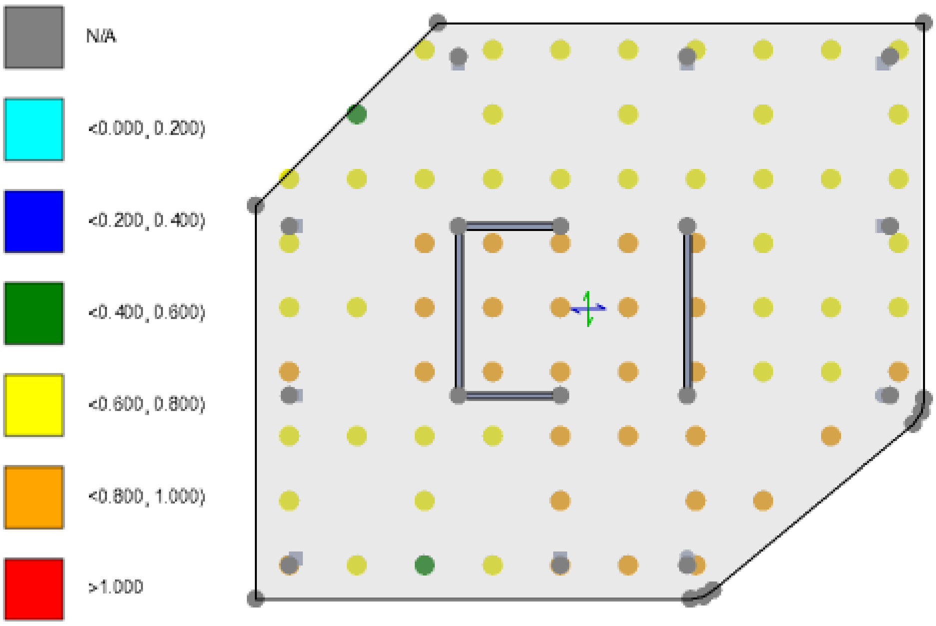
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyze All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

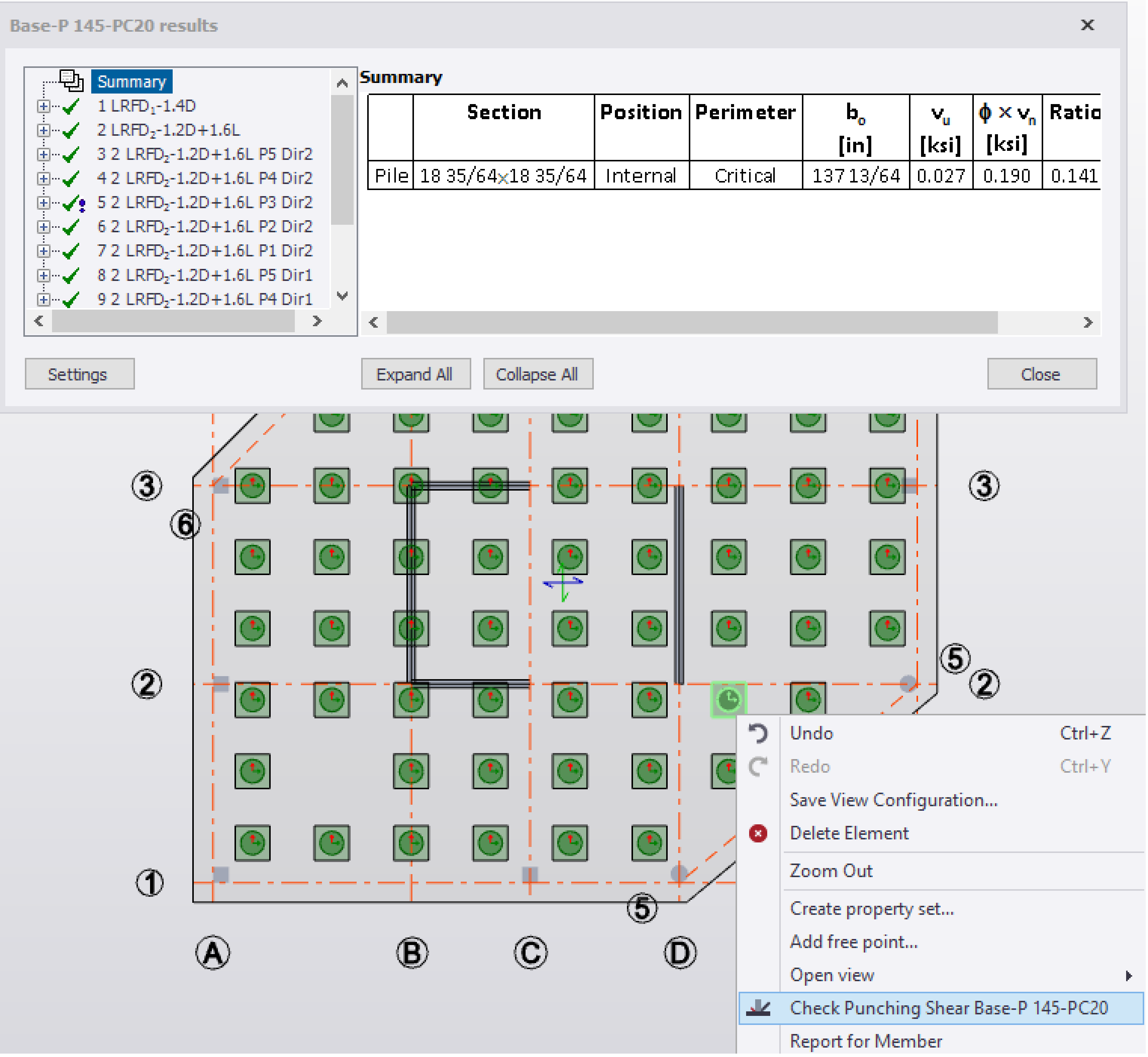
Add and run pile punching checks

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

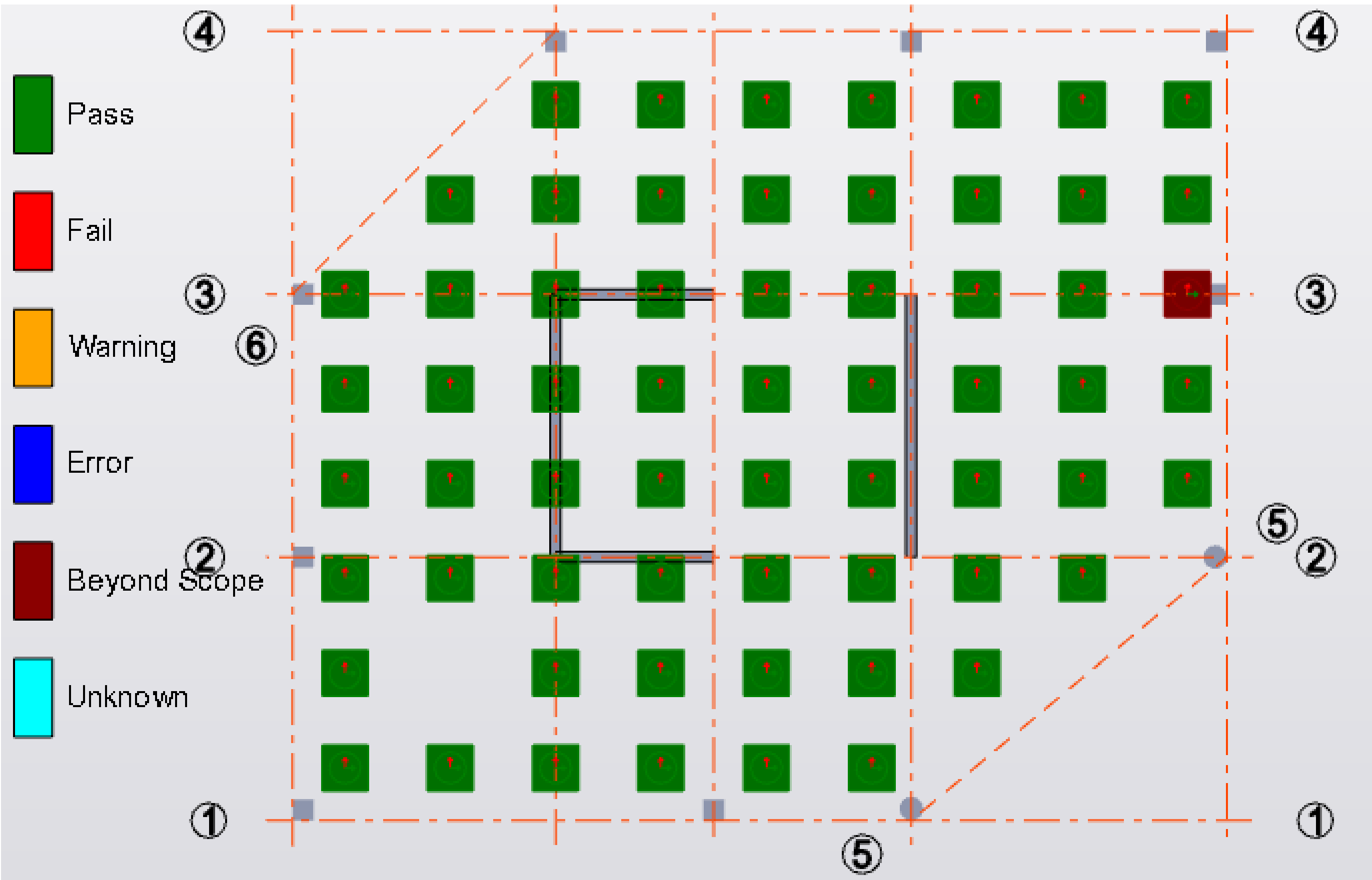
Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See:



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.

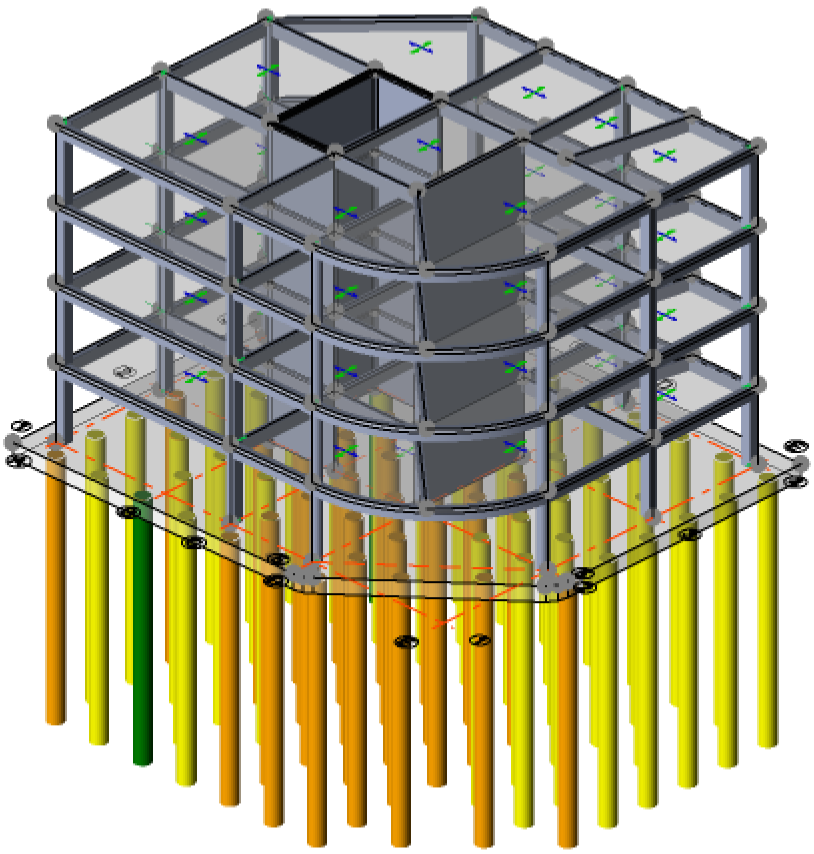
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).
- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (US customary units) (page 583) topic.

