Tekla Structural Designer 2018i

Engineer's Handbooks (Eurocodes)

September 2018
(6.1.06)

Table of Contents

Wind Modeling Handbook

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

- The **Wind Model** method is the most comprehensive - requiring you to first 'clothe' the structure in wind and roof panels and then run the 'Wind Wizard'. The wizard creates wind zone loads that are subsequently decomposed to the structure during analysis.
- Alternatively you might choose to **manually apply wind loads** (thus avoiding the requirement to construct a wind model). For this approach loads can either be applied directly to the structure as Panel, Member, or Structure loads; or they can be applied as **Simple Wind** loads, which are subsequently decomposed to the structure during analysis.

Overview of the Wind Model method

The basic steps required for this method are as follows:

1. Clothe the structure in wind wall and roof panels
2. Perform the gravity design
3. Run the Wind Wizard
4. Review the wind zones
5. Define the wind loadcases
6. Review wind zone loads
7. Combine the wind loadcases into design combinations
8. Perform the static design

Related video

- Wind Loading

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.

You can, should you wish, use Tekla Structural Designer purely for wind assessment – by setting up a model of consisting only of wall panels and roof panels (no members) the software can 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. The results may be compromised if you define many small panels rather than one large one. (Particularly the calculation of the reference height 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 all shaded in the same colour (the one assigned to 'Wind Wall - Front' in Settings > Scene). The inward faces will all be shaded in a different colour. To correct any mistakes, choose the **Reverse** command (located on the **Edit** toolbar) 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 Wind tab of the Project Workspace, with affected panels being marked thus: ( ).

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.

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** - 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** - 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.

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 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.

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.

- Wind Zone Editing for Multibay Structures (Eurocode and BS only)

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.

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.

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.

EC1991 1-4 Wind Wizard

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).

Design Codes and References

Unless explicitly stated all calculations in the EC1991 1-4 Wind Wizard are in accordance with the relevant sections of EC1991 1-4 (Ref. 3) 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)
- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings. (Ref. 7)

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)

Unless explicitly noted otherwise, all clauses, figures and tables referred to in this section of the handbook are from reference 3.

Using the EC1991 1-4 Wind Wizard with BREVe data

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

Data Source page

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

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

The remaining choices on the Data Source page are:

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.

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.

If working to the Sweden NA, Obstructions are not considered.

Next

Clicking **Next** takes you to the Basic Data page.

BREVe location page

This page allows you to define the location of the site using the BREVe database, and also the orientation if known.

Building details

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.

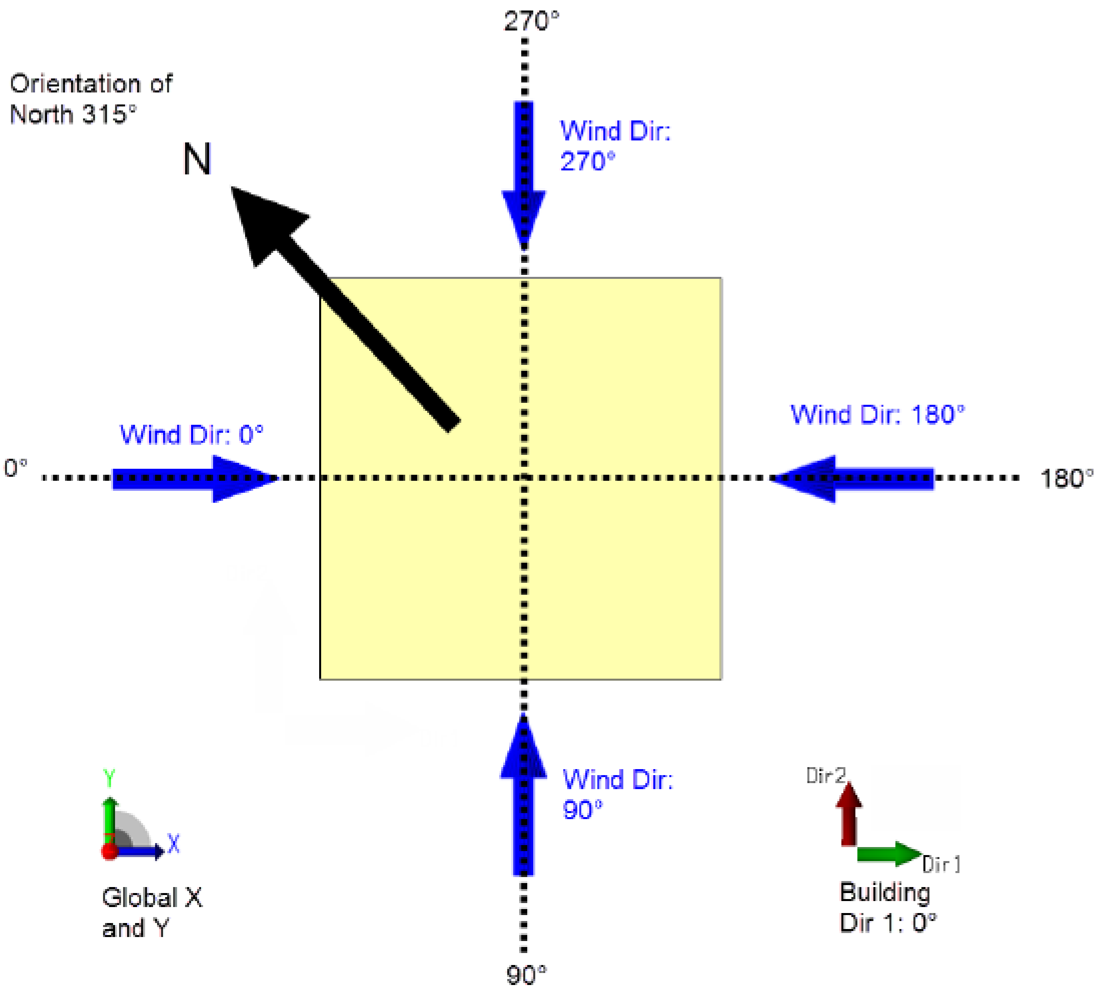
Orientation of North

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:

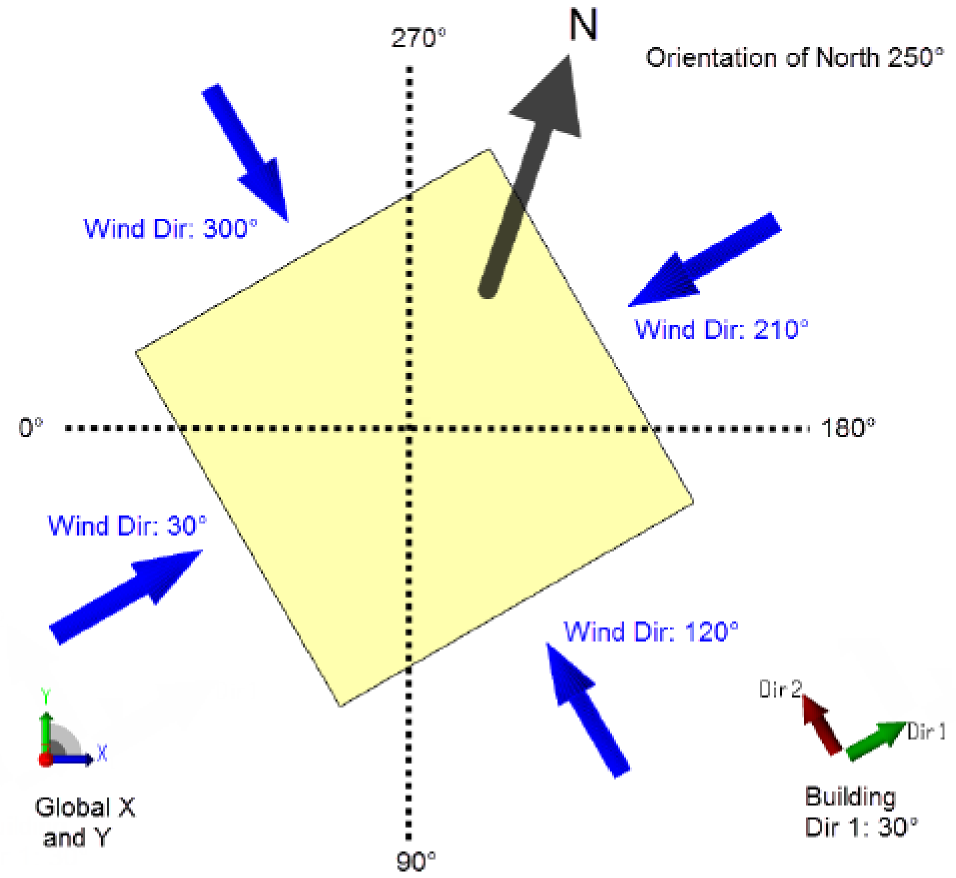
In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°.

The resulting relation between the building axes and North is as shown below:



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:



BREVe information

Using BREVe, there are 2 methods available for you to define the site location:

Site By Ref...

You can define the grid reference of the site.

You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid reference from a postcode or given location, for example www.streetmap.co.uk or www.multimap.co.uk,

Site By Map...

You can pick the site from a Land / Town Map,

- You can pick the site from a Orography Map.
- You can pick the site from a ground roughness Category Map,

The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).

Next

Click **Next** to go to the Basic Data (BREVe) page.

Basic Data page

This page is used you to define the site details.

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.

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 (BREVe) page

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

Terrain Category

- ***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.**

Next

Click **Next** to go to the Orography (BREVe) page.

Orography (BREVe) page

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.

Orographic Feature (Clause A.3)

- ***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.

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.

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 (BREVe) page.

Using the EC1991 1-4 Wind Wizard with other data

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.

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.

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.

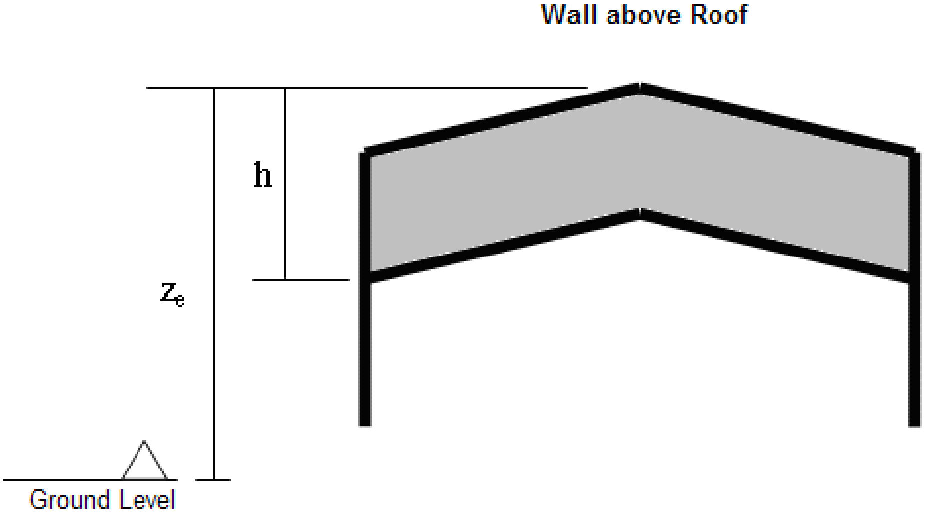
Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no have been encountered.

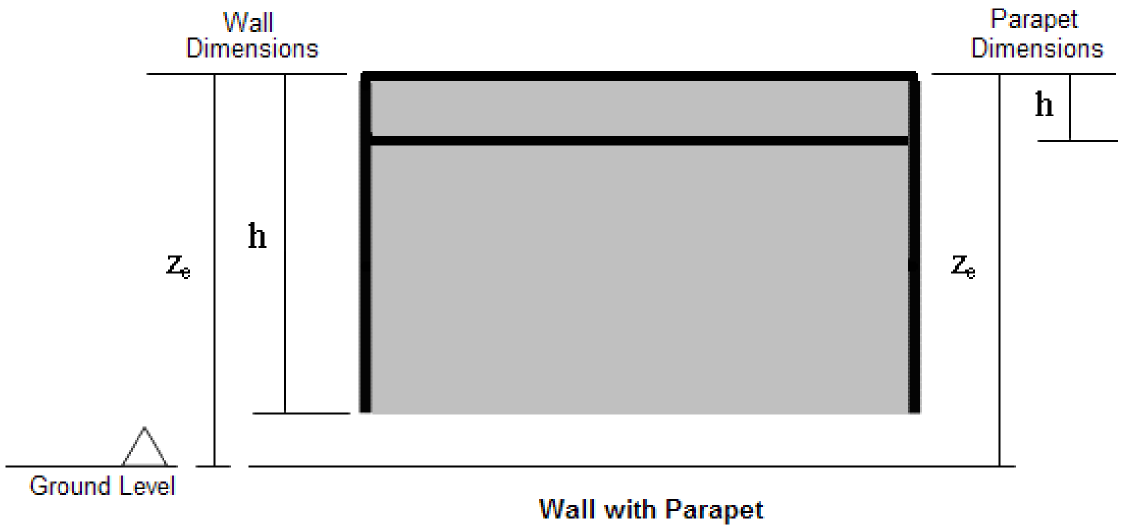
EC1991 1-4 Wind Zones

EC1991 1-4 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.





Wind Model Loadcases

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** toolbar).

The Wind Loadcases dialog is only available once a valid Wind Model has been created using the Wind Wizard.

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

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

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

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

EC1991 1-4 - Creating Wind Loadcases

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** toolbar.

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

- -0.3 for Cpi with -ve roof Cpe; not Overall
- -0.3 for Cpi with +ve roof Cpe; not Overall
- +0.2 for Cpi with -ve roof Cpe; not Overall
- +0.2 for Cpi with +ve roof Cpe; not Overall
- Overall with zero for Cpi; -ve roof Cpe

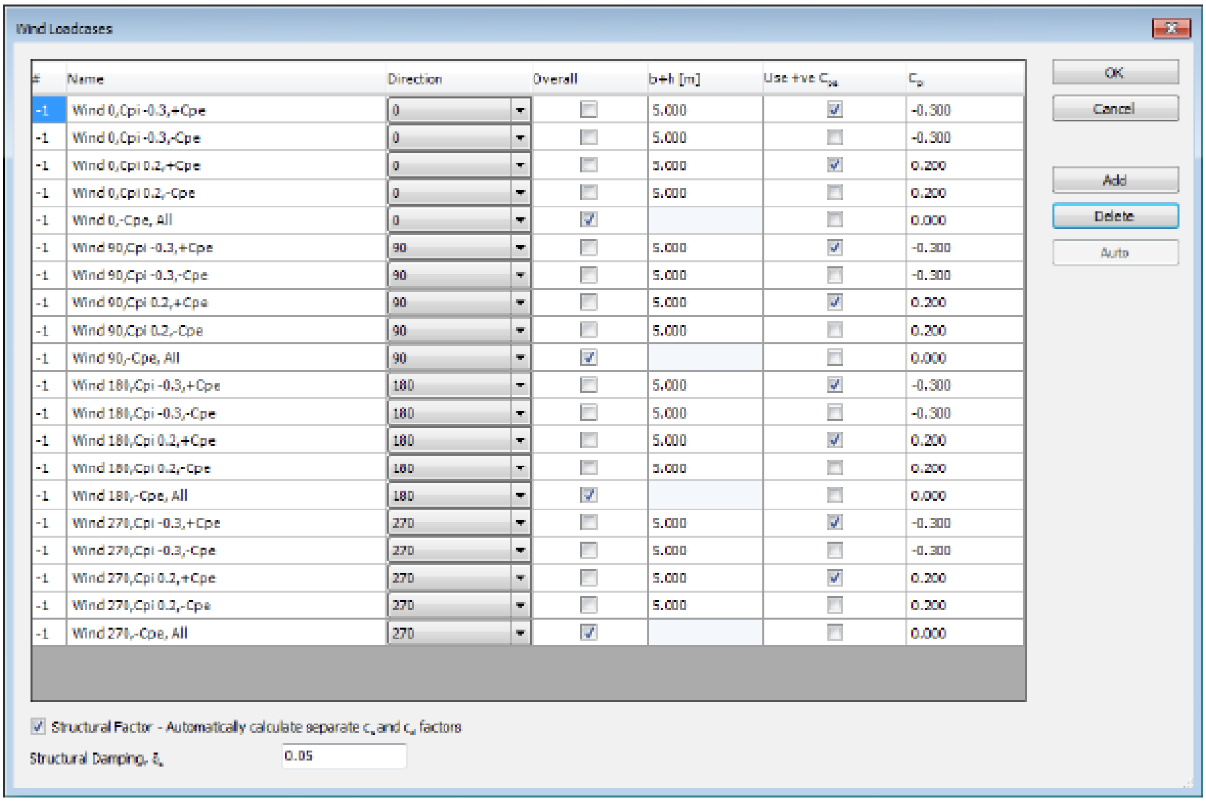
Alternatively you can create loadcases manually using the **Add** button. You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

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

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

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.



#	Name	Direction	Overall	b+h [m]	Use +ve C_{pe}	C_p
-1	Wind 0,Cpi -0.3,+Cpe	0	☐	5.000	☑	-0.300
-1	Wind 0,Cpi -0.3,-Cpe	0	☐	5.000	☐	-0.300
-1	Wind 0,Cpi 0.2,+Cpe	0	☐	5.000	☑	0.200
-1	Wind 0,Cpi 0.2,-Cpe	0	☐	5.000	☐	0.200
-1	Wind 0,-Cpe, All	0	☑		☐	0.000
-1	Wind 90,Cpi -0.3,+Cpe	90	☐	5.000	☑	-0.300
-1	Wind 90,Cpi -0.3,-Cpe	90	☐	5.000	☐	-0.300
-1	Wind 90,Cpi 0.2,+Cpe	90	☐	5.000	☑	0.200
-1	Wind 90,Cpi 0.2,-Cpe	90	☐	5.000	☐	0.200
-1	Wind 90,-Cpe, All	90	☑		☐	0.000
-1	Wind 180,Cpi -0.3,+Cpe	180	☐	5.000	☑	-0.300
-1	Wind 180,Cpi -0.3,-Cpe	180	☐	5.000	☐	-0.300
-1	Wind 180,Cpi 0.2,+Cpe	180	☐	5.000	☑	0.200
-1	Wind 180,Cpi 0.2,-Cpe	180	☐	5.000	☐	0.200
-1	Wind 180,-Cpe, All	180	☑		☐	0.000
-1	Wind 270,Cpi -0.3,+Cpe	270	☐	5.000	☑	-0.300
-1	Wind 270,Cpi -0.3,-Cpe	270	☐	5.000	☐	-0.300
-1	Wind 270,Cpi 0.2,+Cpe	270	☐	5.000	☑	0.200
-1	Wind 270,Cpi 0.2,-Cpe	270	☐	5.000	☐	0.200
-1	Wind 270,-Cpe, All	270	☑		☐	0.000

Fields

Structural Factor - Automatically calculate separate c_s and c_d factors

The UK NA states that the Structural Factor, c_sc_d may be separated into a size factor c_s and a dynamic factor c_d, i.e. it is still acceptable to apply Clause 6.2 (1) a) to d) and set c_sc_d = 1, or use Annex D. Where **Structural Factor - Automatically calculate separate cs and cd factors** is checked, c_d is calculated using Figure NA.9 and c_s using Table NA.3.

The option to automatically calculate c_c and c_d is not available for other National Annexes.

Structural Damping

The δ_s value is used to determine the dynamic factor c_d. It is only visible if the Separate Factors box is checked. See Table F.2.

Name

The loadcase name is auto generated from the other input parameters, but it can be edited if required.

Direction

The direction the loadcase will be created for is selected from the droplist.

Overall

You are able to flag if the loadcase is to be specifically used for examining the overall behaviour of the structure by checking this box.

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 $c_s c_d$), 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 C_{pe}

Where 2 sets of coefficients are given in a BS6399 Table for roof zones, this field indicates if the negative or positive C_{pe} value is to be used.

C_{pi}

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

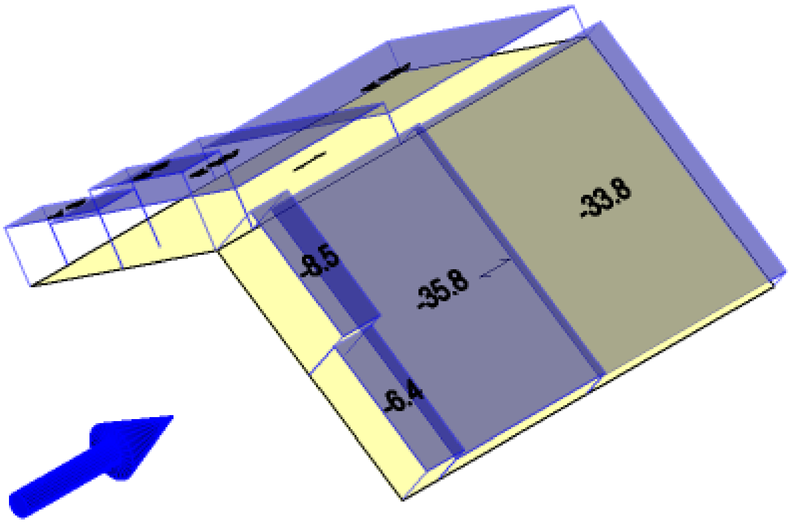
Click this button to create a default set of loadcases in each of the directions.

Wind Model Load Decomposition

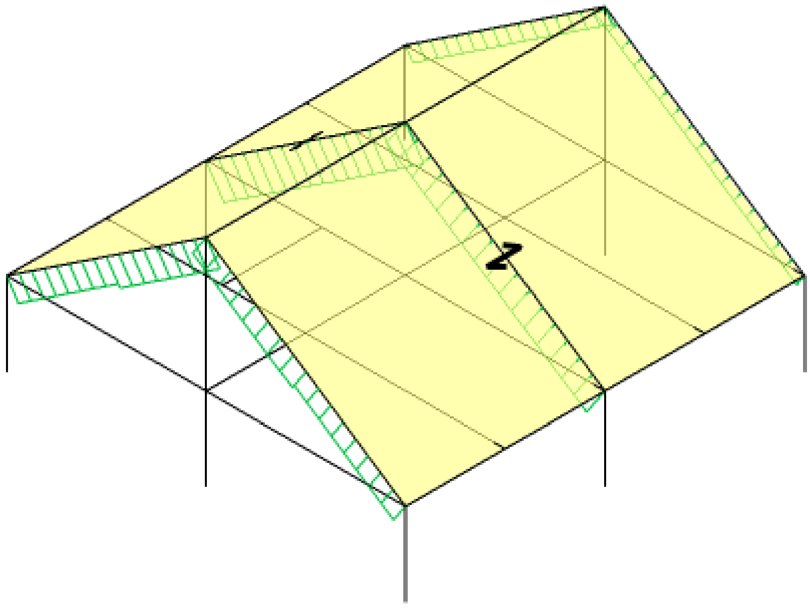
Roof Panel Load Decomposition

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.

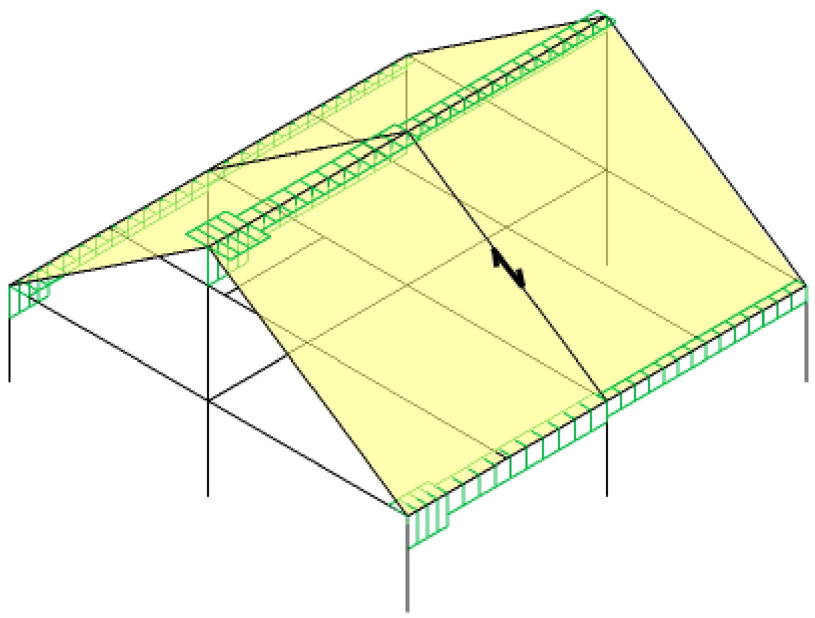
All types of elements within the wind wall plane (except bracing and cold rolled members) are considered during the load decomposition.



Zone Loads on Roof



Roof Panel Decomposition (rotation angle 0 degrees)



Roof Panel Decomposition (rotation angle 90 degrees)

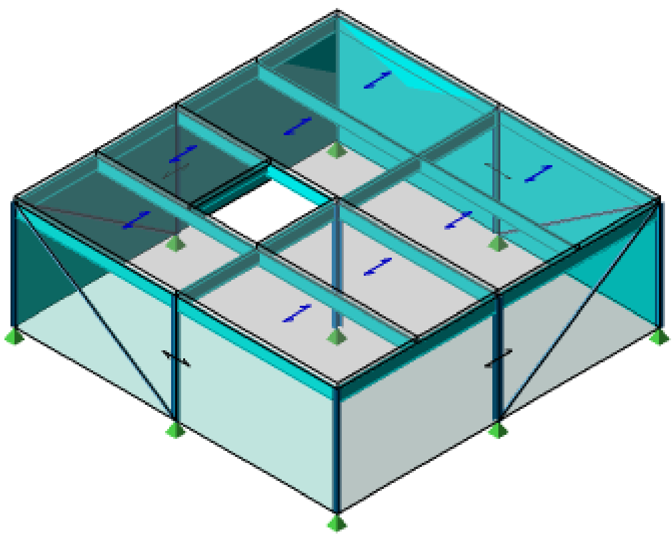
Wind Wall Panel Load Decomposition

Decomposition Options

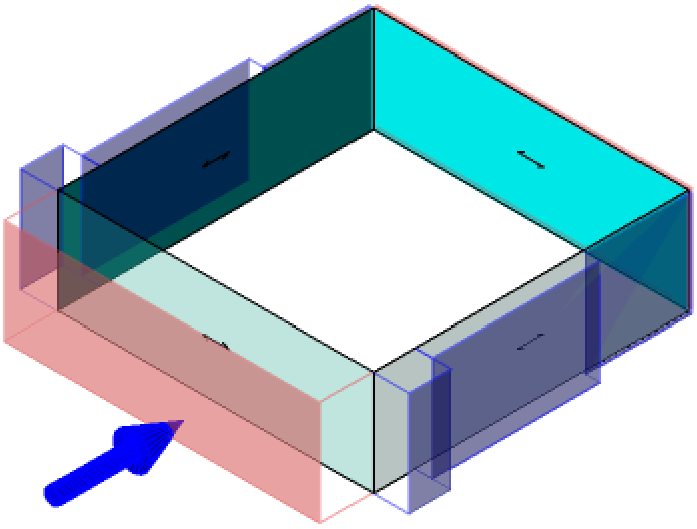
Wind zone loads are decomposed on to the structure according to the setting of the 'Decompose to' wind wall property, which can be set to:

- Members,
- Nodes,
- Rigid Diaphragms

To demonstrate the effect of the different Decompose to' settings, consider the braced steel frame clad in wind walls shown below:



For wind direction 0 the wind model produces Zone Loads as follows:



The related topics below illustrate how the above zone loads are decomposed for the three different options.

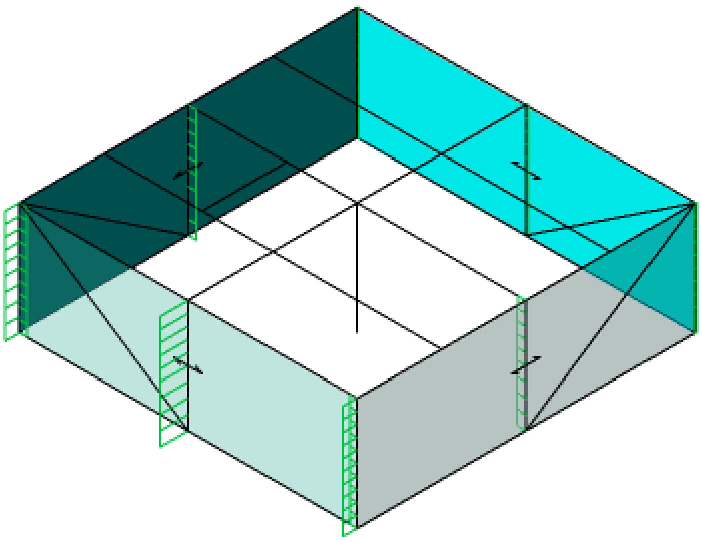
Decompose to Members

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

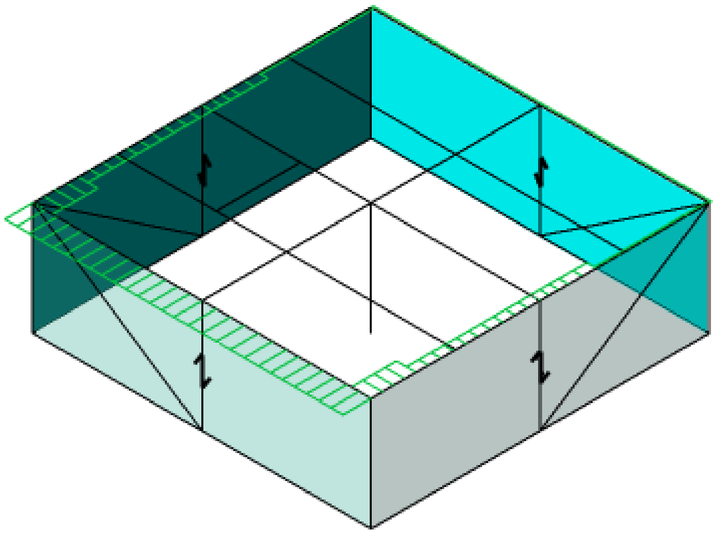
All types of elements within the wind wall plane are considered except bracing and cold rolled members.

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

So for the model shown in [Decomposition Options](), choosing 'decompose to members' would produce the following loads:



Decomposition to Members (rotation angle 0 degrees)



Decomposition to Members (rotation angle 90 degrees)

Decompose to Nodes

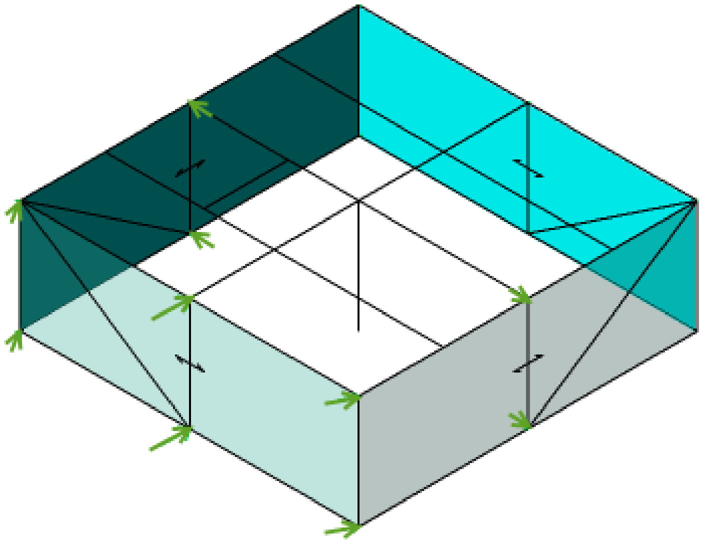
Decomposition to nodes is the default setting and results in nodal loads on the supporting members. This setting is generally appropriate to avoid lateral loads on simple beams.

All types of elements within the wind wall plane are considered except bracing and cold rolled members.

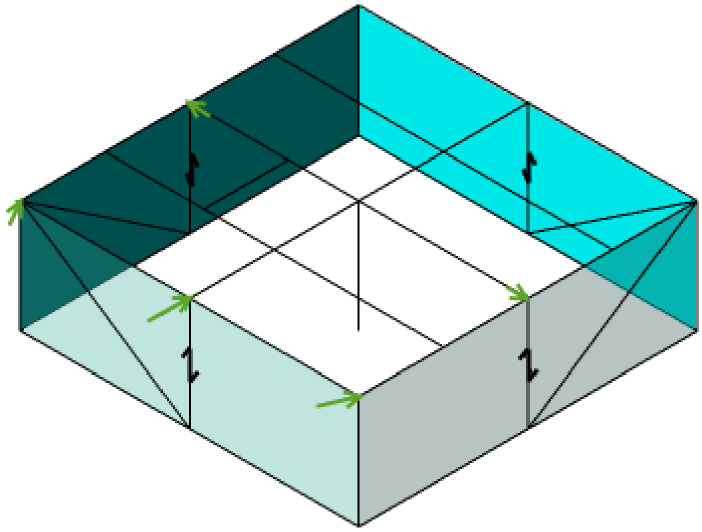
The initial decomposition is the same as for members, with the direction of the one way decomposition being specified by the span direction of the panel. A second decomposition stage is then undertaken to convert the member loads to nodal loads:

- 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 in Decomposition Options, choosing 'decompose to nodes' would produce the following loads:



Decomposition to Nodes (rotation angle 0 degrees)



Decomposition to Nodes (rotation angle 90 degrees)

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

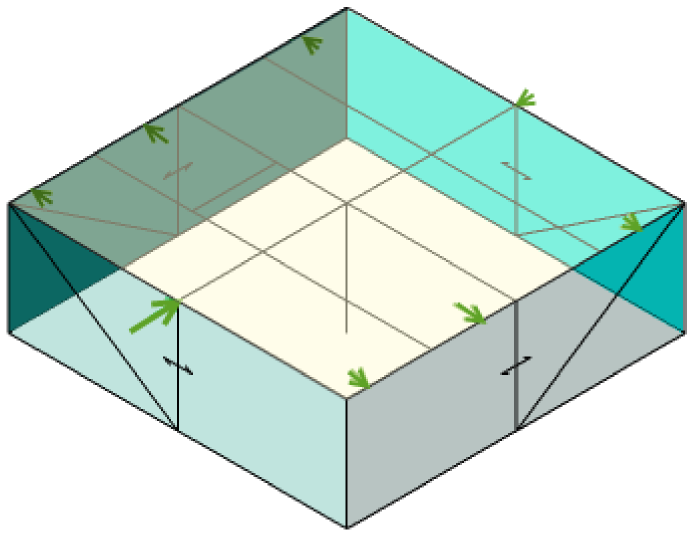
Decompose to Rigid Diaphragms

Decomposition to rigid diaphragms does not need to 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 'member' or 'node' decomposition methods require supporting members that may not exist in the model.

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 in Decomposition Options, choosing 'decompose to rigid diaphragms' would produce the following loads:



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 for wind wall panels set to decompose to rigid diaphragms:

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

Alternative decomposition methods for complex models

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

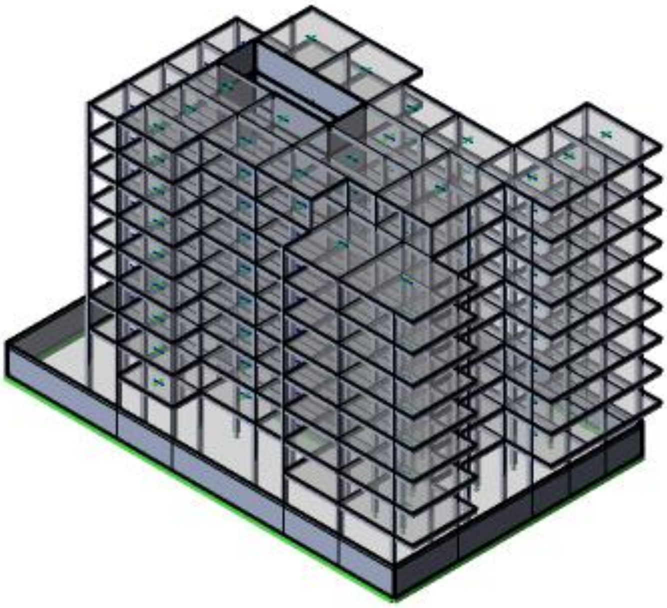
In such situations (provided the model has suitable rigid diaphragms) the following alternatives offer quicker and simpler ways to apply approximate wind loads:

- use engineering judgement to clothe the structure with an arrangement of Simplified Wind Wall Panelsaround its bounding box, set the wind walls to decompose to rigid diaphragms, and proceed with the Wind Model method

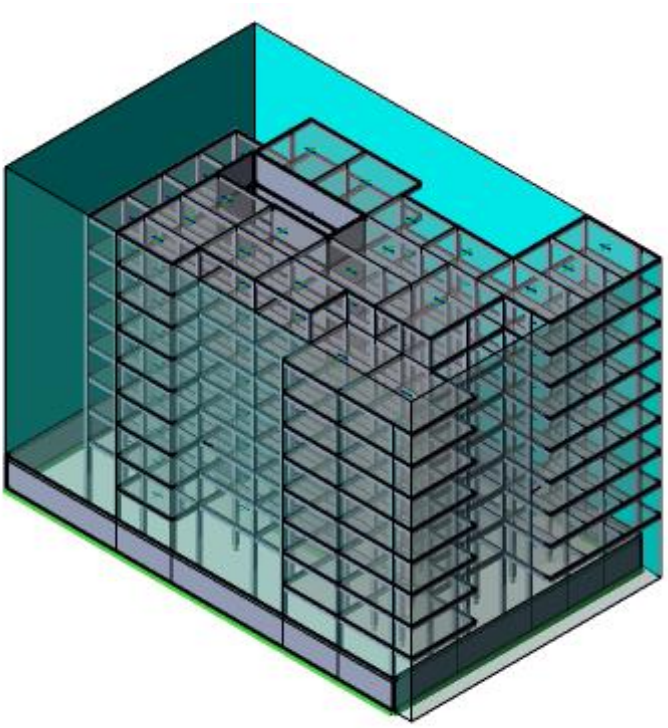
- don’t apply wind walls - consider Application of manual wind loads instead

Simplified Wind Wall Panels

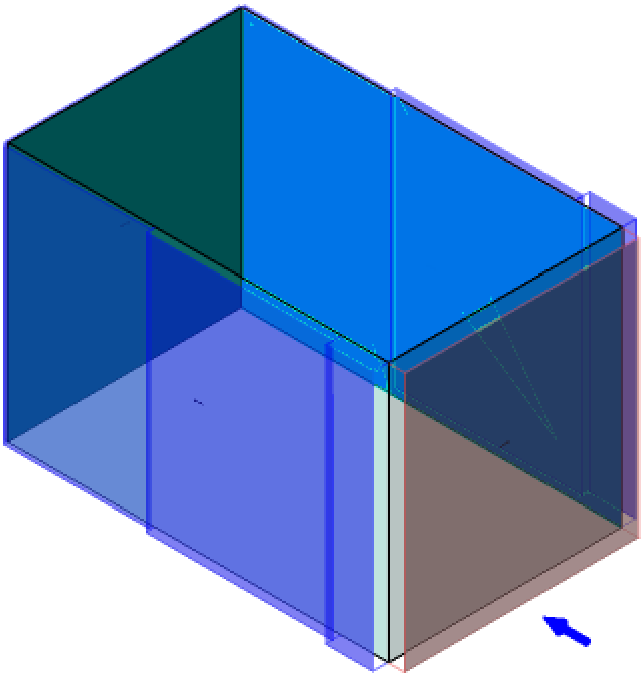
In theory, because wind walls set to decompose to rigid diaphragms don’t have to be physically connect to the diaphragms, even a very complex model could be clothed with just four simplified wind walls defined along its bounding box, as illustrated below:



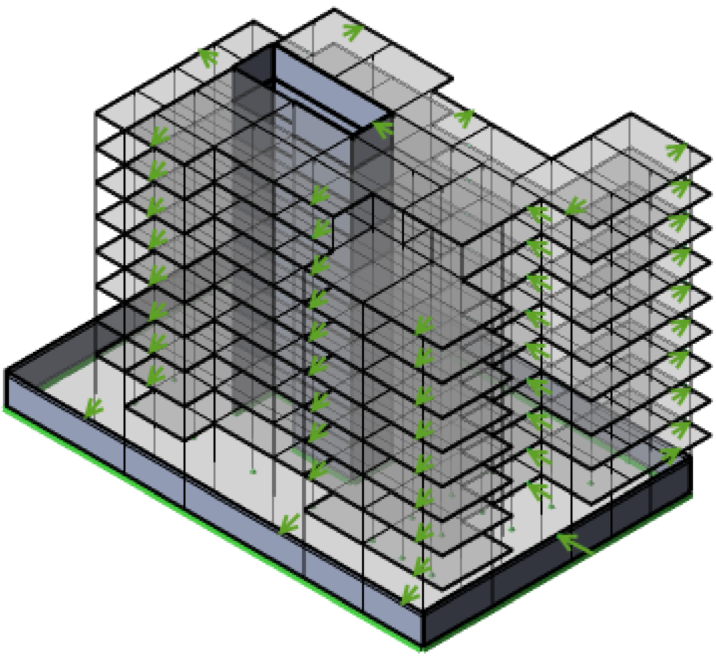
Complex Model



Simplified Wind Walls around the ‘bounding box’



Zone Loads for the Simplified Wind Wall Model



Zone Loads Decomposed to Rigid Diaphragms

Such an approach could be used to rapidly establish approximate wind loads (based on the rectangular building envelope) which could then be refined at a later stage if necessary.

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 and consider Application of manual wind loads instead.

Application of manual wind loads

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

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

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

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

Simple Wind

Overview of Simple Wind

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

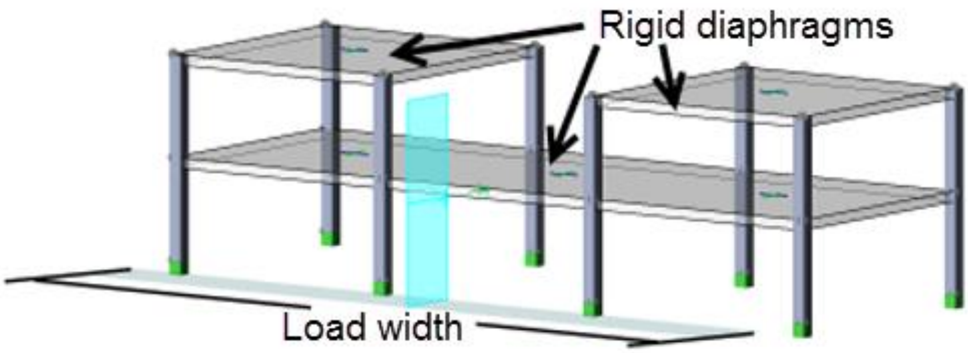
1. Define the structure ensuring that slabs have been created with the diaphragm option set to rigid as opposed to semi-rigid.
2. In order access Simple Wind, you must first create and then select the loadcase into which the Simple Wind loads are to be added.
3. Click **Simple Wind** on the Load ribbon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

Simple Wind application and decomposition

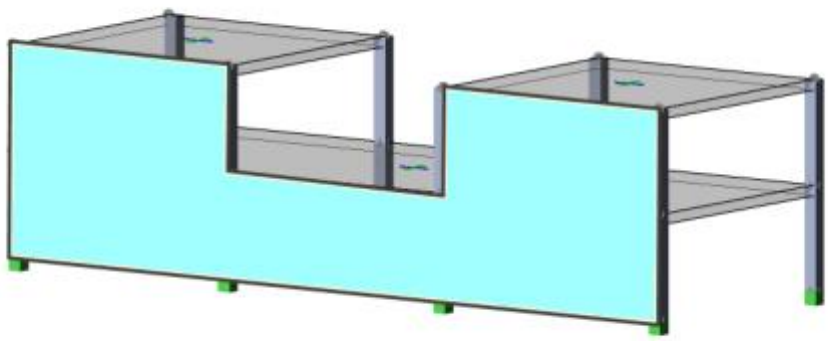
Application

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

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



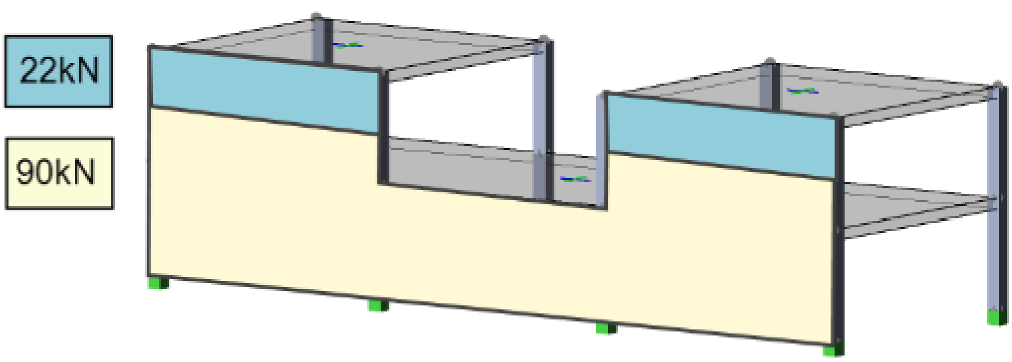
- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation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



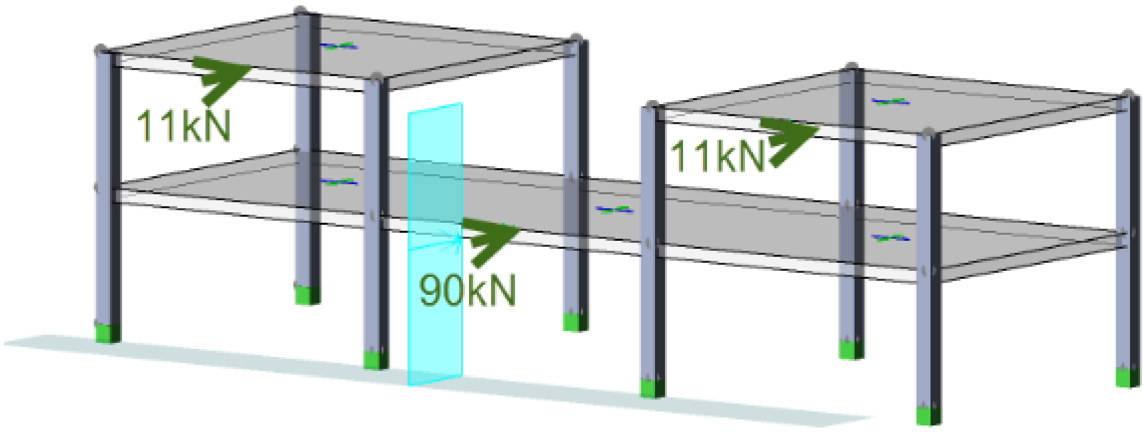
Decomposition

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

- All rigid diaphragms on the top or bottom level or anywhere in-between are considered, with the area load being divided between the levels before it is decomposed.



- The loads are then decomposed to the diaphragms at each level in proportion to the width of each diaphragm. Each load being applied as a nodal load in the direction of the Simple Wind load at the mid point of the projected load.



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

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

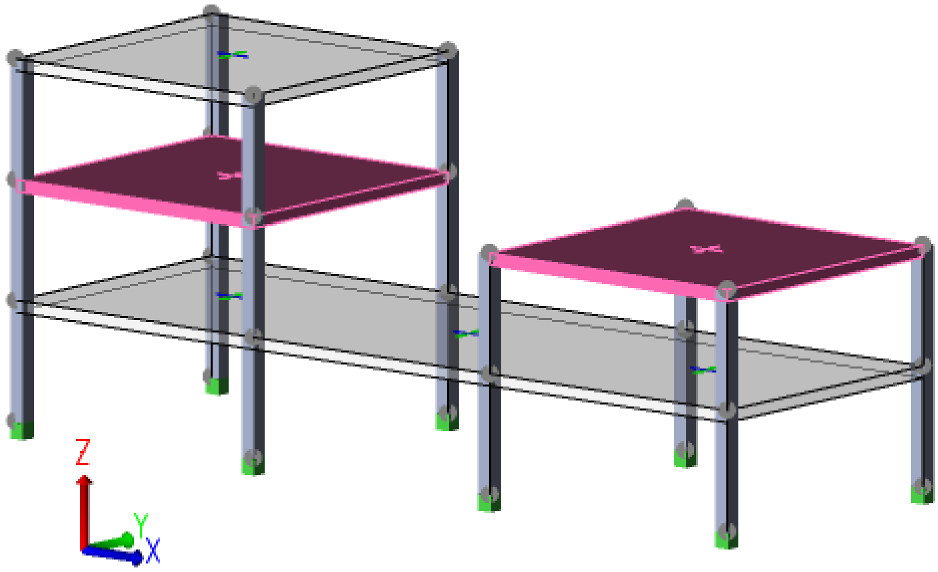
Limitations of wind decomposition to diaphragms

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

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

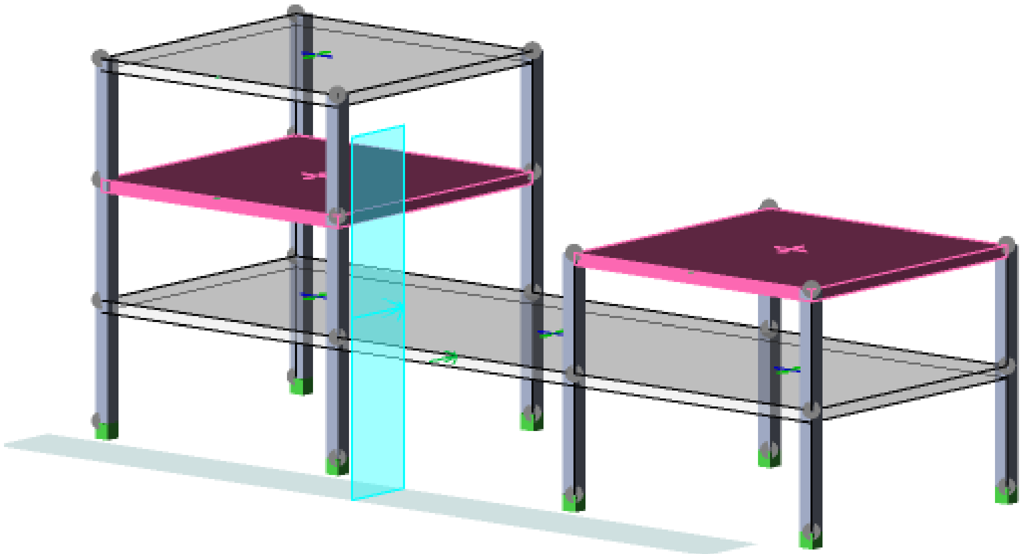
Wind load perpendicular to disconnected diaphragms

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

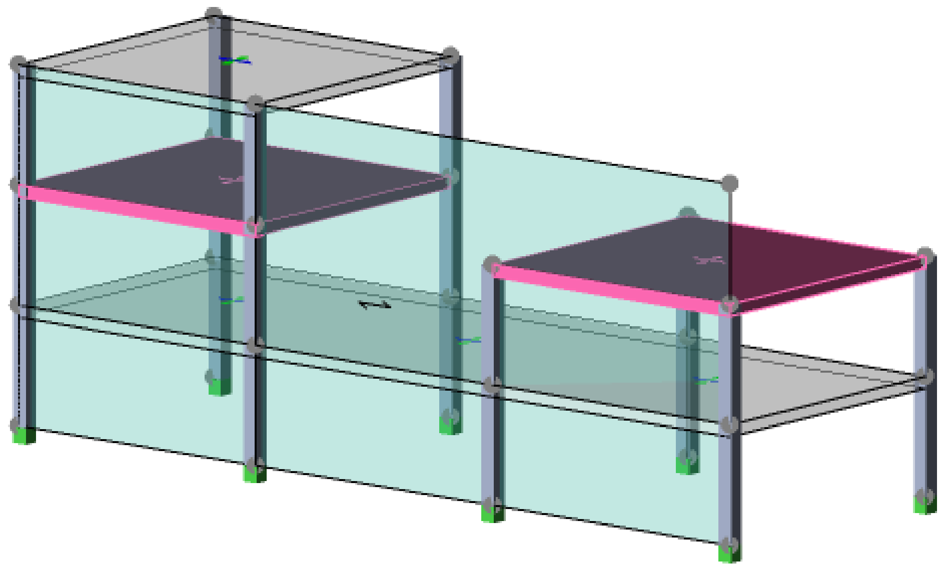


Disconnected diaphragms at second floor level

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

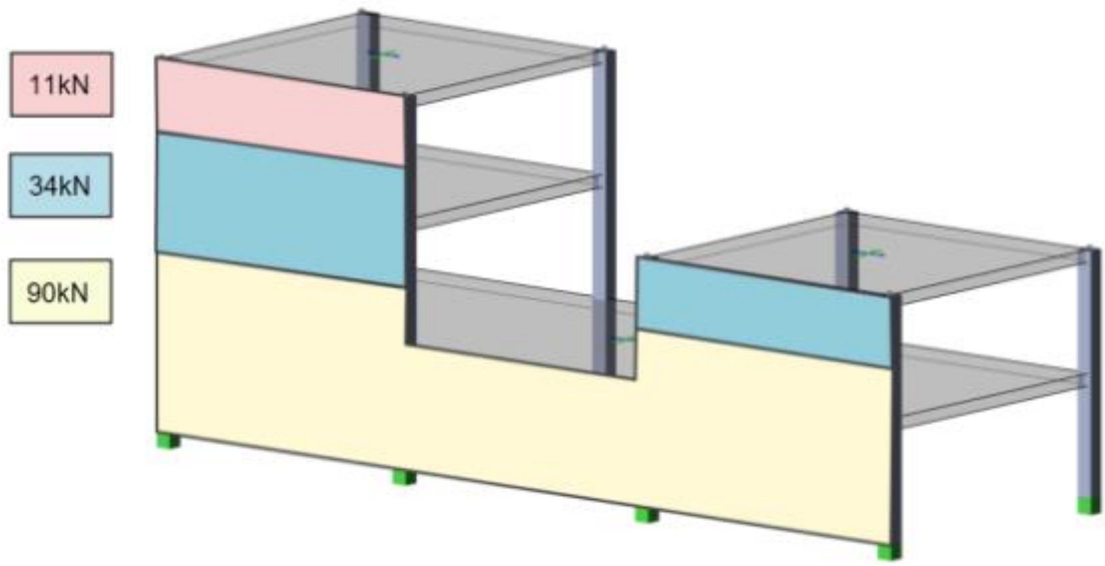


Load applied via a Simple Wind Load



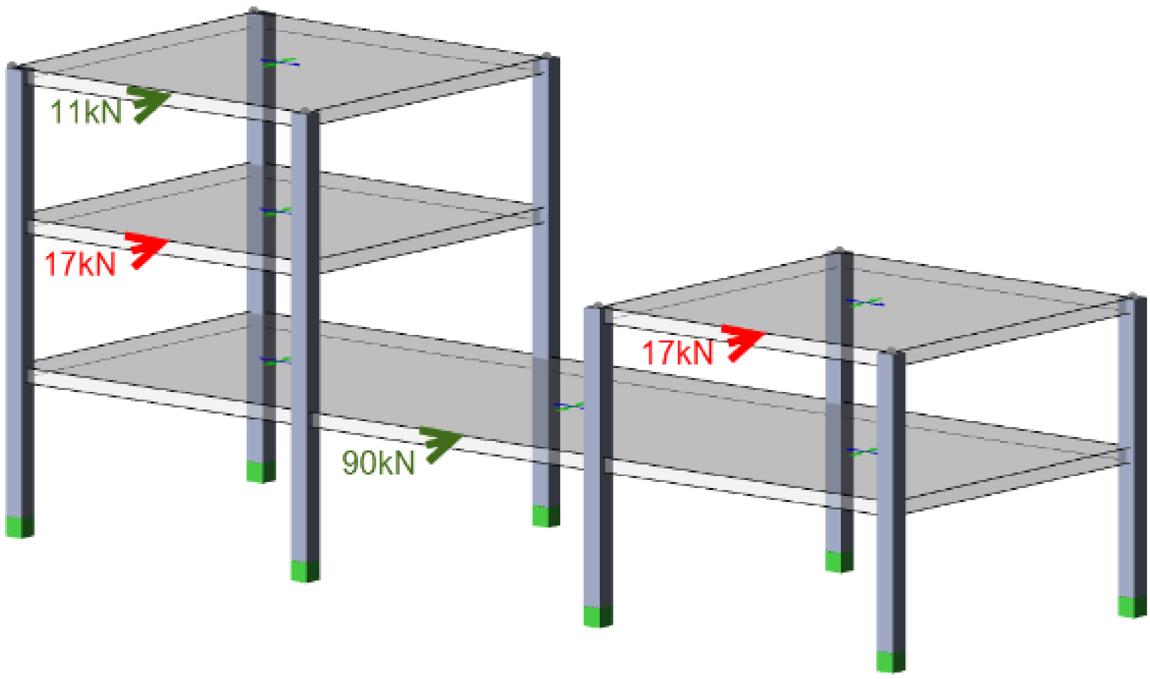
Load applied via a Wind Wall Panel

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

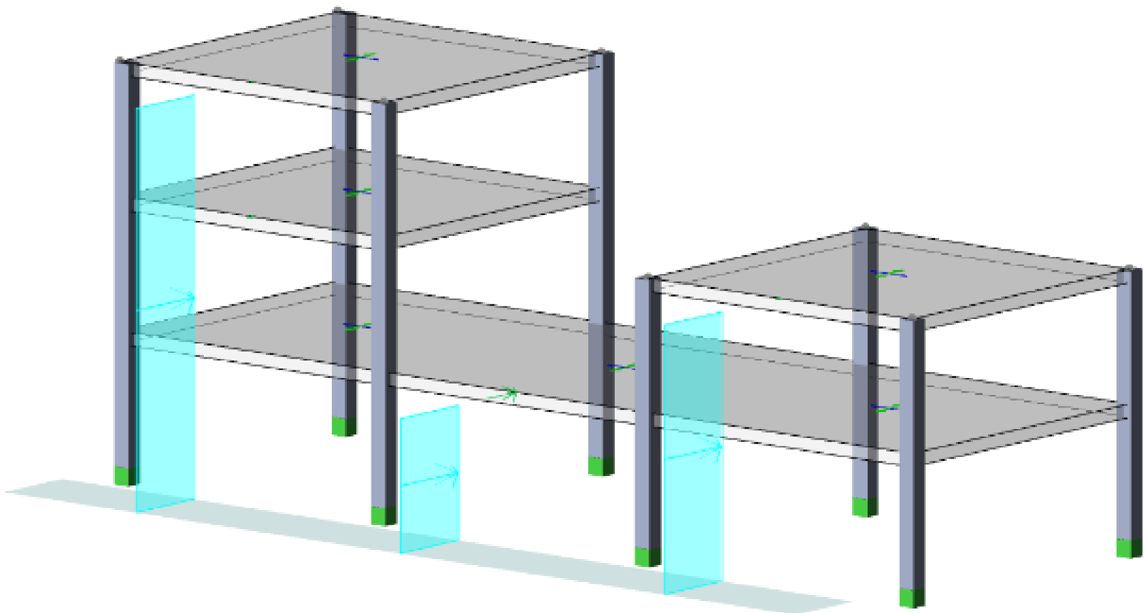


The load is then decomposed to the diaphragms at each level **in proportion to the width of each diaphragm**.

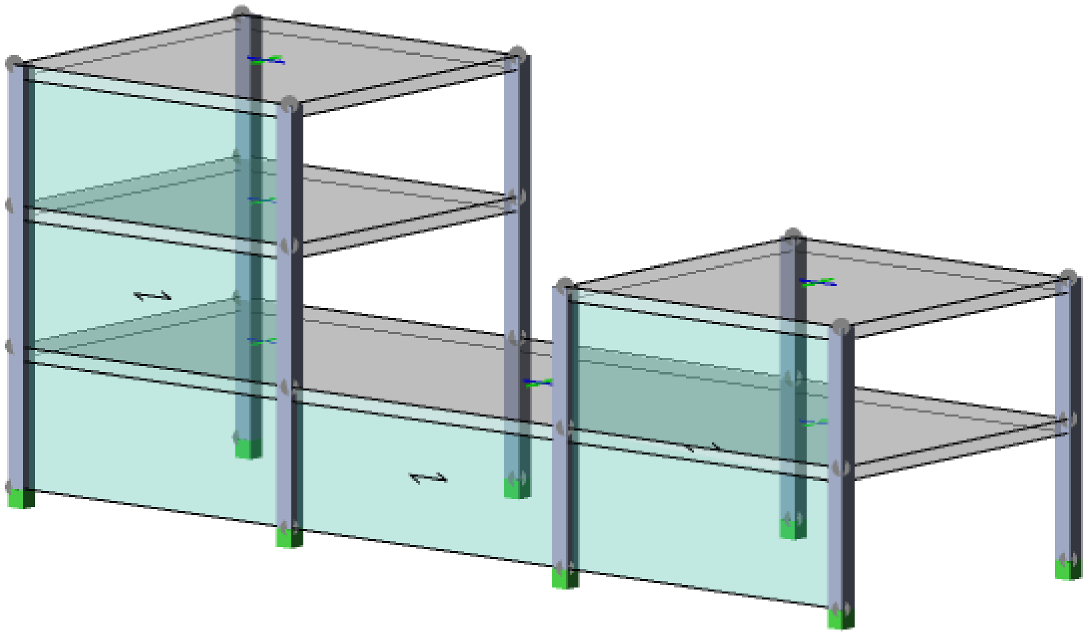
In this case because both diaphragms the second floor level are of equal width, the load is shared equally between them. This is not satisfactory as more of the load should have been applied to the left hand diaphragm in this case.