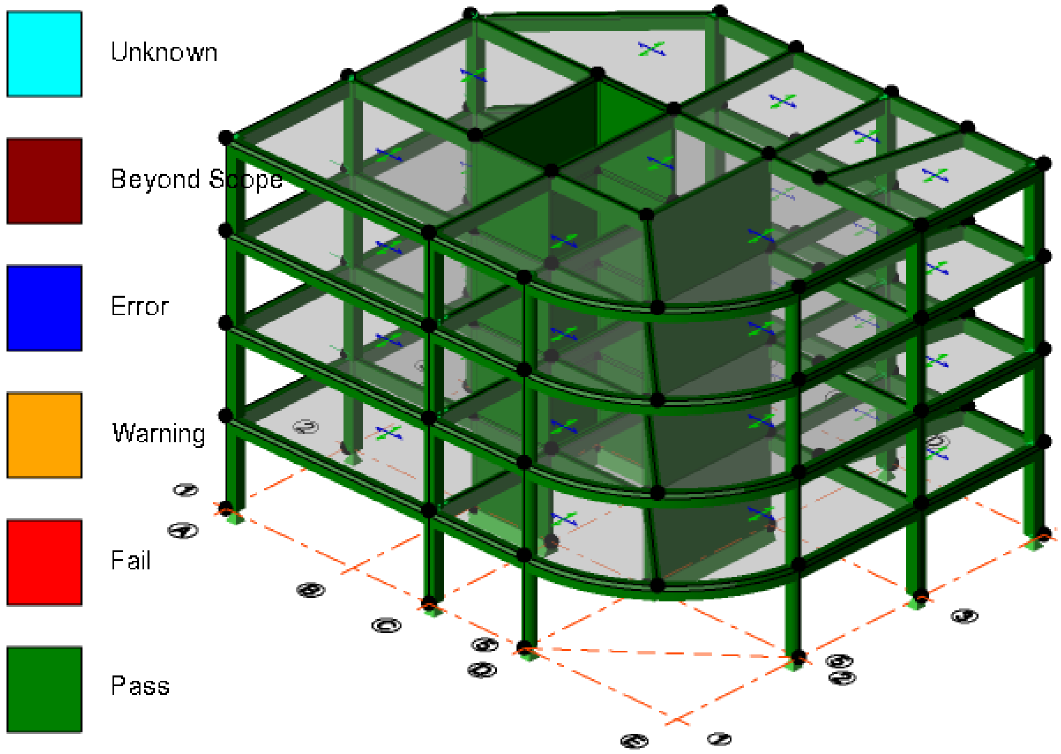
Piled mat foundation design workflow (metric units)

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



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

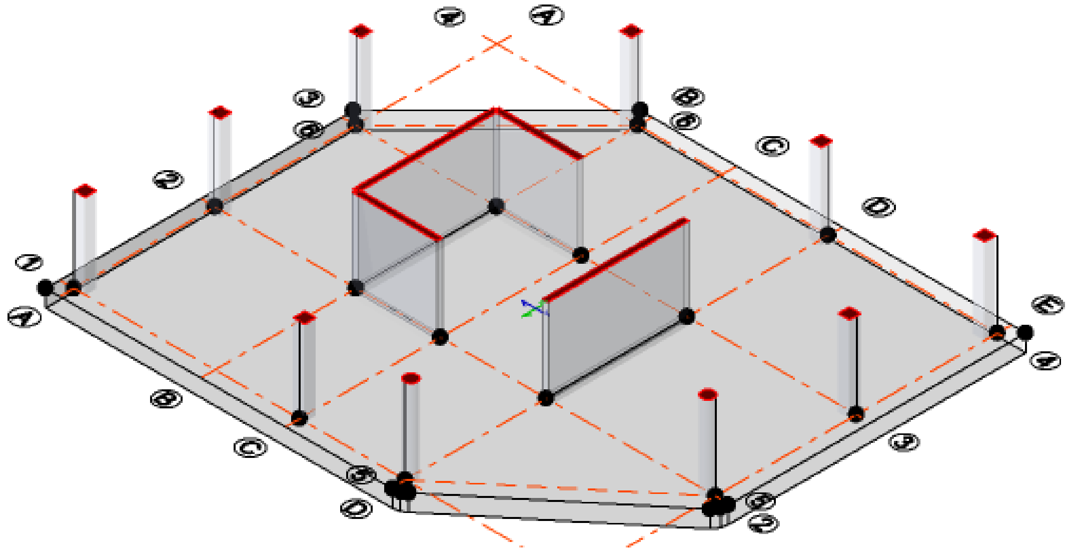
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 1.0m
- Mat thickness 600mm
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

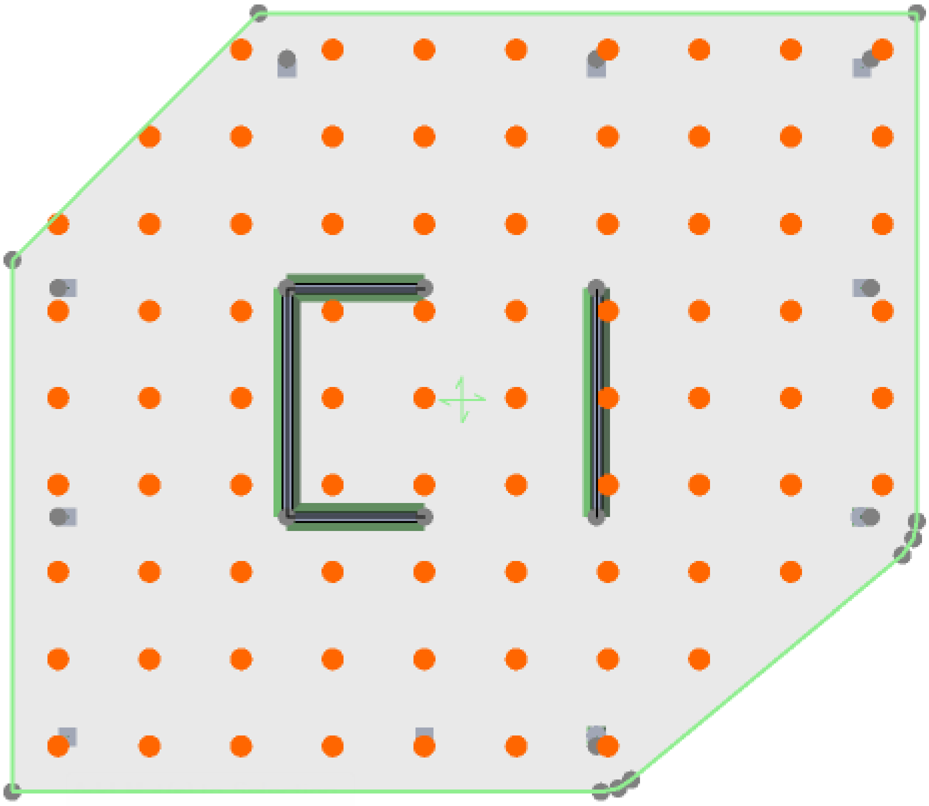
NOTE In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

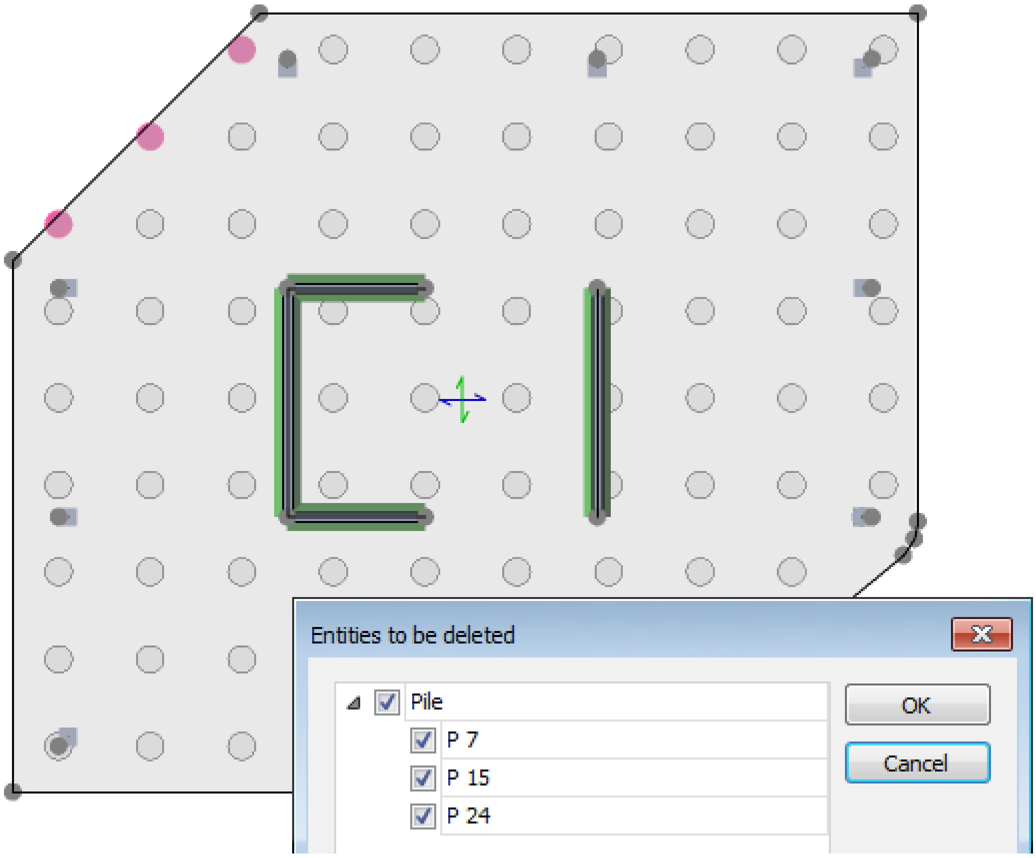
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