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

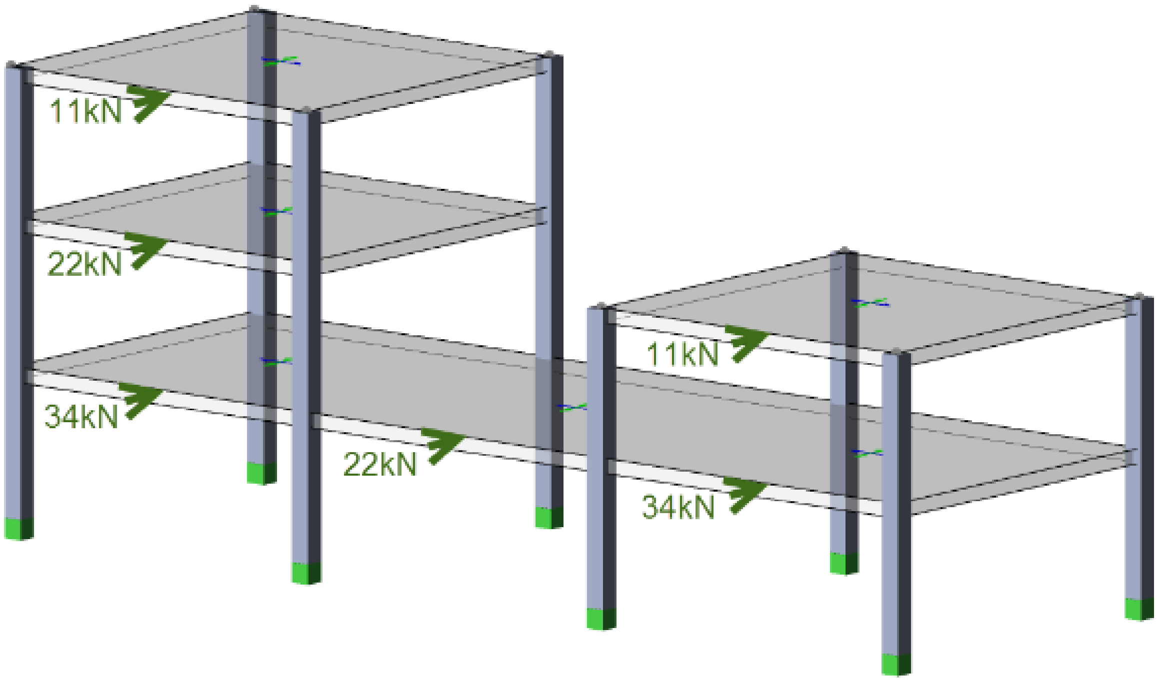


Simple Wind Workaround - Load re-applied as 3 Simple Wind Loads



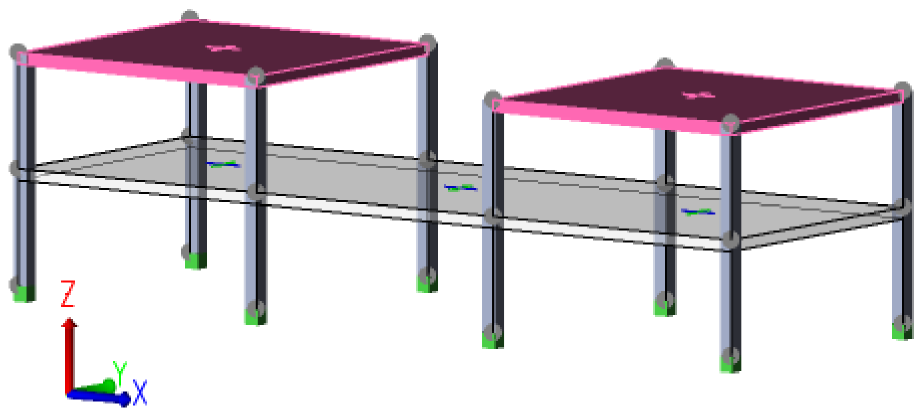
Wind Wall Workaround - Load re-applied via 3 Wind Wall Panels

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



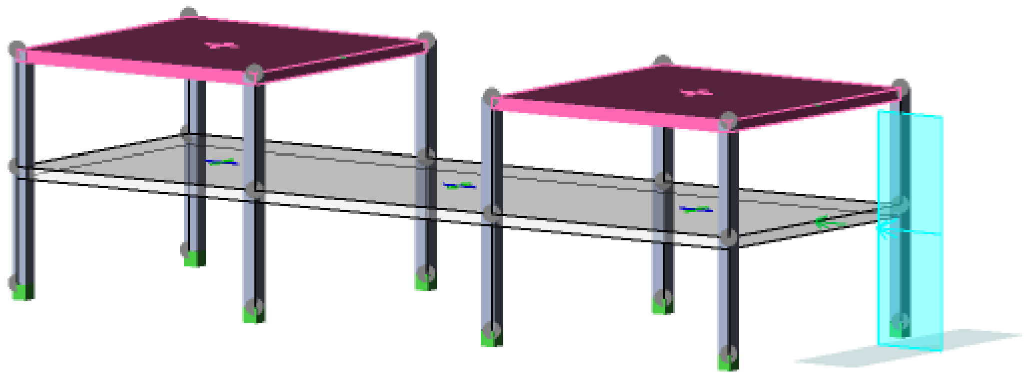
Wind load parallel to disconnected diaphragms

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

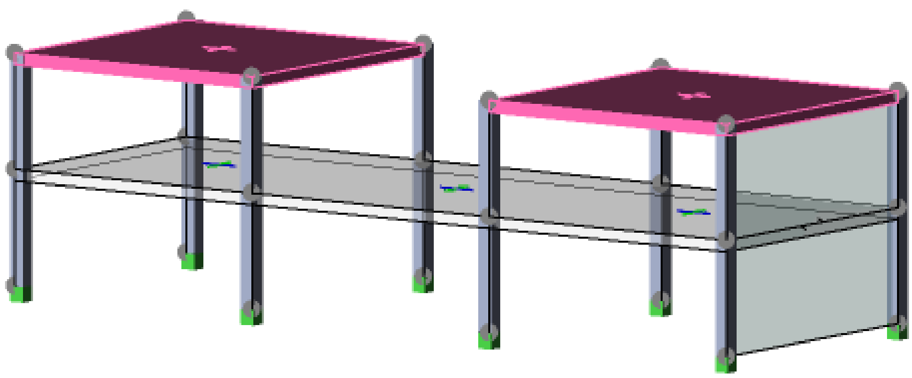


Disconnected diaphragms at second floor level

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

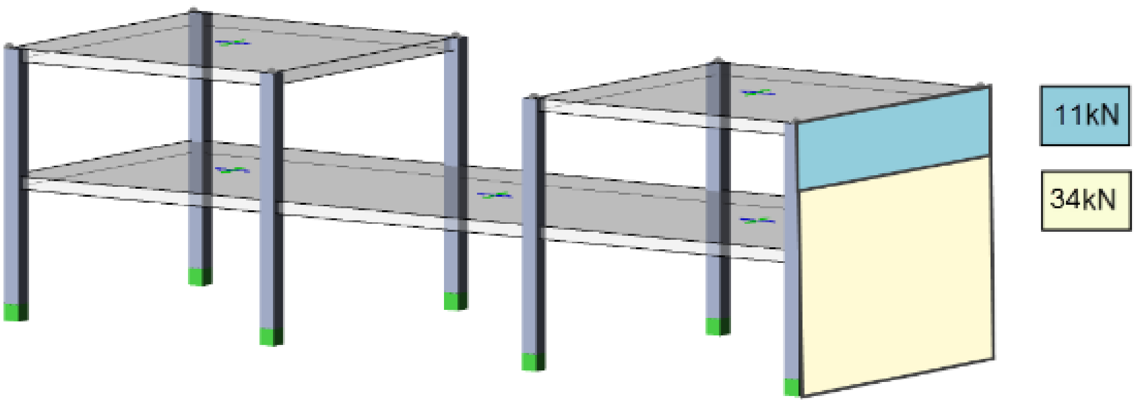


Load applied via a Simple Wind Load

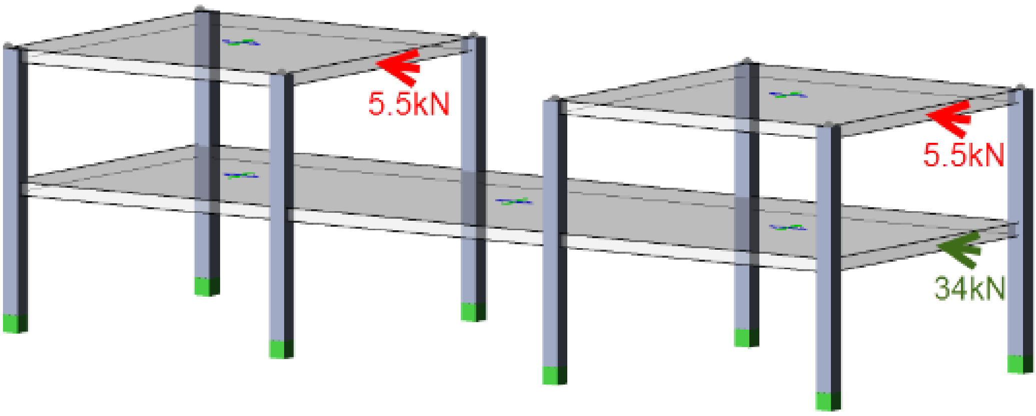


Load applied via a Wind Wall Panel

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



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



To avoid the load being shared equally:

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

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**.

Snow Loading Handbook

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

- The **Snow Wizard** can be run to input basic snow data and set up the required snow loadcases. Provided that roof panels have been modelled some of these loadcases will be populated with Uniform Snow automatically; the remaining loadcases then have Uniform Snow, Valley Snow and Local Drift Snow applied manually as required.

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

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

Overview of Snow Loading

The intensity of snow load is based upon geographic location, building/roof geometry, environmental factors and local roof factors.

All snow loading falls into three categories

- Uniform Snow load (the first fall of snow)
- Drifted uniform snow (the first fall of snow blown into uneven uniform loading)
- Drift loading (local build-up of snow load behind steps, objects, parapets)

The Snow Wizard

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

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

From this information all the required snow loadcases are automatically set up and the loads in the undrifted (or balanced) snow load case are created.

Following the Wizard, you then manually define the drift cases. To do this select the relevant snow load case, then define the key attributes for the drift load prior to placing the load in the relevant position on the roof of the building.

The end result is a series of snow loadcases ready to be combined in the Combination Generator with other load cases.

The basic steps required for this method can be summarised as follows:

1. Apply roof panels to structure
2. Run the Snow Wizard
3. Apply snow loads to the drift loadcases set up by the wizard
4. Combine snow loadcases into design combinations
5. Perform the static design

Roof Panel Types

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

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

ASCE7 Snow Wizard

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

EC1991 1-4 Snow Wizard

EC1991 1-4 Snow Wizard

To access this configuration of the Snow Wizard the Snow Loading Code has to be set to EC1991 1-4.

Once the roof panels are in place, you use the *Snow Wizard* on the **Load** toolbar to define sufficient site information to calculate the snow loadcases.

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

Page 1 - Basic Data (Plain Eurocode, Ireland and Sweden NA)

The following basic data is required.

Exposure

> **Exposure Coefficient**
>
> (default 1.0)
>
> **Thermal Coefficient**
>
> (default 1.0)
>
> **Coefficient for Exceptional Snow Loads**
>
> (default 1.0)

Snow Density

> **Snow weight density**
>
> (default 2.0kN/m2)

Snow Load

> **Characteristic Ground Snow Load**
>
> sk (kN/m2)

Next

Clicking **Next** takes you to Page 2 - Snow Load Cases (Plain Eurocode, UK, Ireland and Norway NA) or Page 2 - Snow Load Cases (Sweden NA)

Page 1 - Basic Data (UK NA)

The following basic data is required.

Exposure

Exposure Coefficient

(default 1.0)

Thermal Coefficient

(default 1.0)

Coefficient for Exceptional Snow Loads

(default 1.0)

Snow Density

Snow weight density

(default 2.0kN/m2)

Snow Load

Zone Number

(default 2.0kN/m2)

Altitude

(default 1.0)

Characteristic Ground Snow Load

sk = (0.15 + (0.1 * Z + 0.05)) + ((A - 100)/525)

Next

Clicking **Next** takes you to Page 2 - Snow Load Cases (Plain Eurocode, UK, Ireland and Norway NA).

Page 1 - Basic Data (Finland NA)

The following basic data is required.

Exposure

Topography

- Windswept,
- Normal (default),
- Sheltered

Exposure Coefficient
(default 1.0)

Thermal Coefficient
(default 1.0)

Coefficient for Exceptional Snow Loads
(default 1.0)

Snow Density

Snow weight density
(default 2.0kN/m2)

Snow Load

Characteristic Ground Snow Load
sk (kN/m2)

Next

Clicking **Next** takes you to Page 2 - Snow Load Cases (Finland NA).

Page 1 - Basic Data (Norway NA)

The following basic data is required.

Exposure

Topography

- Windswept,
- Normal (default),
- Sheltered

Exposure Coefficient
(default 1.0)

Thermal Coefficient
(default 1.0)

Snow Density

Snow weight density
(default 2.0kN/m2)

Snow Load

Altitude - H

H

Basic Reference Altitude - Hg

H

Basic Snow Load - sk0

H

Δsk

H

Δsk,maks

H

Characteristic Ground Snow Load

sk (kN/m2)

Next

Clicking **Next** takes you to [Page 2 - Snow Load Cases (Plain Eurocode, UK, Ireland and Norway NA)].

Page 2 - Snow Load Cases (Plain Eurocode, UK, Ireland and Norway NA)

On this page you specify the design situations to be considered by checking the box against each loadcase to be applied.

Loadcases to Apply

Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3)

- Loads in these cases are generated automatically

Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4)

- You must create the loads in these cases manually

The EC/NA recommends which loadcases to apply.

Loadcase Types

Each Loadcase also has a Loadcase type which can be set as:

- Snow
- Snow > 1000m
- Snow Drift

Refer to EC 1991-1-3 Annex A, Table A1.

Number of wind directions to be considered for drifted snow

You can select between 1 and 4 separate loadcases to be generated for each ‘Drifted’ loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name).

For example if you have checked the box to apply loadcase “Snow Load - Case A - 2) Drifted” and selected 2 wind directions, the following loadcases will be created:

- Snow Load - Case A - 2) Drifted 1
- Snow Load - Case A - 2) Drifted 2

Finishing the Snow Wizard

When you click **Finish**, the *Snow Wizard* sets up the loadcases accordingly.

Page 2 - Snow Load Cases (Finland NA)

On this page you specify the design situations to be considered by checking the box against each loadcase to be applied.

Loadcases to Apply

Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3)

- Loads in these cases are generated automatically

Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4)

- You must create the loads in these cases manually

The EC/NA recommends which loadcases to apply.

Loadcase Types

Each Loadcase also has a Loadcase type which can be set as:

- Snow
- Snow Sk > 2.75 kN/m2
- Snow Drift
- Ice

Refer to EC 1991-1-3 Annex A, Table A1.

Number of wind directions to be considered for drifted snow

You can select between 1 and 4 separate loadcases to be generated for each ‘Drifted’ loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name).

For example if you have checked the box to apply loadcase “Snow Load - Case A - 2) Drifted” and selected 2 wind directions, the following loadcases will be created:

- Snow Load - Case A - 2) Drifted 1
- Snow Load - Case A - 2) Drifted 2

Finishing the Snow Wizard

When you click **Finish**, the *Snow Wizard* sets up the loadcases accordingly.

Page 2 - Snow Load Cases (Sweden NA)

On this page you specify the design situations to be considered by checking the box against each loadcase to be applied.

Loadcases to Apply

Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3)

- Loads in these cases are generated automatically

Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4)

- You must create the loads in these cases manually

The EC/NA recommends which loadcases to apply.

Loadcase Types

Each Loadcase also has a Loadcase type which can be set as:

- Snow
- Snow Sk > 2 kN/m2
- Snow Sk > 3 kN/m2
- Snow Drift

Refer to EC 1991-1-3 Annex A, Table A1.

Number of wind directions to be considered for drifted snow

You can select between 1 and 4 separate loadcases to be generated for each ‘Drifted’ loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name).

For example if you have checked the box to apply loadcase “Snow Load - Case A - 2) Drifted” and selected 2 wind directions, the following loadcases will be created:

- Snow Load - Case A - 2) Drifted 1
- Snow Load - Case A - 2) Drifted 2

Finishing the Snow Wizard

When you click **Finish**, the *Snow Wizard* sets up the loadcases accordingly.

Snow Loadcases

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

All loads and lengths of loads are assumed to act on a horizontal projection of the roof surface.

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

ASCE7 Snow Loadcases

EC1991 1-4 Snow Loadcases

In the Snow Wizard you choose the Snow Loadcases to be set up from the following list:

- Snow Load - Case A - 1) Undrifted
- Snow Load - Case A - 2) Drifted*****
- Snow Load - Case B1 - 1) Undrifted Snow Load - Case B1 - 2) Drifted*****
- Snow Load - Case B1 - 3) Undrifted (Acc)
- Snow Load - Case B1 - 4) Drifted (Acc)*****
- Snow Load - Case B2 - 1) Undrifted
- Snow Load - Case B2 - 2) Drifted*****
- If any drifts from Annex B are selected
    - Snow Load - Case B2 - 3) Drifted (Annex B) (Acc)*****
- Snow Load - Case B3 - 1) Undrifted
- Snow Load - Case B3 - 2) Drifted*****

- Snow Load - Case B3 - 3) Undrifted (Acc)
- If any drifts from Annex B are selected
  - Snow Load - Case B3 - 4) Drifted (Annex B) (Acc)*

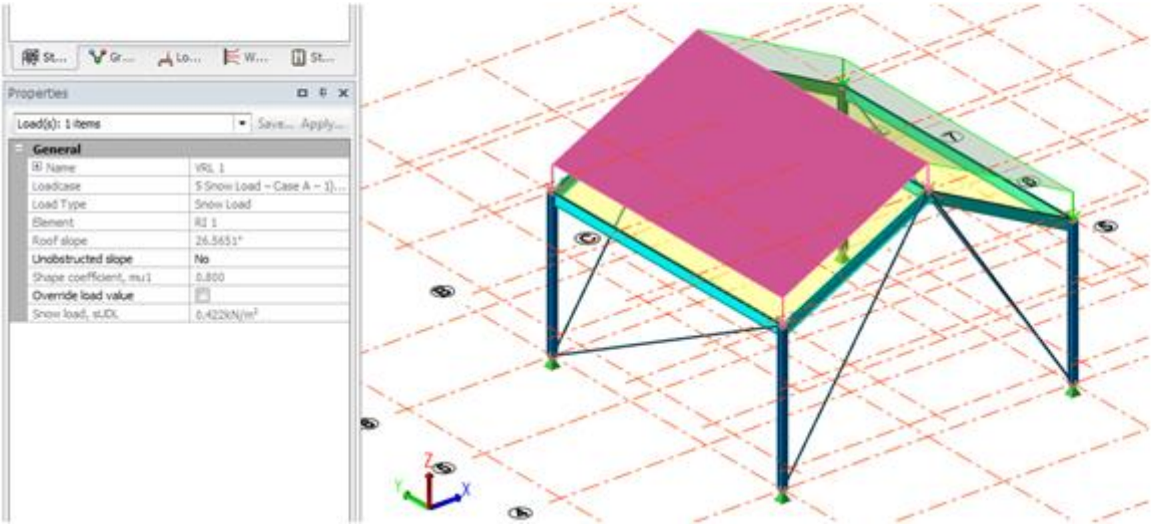
For cases marked ***** *- the number of cases actually set up will depend on the number of wind directions that are asked for.*

The Eurocode / National Annexe recommends which loadcases to generate.

Undrifted loadcases

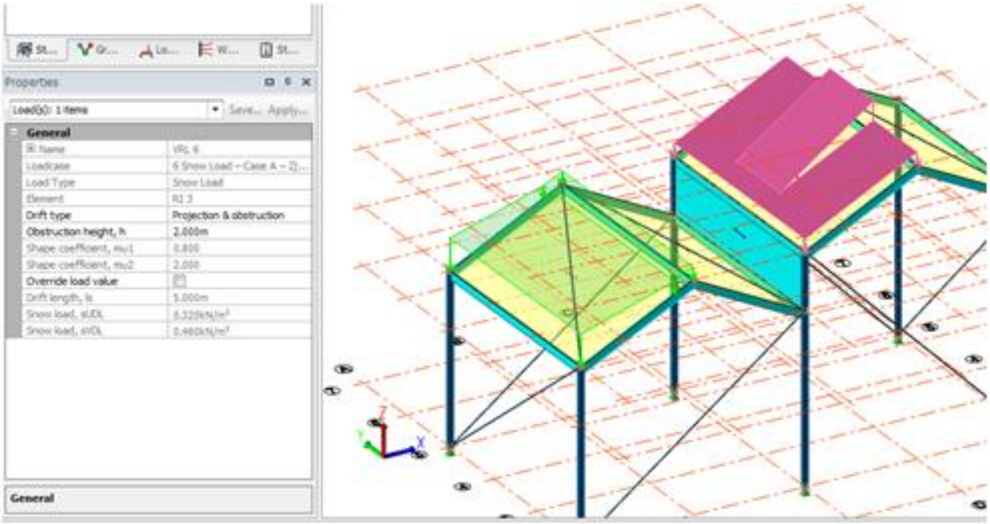
Any undrifted loadcases that have been set up in the Snow Wizard are automatically populated with uniform loading on completion of the wizard

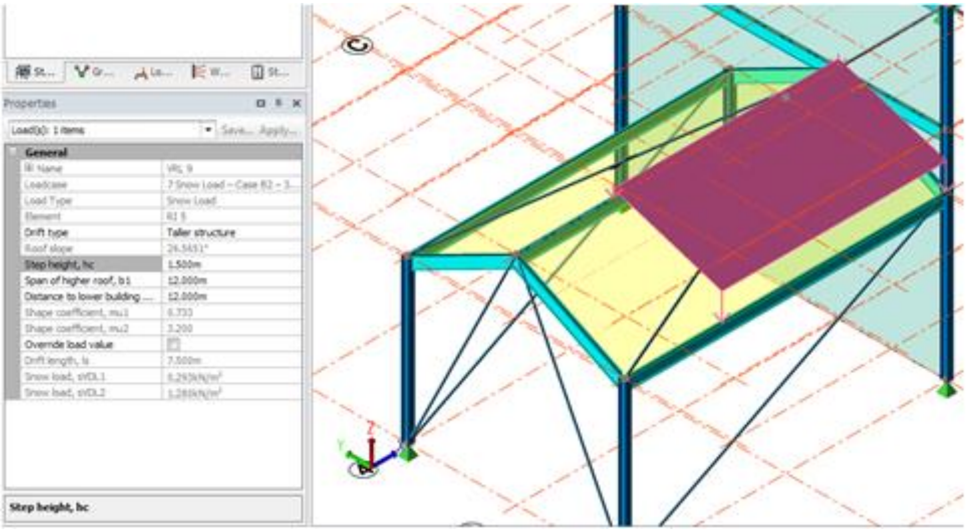
Undrifted loadcase A- 1)

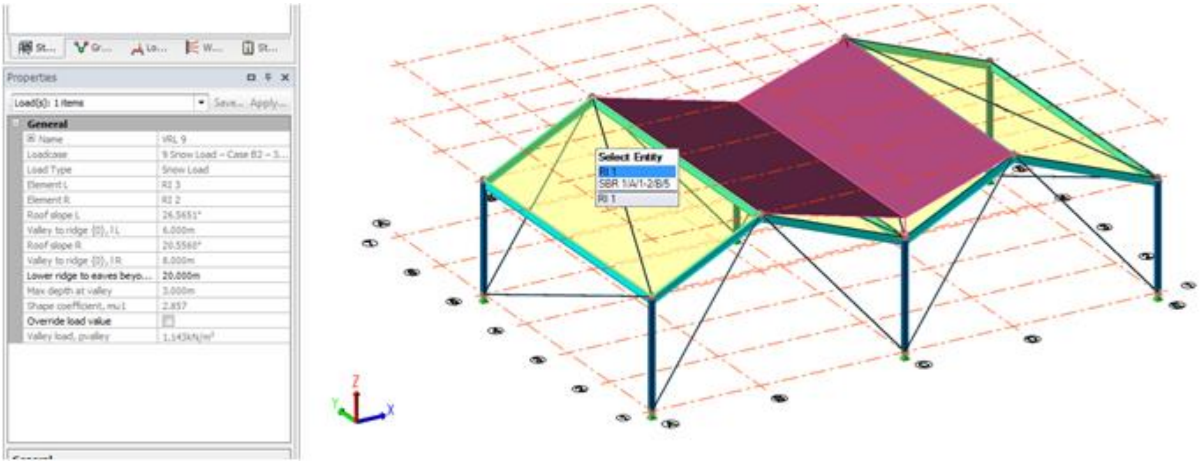


Drifted Loadcases

All drifted loadcases that have been set up in the Snow Wizard would need to be have their snow loads manually applied on completion of the wizard.







Related topics

- How do I manually apply a Uniform Snow load?
- How do I manually apply a Valley Snow load?
- How do I manually apply a Local Drift Snow load?

Snow Load Decomposition

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

Manual application of snow loading

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

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

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

References

1. **ASCE/SEI 7-10.** *Minimum Design Loads for Buildings and Other Structures*. **ASCE, 2010. ISBN: 978-0-7844-1085-1.**

2. **British Standards Institution (September 2008)**. *UK National Annex to Eurocode 1: Actions on structures. NA to BS EN 1991-1-4:2005.*

3. **British Standards Institution**. *Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009.*

Stability Requirements Handbook

Introduction to stability requirements

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

1. Global second-order (P-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 effects

Choice of analysis type (Eurocode)

First or second order analysis?

You have the choice of three analysis types on the Analysis page of the Design Options dialog. 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: When should a building be classed as sway sensitive?

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). Full second-order analysis is preferred for steelwork and since it is not precluded by EC2 it can be used for concrete.

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

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

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

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

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.

- Calculation of the elastic critical load factor

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?)

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 that should be used 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 full second-order, or the amplified forces method, perform **Design All (Static)**

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.

If you use the 'Second-order analysis - Amp. forces method' be aware that EC3 classes certain structures outside of its scope (see Validity of the amplified forces method). 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.

First or second order analysis?

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

[Amplified forces (kamp) method]

Sway sensitivity assessment (Eurocode)

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:).

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).

A suggested approach for assessing sway sensitivity and considering global second order effects is given in the topic: [A practical approach to setting the analysis type]

Calculation of the elastic critical load factor

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

Derivation of the kamp 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,

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

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*.

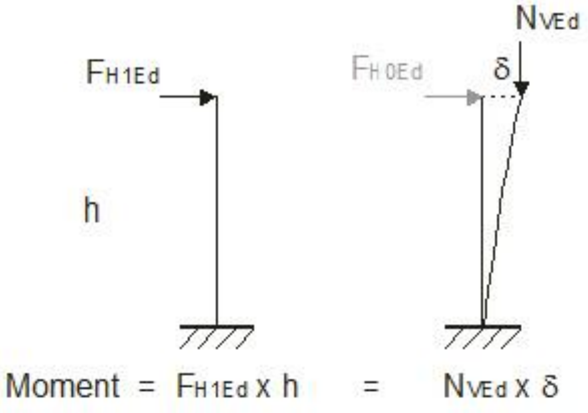
Instead, recourse is made to determining the level of second-order effect using Annex H.2. Using this approach, by rearranging **Equation H.8** it is possible to provide a 'stability coefficient' $1/\alpha_{cr}$ which can be applied as the change point between non-sway and sway sensitive structures.

$F_{HEd} \quad = \quad F_{H0Ed} /(1-F_{H1Ed}/F_{H0Ed})$ 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:

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

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

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.

What are the twist results?

A ‘measure’ of twist is also tabulated for each column - this indicates the degree to which if you push the column one way, how much it moves orthogonally as well. If you have a building where the 'lateral load resisting system' is not well dispersed then pushing one way can cause significant movement in the other direction.

The twist is reported as a ratio of: distance moved in the direction of loading/absolute distance moved.

EG

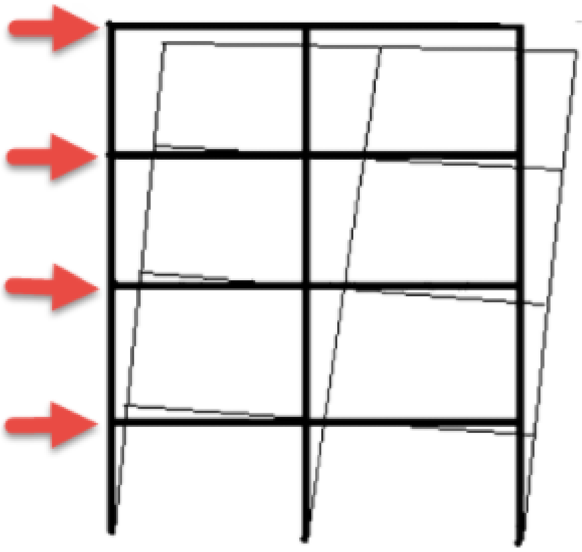
When a column node moves in X and Y then the 'total' deflection is SQRT(delta_x^2 + delta_y^2) in other words the diagonal of the triangle and not either of the sides. So if a node moves say 10mm in X and 2mm in Y, its diagonal i.e. absolute deflection in this plane is SQRT(100 + 4) = 10.198. Hence its twist is what it should have been with just X loading i.e. 10mm divided into what it actually moved i.e. 10.198. So Twist = 1.0198.

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 that will induce the same effect. Basically this means applying horizontal forces = 0.2 to 0.5% of the vertical forces in any combination.



In *Tekla Structural Designer* we use the second option.

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

Member imperfections

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

Wind drift

The Wind Drift check is performed during the structure static design and also when any 3D analysis is run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are considered in the drift checks, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

The check is performed for (wind) combinations using the combination SLS (Service Level) factors - (which can be < 1.0).

By default the check is applied to all columns (of all materials), and all walls.

Wind drift check design options

In [Design Options-Sway & Drift Checks](#) there is a **Check wind cases only** option (default On) that applies to the check as follows:

- With the setting On, the Wind Drift check only considers the effects of the wind load case(s) in wind combinations.
- With the setting Off, the check considers the effects of all load cases in wind combinations (which would include drift induced by gravity loads).

On the same page of the Design Options there is also a **Merge short stacks** option (default Off) that applies to the check as follows:

- With the setting Off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting On, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged stack length. The check is not performed on single stack columns/walls that are less than this limit.

Manually excluding an entire column or wall or an individual column stack or wall panel from the check

An entire column/wall can be excluded from the check by unselecting **Wind Drift check** located under **All stacks/panels>Sway and Drift Checks** in the column/wall properties window.

Similarly an individual stack/panel can be excluded from the check by unselecting **Wind Drift check** located under **Sway and Drift Checks** for the specific stack/panel in the column/wall properties window.

Manually adjusting the automatically determined stack lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

In order to do this you would have to select **Merge with stack below** located under **Sway and Drift Checks** for the specific stack/panel in the column/wall properties window.

Wind drift calculations

For those stacks to which the check has been applied, the lateral drift in each direction (i.e. the difference between top and bottom deflection of the stack) is determined for each wind load case and wind service combination. This drift is then compared against a user-defined limit (the default is 1/300 of the storey height, in line with Eurocode 3 recommendations, but you are free to specify a limit of your choice). Different limits can be applied to different stacks if required.

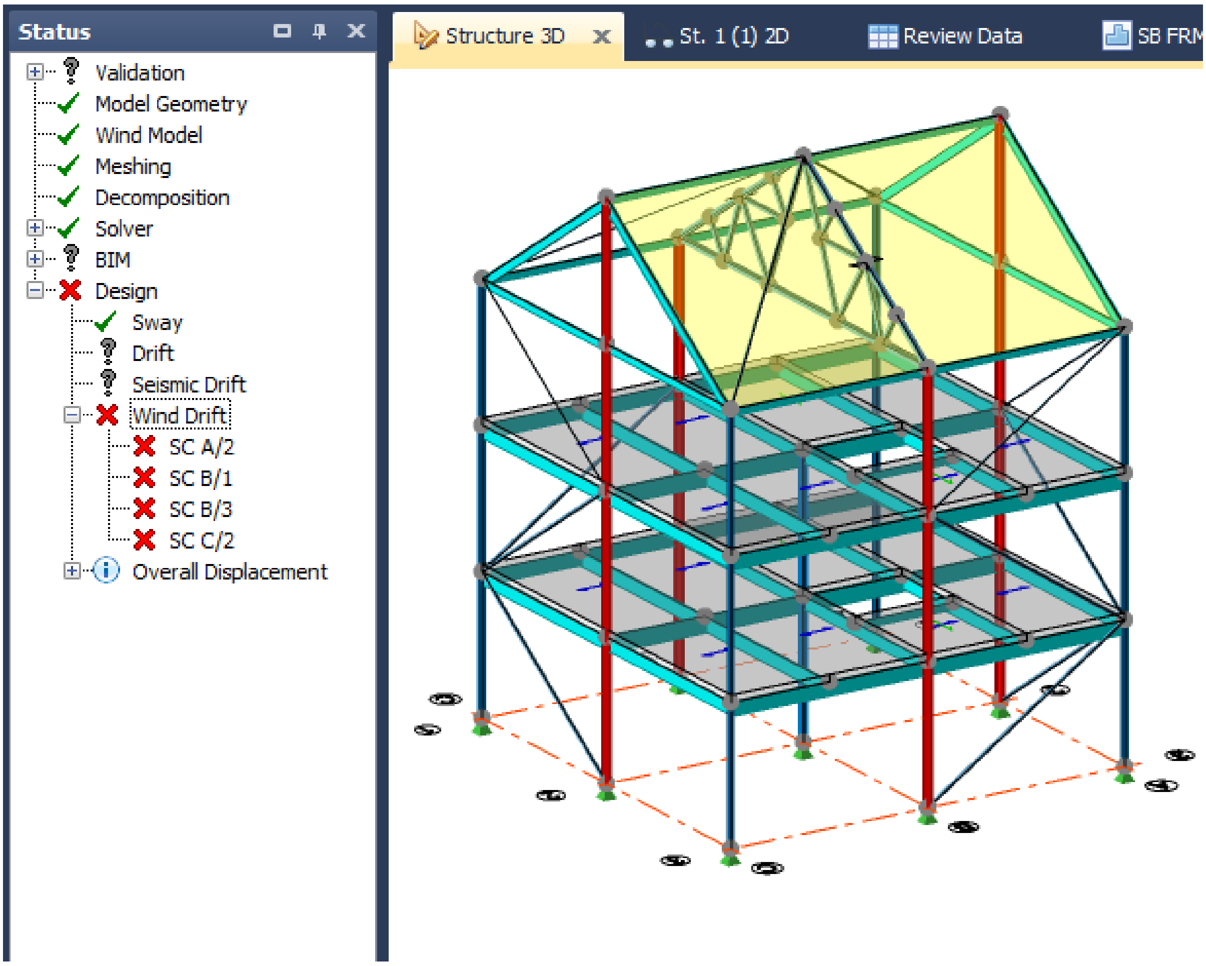
The checks are performed using results from a 1st order linear analysis (with no Reduced stiffness factor), which are generated by running any of the **Design (Gravity)**, or **Design (Static)** commands from the Design ribbon.

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

Review of wind drift checks

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

- Double clicking on a failing column in this list causes it to be highlighted in red in the current view.
- Double clicking on the Wind Drift heading at the top of the list causes all failing columns to be highlighted in red..



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

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

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

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

[Sway checks](#)

-

Overall displacement

By expanding the Design branch of the Project Workspace Status Tree, you are able to review the maximum positive and negative overall displacement results from the **3D Analysis** for both Strength and Service combinations.

Displacements can also be viewed graphically in the Results View by using the buttons on the Deflections group.

Solver Models Handbook

Solver models

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

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

Working Solver Model

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

Although 1D elements and solver nodes are displayed, 2D elements are not (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis.

Solver Model used for 1st Order Linear

This solver model is in the form of a 3D Analysis model.

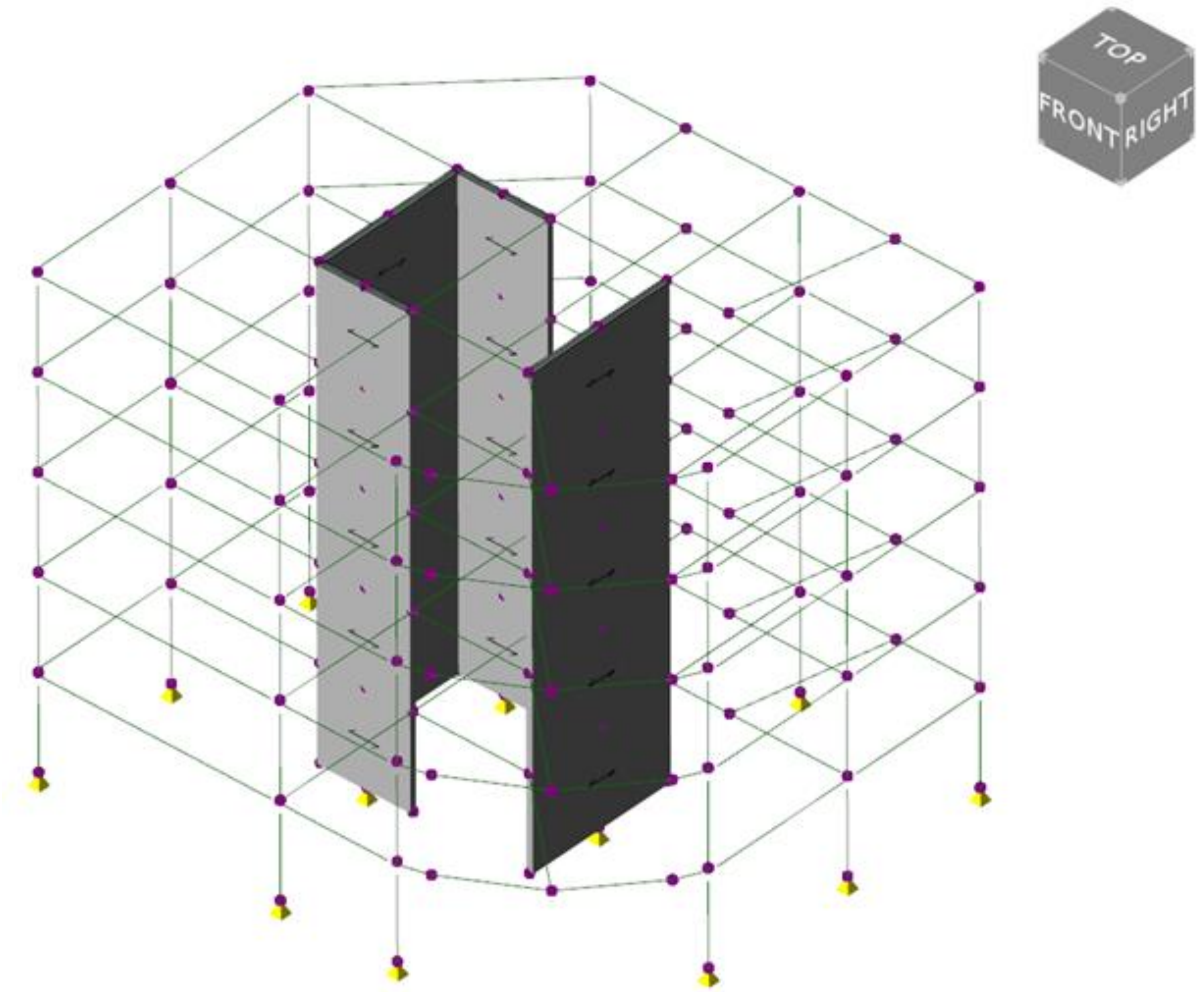
Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order linear analysis.

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

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

Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis).
Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

3D Analysis model



The 3D analysis model consists a mix of 1D analysis elements and FE meshes as follows:

- beams and columns are modelled as 1D analysis elements
- walls are either mid-pier analysis elements, or FE meshes
- slabs (optionally) form rigid diaphragms in floors
- 1-way slabs have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs are (typically) not meshed, in which case they will also have their loads decomposed on to supporting members at a preliminary stage of the analysis - see: Why slab load decomposition is required
- 2-way slabs (optionally) can be meshed
    - Recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
- supports are user defined

2-way slabs meshed

Optionally you can choose to mesh all 2-way slabs – making a fully meshed model (both walls and floors) possible.

This is generally not recommended as it will inevitably increase the model size, (and potentially the time to solve for large models), although it might be considered that a fully meshed model behaves more “correctly” where slabs are considered to be part of the lateral load resisting system of the structure.

It is more likely that you will choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

Solver Model used for 1st Order Non Linear

This solver model is in the form of a 3D Analysis model.

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

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

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

Results are still displayed for the “old” solver model until the working solver model is updated to reflect the changes (by running an analysis).
Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Linear

This solver model is in the form of a 3D Analysis model.

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

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

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

Results are still displayed for the “old” solver model until the working solver model is updated to reflect the changes (by running an analysis).
Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Non Linear

This solver model is in the form of a 3D Analysis model.

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

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

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

> *Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis).*
> *Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.*

Solver Model used for 1st Order Vibration

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

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

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

> *Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis).*
> *Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.*

Solver Model used for 2nd Order Buckling

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

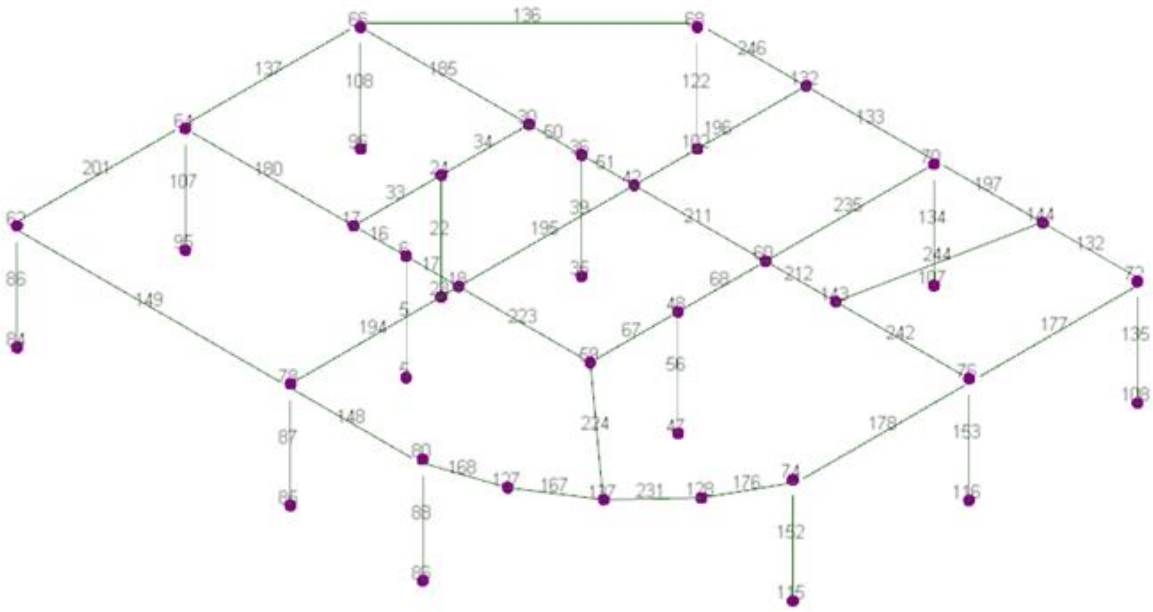
If the analysis has yet to be run, the current mesh parameters are applied.

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

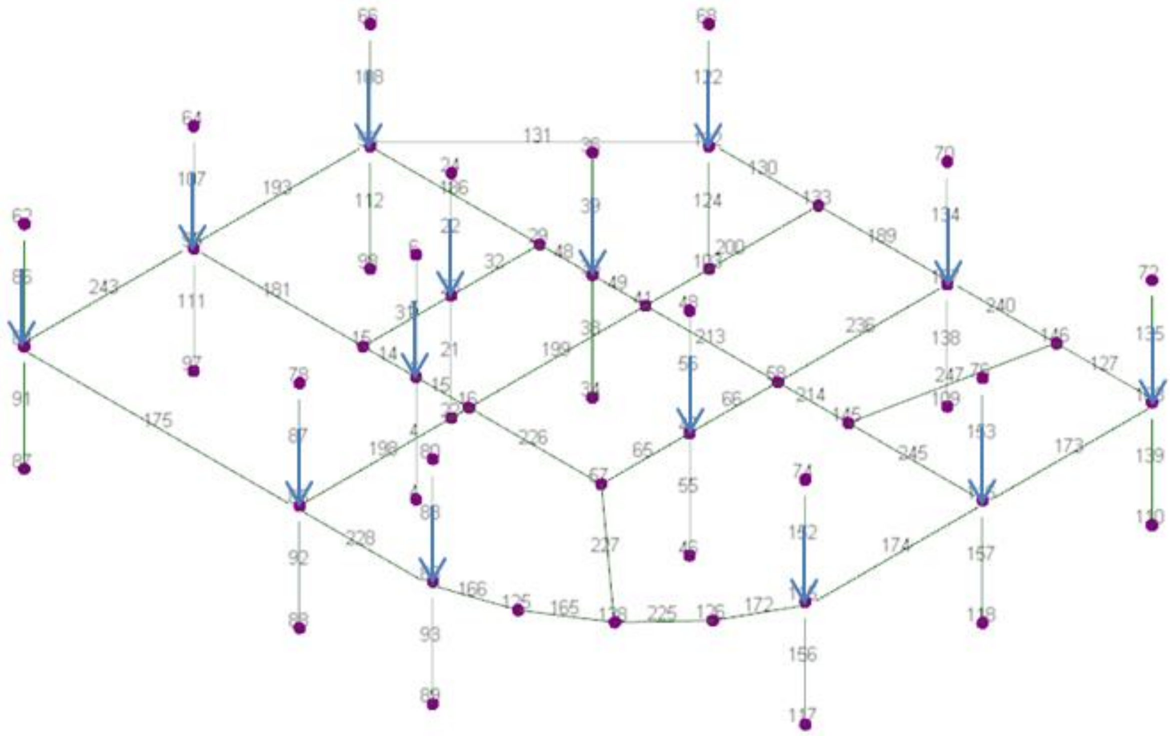
Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis).
Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

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



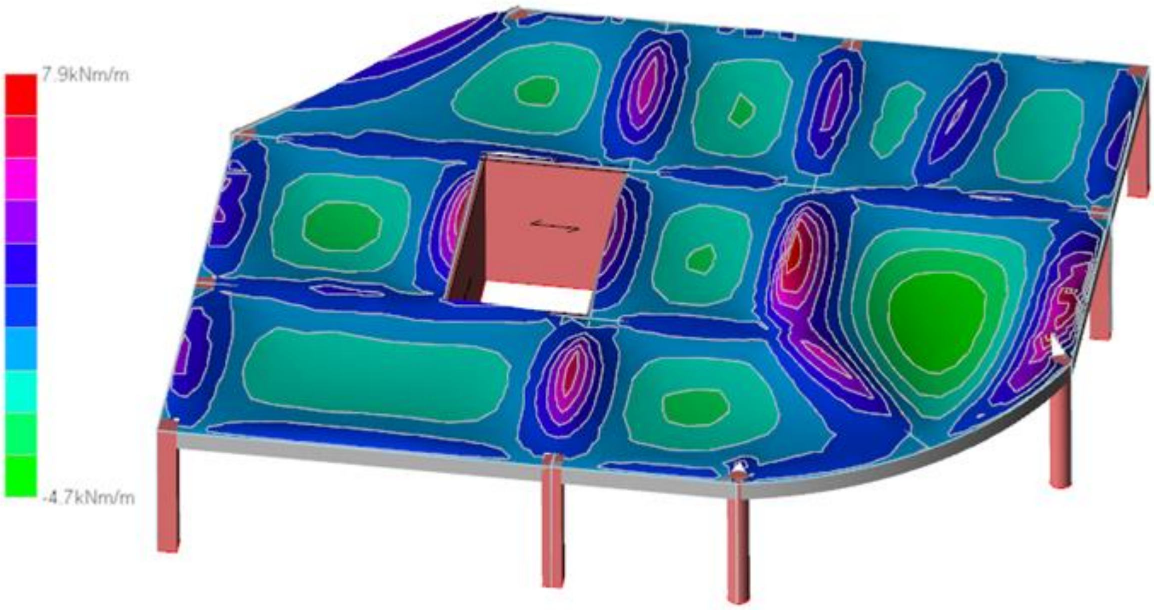
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

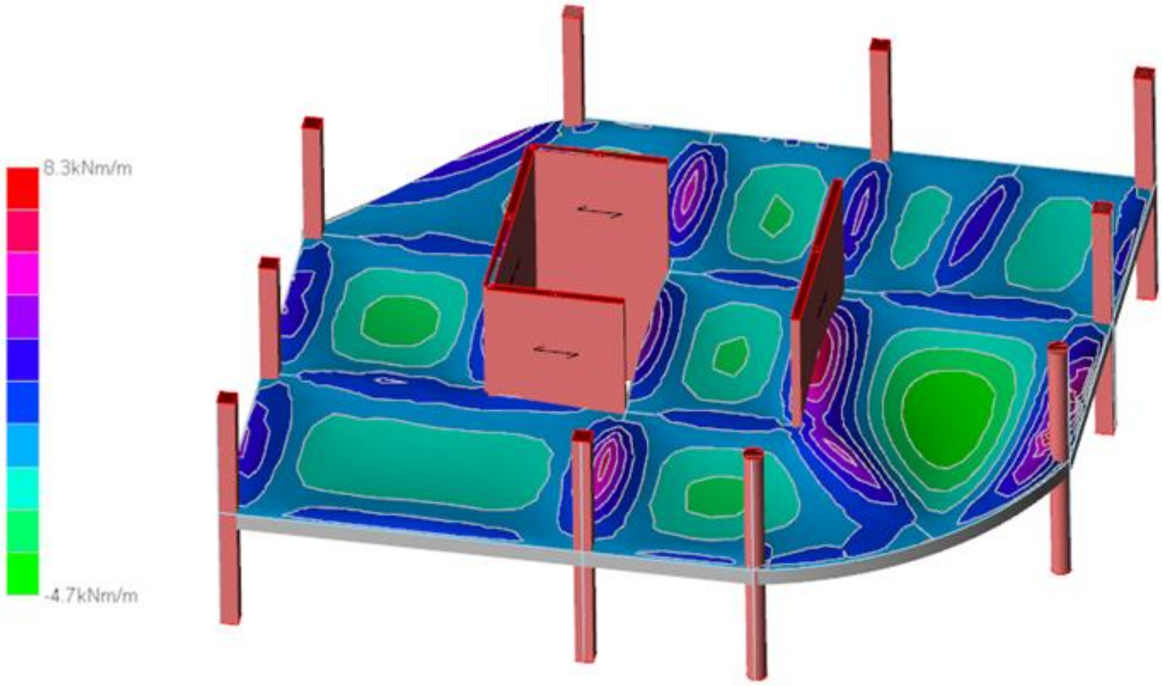


Two-way slabs are only meshed in grillage sub-models at those levels where they have been set as meshed for 3D Analysis. For all other slabs Why slab load decomposition is required is carried out prior to the analysis.

Solver Model used for FE Chasedown

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

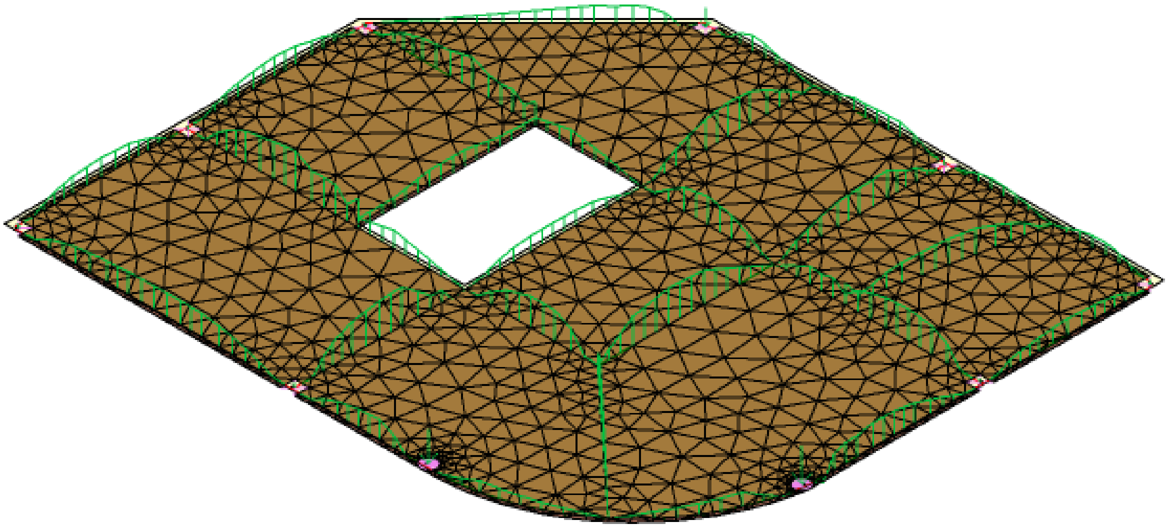




Solver Model used for Load Decomposition

At each level, (provided you have not checked the **Mesh 2-way Slabs in 3D Analysis** option), a solver model is created solely for the purpose of decomposing slab and panel loads back on to the supporting members. As these load decomposition models are only used during the pre-analysis stage, by default they are not retained.

However, if you want to examine the load decomposition model used at a given level this is possible by editing the level properties prior to analysis and selecting **keep solver model**.



Solver Model used for Load Decomposition

Refresh Solver Model

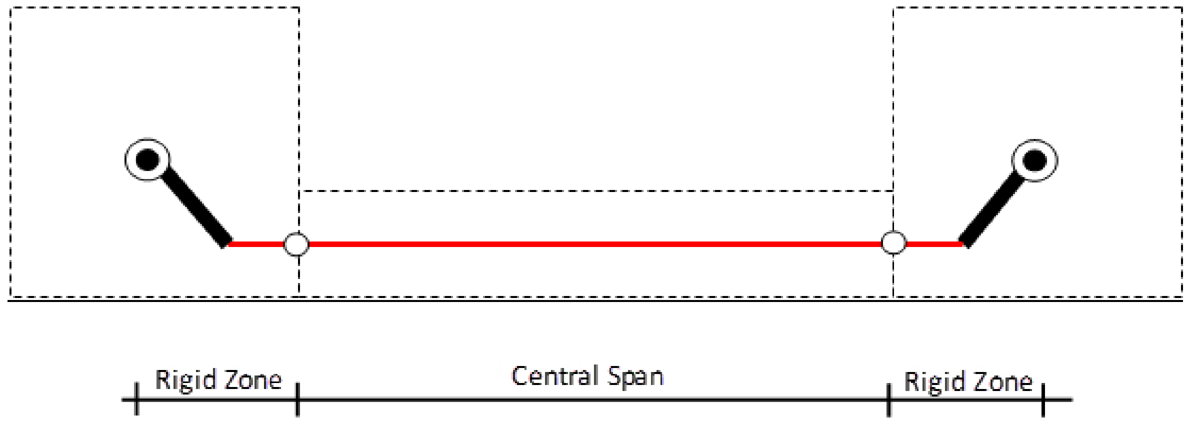
The solver model currently displayed is updated to reflect changes that have occurred in the working solver model since the last analysis. The 2D element mesh is also updated to reflect the current mesh parameters. Previous analysis results are also erased for this solver model.

Solver models created for concrete members

Rigid zones

Application of rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections. The proportion of the zone which is modelled as rigid (the thick black line shown below) is specified as a percentage, the remaining portion of the rigid zone (the red line inside the rigid zone) remains elastic. The proportion of the rigid zone that is rigid is specified in Model Settings and can vary between 0 - 100%



As shown above, the elastic portion of the rigid zone is aligned with the central span solver element.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the **Rigid zones not applied** option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the **Apply rigid zones** option that is provided in the column properties under the **Design control** heading.

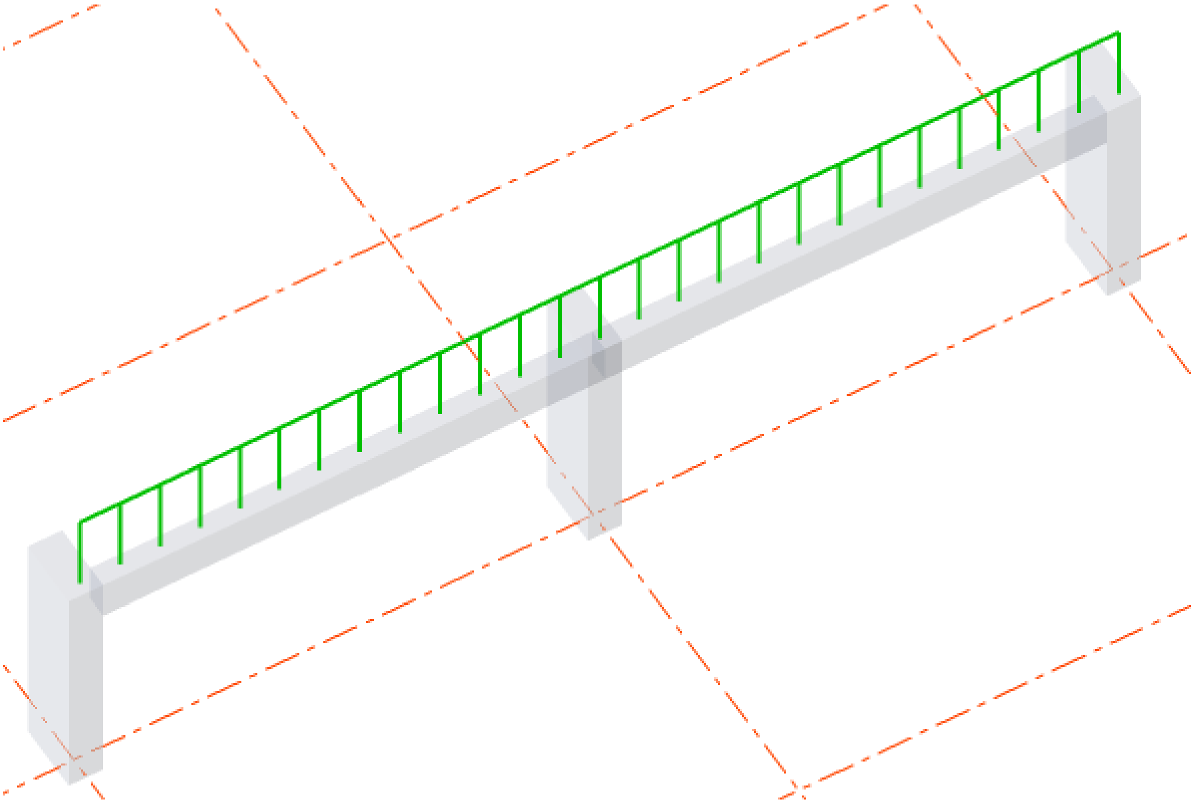
- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

There is a significant difference between Rigid Zones Not Applied and Rigid Zones Applied with 0% rigidity. The total elastic length of a member is the same in the two models, but the position of releases and start/end of design members will be different.

Rigid zones should not be confused with rigid offsets which are used to ensure that the analysis model is properly connected, i.e. it is possible to have rigid offsets in the model even if rigid zones are turned off.

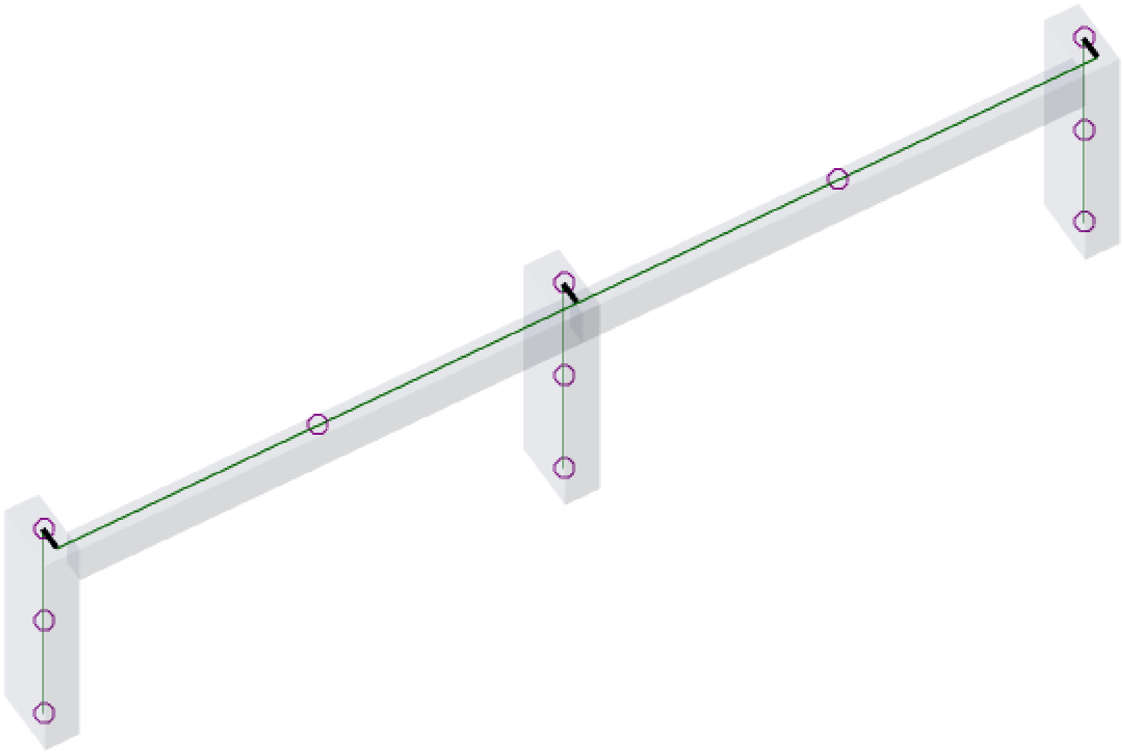
Rigid zones example 1 - fixed ended beam

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

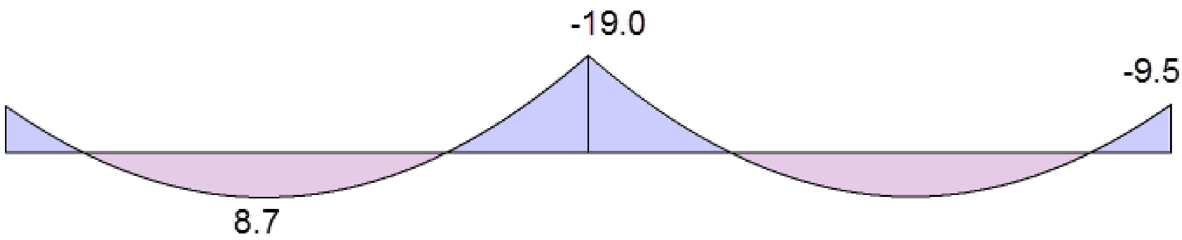


Rigid Zones Not Applied

The analysis model is simply constructed from the solver elements with rigid offsets applied as necessary to connect the beam solver elements to the column solver elements.