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

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

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

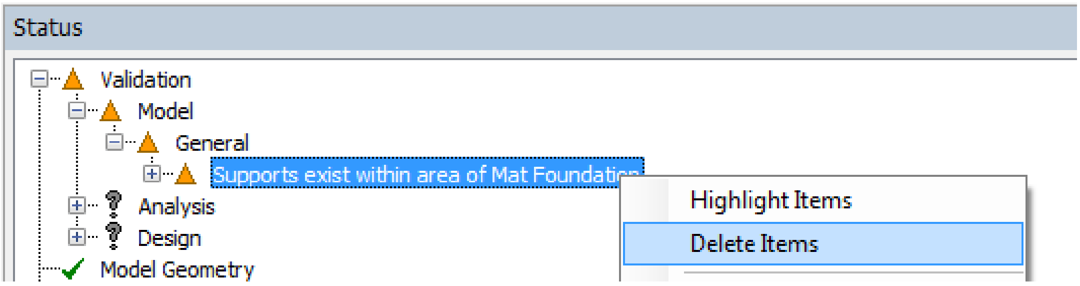
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

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

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

Perform the pile design

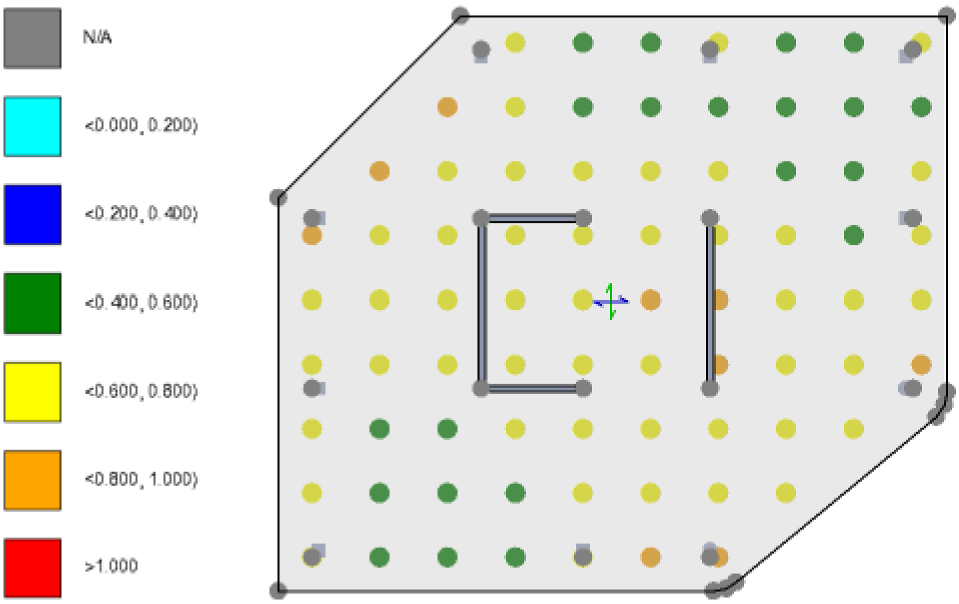
The piles are checked (and the mat is designed) by running Design Mats from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process

NOTE In the current release lateral stability of piled foundation systems is not considered in software design checks.

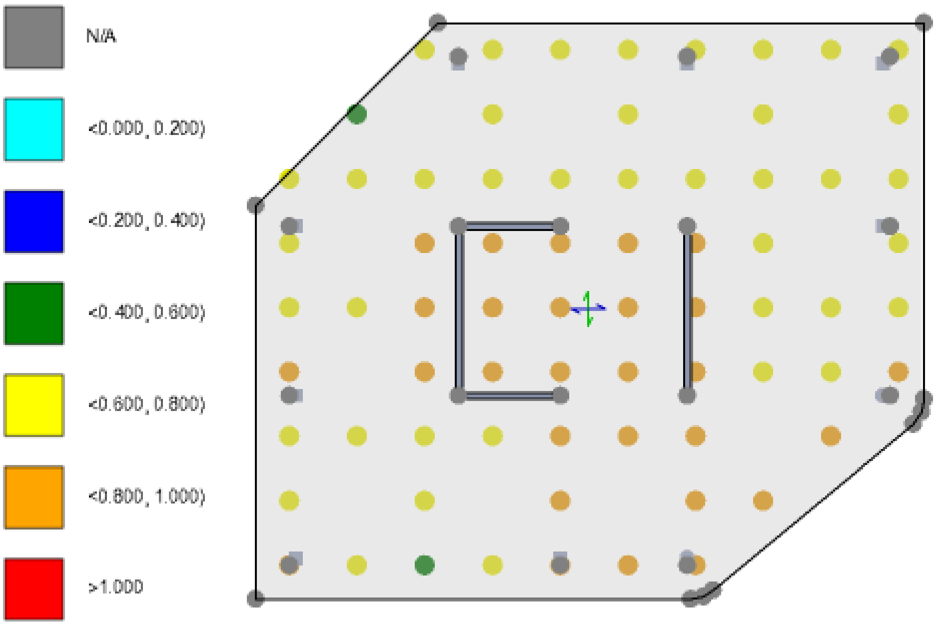
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyse All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

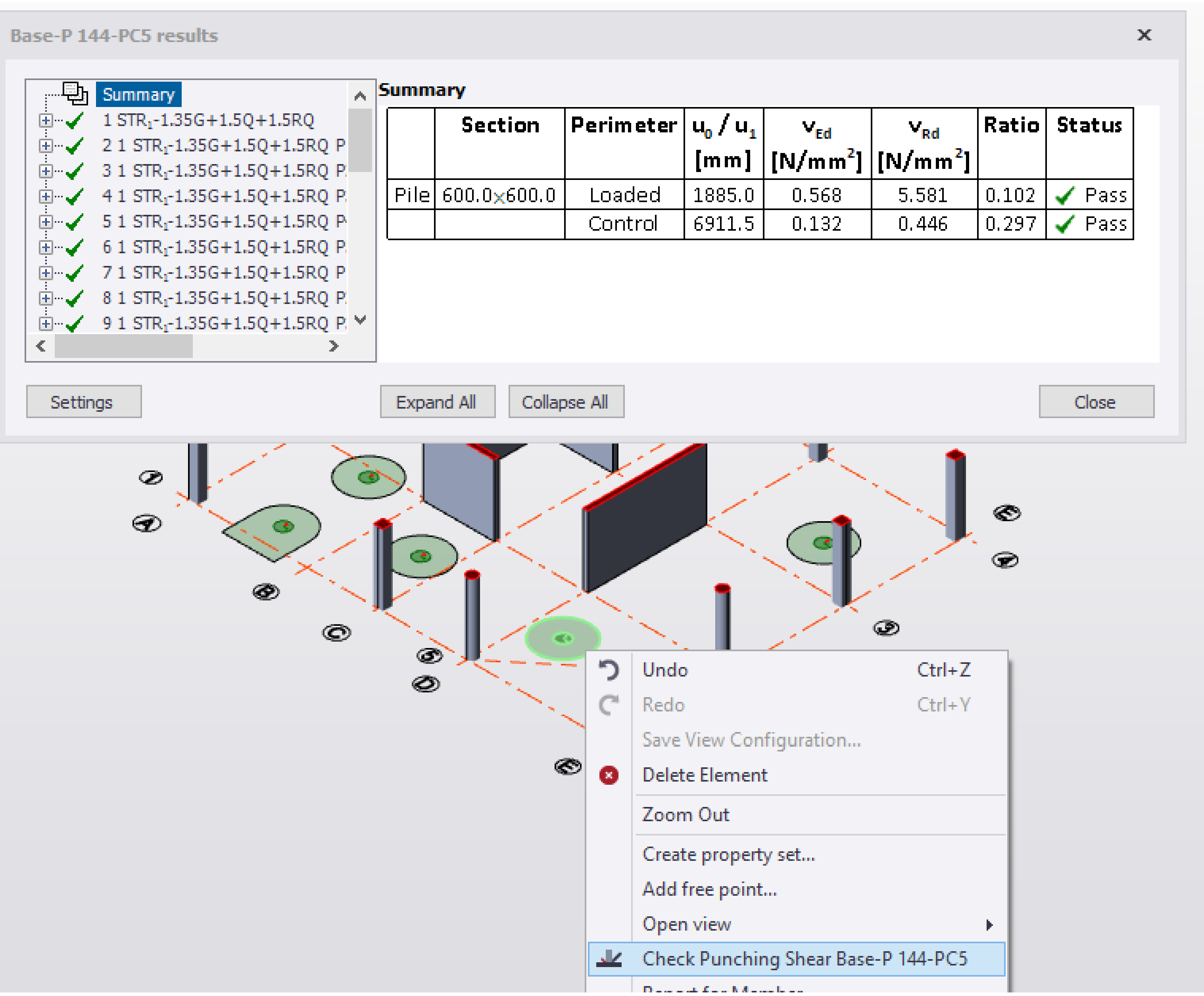
Add and run pile punching checks

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

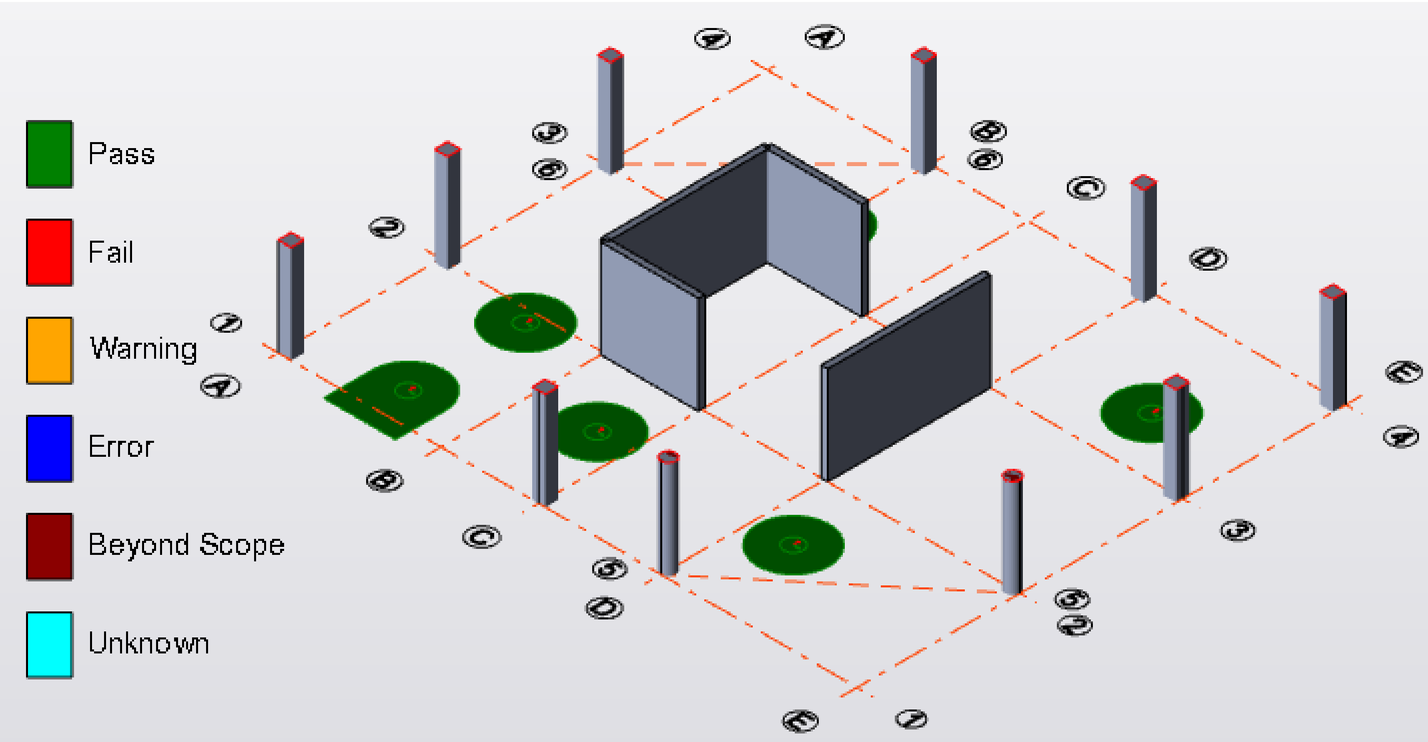
Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See:



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).

- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (metric units) (page 569) topic.

1.11 Sustainability and Tekla Structural Designer

Click the link below to find out about the carbon impact of structures and how Tekla Structural Designer helps engineers to assess this for their projects:

- Measuring the carbon impact of a structure (page 618)
- Embodied carbon workflow (page 620)

Measuring the carbon impact of a structure

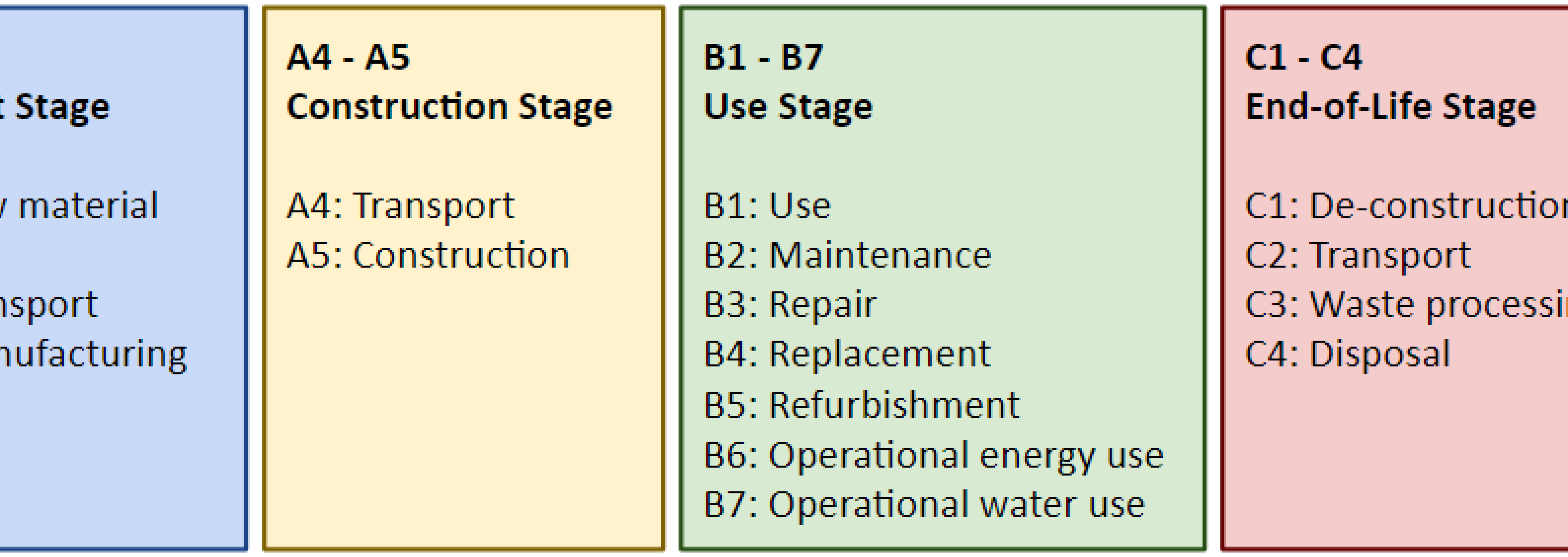
Global impact of construction industry

According to The Institution of Structural Engineers (IStructE):

- Buildings and construction account for about **40%** of energy related CO2 emissions
- Manufacture and disposal of materials used in structures accounts for about **11%** of all greenhouse gas emissions
- Analysis suggests that inefficiency in material use of up to **50%** is common

Typical emissions at each stage of the structure's life

In order to reduce carbon emissions it is first necessary to measure them. This can be done in different ways - in the UK for example by using BS EN 15978[1] the life cycle of the structure can be split into four stages as shown below, and the emissions assessed at each stage.



The distribution of emissions at each stage breaks down in approximately in these proportions:

Stage	Typical distribution of emissions
Product Stage	50%
Construction Stage	5%
Use Stage	43%
End-of-Life Stage	2%

It can be seen that the largest portion of emissions occurs at the Product Stage, and this is the stage that can be targeted by Tekla Structural Designer to make savings.

1. BS EN 15978:2011: Sustainability of construction works. Assessment of environmental performance of buildings. Calculation method. London: BSI, 2011

Measuring Product Stage Carbon

In the UK, the Product Stage is split into modules A1-A3, which are:

- Raw material extraction
- Transport of raw materials to manufacturing facilities
- Manufacture

The carbon impact of materials used in manufacture can then be measured by using an Embodied Carbon Factor (ECF).

This factor is specific to each material and is multiplied by the quantity of that material in order to give a quantity of CO_2 emitted in the Product Stage.

Many construction materials have standard ECF values that the engineer can use, and in addition manufacturers may also provide third-party verified Environmental Product Declarations (EPDs) which can be used when the engineer knows exactly which manufacturer the material will be sourced from.

Reporting and export of embodied carbon data

Tekla Structural Designer includes an automated and comprehensive built-in calculation of *Embodied Carbon* quantities, together with powerful options for graphical review/optimization and reporting of this aspect of design. For more information, see Embodied carbon workflow (page 620).

In addition Tekla Structural Designer also enables model material data to be exported to One Click LCA, which:

- is a widely used cloud-based software for life-cycle assessment and the calculation of embodied carbon.
- complies with a large number of standards including BREEAM and LEED.
- includes a huge database of generic and manufacturer-specific Environmental Product Declarations (EPDs)

The data can either be exported to a spreadsheet, or directly to the One Click LCA service, from where the environmental impact of the design can be reviewed and reports can be created.

NOTE The One Click LCA service can only be used if you have a One Click LCA account.

Both of the above features enable the Engineer to quickly and easily determine the embodied carbon for the part of a project they are responsible for. This also helps them to develop and compare different scheme options. For a selected scheme they can then track progress as they make refinements to drive down the carbon impact.

See also

Embodied carbon workflow (page 620)

Embodied carbon workflow

The basic steps required to calculate, optimize and report embodied carbon content are as follows:

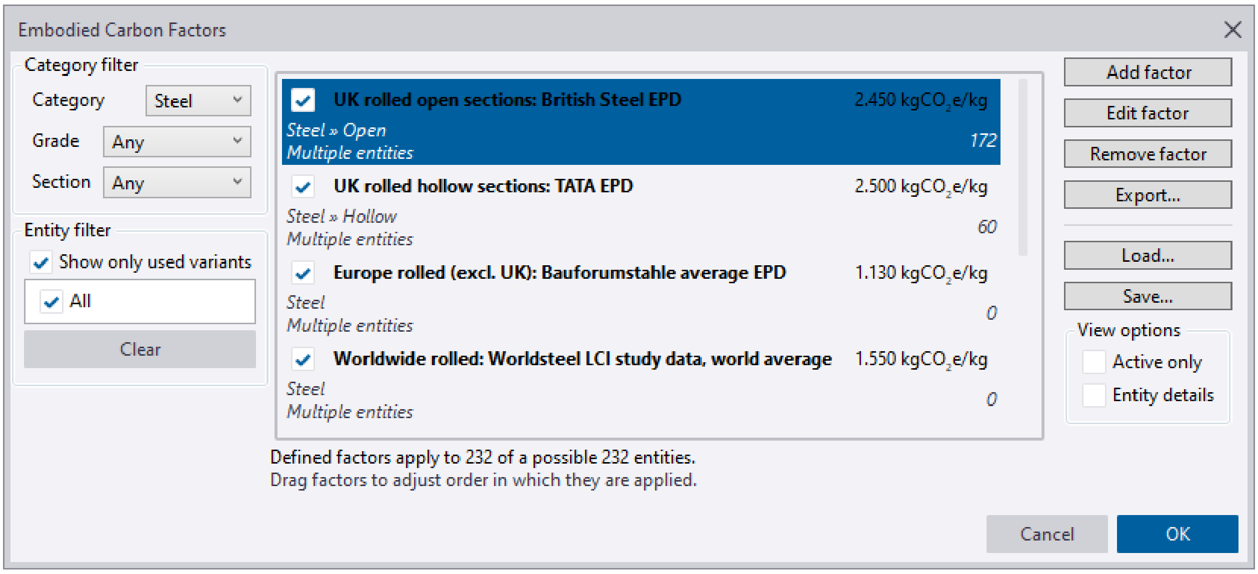
Embodied carbon assessment (video)

Embodied carbon assessment in Tekla Structural Designer (eLearning)

Set up embodied carbon factors

The factors to be assigned to model entities are set up from the which is accessed from the **Home** ribbon by clicking:

	Embodied Carbon Factors



Setting up and managing the embodied carbon factor (ECF) values is the engineer's responsibility, although some sample ECF definitions have been added to the settings sets for all countries/ codes for the following categories; concrete, metal deck, reinforcement, steel, timber.

NOTE There is potentially considerable variability/ uncertainty in ECFs - for example for concrete, many mix variations are possible for every strength grade and all could be unique. Hence the sample ECFs are included principally for guidance and are not a complete set - the engineer is advised to review, edit and add to them as they need.

Many construction materials have ECF values that the engineer can use, and in addition manufacturers may also provide third-party verified Environmental Product Declarations (EPDs) which can be used when the engineer knows exactly which manufacturer the material will be sourced from.

Standard ECF values should be saved in global settings so that they can be applied to future models also. However there are some project specific ones (like cladding for walls and roofs) that you would not want to make available for all models.

NOTE If you open an old model (i.e. one created before you established a global set of ECF values), initially the won't contain any values, but you can click the **Load** button in order to inherit any factors that exist as global settings.

Related tasks

•

Manage how factors are applied to the model

Once the factors have been set up, the same dialog is used to manage how the ECFs in each category are applied to entities in the model.