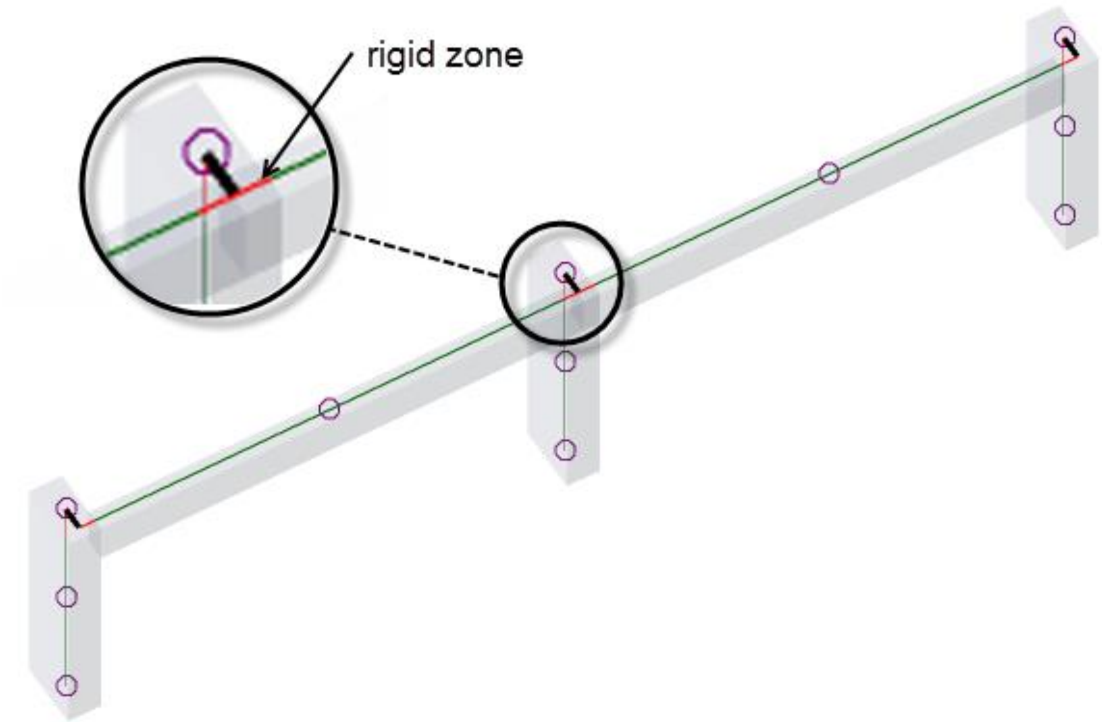
The resulting beam bending moment diagram is as follows:



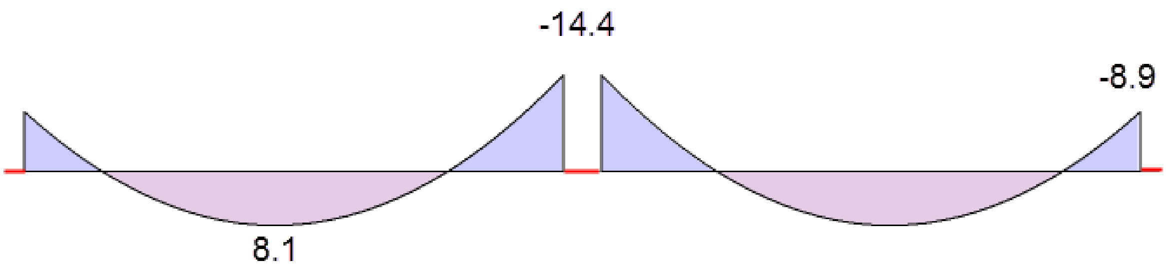
Rigid Zones Applied (default 50%)

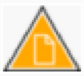
Rigid Zones are activated via Model Settings, and this is also where the percentage of rigidity of the zones can be specified. Initially for this example it will be left at the default of 50%.

The revised solver model is as shown below, note the rigid zones that have been formed where the columns and beams connect:



The beam bending moment diagram for the revised model is as shown below.



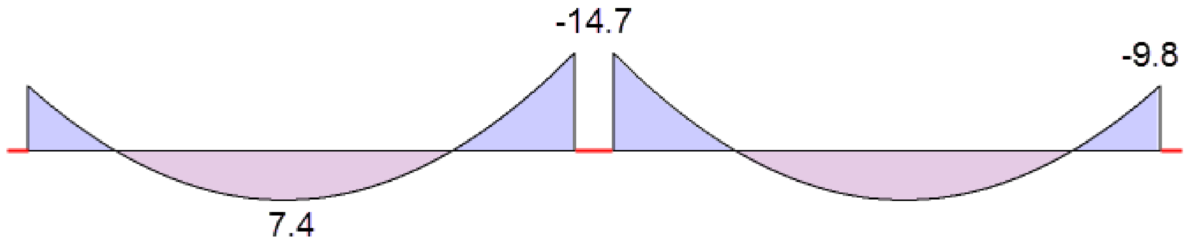


The above diagram was generated from a Results View to illustrate that a "gap" is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

We might expect the extra stiffness introduced at the supports to increase the hogging moments and reduce the sagging moments, however because the element end forces are now reported at the rigid zone boundaries (rather than the ends of the solver elements) - in this example the main effect is that the hogging moment over the central column support is substantially reduced.

Rigid Zones Applied (100%)

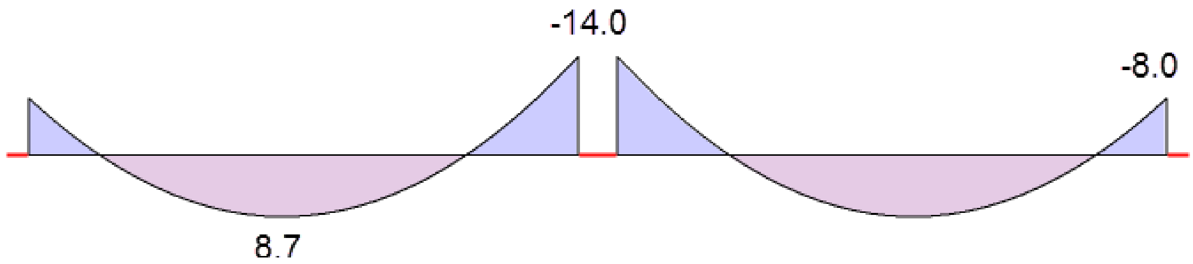
To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:



As expected the extra stiffness introduced at the supports increases the hogging moments and reduce the sagging moments in comparison to the run at 50%.

Rigid Zones Applied (0%)

If the percentage rigidity is reduced to 0% the bending moment is as shown below:



If this result is compared to the run in which rigid zones were not applied, it is clear that although the sagging moments are identical, the hogging moments that are reported are not the same. This is because, although the total elastic length of a member is the same in the two models, the position of the start and end of design members is different (being taken at the rigid zone boundaries when applied).

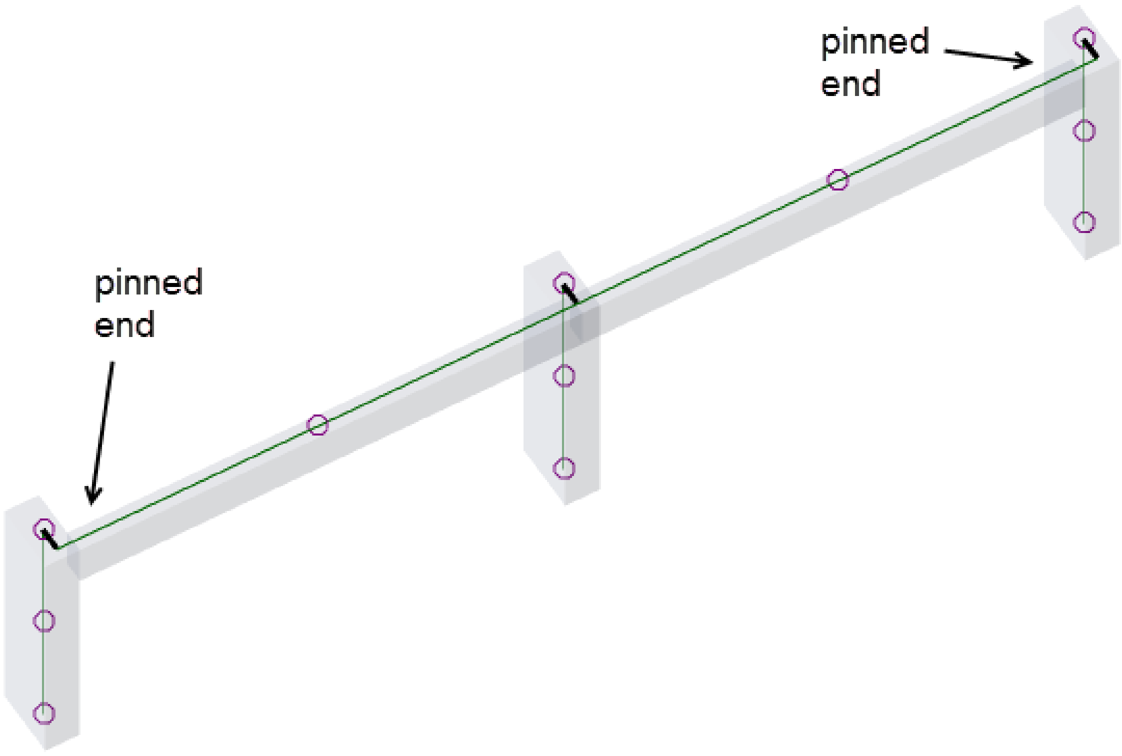
Rigid zones example 2 - pin ended beam

When rigid zones are applied to a pin ended member, the end release is shifted from the end of the solver element to the rigid zone boundary.

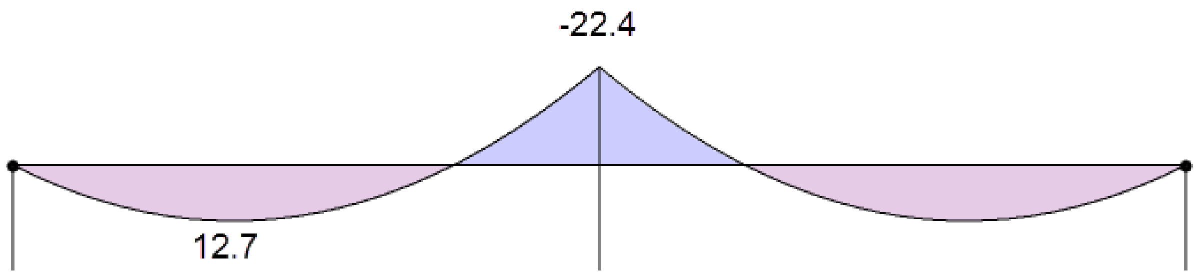
To illustrate this effect the model illustrated in Rigid zones example 1 - fixed ended beamis modified to have pinned connections introduced at the two remote ends of the beam.

Rigid Zones Not Applied

The analysis model is constructed from the solver elements with rigid offsets applied to connect the beam and column solver elements. Releases are formed at the two remote ends of the beam solver elements.



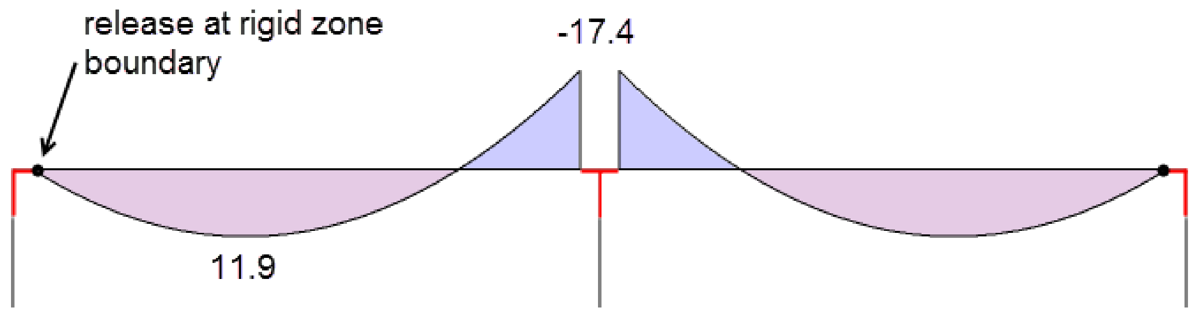
The beam bending moment diagram is as follows:



Rigid Zones Applied (0%)

For comparison, rigid zones are then introduced, (with 0% rigidity in order to keep the total elastic length of the beams the same in both models).

Because the releases are moved to the rigid zone boundaries, this has the effect of reducing the moments in the beams.



Concrete wall openings and extensions

Concrete wall openings

Limitations of wall openings

1. If you have specified a door or window opening in a wall panel you must model the wall using FE elements, otherwise a “Walls with openings have a mid-pier” validation error is displayed and the analysis will not proceed.

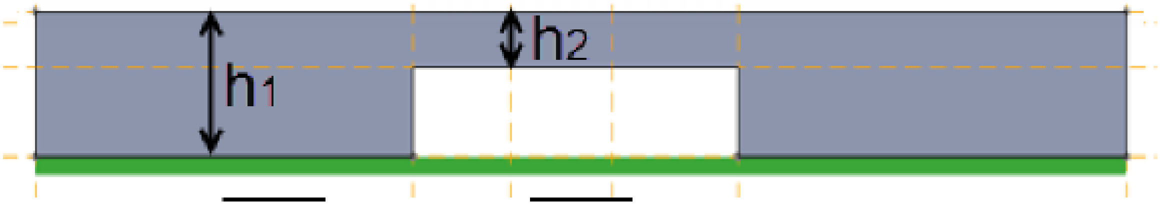
2. Assuming the wall has been modelled using FE elements, the analysis will still not proceed if you have applied a wind wall panel over the top of the wall. In this situation a “Panel is not surrounded by load carrying members” validation error is displayed. This error can only be cleared by deleting the openings from the affected walls.

3. Given that the analysis has been able to complete; a “Panel contains openings - these are ignored in design” warning will always be issued when a wall containing openings is designed. When you encounter this warning, as well as taking stock of the design implications; you need also to consider if the analysis model is appropriate, as potentially it may not reflect your original intention. In certain situations the Alternative model for wall openings may prove to be a better solution.

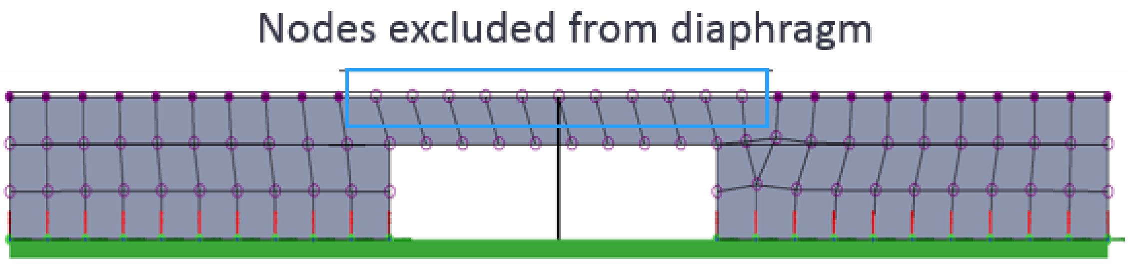
Analysis model applied to meshed wall panels with openings

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

- wall beam properties in the lintel use the lintel depth (h_2), rather than the panel depth (h_1)



- wall beam nodes in the lintel are removed from the slab diaphragm



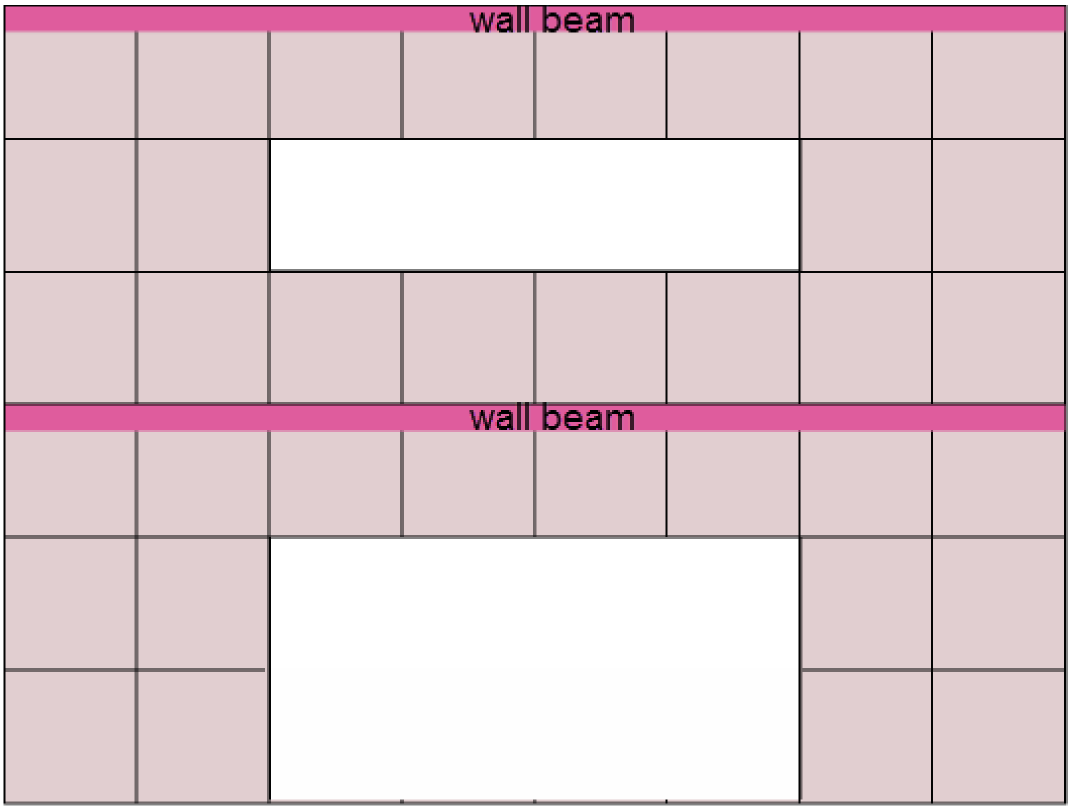
Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

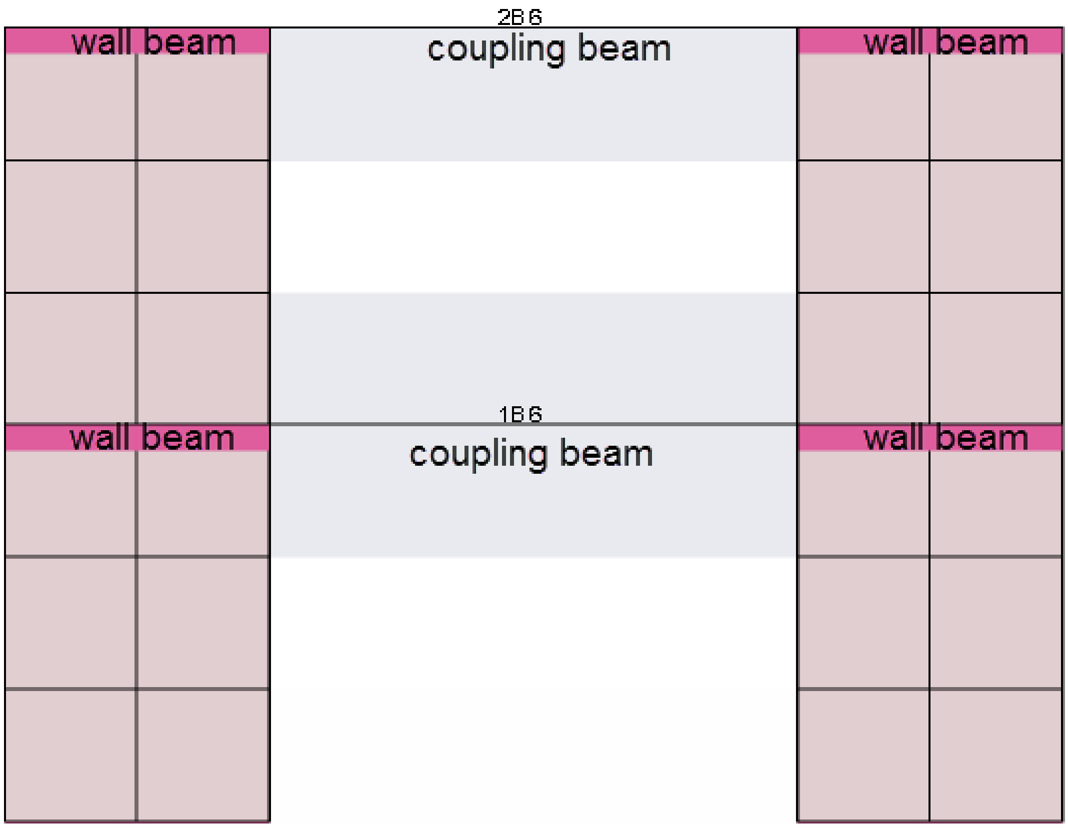
If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam ('coupling beam').

This method can be demonstrated by considering the below example, consisting of a two storey wall with a large opening at each level.

If the openings were to be created as a window and door the resulting model would be as shown:



However, by separating the wall into discrete panels and inserting coupling beams you obtain an alternative model as below:



Such an idealisation enables the panels either side of the openings to be designed for their respective forces and enables the strips between the openings to be designed as beams.

Of course, this approach will require some additional detailing, but that would have been the case anyway had the openings been added and subsequently ignored by the design.

Concrete wall extensions

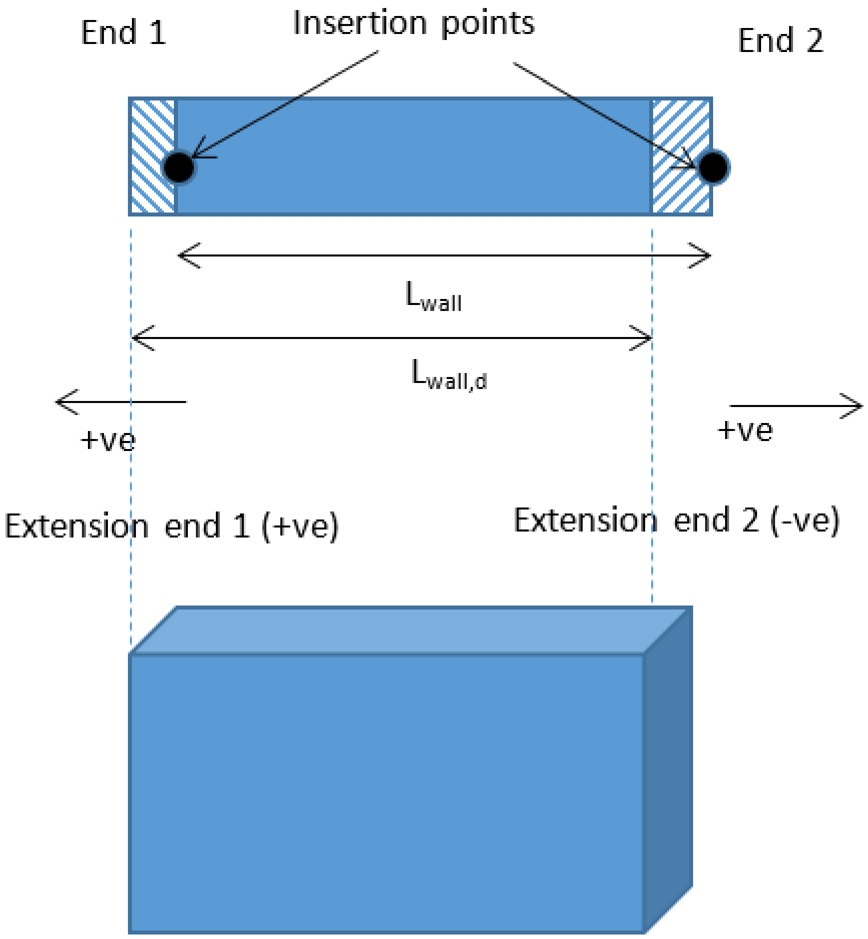
Use of concrete wall extensions

Wall extensions (End 1/End 2) are applied in order to remove physical overlaps with adjoining walls and columns without compromising the integrity of the underlying analysis model.

Negative extensions can be created automatically where appropriate. Extensions can also be defined manually if required, in which case they can be input with either positive or negative values:

- A positive extension extends the wall length beyond its insertion point.
- A negative extension trims the wall back from the insertion point.

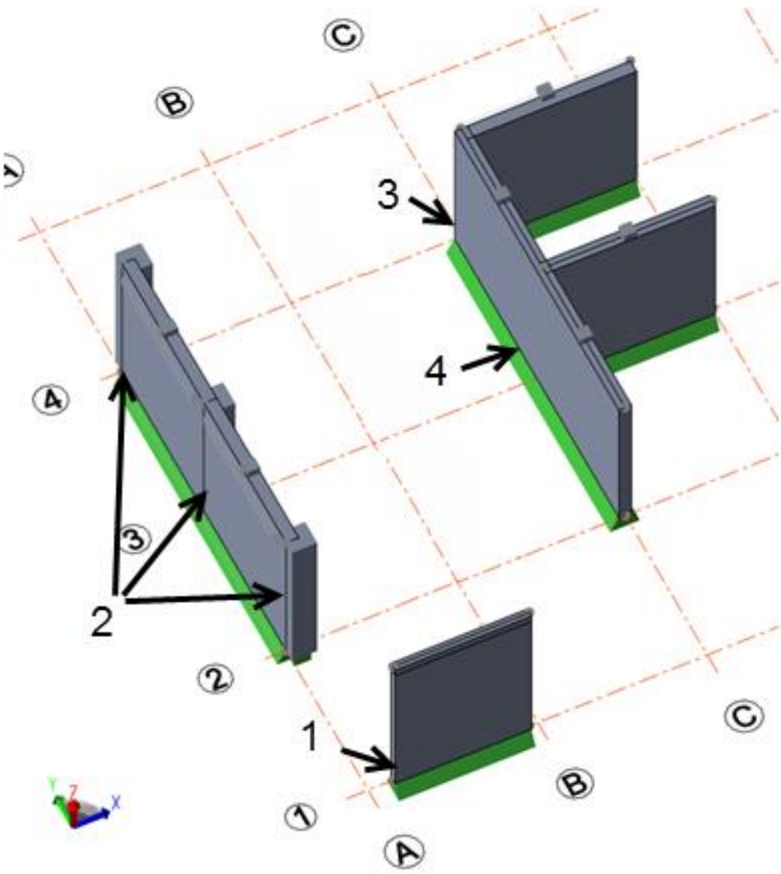
The example below shows the effect of a positive extension at end 1 and a negative extension at end 2.



Although the length of the wall used in the analysis model (L_{wall}) is unchanged, the wall length that is used in the design, quantity reporting and drawings changes to $L_{wall,d}$

Concrete wall extension examples

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



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

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

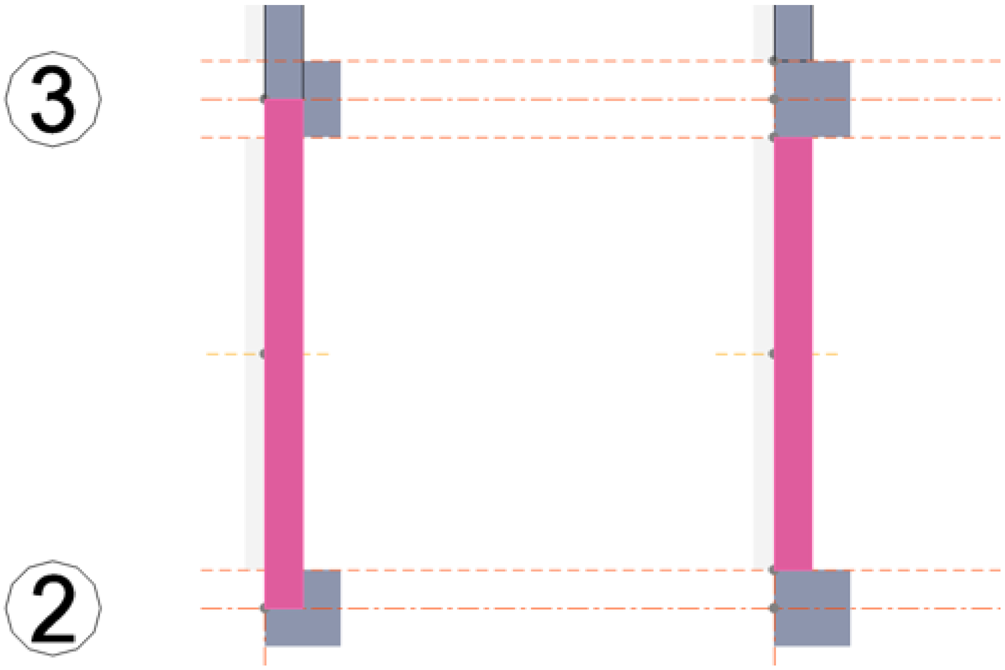
2. Where wall end overlaps a column - a negative extension can be applied automatically.

3. Where two wall ends meet - a negative extension can be applied automatically.

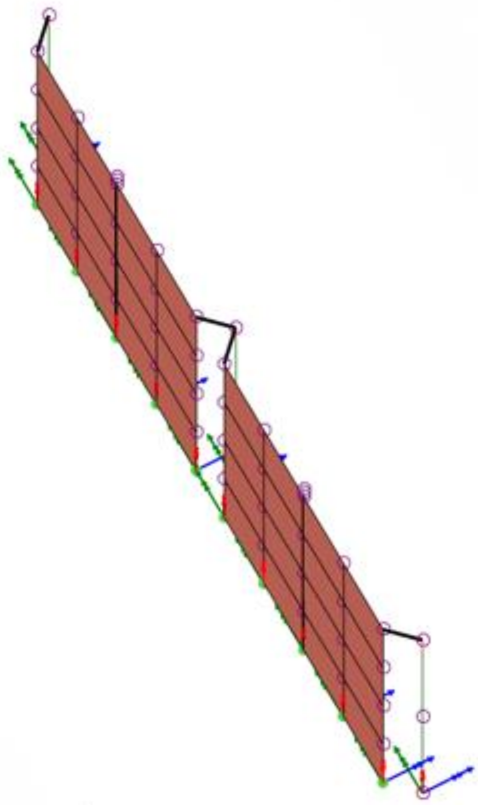
4. Where a wall end meets another wall part way along its length- a negative extension can be applied automatically.

Wall and column overlap

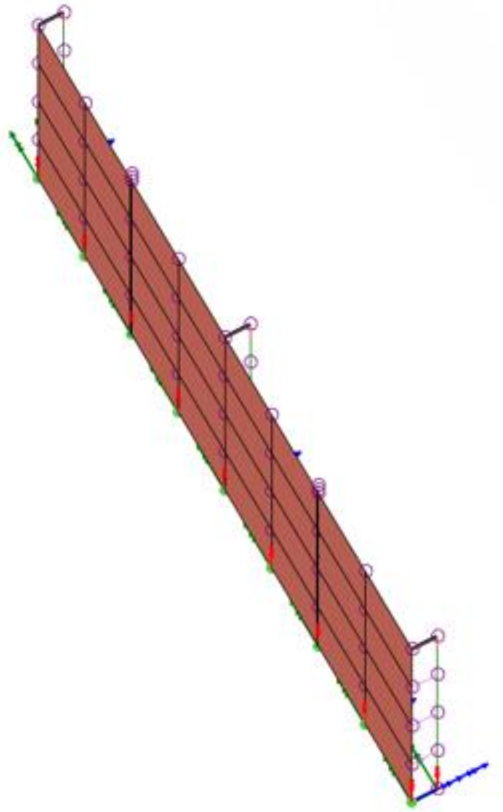
Consider case 2 above, where the wall ends overlap the columns.



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

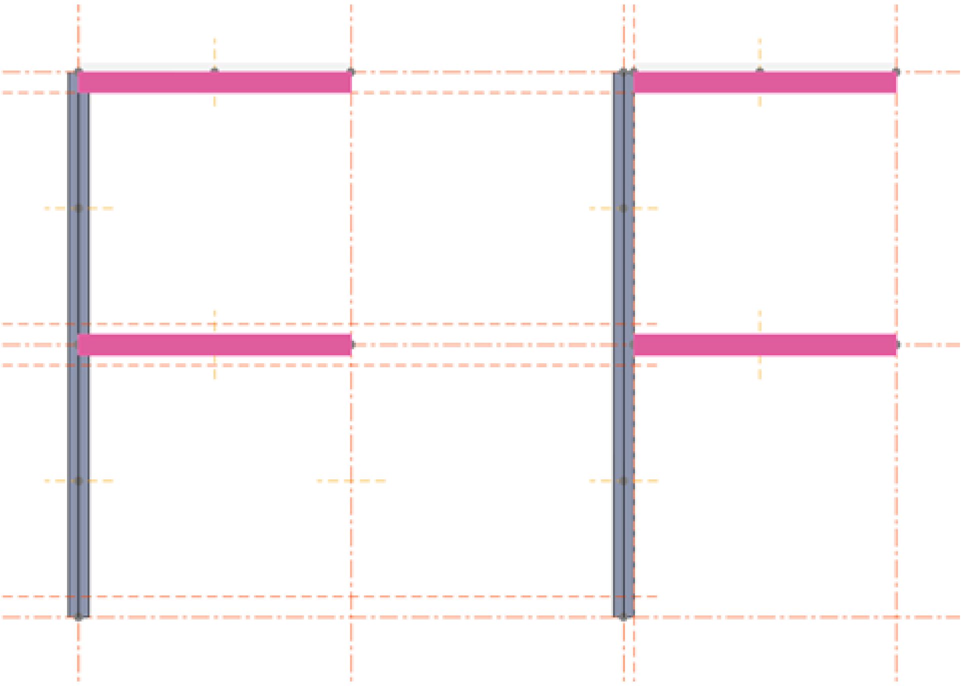


If negative wall extensions are employed instead, the analysis model is much better.



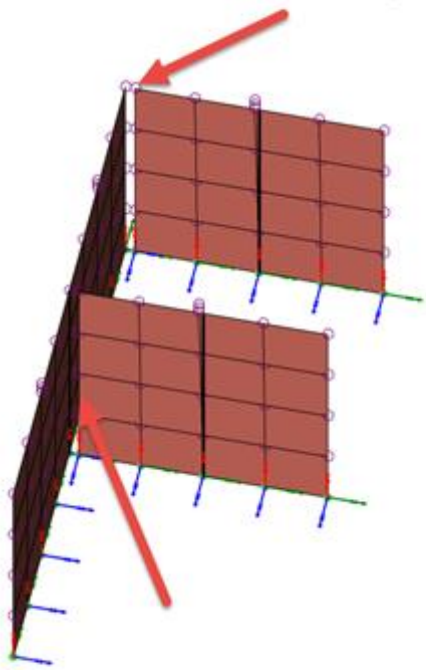
Wall overlaps with another wall

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

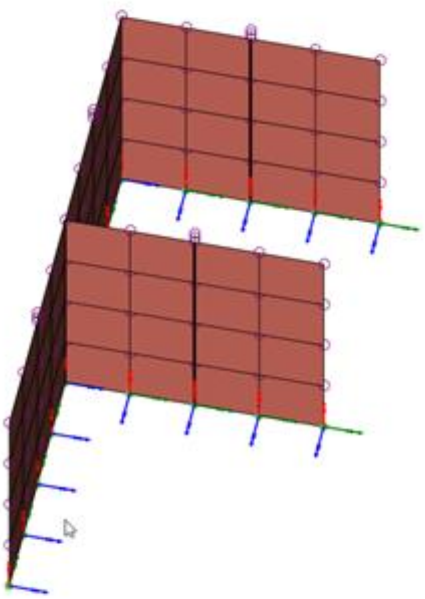


If negative extensions are not automatically applied you will see the overlap of walls (as shown above left). The main problem with this is that from a design point of view the concrete overlaps would result in a duplication of reinforcement in the overlapped areas.

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



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



Diaphragms and floor meshing

Diaphragm types

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

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

- Rigid
- Semi-rigid
- None

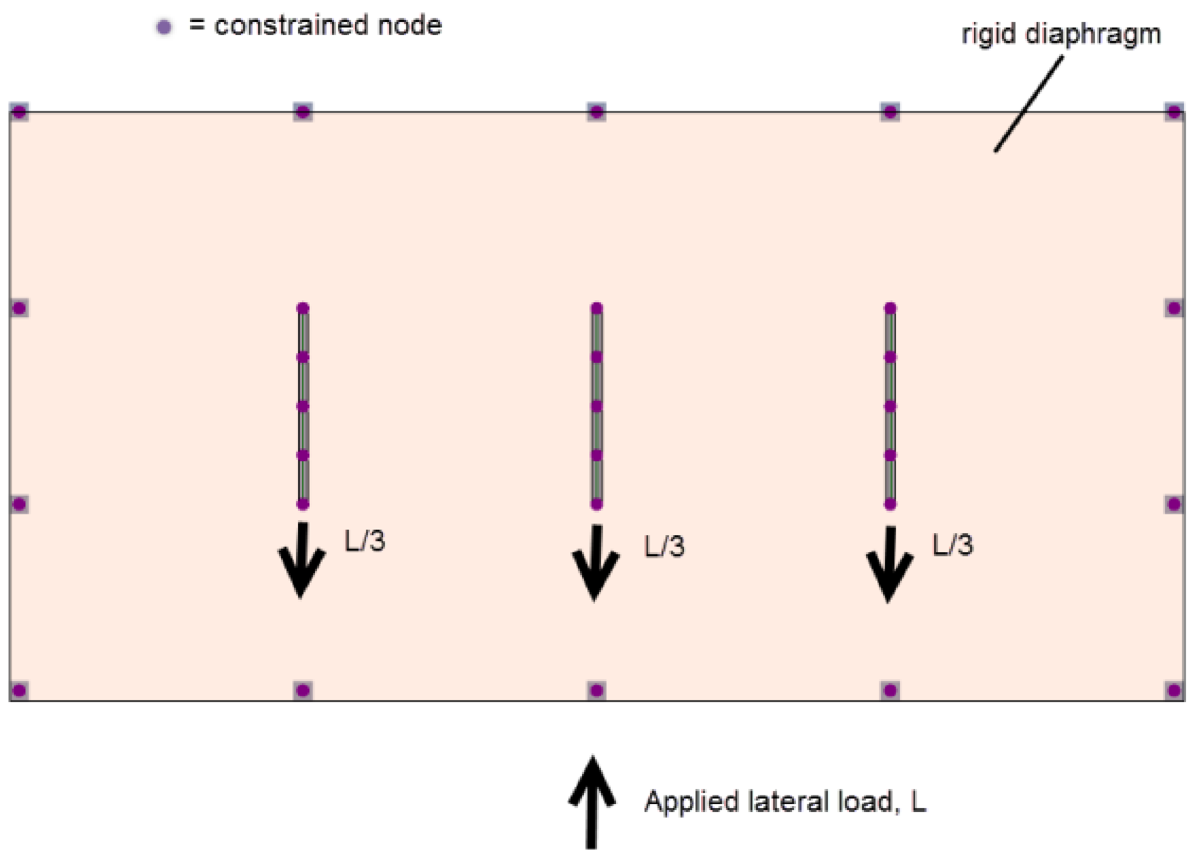
For roof panels the diaphragm type is controlled by the **Include in diaphragm** roof property, which when selected creates a semi-rigid diaphragm.

Roof panels cannot be used as rigid diaphragms.

Rigid

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

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

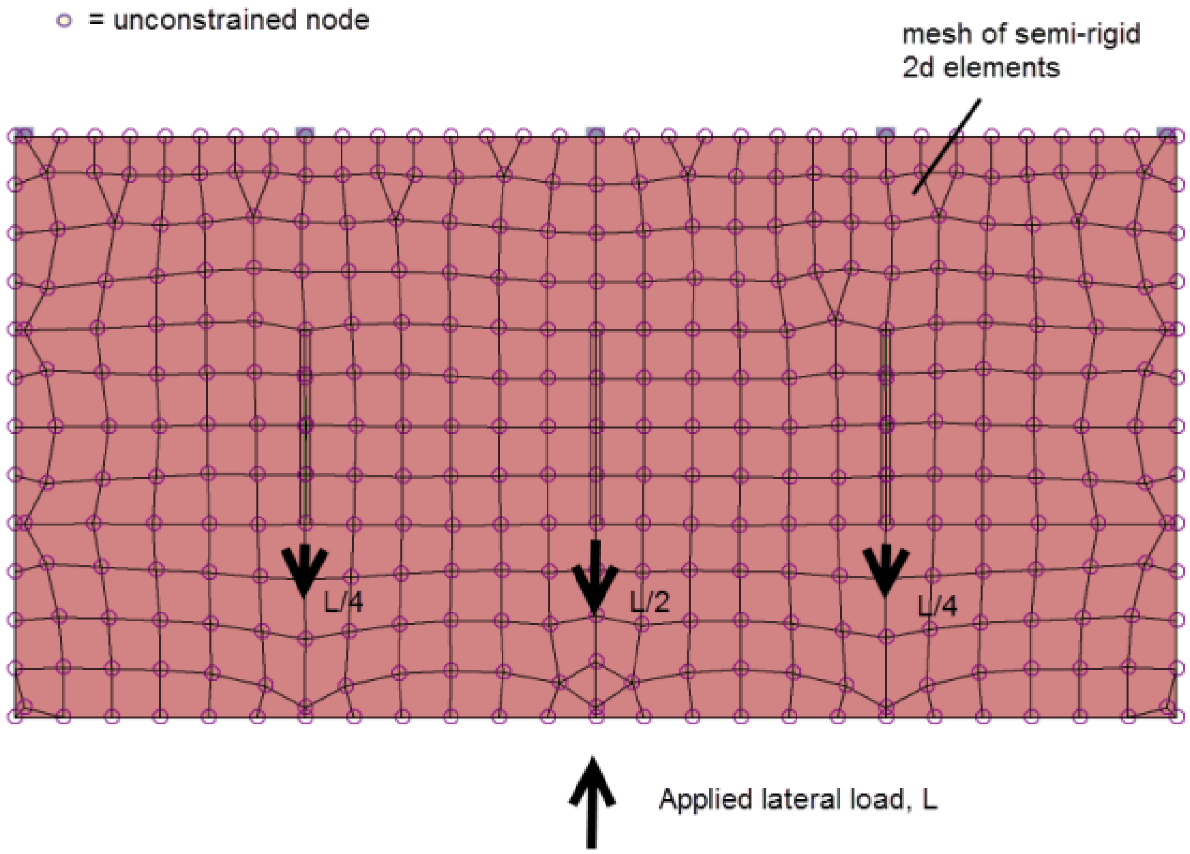


In the Solver View:
- within the rigid diaphragm boundary: *solid nodes are constrained; hollow nodes have been manually excluded and are therefore unconstrained.*
- outside the rigid diaphragm boundary: *both solid nodes and hollow nodes are unconstrained.*

Semi-rigid

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

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



Diaphragm constraint and mesh type configurations

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

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

Diaphragm option

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

- Rigid
- Semi-rigid
- None

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

Decomposition

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

- Composite Slab - One-Way only

- Precast Slab - One-Way only
- Slab on Beams - Two-Way or One-Way
- Flat Slab - Two-Way only
- Steel Deck - One-Way or Two-Way
- Timber Deck - One-Way only

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

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

Two-way spanning slabs always adopt a mesh of shell elements for the FE chasedown and FE Load Decomposition solver models. However for the 3D Analysis and Grillage chasedown solver models they are:

- unmeshed - if the **Diaphragm option** is set to None,
- meshed with semi-rigid elements - if the **Diaphragm option** is set to Semi-rigid, or,
- meshed with shell elements - if the **Diaphragm option** is set to Rigid.

Semi-rigid 2-way spanning slabs can be designed, but only using the results from the FE Chasedown, not the 3D Analysis or Grillage Chasedown.

Mesh 2-way Slabs in 3D Analysis

Two-way spanning slabs are always meshed with shell elements in FE Chasedown and Load decomposition solver models. When the **Mesh 2-way Slabs in 3D Analysis** property (specified in the **Level**, or **Slope** properties) is checked the same meshing parameters are then extended to the Grillage chasedown and 3D Analysis solver models.

Summary of diaphragm constraint and mesh type configurations

The configurations of mesh and nodal constraints applied to each solver model resulting from the different permutations of the “decomposition”, “diaphragm option”, and, “mesh 2-way slabs in 3D analysis” properties are summarised in the table below.

Decomposition	Diaphragm Option	Mesh 2-way slabs in 3D Analysis	FE Load Decomposition & FE Chasedown Models	Grillage Chasedown & 3D Analysis Models
1-way	None	Not Applicable	No mesh; no nodal constraints	No mesh; no nodal constraints

	Semi-Rigid	Not Applicable	Semi-Rigid mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Not Applicable	No mesh; Nodal constraints	No mesh; Nodal constraints
2-way	None	Yes	Shell Mesh; no nodal constraints.	Shell Mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Yes	Shell mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
		No	Shell mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Yes	Shell Mesh; Nodal constraints	Shell Mesh; Nodal constraints
		No	Shell Mesh; Nodal constraints	No Mesh; Nodal constraints

Other slab properties affecting the solver models

Rotation Angle

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

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

Include in Diaphragm

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

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

Where 2 items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

Divide stiffness by

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

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

Mesh parameters

Slab Mesh

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

Semi-Rigid Mesh

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

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

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

Releases

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

Column releases

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

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

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

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

User Defined

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

1. Right-click the column to display the context menu.
2. Choose **Edit**
3. From the Column Property Dialog open the Releases page and choose the stack to edit.
4. Uncheck the My or Mz degree of freedom at the desired end as required.

Wall releases

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

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

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

Beam releases

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

- **Fully fixed (free end)** - Denotes a cantilever end. It is achieved by selecting **Free end**. (In a single span beam this box can only be checked if the opposite end is fully fixed.)
- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Moment** - Major axis moment connection, and pinned about the minor axis.
- **Fully fixed** - Encastré, all degrees of freedom fixed.
- **Continuous** - This setting is automatically applied when a continuous beam is created and effectively creates a non-editable fully fixed connection between the spans of the continuous member. The connection can only be edited by splitting the beam.
- **User defined** - This setting appears if the connection is pinned for major axis bending (My released) but remains fixed for minor axis bending (Mz).

In addition to the above release options you are also able to apply a **torsional release** at either end by checking the appropriate box. Similarly an **axial release** can be applied to beams of all materials apart from concrete.

User Defined

The User defined option is not available in the Properties Window and can only be specified as follows:

1. Right-click the beam to display the context menu.
2. Choose **Edit**
3. From the Beam Property Dialog open the Releases page.
4. Check the Mz and uncheck the My degree of freedom at the desired end as required.

Brace releases

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

An option is provided to include force eccentricity moment.

Supports

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

Supports can also be placed manually at other locations.

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

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

Support degrees of freedom

Each support has six possible degrees of freedom:

- translational (Fx, Fy, and Fz)
- rotational (Mx, My, and Mz)

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

The options for a support that is rotationally free in Mx or My are:

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

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

- Release
- Spring Linear
- Spring Non-linear

The options for a support that is translationally free in Fx, Fy, or Fz are:

- Release
- Spring Linear
- Spring Non-linear

Non linear spring supports

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

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

Compression only ground spring

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

Type: Spring Non-Linear

Stiffness -ve: 0

Fmax -ve (tension): 0

Stiffness -+e: your choice of ground spring stiffness value

Fmax +ve (compression): your choice of capacity

Partial fixity of column bases

Two additional types of rotational linear spring are provided to allow partial fixity to be modelled, these are:

- Nominally pinned
- Nominally fixed

These are specifically provided for supports under columns (of any material), but will result in a validation error if placed under walls (meshed or mid-pier), or if they are used for any other supports.

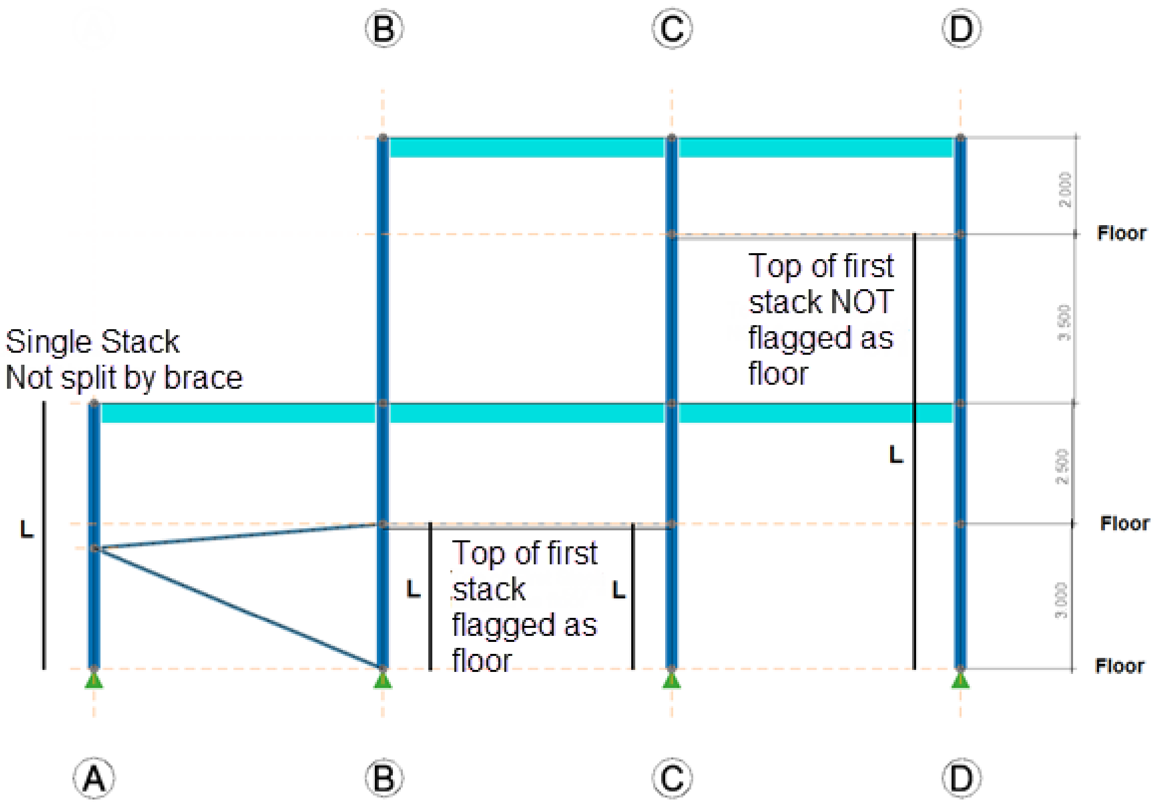
The support stiffness is based on the column properties (E*I/L)

- E = Young's Modulus of the column
- I = relevant bending stiffness (Ixx or Iyy) of the column
- L = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialogue, i.e. combined length of all the stacks until a floor is found.

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

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

- Nominally pinned (spring stiffness) - x% * 4 * E*I/L (default x% = 10%)
- Nominally fixed (spring stiffness) - x% * 4 * E*I/L (default x% = 100%)



Since the spring stiffness is dependent upon stack height and column stiffness (E and I), the spring stiffness will change if any changes are made to column stack height, column E or I values.

In addition, since for steel, Auto Design can change the column size (and hence I value) the spring stiffness will change with any change in column size.

Steel Design Handbook

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

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

Composite beam design
Steel connection design

General design parameters

Material type

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

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

Autodesign (steel)

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

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

When **Autodesign** is not selected (i.e. check mode), you assign your desired section size to the member and *Tekla Structural Designer* determines if the section is sufficient.

When **Autodesign** is selected the section type to be used is specified from a Design Section Order and *Tekla Structural Designer* attempts to automatically determine a suitable size.

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

- Size Constraints
- Steel beam deflection limits

- Steel beam camber (in the case of beams)

Design Section Order

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

The design process commences by starting with the first section in the chosen order file. Any section that fails any of the design conditions is rejected and the design process is then repeated for the next available section in the list.

On completion of the design process, the first satisfactory section from the Section Designation list is assigned to the member.

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

1. Edit the properties of the member.
2. Click the **Design section order** drop list and select **<New\Edit>...**
3. Choose a section order from the available list and then click **Edit...**

The sections contained within the chosen order file appear in the **Sections in use** list on the right of the page.

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

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

Limiting the choice of sections by unchecking a section within an order file is a global change that affects ALL projects, (not just the currently open one). It is typically used therefore to eliminate unavailable or non-preferred sections from the design process. If design requirements for an individual member require section sizes to be constrained, (due to, for example depth restrictions), then the choice of sections should be limited instead by using Size Constraints, (as these only affect the current member).

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

While viewing the list of sections you can:

1. Select a property from the **Sort By** droplist
2. Click **Sort** to re-order by the chosen property.
3. Having sorted, if you don't want to subsequently move individual sections up or down the list, check **Keep sorted** to de-activate **Move Up** and **Move down**.

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

How do I specify that a section is non-preferred?

Some sections might be more expensive or difficult to obtain; you might therefore want other sections to be chosen in preference to them, (whilst still keeping them available). You can achieve this by moving the "non-preferred" sections further down the design order list.

To move a section up or down the list:

1. If **Keep sorted** is checked, you must uncheck it in order to activate **Move Up** and **Move Down**.
2. Highlight the section in the **Sections in use** list and then click **Move Up** or **Move Down** to promote or demote it.

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

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

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

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

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

How do I create a new Design section order?

If you want to create a completely new design section order you can do so as follows:

1. Edit the properties of a member.
2. Click the **Design section order** drop list and select **<New\Edit>...**
3. In the Select a Section Order dialog, click **Add...**

4. Enter a unique name for the new design section order.
5. Select the **Country** and the **Section Group** required.
6. Highlight sections in the **Available Sections** list and add, then sort them as required.
7. When the list of sections in use is as you want it, click OK

   The new design section order appears on the list of available section orders.

Size Constraints

Size Constraints are only applicable when **Autodesign** is checked. They allow you to ensure that the sections that *Tekla Structural Designer* proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm. If so you would simply enter this value as the Minimum width, and *Tekla Structural Designer* would not consider sections with flanges less than this width for the design of this beam.

Gravity only design

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

- Gravity only - designed for gravity combinations and seismic combinations
- Lateral and Gravity - designed for all combinations types - gravity, lateral and seismic

Setting columns that do not help resist lateral loads to be designed for gravity loads only will reduce the overall design time.

Engineering judgement will however be required when identifying members as being 'gravity only'.

For example:

- if an inclined braced member connects to a simple/composite beam, axial force in the brace (from both gravity and lateral loads) puts the beam into bending and therefore the beam should be designed for both gravity and lateral loads.
- potentially, simple beams in a sloping roof would also need to be designed for both gravity and lateral load

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

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because there are many smaller members or sheeting, for example, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier can be applied to steel beams, composite beams and steel columns. It is defined in the properties window by selecting **Prevent out of plane instability** and then entering a suitable value in the **Instability factor** field.

This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, there is no amplification of the minor axis inertia in the design of the member.

Steel beam design

- Web Openings to SCI P355

Steel beam torsion

Steel beam design properties

Web Openings to SCI P355

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.

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...)

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

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 >= $0.1l_o$ 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	- - - -	<= $1.5h_o$ high shear* <= $2.5h_o$ low shear <= $2.5h_o$ high shear* <= $4h_o$ low shear
Min. width of web post: - Low shear regions -High shear regions	>= $0.3h_o$ >= $0.4h_o$	>= $0.5l_o$ >= l_o
Corner radius of rectangular openings:	-	r_o >= $2t_w$ but r_o >= 15 mm
Min. width of end post, s_e:	>= $0.5h_o$	>= l_o and >= h

Min. horizontal distance to point load: - no stiffeners - with stiffeners	 >= 0.5h >= $0.25h_o$	 >= h >= $0.5h_o$

** 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

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.

Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Composite beam design

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 Imposed **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.

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.*

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, Imposed or Wind.

For each load that you add to an Imposed 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 design properties

Properties common to composite and non-composite beams

The related topic links below describe those properties that are shared by composite and non-composite beams.

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**.

Floor construction

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 centre-line of the beam, and at the position of the lap (if known). If the position of the lap is not known, then the default value of 0mm should be used (that is the lap is at the centre-line of the beam) as this is the worst case scenario.

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.

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).

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.

Further details of stud optimization and the partial interaction rules are provided in the Steel Design Reference Guides. Refer to the Composite Beam Section in the guide appropriate to the code being designed to.

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.

'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 rationalie 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.

EG

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) 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.

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.

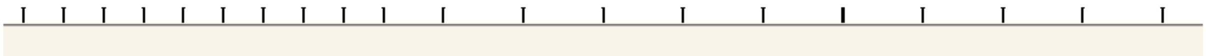
The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the **Stud strength** page.

Non-uniform

If optimization has been checked (see Optimize shear interaction) 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

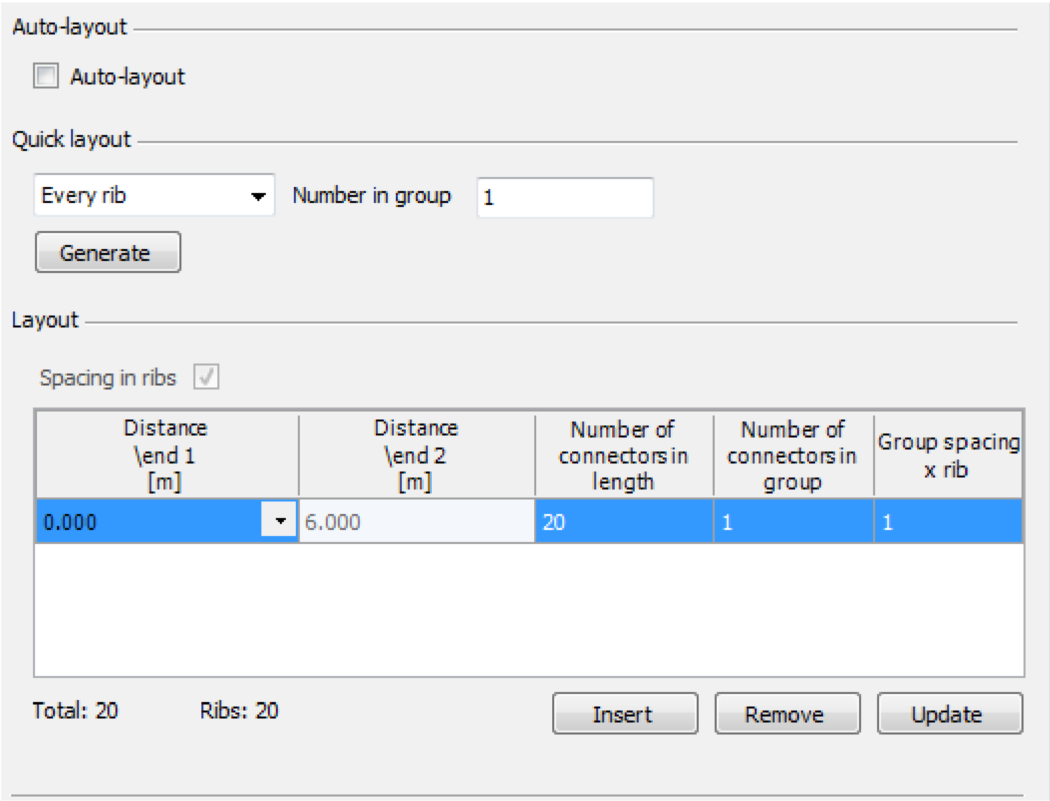
You may prefer to manually define/adjust the group spacing along the beam. This can be achieved by unchecking **Auto layout**.