You first select the main category you wish to view/edit at the top left e.g. “Steel”. You can then review for this category the ECF list order, description (which can be any variation of text and numbers etc) value, and the number of assigned entities where applicable. This last figure is key to the management process and is circled in the image below.

- The list of factors in the dialog can be further controlled by additional filters below the main category, those available being dependent on the selected category - e.g. by a grade filter for the material categories and/or by a country filter for categories such as reinforcement and metal deck.

The current assignment of ECFs is constantly active, starting at the top of the list and working down, so the top factor has first priority and so on.

From here you might choose to reorder the list of factors to suit the current model by dragging and dropping factors with the mouse. The assignment of factors updates automatically as edits are made - applicable factors at the top of the list always being assigned wherever possible in preference to factors occurring lower down the list.

If certain factors are inappropriate for the current model they can be deactivated via the checkbox at the left of the factor to prevent them from being applied.

Once an entity in the model has a factor from the list assigned the process stops for that entity and category, hence there is no duplication of factor assignment within a category where lower (in the list) ECF definitions are also applicable.

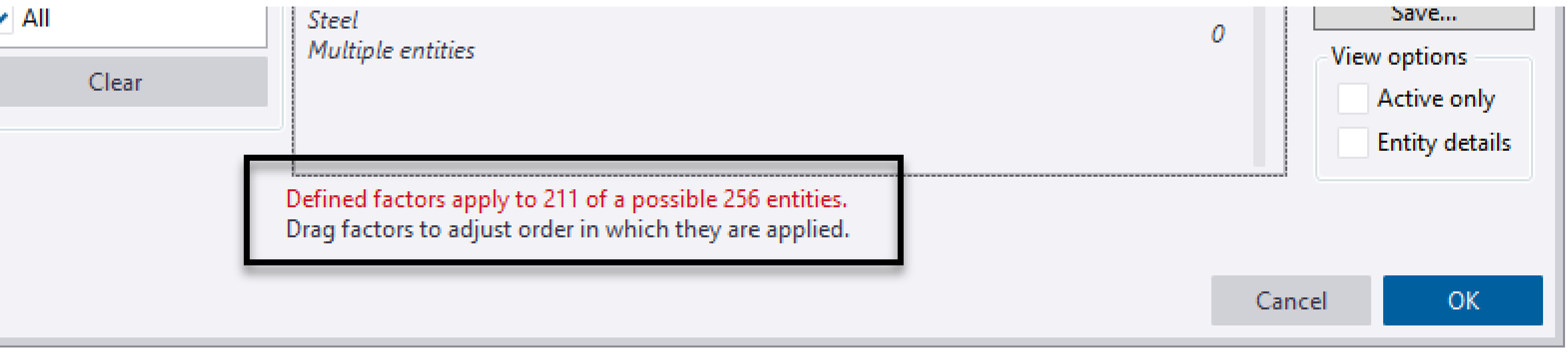
NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

You can edit the description and/or value for ECFs and add additional ones via the Add/Edit factor buttons. While doing this you can also set the category and entity *filters* which control the grades and/or entity types to which the factor applies. For example you could set a factor for the concrete category that applies only to a specific grade and only concrete columns and beams. You can also remove factors.

NOTE The category and entity filters available in the Edit Factors dialog are dictated by the main category filter set at the top left of the dialog - e.g. "Concrete", "Steel", "Cladding" etc. Thus for example only steel entity types are listed in the Edit Factor dialog's entity filter list when "Steel" is set as the main dialog category.

NOTE Any amendments will only apply to the current model unless you choose to save them back to the global settings set.

Note that as shown below, a message will be displayed in red text until every entity in the model has had a factor assigned, the purpose of the message being to highlight that the total embodied carbon mass of the structure will be incomplete.



To help resolve this issue you can graphically review the carbon factors, as described in the following **Review how factors have been applied** section.

Related tasks

•

Review how factors have been applied

Probably the easiest way to review how the ECFs have been applied to the model is graphically in a Review View.

By stepping through the applicable carbon source categories you can quickly review how each of the factors have been applied.

An option is also provided here to apply overrides if required, (although preferably you should consider adding new factors instead using the). Setting overrides is primarily recommended for entities that require different factors but are indistinguishable in terms of their data (and so cannot be assigned automatically) such as Roof/ Wind panels (in the “Cladding” category).

NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

Related task

•

Examine tabular overview/details

At an early stage you might choose to display the Embodied Carbon Overview in a Review Data Table. This is very useful because it rapidly provides a grand total of the embodied carbon mass for the entire structure. This can be summarized for the whole model either on a by plane basis....

Review Data

Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	35173	223	157.9
St. 2 (2nd Floor)	8.000	35173	223	157.9
St. 1 (1st Floor)	4.000	35173	223	157.9
St. Base (Base)	0.000	0	0	-
FRM 1	-	0	-	-
Other	-	14257	-	-
Total	-	**119776**	-	-

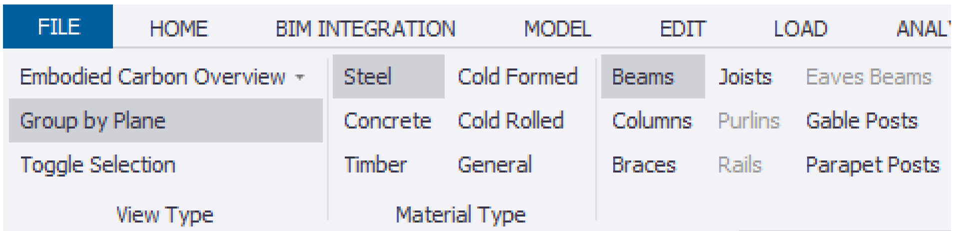
or by construction type...

Review Data

Embodied Carbon Overview

Construction Type	Embodied Carbon Mass [kgCO_2e]
Composite Slabs	54013
Steel Beams Rolled Composite	26688
Steel Beams Rolled Non-Composite	24668
Steel Columns Rolled	12010
Steel Braces Rolled	2248
Wind Walls	0
Total	**119626**

The overview can also be filtered in order to see the totals for specific construction types only, such as steel beams, as shown below.

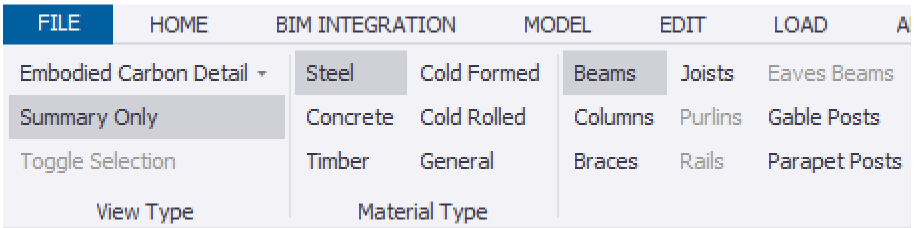


Review Data

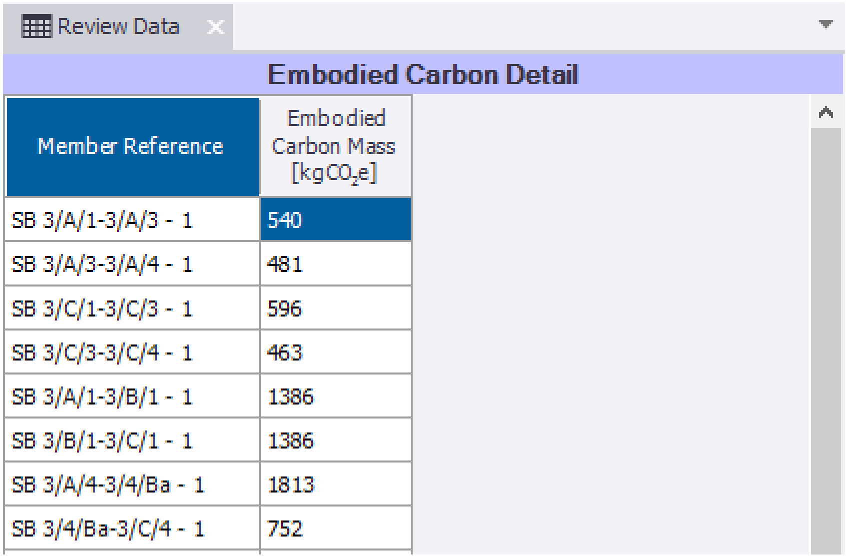
Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [kgCO_2e]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [kgCO_2e/m^2]
St. 3 (Roof)	12.000	17168	223	77.1
St. 2 (2nd Floor)	8.000	17168	223	77.1
St. 1 (1st Floor)	4.000	17168	223	77.1
St. Base (Base)	0.000	0	0	-
FRM 1	-	0	-	-
Other	-	0	-	-
Total	-	**51505**	-	-

If you want to investigate where the above values have come from, you also have the option to display an Embodied Carbon Detail Review Data Table for individual construction types.



This can be displayed as a summary...



Member Reference	Embodied Carbon Mass [kgCO_2e]
SB 3/A/1-3/A/3 - 1	540
SB 3/A/3-3/A/4 - 1	481
SB 3/C/1-3/C/3 - 1	596
SB 3/C/3-3/C/4 - 1	463
SB 3/A/1-3/B/1 - 1	1386
SB 3/B/1-3/C/1 - 1	1386
SB 3/A/4-3/4/Ba - 1	1813
SB 3/4/Ba-3/C/4 - 1	752

or, broken down to shown the separate components that contribute to the total for each entity...

Review Data

Embodied Carbon Detail

Member Reference	Material	Quantity	Unit	Carbon Factor	Embodied Carbon Mass [kgCO_2e]
SB 3/A/1-3/A/3 - 1	Steel	209.97	kg	UK rolled open sections: British Steel EPD (2.450)	514
SB 3/A/1-3/A/3 - 1	Connection End 1	5.25	kg	UK rolled open sections: British Steel EPD (2.450)	13
SB 3/A/1-3/A/3 - 1	Connection End 2	5.25	kg	UK rolled open sections: British Steel EPD (2.450)	13
SB 3/A/1-3/A/3 - 1	Coating	6.5	m^2	<not set>	0
SB 3/A/3-3/A/4 - 1	Steel	186.88	kg	UK rolled open sections: British Steel EPD (2.450)	458
SB 3/A/3-3/A/4 - 1	Connection End 1	4.67	kg	UK rolled open sections: British Steel EPD (2.450)	11
SB 3/A/3-3/A/4 - 1	Connection End 2	4.67	kg	UK rolled open sections: British Steel EPD (2.450)	11
SB 3/A/3-3/A/4 - 1	Coating	6.4	m^2	<not set>	0
SB 3/C/1-3/C/3 - 1	Steel	231.51	kg	UK rolled open sections: British Steel EPD (2.450)	567
SB 3/C/1-3/C/3 - 1	Connection End 1	5.79	kg	UK rolled open sections: British Steel EPD (2.450)	14
SB 3/C/1-3/C/3 - 1	Connection End 2	5.79	kg	UK rolled open sections: British Steel EPD (2.450)	14

NOTE For steel members with pinned or fixed connections, the carbon mass of the connections at each end is determined as a percentage of that of the member. This percentage can be adjusted via

Related tasks

- Create embodied carbon overview tabular results
- Create embodied carbon detail tabular results

Review utilization and embodied carbon

Once the model has been designed you can investigate the potential for optimizing utilization and embodied carbon in a Review View.