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:

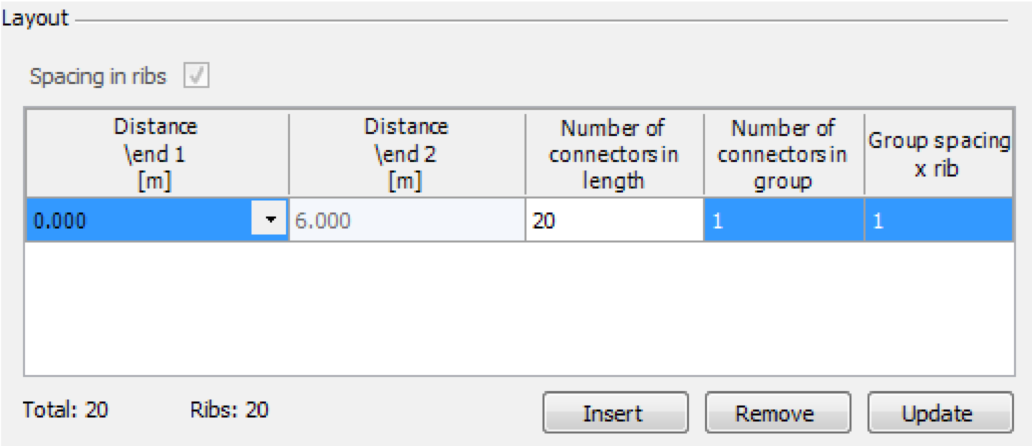


To use **Quick layout,** proceed in one of two ways:

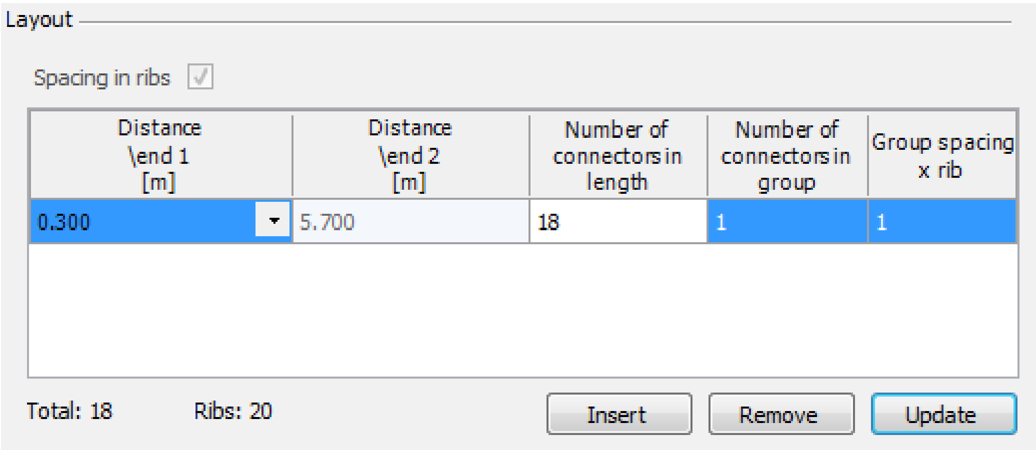
- Choose to position groups in either every rib, or alternate ribs, then specify the number of studs required in the group and click **Generate**.
- Alternatively: specify the total number of studs, then when you generate, if the number specified is greater than the number of ribs, one will be placed 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:

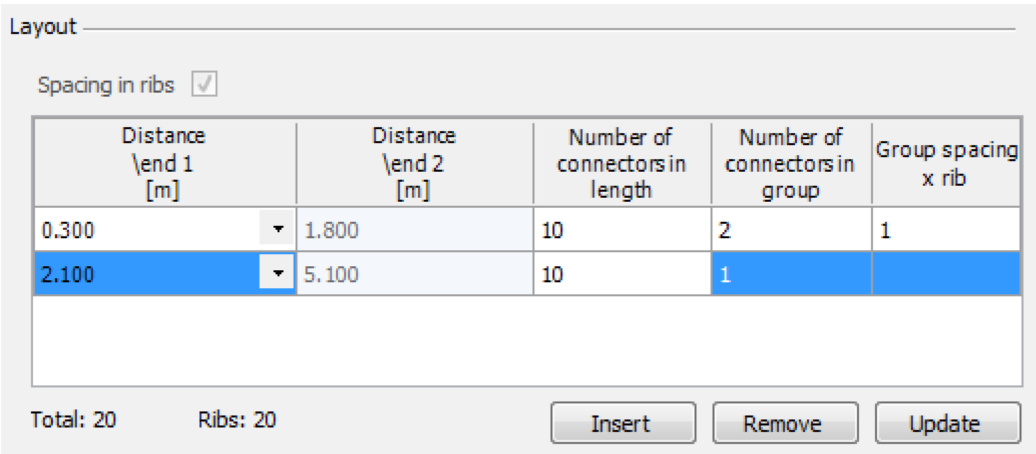
- For each segment you should define the following parameters: **No. of connectors in length** and **No. of connectors in group**; **Group spacing x rib**.



- Your input for these parameters is used to automatically determine **Distance end 2** - this latter parameter cannot be adjusted directly, hence it is dimmed.

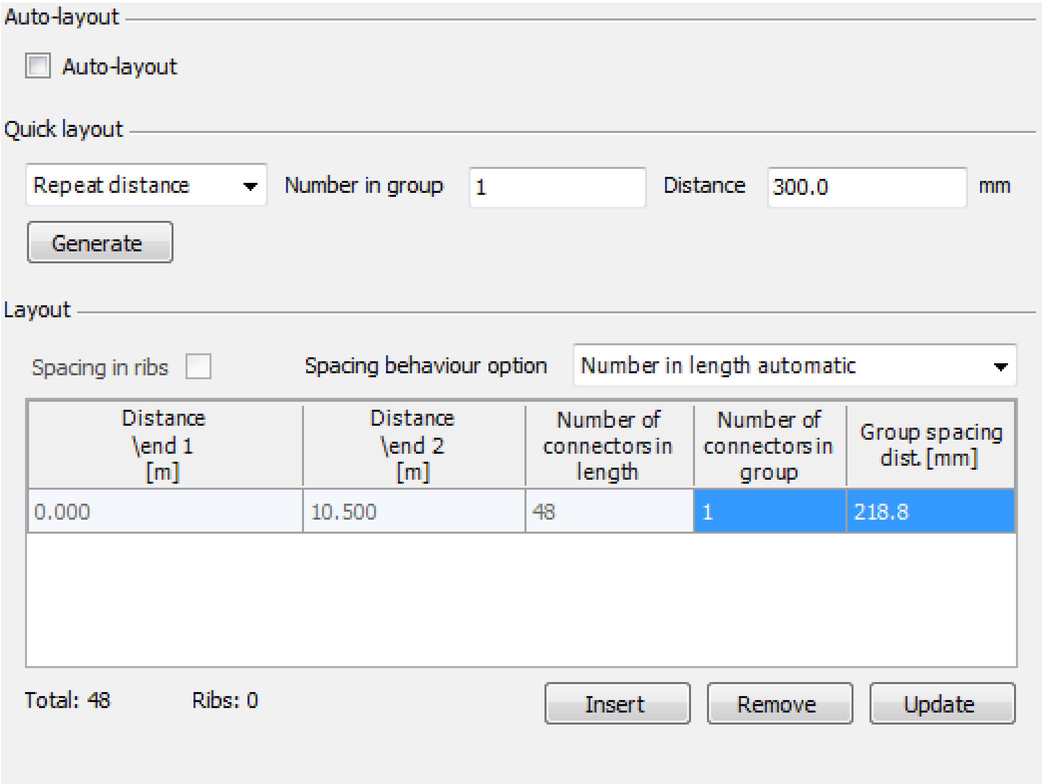


- If required click **Insert** to divide the beam into additional segments. (Similarly **Delete** will remove segments). You can then specify a different stud layout for each segment.
- We would advise that having entered No. of studs in length, group and spacing and ignoring Distance ends 1 and 2 you 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.

Steel column design

Limitations for sloping columns

The following limitations apply:

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

Steel column design properties

Simple columns

A steel column can be designated as a `simple column' - in which case specific design rules are required.

A simple column should not have any applied loading in its length. Simple columns are subject to axial forces and moments due to eccentricity of beam reactions. In order to prevent end fixity moments you would have to manually pin the ends of the column.

The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as "beyond scope" when designed.

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 take account of the P-δ moment at the position as well as the forces from the analysis.

Each lift (length between splices) of a general column can be of different section size and grade. Different section types within the same column are not allowed due to the particularly complex design routines that general columns require. You are responsible for guaranteeing that the splice detail ensures that the assumptions in the analysis model are achieved and that any difference in the size of section between lifts can be accommodated practically.

Column Splice Report

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

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

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

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

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

Columns

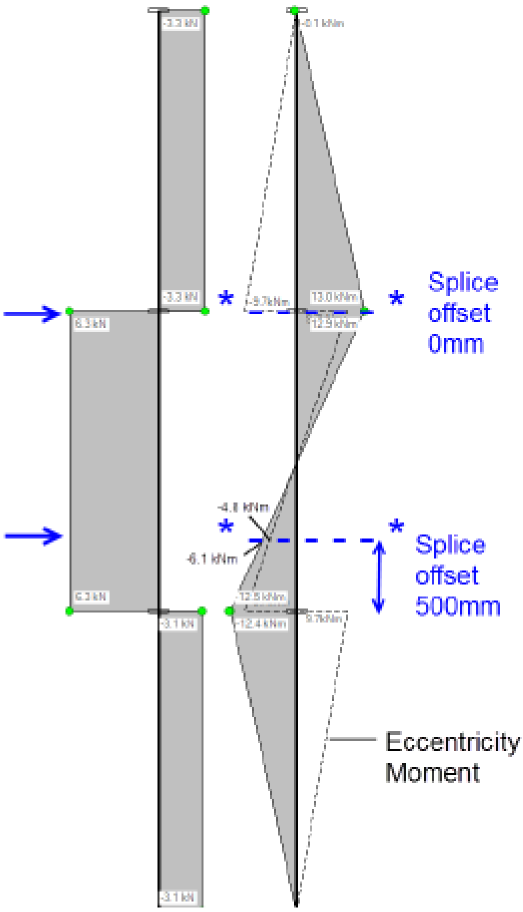
Column Splice Loads

Reference	Lift	Location	Axial Force [kN]	Shear [kN]		Moment [kNm]		Ecc. Moment [kNm]	
				Major	Minor	Major	Minor	Major	Minor
SC C/3	1	Bottom	123.0	-3.1	0.0	0.0	0.0	0.0	0.0
		Top	61.5	6.3	0.1	-6.1	0.0	-4.8	-0.8
	2	Bottom	61.5	6.3	0.1	-6.1	0.0	-4.8	-0.8
		Top	61.5	6.3	0.1	12.9	0.4	9.7	1.6
	3	Bottom	0.0	-3.3	-0.1	13.0	0.4	-9.7	-1.6
		Top	0.0	-3.3	-0.1	-0.1	0.0	0.0	0.0

Columns

Column Splice Loads

Reference	Lift	Location	Axial Force [kN]	Shear [kN]		Moment [kNm]		Ecc. Moment [kNm]	
				Major	Minor	Major	Minor	Major	Minor
SC C/3	1	Bottom	123.0	-3.1	0.0	0.0	0.0	0.0	0.0
		Top	61.5	6.3	0.1	-6.1	0.0	-4.8	-0.8
	2	Bottom	61.5	6.3	0.1	-6.1	0.0	-4.8	-0.8
		Top	61.5	6.3	0.1	12.9	0.4	9.7	1.6
	3	Bottom	0.0	-3.3	-0.1	13.0	0.4	-9.7	-1.6
		Top	0.0	-3.3	-0.1	-0.1	0.0	0.0	0.0



Steel brace design

Input method for A and V Braces

A and V Braces should be modelled using special tools which can be found on the 'Steel Brace' drop list in the 'Steel' section on the 'Model' tab.

Although it is also possible to model the exact same brace arrangement using individual elements created using the simple 'Steel Brace' command, it is important to note that whilst the 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 EHF calculations with the result that the calculated EHFs are not correct.

Steel truss design

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

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

Truss members can either be defined manually, or the process can be automated using the ***Truss Wizard***. Irrespective of the method used the resulting truss members will be one of four types:

- Internal
- Side
- Bottom
- Top

Depending on the type, different design procedures are adopted.

Internal and Side Truss Members

Internal and side truss members are designed as steel braces (for axial forces only).
See: Steel brace overview

Top and Bottom Truss Members

Top and bottom truss members are designed as steel beams, with the exception that seismic forces are not considered.
See: Steel beam design overview

Steel joist design

Steel joists are a specific type of member used in the United States. They are constrained to standard types specified by the US Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity.

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

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

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

Standard types

Steel joists are constrained to standard types specified by the Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity. There are four standard types of steel joist available in *Tekla Structural Designer*.

- K series joists -open web, parallel chord steel joists - depths 8" to 30" with spans up to 60ft.
- 2.5 K series joist substitutes - a depth of 2.5in, intended to be used for spans up to 10ft.
- KCS series joists - K series adapted and specially designed for constant moment/shear along length (position of point loads become irrelevant).
- LH series joists - long span joists - depths 18" to 48" for clear spans up to 96ft.
- DLH series joists - deep long span joists - depths 52" to 72" for clear spans up to 144ft.

Special Joists

"SP" suffixes can be added to K, LH and DLH Series joists. Special Joists can handle 'non-uniform' loading situations. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed. Load diagrams for the relevant joist can be output to forward to the fabricator for designing.

Joist Girders

These are provided as an option to support steel joists. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Joist Analytical Properties

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

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

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

Performing steel structure design

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

Gravity design

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

The gravity design stage enables you to design or check the simple beams, composite beams and Gravity only design simple columns for the designated gravity combinations (this will include the Construction Stage combination). Other members will also be designed or checked for these combinations but the resulting section sizes are less useful and are likely to require increasing in later stages of the analysis/design process. This approach is intended to speed up the design process.

Gravity design is initiated by clicking Design Steel (Gravity).

After the gravity design has been completed, by default all steel members are reset to check design mode. You may therefore decide to reset certain members to auto design e.g. columns and beams in 'moment frames'. In such cases, when the full design is performed member 'pre-sizing' will take place and for members resisting lateral loads this section size will be used if it is larger than that which resulted from the Gravity design.

Full design

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

Full design is initiated by clicking Design Steel (Static), or Design All (Static).

As part of the full design process a 3D Analysis is performed, for which you must select (via Design Options) the analysis type. The choice of analysis type will depend on the code being designed to. (See: Allowing for global second-order effects)

Concrete Design Handbook

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

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

Design Concrete

Features common to concrete beam, column and wall design

Analysis types performed in the Design Concrete process

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

3D Analysis

This analysis type is always performed.

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

Grillage chasedown analysis

This analysis type is also always performed.

FE chasedown analysis

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

Pre-design considerations

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

1. **Grouping** - decide if you want to make use of Design and detailing groups (concrete).

2. **Concrete Design Options** - check the concrete Design Options are appropriate for your design.

3. **Member properties and Autodesign settings** - review the design related properties that have been assigned to individual members.

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

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

Nominal cover

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

For beams and columns

The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

For walls

For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.

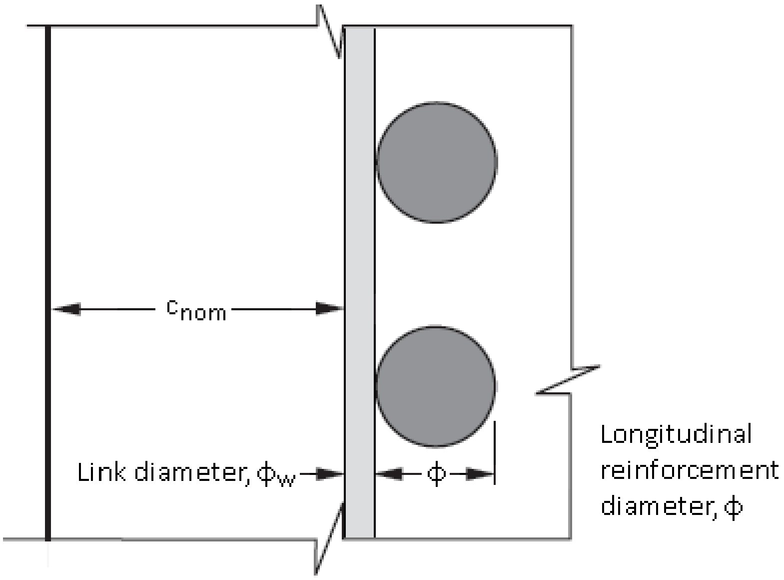
For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/confinement transverse reinforcement that may be present.

Eurocode: Minimum Cover

You are required to set a minimum value for the nominal cover, $c_{nom,\,u}$, which is then checked against the nominal limiting cover, $c_{nom,lim}$ which depends on the diameter of the reinforcement plus an allowance for deviation, Δc_{dev} (specified in **Design Options**).

Generally, the allowance for deviation, Δc_{dev} is a NDP. (Refer BS EN 1992-1-1:2004 cl 4.4.1.3 (1)P). The recommended value is 10mm, but under strict controls it can be reduced to 5mm..



Assume cracked

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

Assuming concrete sections are cracked has a direct affect on the analysis; smaller Use of Modification Factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway sensitivity calculations are also influenced by this assumption.

Design parameters

The following design parameters relating to shrinkage and creep can be specified individually as part of each member's properties set.

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 + ψ_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 ψ_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

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.

The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Reinforcement Parameters

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

Eurocode: Reinforcement Anchorage Length Parameters

Max. Bond Quality Coefficient

Acceptable input range 0.5 to 1.0

In the bond stress calculation (Cl 8.4.2), the bond quality coefficient η1 can be either 1.0 or 0.7 depending on section depth. Where 0.7 is used the bond strength is reduced and laps are extended.

Specifying a maximum of 1.0 for the Bond Quality Coefficient allows the coefficient to vary between 0.7 and 1.0 as required, hence lap lengths will vary accordingly.

Some users may prefer to specify a maximum of 0.7 (which actually fixes the coefficient at 0.7), the effect is to standardise on the use of extended lap lengths throughout. Further conservatism can be introduced in all lap lengths by using a value as low as 0.5.

Plain Bars Bond Quality Modifier

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for plain bars.

In *Tekla Structural Designer* a factor “T” has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for plain bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.5 would be reasonable.)

Type-1 Bars Bond Quality Modifier

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for Type 1 bars.

In *Tekla Structural Designer* a factor “T” has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for Type 1 bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.8 would be reasonable.)

Design and detailing groups (concrete)

Why use concrete design and detailing groups?

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

For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

A fixed set of rules are used to determine the automatic member grouping: for example beams must be of similar spans, columns must have the same number of stacks, bases must be of similar lengths in X and Y, and similar depths etc. The same rules also constrain manual group editing.

For design and detailing purposes - to reduce the processing time and also reduce the volume of output created.

In a manual process, the Engineer might select a number of sufficiently similar members to form a “design group” to carry out a single design that is sufficient for all members in the group. Using this single set of design results, they would then create sub-groups of the members in the design group to produce a set of output details for each of these sub-groups.

In *Tekla Structural Designer*, concrete design groups are analogous to the manually created design groups described above. Concrete detailing groups are analogous to the sub-groups.

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

What happens in the group design process?

When the option to design a specific concrete member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the reinforcement so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the reinforcement is adequate for each and if this proves not to be the case, it is increased as necessary and the revised reinforcement is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilisations are established.

Concrete design group requirements

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

Concrete beam design groups

- A beam element may be in only **one** design group.
- Design groups may be formed from single span or multi-span continuous beams.
- All beam elements in the group must have an identical number of spans.
- For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length.
- All beam elements in the group must have identical material properties and nominal cover.

- All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.

Concrete column design groups

- A column element may be in only **one** design group.
- All column elements in the group must have an identical number of stacks.
- For each individual stack all column elements in the group must have an identical cross section, and stack length.
- All column elements in the group must have identical material properties and nominal cover.

Pad base design groups

- A pad base may be in only **one** design group.
- Each base in the group must have an identical cross section and depth.
- Each base in the group must have identical eccentricities in X and Y.
- Each base in the group must have identical material properties and nominal cover.

Pile cap design groups

- A pile cap may be in only **one** design group.
- Each pile cap in the group must have an identical cross section and depth.
- Each pile cap in the group must have identical eccentricities in X and Y.
- Each pile cap in the group must have identical material properties and nominal cover.

Detailing group requirements

Each parent design group is sub-divided into one or more detailing groups.

Although there can be a “1 to 1” relationship between a design group and a detailing group, in practice there will often be a “1 to many” relationship as each design group is likely to require several detailing groups to allow for differences in the connected geometry.

Detailing groups are formed for the different concrete member types based on the following rules:

Concrete beam detailing groups

- A detailing group may be associated with only **one** parent design group.
- A beam element may be in only **one** detailing group.
- Detailing groups may be formed from single span or multi-span continuous beams.
- All beam elements in the group must have an identical number of spans.

- The cross section, including flange width where appropriate, span length and material properties in span
- *"i"* of **<u>all</u>** beam elements in the group **<u>must be identical</u>**.
- All beam elements in the group must have identical plan offsets.
- All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.
- All beam elements in the group must have identical inclinations.
- The support types and sizes, including the attached structure above and below the beam element, must be identical in all beam elements in the group however different support types and sizes in individual multi-span continuous beams are acceptable i.e. support *i* in beam element *j* must be identical to support *i* in all other beam elements in the group but supports *i* and *i+1* in beam element *j* may be different.

Concrete column detailing groups

- A detailing group may be associated with only one parent design group,
- A column may only be in one detailing group,
- All columns in the detailing group must have an identical number of stacks,
- All columns in the group must have an identical cross-section, rotation and alignment/snap levels/offsets in stack 'i'. In a multi-stack column, the cross-section may be different in each stack, i.e. the cross-section in span 'i' may be different to that in span 'j'.
- Stack 'i' and stack 'i+1' must be co-linear for all columns, OR must be non-co-linear with an identical degree of non-co-linearity for all columns. The exact inclination must be the same for stack 'i' in all columns.
- At every level each column is considered to be either "internal" or "external" (depending on if it has beams framing into it on all four sides, or not). These settings do not have to be identical for columns to be in the same group, but only if you have selected the option: **Provide ties through floor depth for internal columns** in **Design Options > Concrete > Column > Detailing Options**.

Pad base detailing groups

- A detailing group may be associated with only **<u>one</u>** parent design group.
- A base may be in only **<u>one</u>** detailing group.
- The attached column cross-section above the base must be identical for all bases in the group however different support types are acceptable.

Pile cap detailing groups

- A detailing group may be associated with only **<u>one</u>** parent design group.
- A pile cap may be in only **<u>one</u>** detailing group.
- The attached column cross-section above the base must be identical for all pile caps in the group however different support types are acceptable.

Group management

Automatic Grouping

Concrete beams and columns are grouped automatically.

In Model Settings the user defined **Maximum length variation** is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

Detailing groups cannot be edited manually.

> *When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.*

Regroup ALL Model Members

If you have made changes in Design Options that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking **Re-group ALL Model Members**.

> *Any manually applied grouping will be lost if you elect to re-group!*

Model Editing and Group Validity Checks

When new beam elements are created when a “split” or “join” command is run the resulting beam elements are automatically placed in existing design and detailing groups [or new groups created].

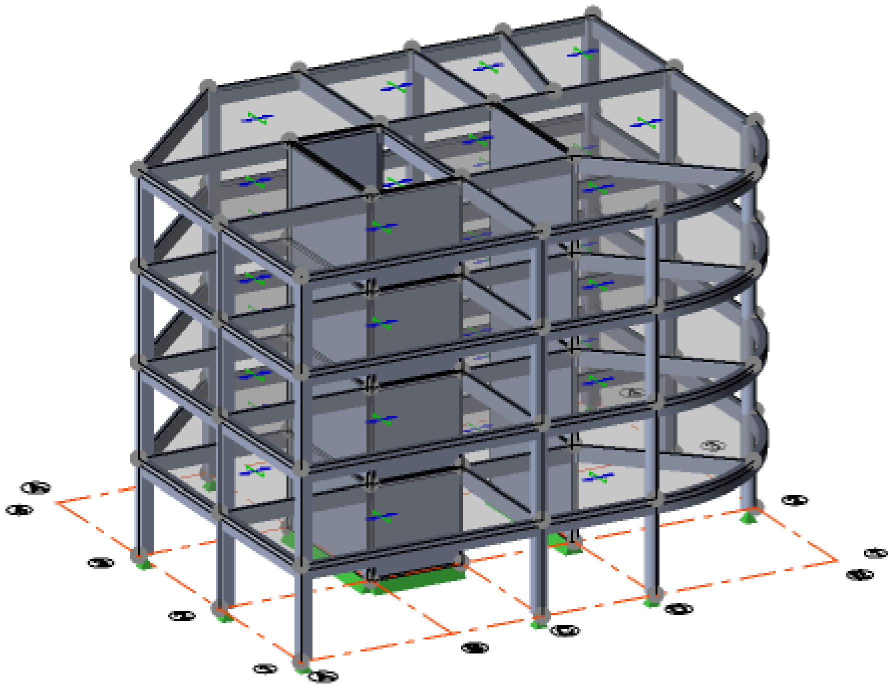
How is grouped design and detailing de-activated for concrete members?

1. Click **Design** > **Options...** ( )
2. Click **Design Groups**

3. Clear the check box adjacent to each concrete member type for which you want to de-activate design grouping.

Typical Design Concrete workflow

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

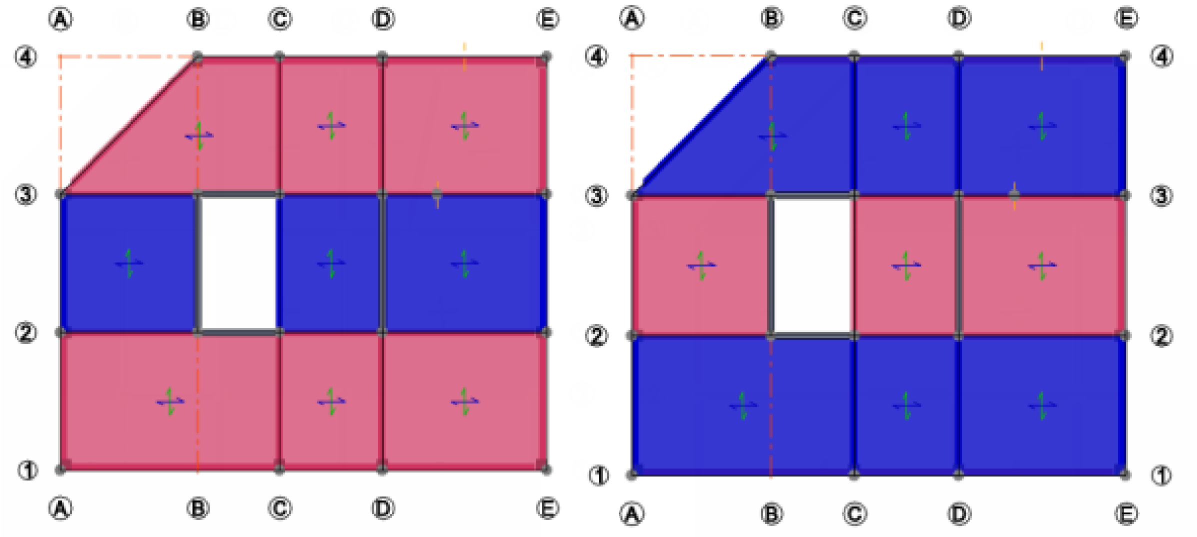


The example has been broken down into the following main steps:

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

Set up Pattern Loading

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Load Patterns** on the Load ribbon.

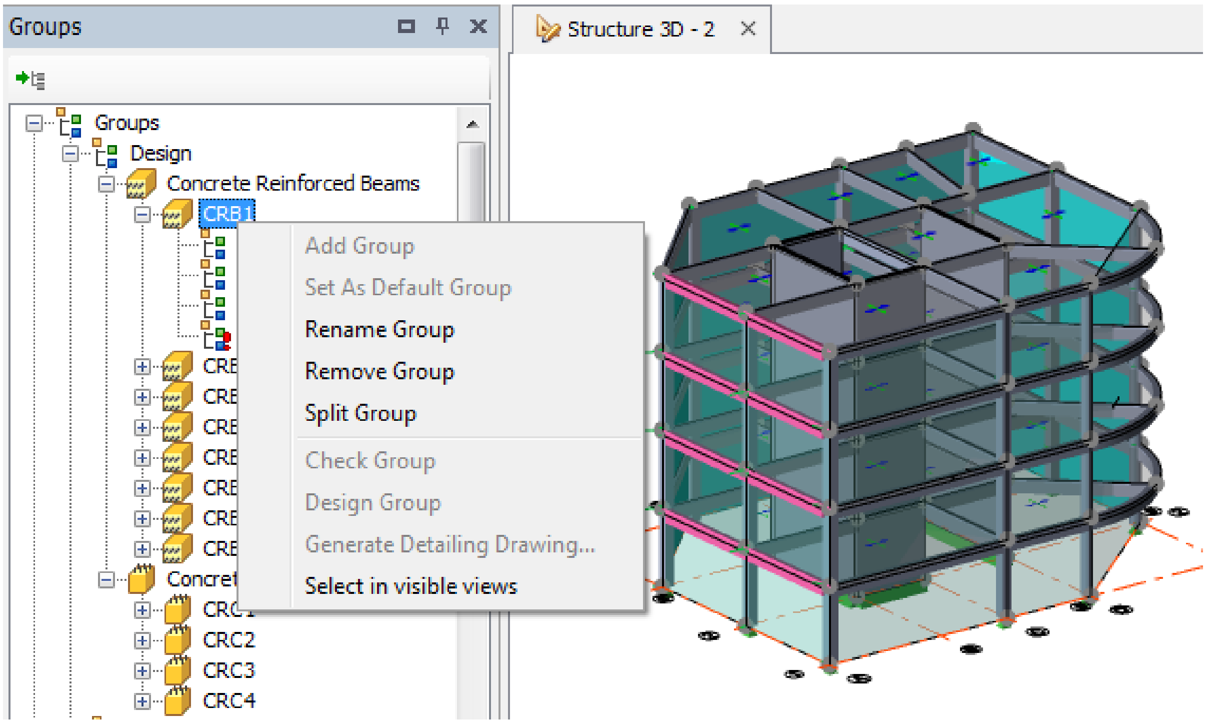


Set all beams columns and walls into autodesign mode

For the first pass, in order to get an efficient design at the outset, it is suggested that you set all members to “autodesign” with the option to select bars starting from **Minima**.