This view shows the embodied carbon usage for all, or selected entity types. It also has filters that can be configured to locate any parts of the model where high carbon usage coincides with low utilization.

NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

Having rapidly identified inefficiency in this way, you can then consider the potential for optimizing your design in some way.

Related tasks

•

Create reports

Embodied carbon mass is automatically included in the .

Additionally, a separate can be created which summarizes the embodied carbon in the model by plane and by construction type.

Tekla Structural Designer	Project				Job Ref. 123	
	Structure Model 1				Sheet no. Page 1/1	
	Calc. by	Date 09/03/2021	Chk'd by	Date 09/03/2021	App'd by	Date 09/03/2021

Embodied Carbon Overview by Plane

Reference	Level [m]	Embodied Carbon Mass [kgCO$_2$e]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [kgCO$_2$e/m^2]
St. 3 (Roof)	12.000	35173	223	157.9
St. 2 (2nd Floor)	8.000	35173	223	157.9
St. 1 (1st Floor)	4.000	35173	223	157.9
St. Base (Base)	0.000	0	0	
FRM 1		0		
Other		14257		
Total		119776		

Embodied Carbon Overview by Construction Type

Construction Type	Embodied Carbon Mass [kgCO$_2$e]
Composite Slabs	54013
Steel Beams Rolled Composite	26838
Steel Beams Rolled Non-Composite	24668
Steel Columns Rolled	12010
Steel Braces Rolled	2248
Wind Walls	0
Total	119776

NOTE You can also include a table of Embodied Carbon Factors in the same report If required.

Related tasks

- Configure and display a Material Listing report
- Configure and display an Embodied Carbon report

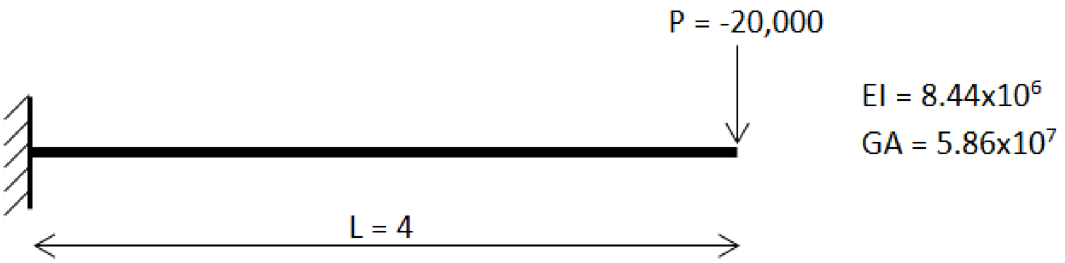
1.12 Analysis verification examples

A small number of verification examples are included in this section. Our full automatic test suite for the Solver contains many hundreds of examples which are run and verified every time the Solver is enhanced. These verification examples use SI units unless otherwise stated.

1st order linear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0%
Support Moment	PL	-80,000	-80,000	0%
Tip Deflection	$\frac{PL^3}{3EI} + \frac{PL}{GA}$	-0.0519	-0.0519	0%

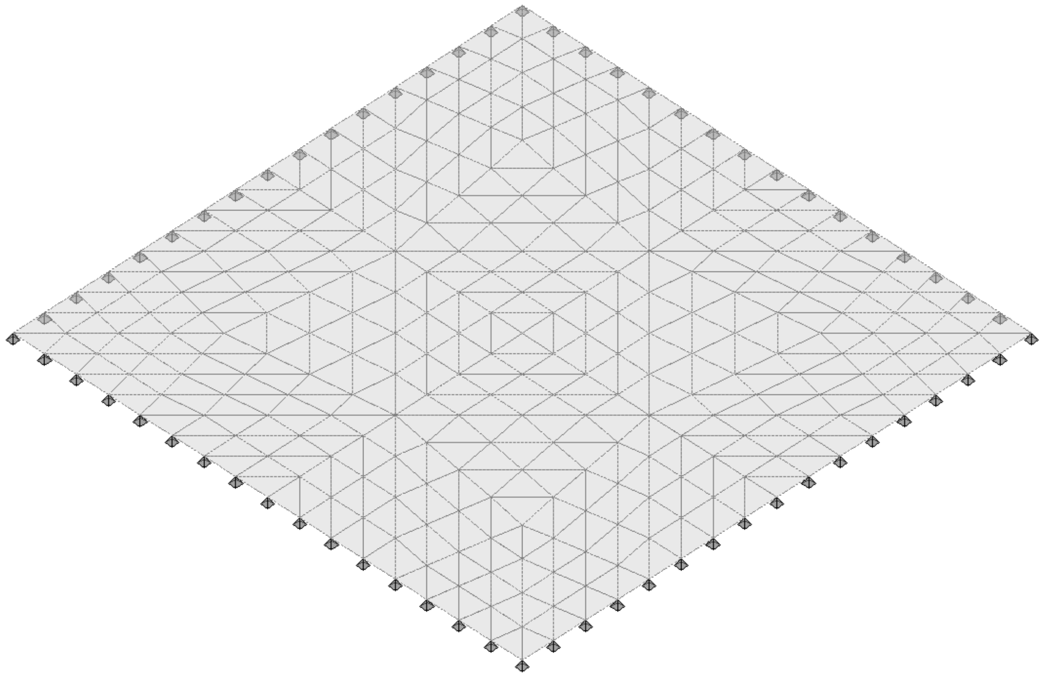
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order linear - Simply supported square slab

Problem definition

Calculate the mid span deflection of an 8x8 simply supported slab of 0.1 thickness under self-weight only. Take material properties $E=2x10^{11}$, $G=7.7x10^{10}$ and ρ=7849.



Assumptions

A regular triangular finite element mesh is used with sufficient subdivision. Flexural and shear deformation is included, and the material is assumed to be isotropic.

Key results

The mid-span deformation is calculated using Navier's Method.

$$w = \frac{16q_0}{\pi^6 D}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

$$M_x = M_y = \frac{16q_0}{a^2\pi^4}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{m^2\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2} + \nu\frac{16q_0}{b^2\pi^4}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{n^2\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

Result	Theoretical value	Comparison 1	Solver value	% error
Mid-span deflection	7.002×10^{-3}	6.990×10^{-3}	7.031×10^{-3}	0.43%
Mid-span Moment	23616	23708	23649	0.14%

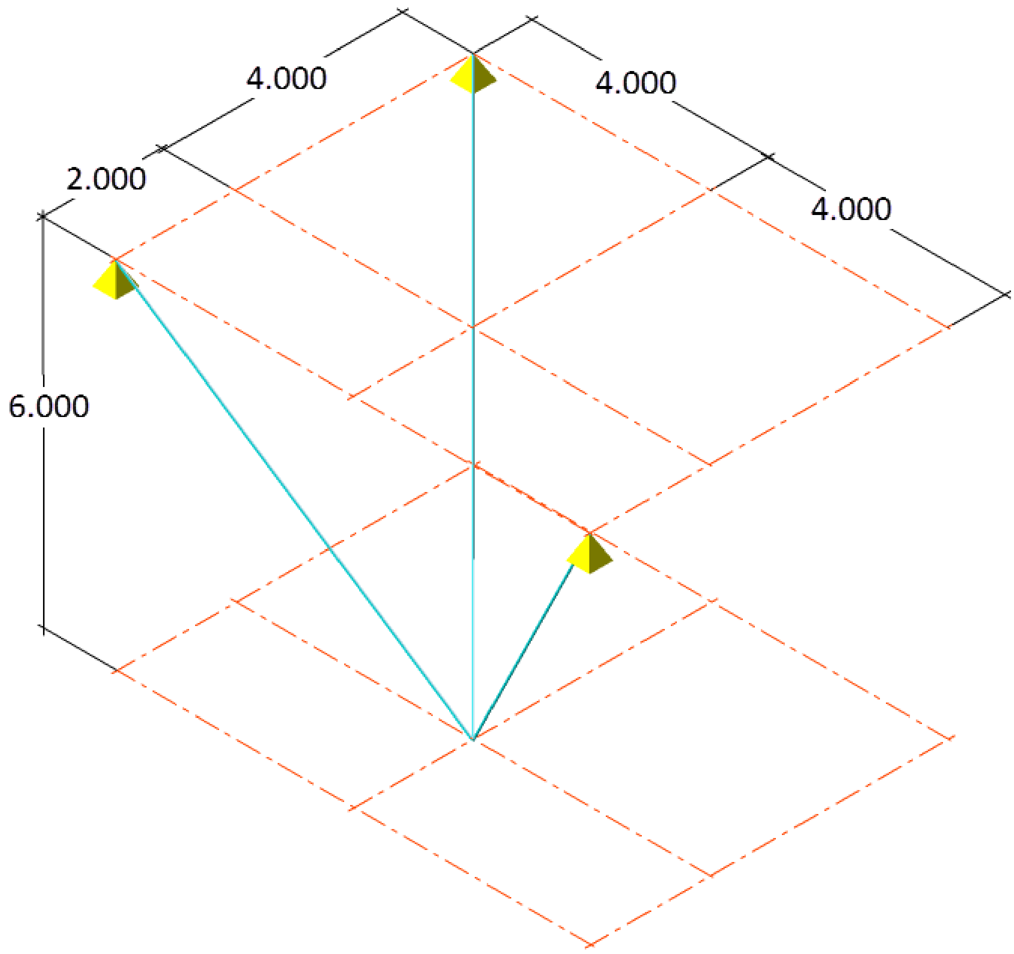
Conclusion

An acceptable match is observed between the theoretical values and the solver results. An acceptable match is also observed between the solver results and those obtained independently.

1st order linear - 3D truss

Problem definition

Three truss members with equal and uniform *EA* support an applied load of -50 applied at the coordinate (4, 2, 6). The start of each truss member is fixed and are located at (0, 0, 0), (8, 0, 0) and (0, 6, 0) respectively. Calculate the axial force in each element.



Key results

The results for this problem are compared against those published by Beer and Johnston, and against another independent analysis package.

Result	Beer and Johnston	Comparison 1	Solver value	% error
(0, 0, 0) - (4, 2, -6)	10.4	10.4	10.4	0.00 %
(8, 0, 0) - (4, 2, -6)	31.2	31.2	31.2	0.00 %
(0, 6, 0) - (4, 2, -6)	22.9	22.9	22.9	0.00 %

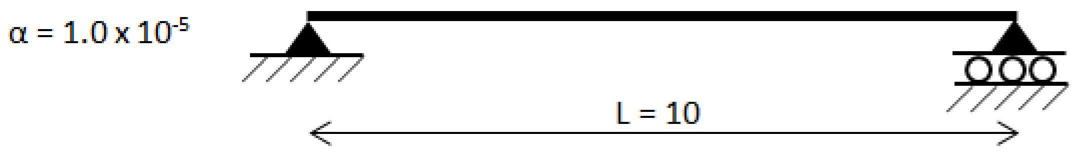
Conclusion

An exact match is observed between the values reported by the solver and those reported by Beer and Johnston.

1st order linear - Thermal load on simply supported beam

Problem definition

Determine the deflection, *U*, due to thermal expansion at the roller support due to a temperature increase of 5. The beam is made of a material with a thermal expansion coefficient of 1.0 x 10^{-5}.



Assumptions

The roller pin is assumed to be frictionless.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Translation at roller	$U = \Delta T \times \alpha \times L$	5x10^{-4}	5x10^{-4}	0%

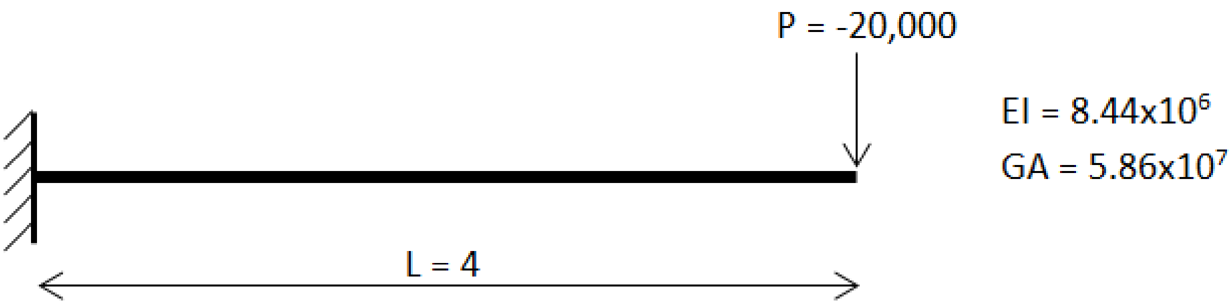
Conclusion

An exact match is shown between the theoretical result and the solver result.

1st order nonlinear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0 %
Support Moment	-PL	-80,000	-80,000	0 %
Tip Deflection		-0.0519	-0.0519	0 %

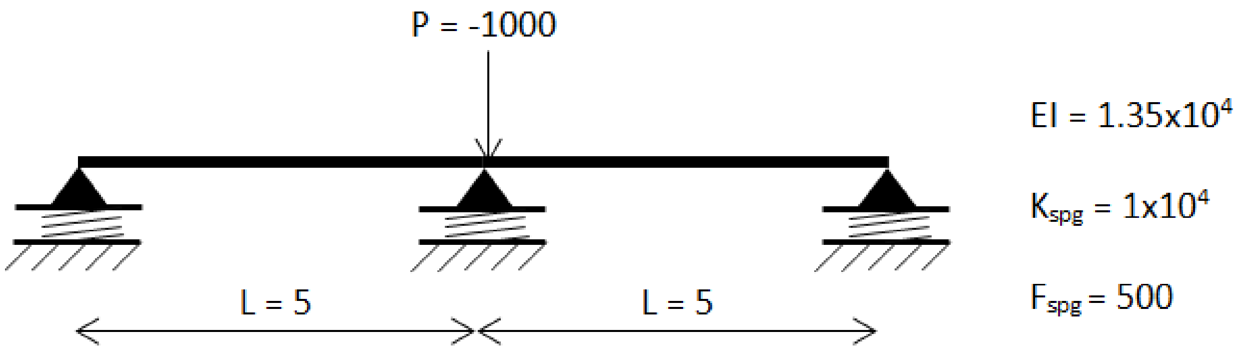
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order nonlinear - Nonlinear supports

Problem definition

A 10 long continuous beam is simply supported by three translational springs as shown. All springs have a maximum resistance force of 500. Calculate the reaction forces and deflection at each support.



Assumptions

Axial and shear deformations are ignored.

Key results

Result	Comparison 1	Solver value
LHS Reaction	250	250
Centre Reaction	500	500
RHS Reaction	250	250
LHS Displacement	-0.025	-0.025
Centre Displacement	-0.797	-0.797
RHS Displacement	-0.025	-0.025

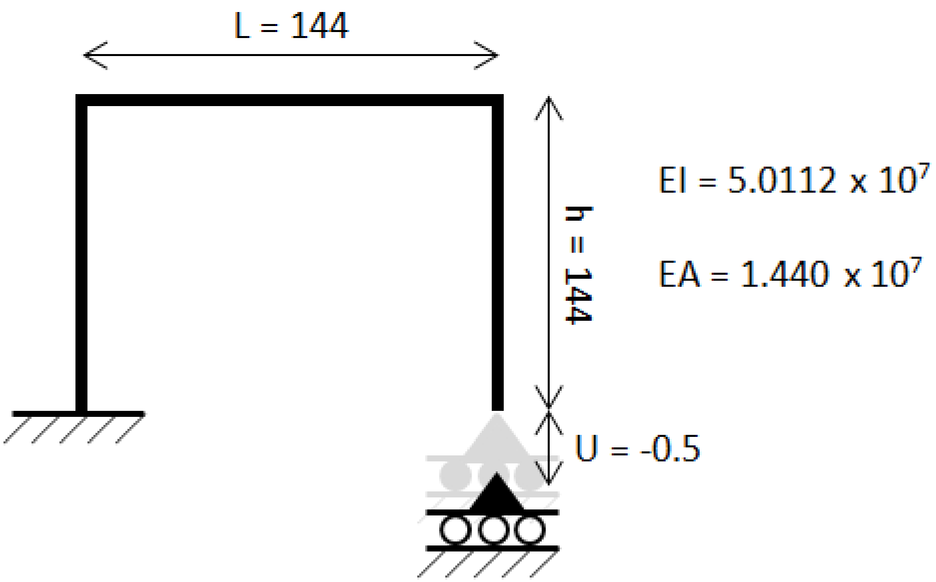
Conclusion

An exact match is shown between the solver and the independent analysis package.

1st order nonlinear - Displacement loading of a plane frame

Problem definition

Calculate the reaction forces of the plane moment frame shown below with the applied displacement *U*.



Assumptions

All elements are constant and equal *EI*. Axial and shear deformations are ignored; to achieve the former - analytically the cross sectional area was increased by a factor of 100,000 to make axial deformation negligible.

Key results

Results were compared with two other independent analysis packages.

Result	Comparison 1	Comparison 2	Solver value
LHS Vertical Reaction	6.293	6.293	6.293
LHS Moment Reaction	-906.250	-906.250	-906.250
RHS Vertical Reaction	-6.293	-6.293	-6.293

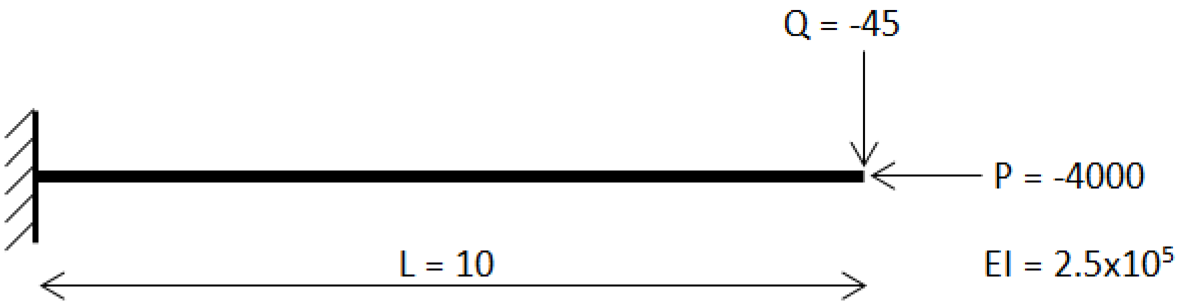
Conclusion

An exact match is shown between the solver and the two independent analysis packages.

2nd order linear - simple cantilever

Problem definition

A 10 long cantilever is subjected to a lateral tip load of 45 and an axial tip load of 4000.



Assumptions

Shear deformations are ignored. Results are independent of cross section area; therefore any reasonable value can be used. Second order effects from stress stiffening are included, but those caused by update of geometry are not. The beam is modelled with only one finite element, (if more elements had been used the result would converge on a more exact value).

Key results

Results were compared with an independent analysis package.

Result	Comparison	Solver value
Tip Deflection	-0.1677	-0.1677
Base Moment Reaction	-1121	-1121

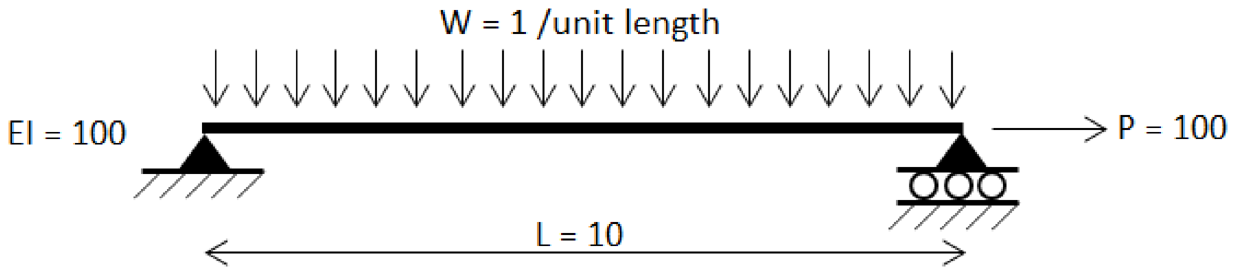
Conclusion

An exact match is observed between the values reported by the solver and the values reported in "Comparison".

2nd order linear - Simply supported beam

Problem definition

Determine the mid-span deflection and moment of the simply supported beam under transverse and tensile axial load.



Assumptions

Shear deformations are excluded. Results are independent of cross section area; therefore any reasonable value can be used. The number of internal nodes varies from 0-9.

Key results

The theoretical value for deflection and moment are calculated as:

$$Y_{max} = -0.115 = \frac{5wL^4}{384EI} \times \frac{\frac{1}{\cosh U} - 1 + \frac{U^2}{2}}{\frac{5}{24}U^4}$$

$$M_{max} = -0.987 = \frac{wL^2}{8} \times \frac{2(\cosh U - 1)}{U^2 \cosh U}$$

Where U is a variable calculated:

$$U = 5 = \sqrt{\frac{PL^2}{4EI}}$$

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
1	-0.116	0.734 %	-0.901	8.631 %
3	-0.115	0.023 %	-0.984	0.266 %
5	-0.115	0.004 %	-0.986	0.042 %
7	-0.115	0.001 %	-0.986	0.013 %

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
9	-0.115	0 %	-0.986	0.005 %

Conclusion

As the element is subdivided the result converges to the correct theoretical value.

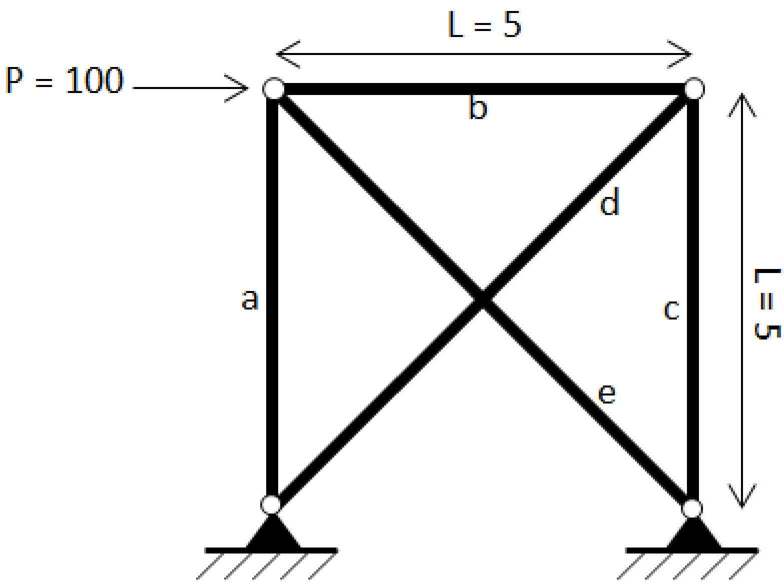
Reference

Timoshenko. S. 1956. *Strength of Materials, Part II, Advanced Theory and Problems*. 3rd Edition. D. Van Nostrand Co., Inc. New York, NY.

2nd order nonlinear - Tension only cross brace

Problem definition

Calculate the axial forces of the elements a-e shown in the 5x5 pin jointed plane frame shown below. Elements d and e can resist tensile forces only.



Assumptions

All elements are constant and equal EA. A smaller value of EA will increase the influence of second order effects, whereas a larger value will decrease the influence.

Key results

Under the applied loading element *e* becomes inactive. The theoretical formulas presented below are obtained using basic statics. Note that a positive value indicates tension. These results assume no 2nd order effects; this requires the value of *EA* to be sufficiently large to make the 2nd order effect negligible.

Result	Theoretical formula	Theoretical value	Solver value	% Error
a	0	0	0	0
b	-P	-100	-100	0
c	-P	-100	-100	0
d	$P\sqrt{2}$	141.42	141.42	0
e	0	0	0	0

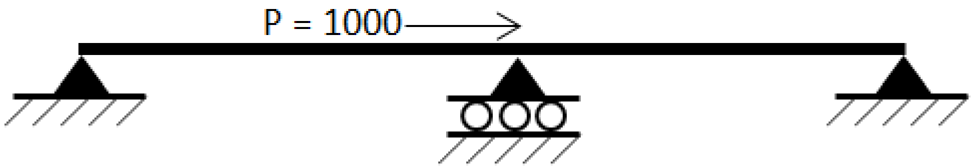
Conclusion

An exact match is observed between the values reported by the solver and the values predicted using statics. A 1st order nonlinear analysis can be used, with any section sizes, to confirm this result without second order effects.

2nd order nonlinear - Compression only element

Problem definition

Calculate the reaction forces for the compression only structure shown below.



Assumptions

All elements are constant and equal *EA*, and can resist only compressive forces

Key results

Under the applied loading the element on the left becomes inactive, therefore all applied loading is resisted by the support on the right.

Result	Theoretical formula	Theoretical value	Solver value
LHS Reaction	0	0	0
RHS Reaction	-P	-1000	-1000

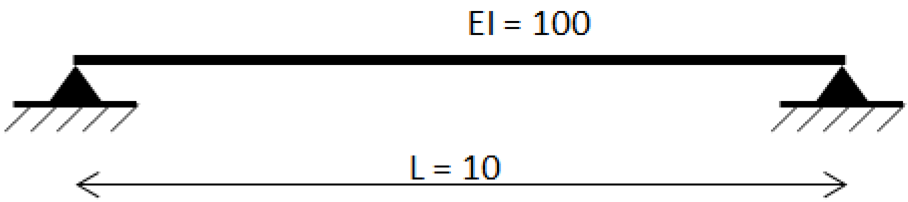
Conclusion

An exact match is observed between the values reported by the solver and the theoretical values.

1st order modal - Simply supported beam

Problem definition

Determine the fundamental frequency of a 10 long simply supported beam with uniform *EI* and mass per unit length equal to 1.0.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5. Consistent mass is assumed.

Key results

The theoretical value for the fundamental frequency is calculated as:

$$\omega = 0.9870 = \sqrt{\left(\frac{\pi}{10}\right)^4 \frac{100}{1}} = \sqrt{\left(\frac{\pi}{L}\right)^4 \frac{EI}{m/L}}$$

Where *m* is the total mass of the beam.

No. internal nodes	Solver value	% error
0	1.0955	10.995 %
1	0.9909	0.395 %
2	0.9878	0.081 %
3	0.9872	0.026 %
4	0.9871	0.011 %
5	0.9870	0.005 %

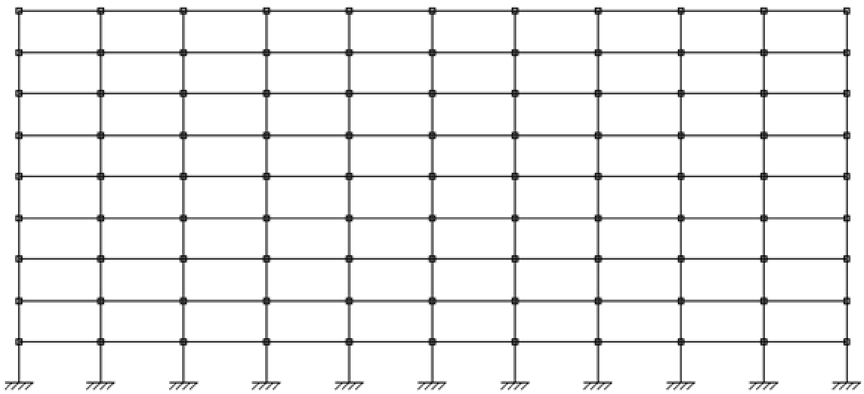
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

1st order modal - Bathe and Wilson eignenvalue problem

Problem definition

A 2D plane frame structure has 10 equal bays each measuring 6.096m wide and 9 stories 3.048m tall. The column bases are fully fixed. All beams and columns are the same section, which have a constant mass/unit length equal to 1.438. Calculate the first three natural frequencies (in Hz) of the structure under self-weight.



Assumptions

Shear deformations are excluded. Each beam/column is represented by one finite element. Consistent mass is assumed.

Key results

The results for this problem are compared with those published by Bathe and Wilson and against an independent analysis package.

Mode	Bathe and Wilson	Comparison	Solver value
1	0.122	0.122	0.122
2	0.374	0.374	0.375
3	0.648	0.648	0.652

Conclusion

The results show a good comparison with the original published results and against the other analysis packages.

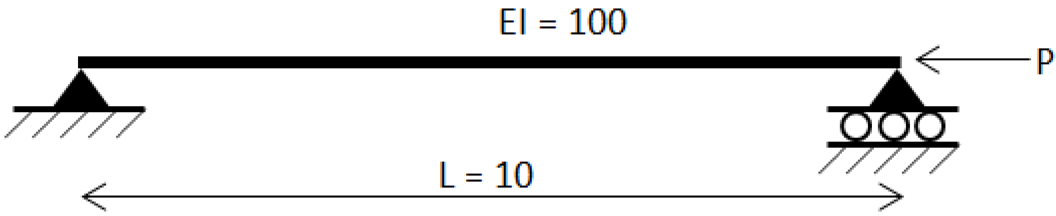
References

Bathe, K.J. and E.L. Wilson. 1972. *Large Eigen Values in Dynamic Analysis.* Journal of the Engineering Mechanics Division. ASCE Vol. 98, No. EM6. Proc. Paper 9433. December.

2nd order buckling - Euler strut buckling

Problem definition

A 10 long simply supported beam is subjected to an axial tip load of P.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5.

Key results

The theoretical value for the first buckling mode is calculated using the Euler strut buckling formula:

$$\lambda = 9.869 = \frac{\pi^2 EI}{L^2}$$

With P = -1.0 the following buckling factors are obtained

No. internal nodes	Solver value	% error
0	12.000	21.59 %
1	9.944	0.75 %
2	9.885	0.16 %
3	9.875	0.05 %
4	9.872	0.02 %
5	9.871	0.01 %

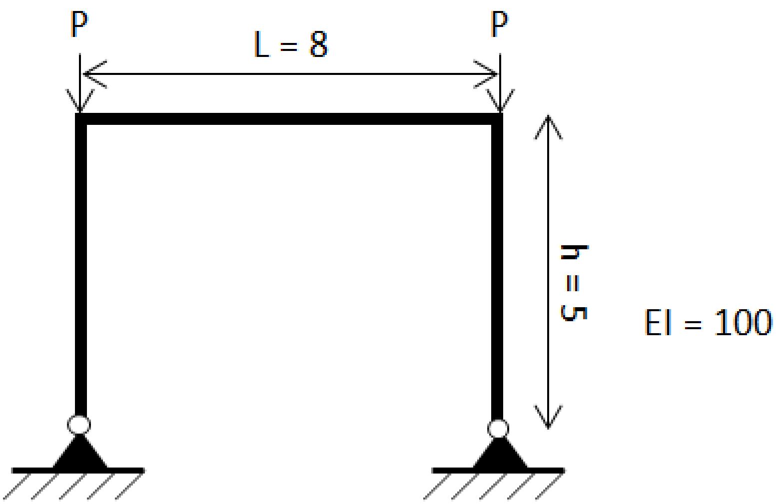
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

2nd order buckling - Plane frame

Problem definition

Calculate the buckling factor of the moment frame shown below.



Assumptions

All elements are constant and equal *EI*. Axial deformations are ignored; to achieve this the cross section area is set to 1000. The number of elements per member is varied between 0 and 5.

Key results

The theoretical buckling load is calculated by

$$P_{cr} = 6.242 = \frac{(kL)^2 EI}{h^2}$$

where

$$kL \tan(kL) = 1.249 = \frac{6h}{L}$$

Which can be solved using Newtons method and five iterations

$$kL_{n+1} = kL_n - \frac{f(kL_n)}{f'(kL_n)} = kL_n - \frac{kL_n \tan(kL_n) - \frac{6h}{L}}{kL_n \sec^2(kL_n) + \tan(kL_n)}$$

No. internal nodes/ members	Solver value	% error
0	6.243	0.17 %
1	6.243	0.01 %
2	6.242	0.00 %
3	6.242	0.00 %

No. internal nodes/ members	Solver value	% error
4	6.242	0.00 %
5	6.242	0.00 %

Conclusion

A good match is shown between the solver and theory. The discrepancy decreases as the level of discretization is increased.

References

Timoshenko, S. and J. M. Gere. 1961. *Theory of Elastic Stability*. 2nd Edition. McGraw-Hill Book Company.

Index

1

2

3

A

B

C

D

T

U

W