Review beam and column design groups

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



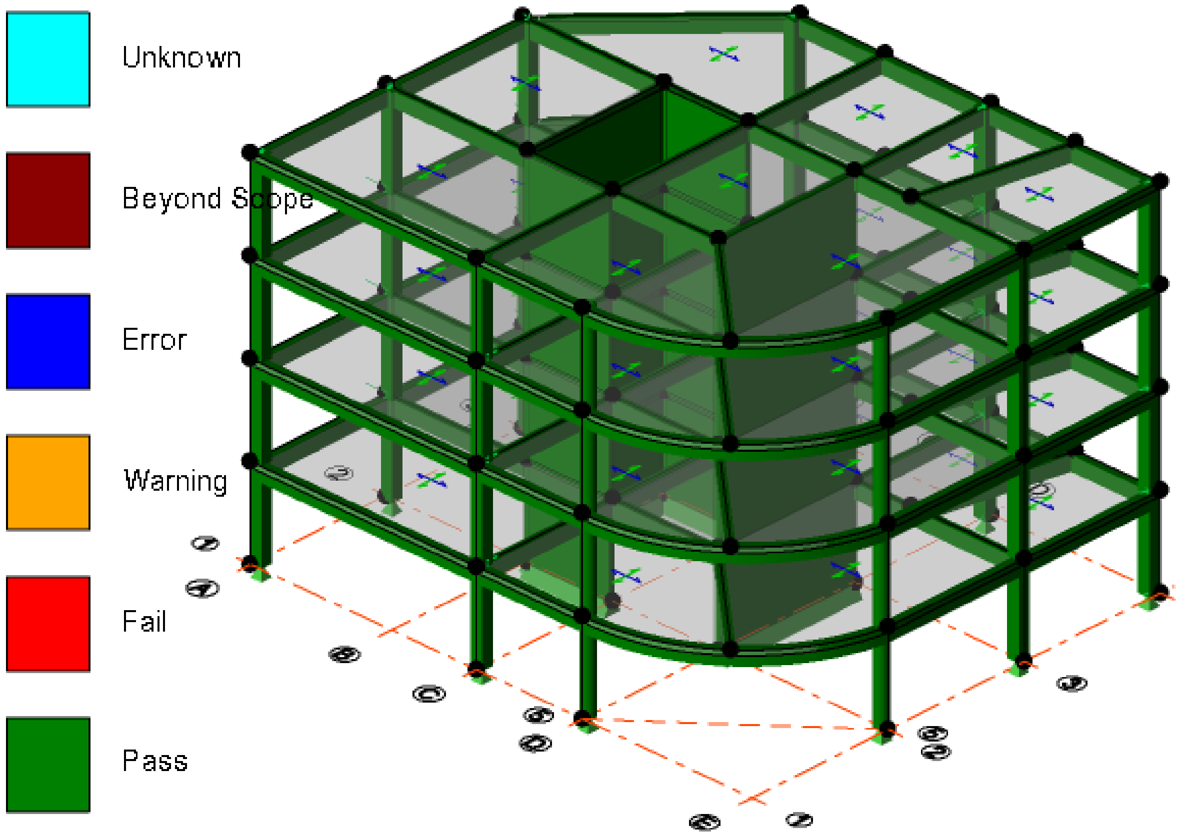
Groups will initially have been established for members sharing the same geometry, but you should consider reviewing and amending them if required.

Review beam, column and wall design parameters and reinforcement settings

The member design parameters and reinforcement settings should be carefully considered prior to running the design.

Perform the concrete design

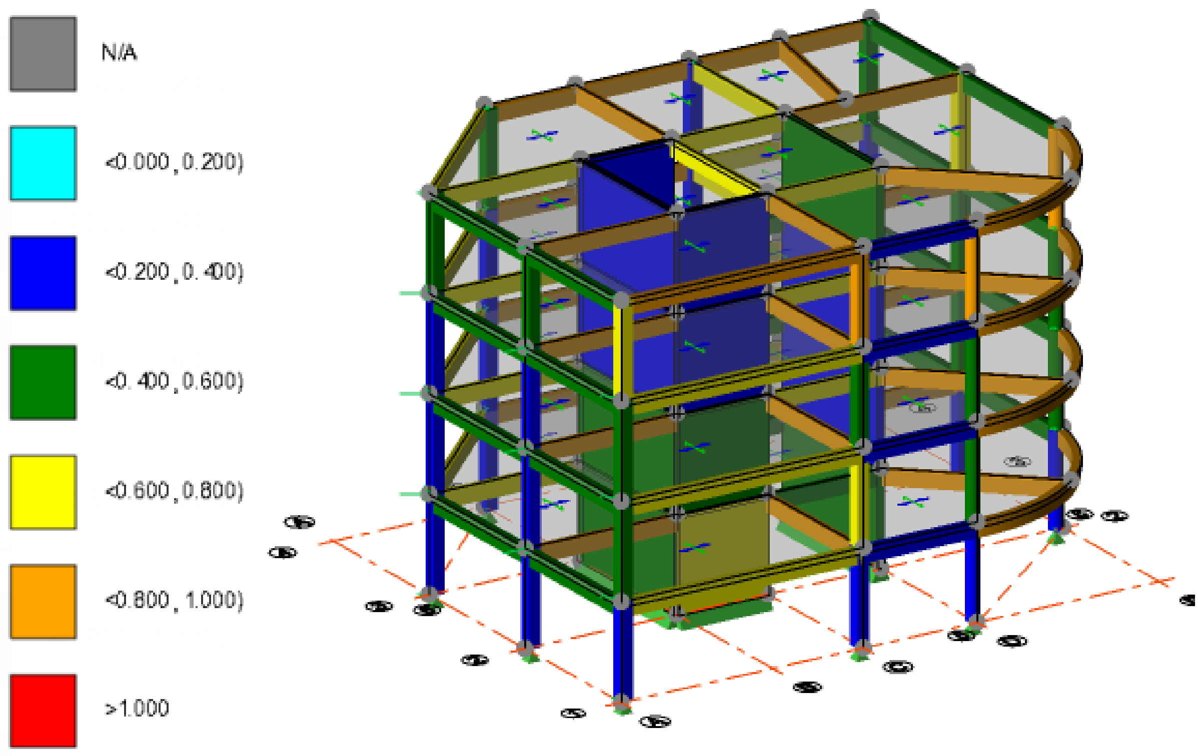
Concrete beams, columns and walls are designed by running **Design Concrete (Static)** from the Design ribbon.



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

Review the design status and ratios

You can display the Design Status and Ratios from the **Review View** in order to determine if any remodelling is required.



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

If you make any changes, to see their effect simply re-run **Design Concrete (Static)** once more.

Create Drawings and Quantity Estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

Print Calculations

Create a model report that includes the member design calculations that have been performed. (The default Member Design Calcs report includes these along with design calculations for other member types in the model).

Reviewing Design Concrete and refining the design of individual members

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

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

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

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

Features of concrete beam design

Analysis types used for concrete beam design

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

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

Autodesign (concrete beam)

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

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

- When **Autodesign** is selected an iterative procedure is used to select longitudinal bars for each bending design region on the beam, both top and bottom. Similarly an iterative procedure is used to select links for each shear design region on the beam.
- When **Autodesign** is not selected (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient.

Select bars starting from

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

The options are:

- Minima (default)
- Current bars

Selecting **Minima** removes the current arrangement and begins with the minimum allowed bar size from the selection order.

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

Rationalisation of Reinforcement

The Auto-design process returns a set of information about the reinforcement to be provided in each design region of the beam. The number and size of the longitudinal bars in the top and bottom of the beam is given as well as the size, number and spacing of the shear links.

This information is then "rationalied" to give an arrangement of longitudinal reinforcement that provides a solution for the beam as a whole whilst still meeting the requirements of the individual design regions.

The rationalisation process is carried out separately for the longitudinal bars in the top of the beam and those in the bottom of the beam.

The arrangement of shear links is not rationalied. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

Deflection control

Tekla Structural Designer controls 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.

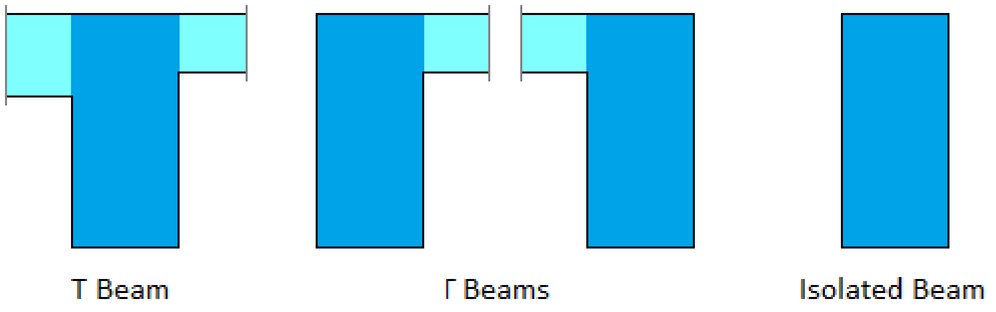
2. Consider flanges

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

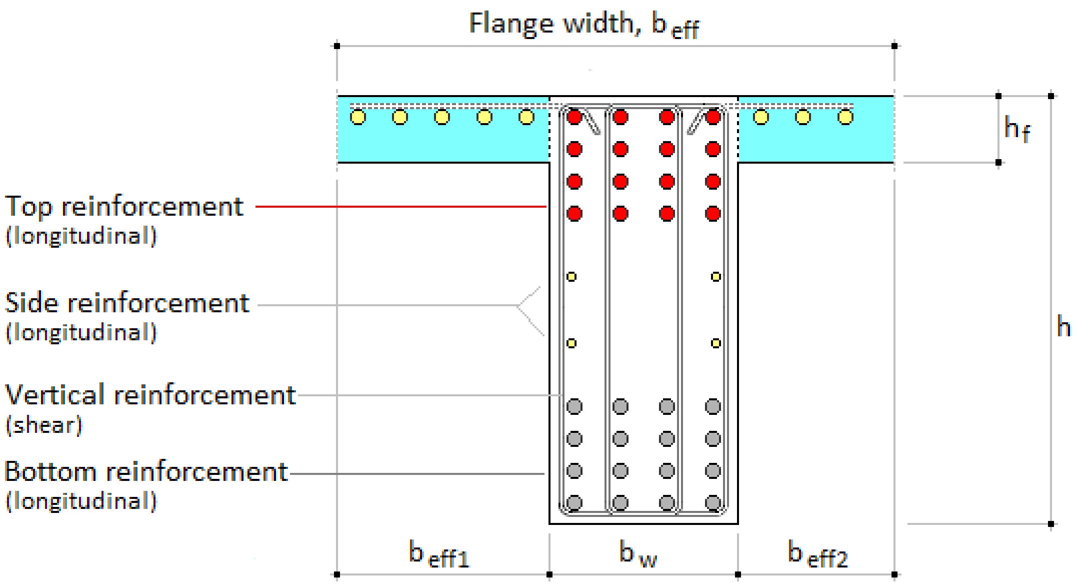
Use of beam flanges

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

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



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



where

h = overall depth including the depth of the slab

h_f = depth of slab

b_w = width of beam

b_{eff1} = flange width side 1

b_{eff2} = flange width side 2

b_{eff} = flange width

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

Use of Flanged Beams

Flanged beam properties can be specified under the **Design Control** heading in beam properties, by selecting **Consider Flanges**.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "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.

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.

Effective Width of flanges

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

= $MIN[0.2*L_0, b_i, (0.2*b_i + 0.1*L_0)] - O_{wi}$

where

L_0 = the distance between points of zero moment as defined above

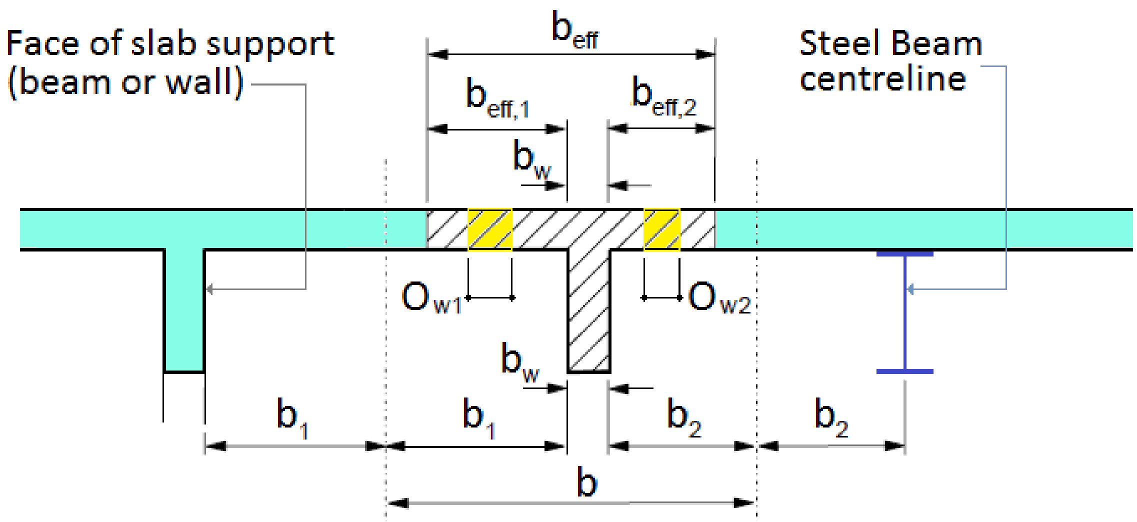
b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centreline of any supporting steel beam

b_w = the width of the beam

O_{wi} = the user specified allowance for an opening

If the slab thickness varies on each side of the beam, the thinner value is used in calculating the beam properties.

This relationship is illustrated below



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.

Adjacent Beams not Parallel

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

Holes/Openings in Calculated Effective Flange Width

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

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

User defined effective flange width and depth

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

Summary of Flange Modelling and Design Choices

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

Consider flanges	Include flanges in	User defined flanges	Outcome

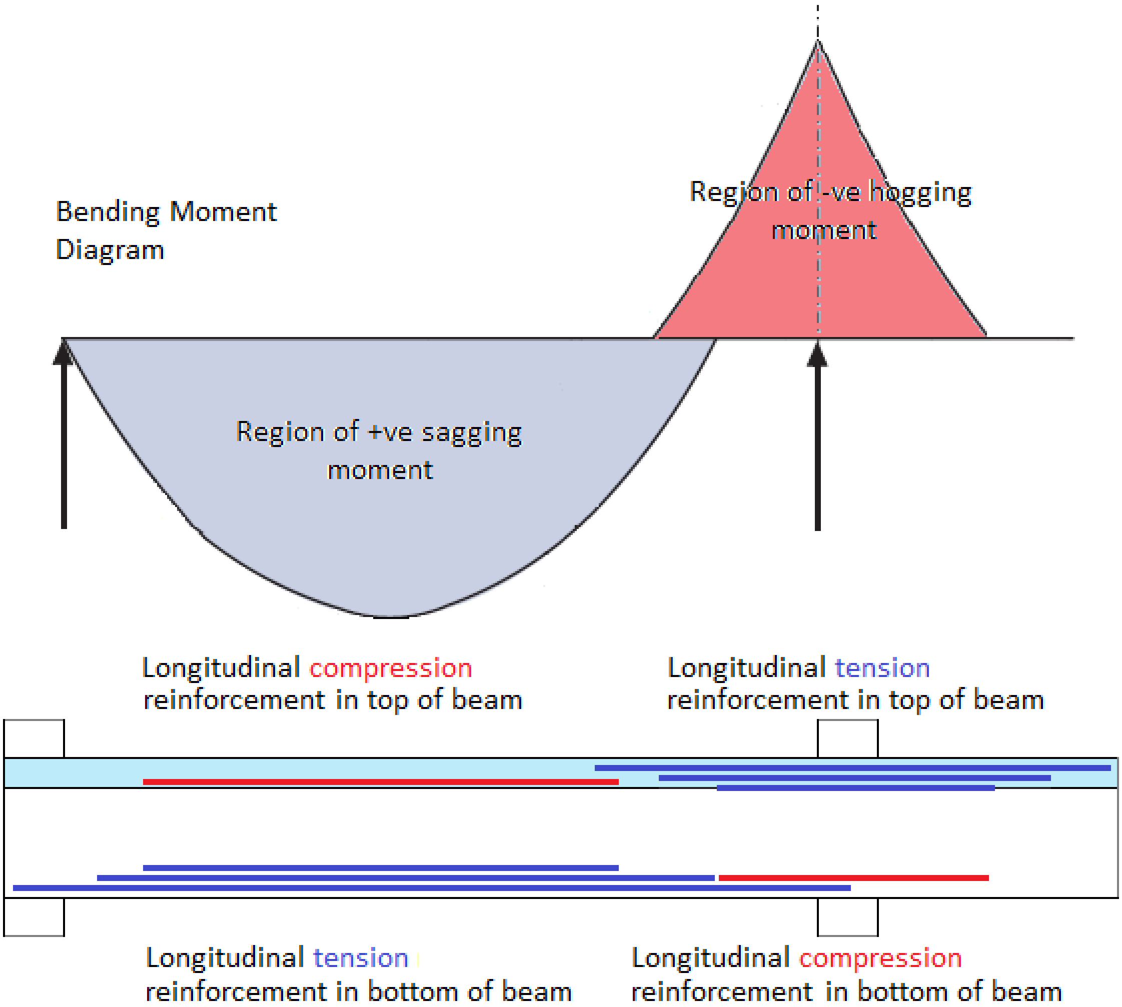
	analysis		
off	N/A	N/A	The beam is analysed and designed as a rectangular beam.
on	off	off	The beam is analysed as rectangular, effective flanges (as defined by the head code) are used in the design.
on	off	on	The beam is analysed as rectangular; user defined flanges are used in the design.
on	on	off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

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

Longitudinal reinforcement

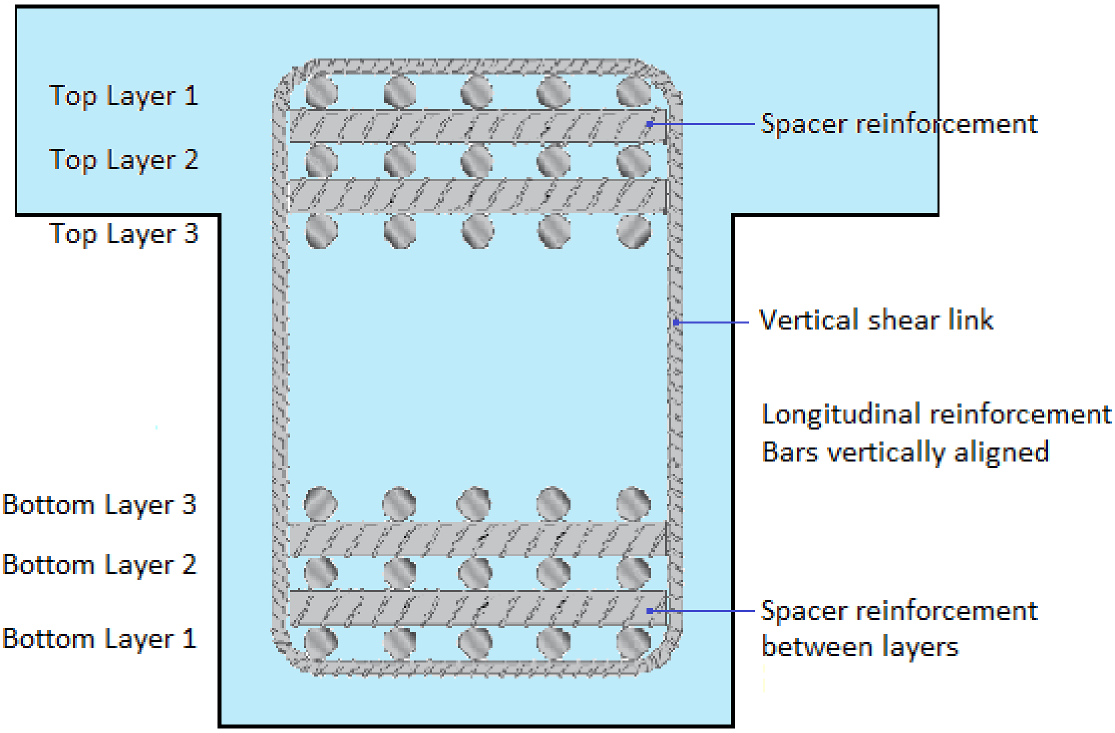
Bar layers

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



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

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



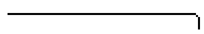
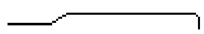
Longitudinal reinforcement in the side of the beam is only provided in beams with a depth greater than a certain value as follows:

h ≥ 1000 mm EC2

Longitudinal Reinforcement Shapes Library

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

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

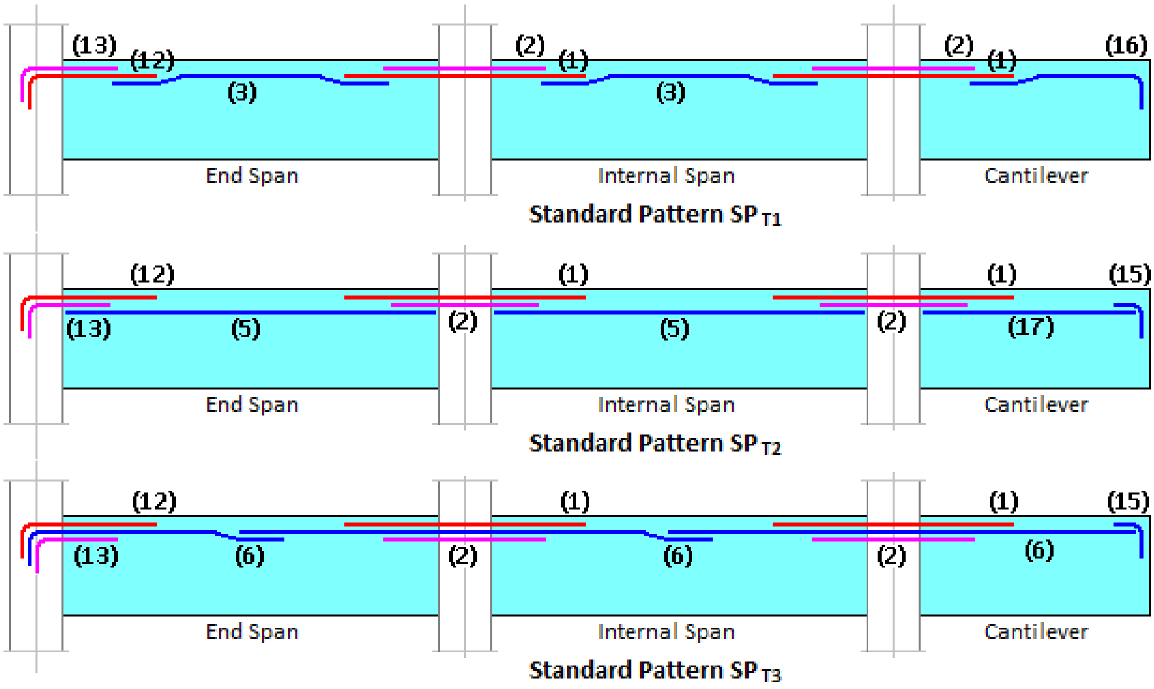
BS8666 Shape Code	Bar Shape	Description
00		Straight bar
26		Single crank
46		Double crank
11		Standard bob
34		Standard bob with crank

11		Extended bob
34		Extended bob with crank
21		U bar
99		U bar with crank

Longitudinal Reinforcement Patterns Library

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

Standard Patterns of Top Reinforcement



The bars used in the Standard Top Patterns are:

(1) Straight bar extending to approximately 25% of each span (end points of this bar are determined by the design region settings)

(2) Straight bar extending to approximately 10% of each span (end points of this bar are determined by the design region settings) – if required by the design

(3) Double cranked bar lapped with bar (1)

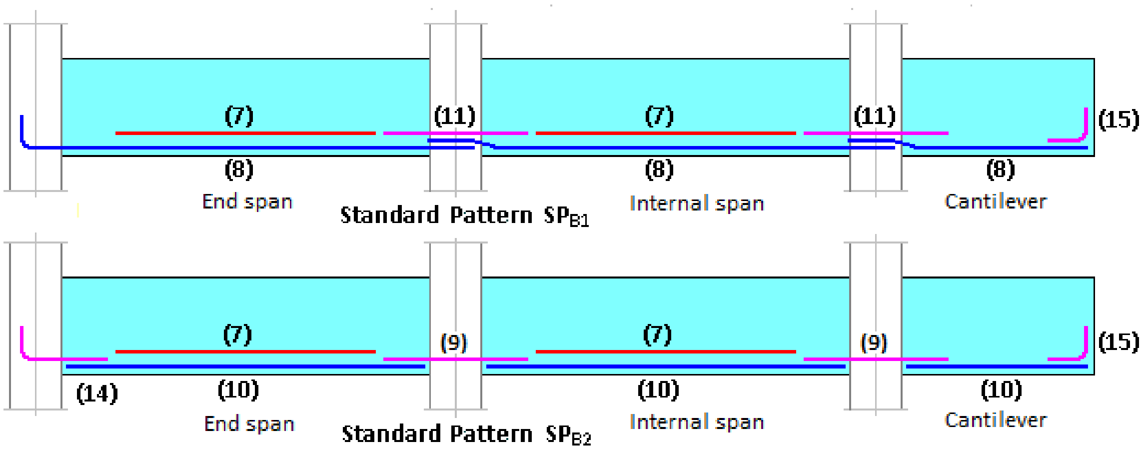
(5) Straight bar running approximately from face to face of beam supports

(6) Single cranked bar running from centre span to centre span with the option to merge bars if they are the same size and number to extend the bar over several spans

(12) Bob bar

(13) Bob bar

Standard Patterns of Bottom Reinforcement



The bars used in the Standard Bottom Patterns are:

(4) Bar with a bob at each end

(7) Straight bar with a length approximately 70% of span – if required by the design

(8) Single cranked bar extending over several spans or over one span only and lapped within a support – with bob if it continues over an end span.

(9) Straight bar

(10) Straight bar running approximately from face to face of beam supports

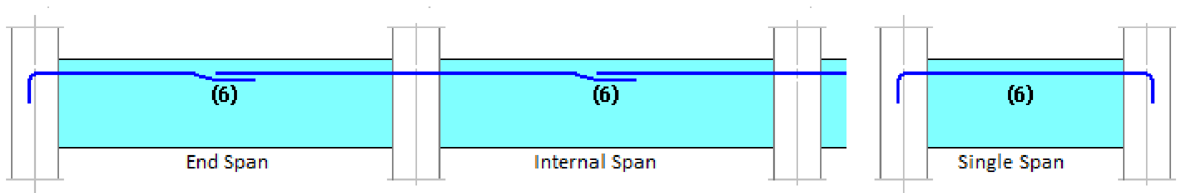
(11) Straight bar

(14) Bob bar

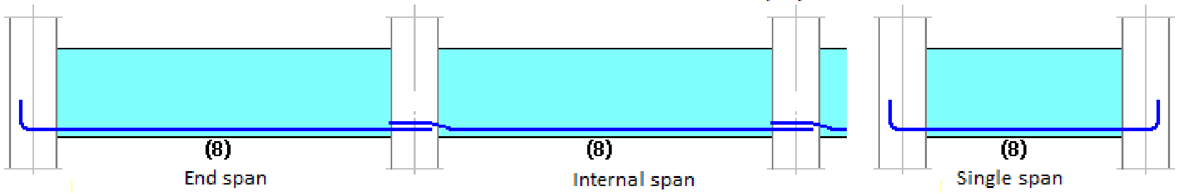
Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in **Design Options > Beam > Reinforcement Settings** and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams



Short Span Standard Patterns SP $_{T1, T2, \& T3}$



Short Span Standard Patterns SP $_{B1 \& B2}$

Longitudinal Reinforcement Regions

Design Check Regions for Bending

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

Top Regions

Three standard patterns are available for defining the top regions:

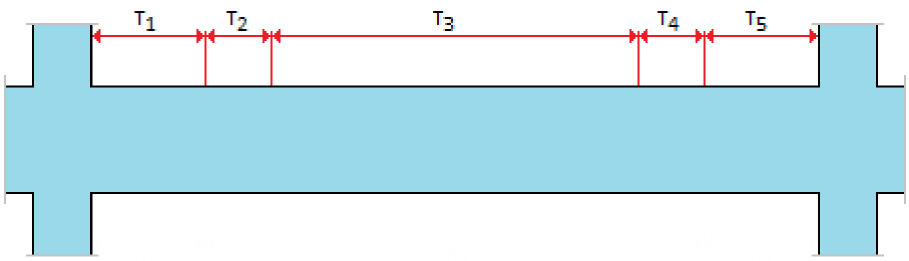
Standard Top 1 – maximum of 3 regions

Standard Top 2 – maximum of 5 regions

Standard Top 3 – maximum of 6 regions

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

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



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

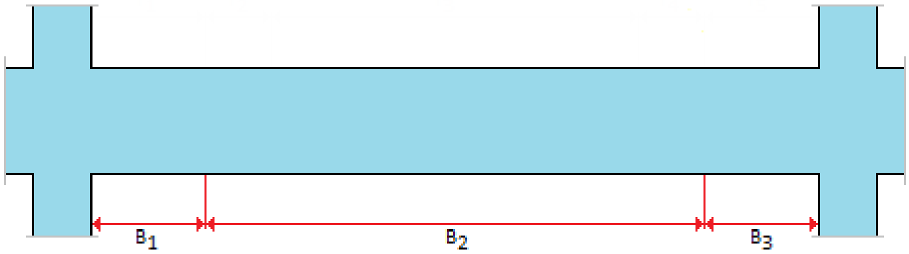
1 Region T1=T5 = 0%; T2=T4= 0%; T3= 100%

3 Regions T1=T5 = 0%; T2=T4 = 100%-T3%; 0% < T3 < 100%

5 Regions T1=T5 > 0%; T2=T4 > 0%; T3> 0%.

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

Bottom Regions



Two standard patterns are available for defining the bottom regions:

Standard Bottom 1 – maximum of 3 regions

Standard Bottom 2 – maximum of 3 regions

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

This enables the following single region, or three possibilities:

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

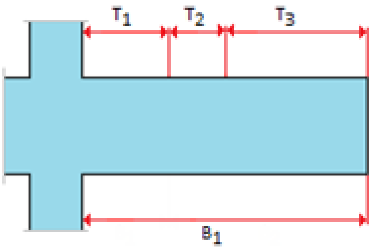
3 Regions B_1=B_3 = RANGE(0%-25%); B_2 = RANGE(50%-100%)

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

Regions for Cantilevers

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

These regions are illustrated below



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

Relationship between Reinforcement Patterns and Design Regions

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

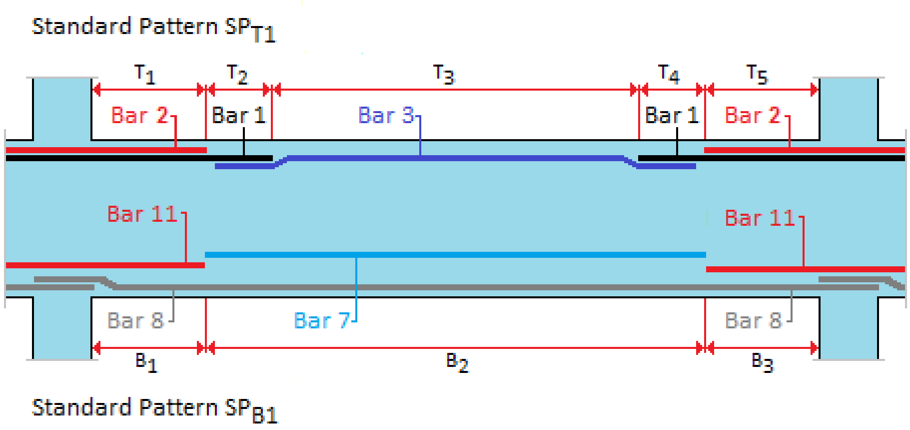
The selection process of Standard Reinforcement Patterns and Design Regions is:

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

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

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

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



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

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

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

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

Design Region T_2 (& T_4) : Bar 1

Design Region T_3 : Bar 3

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

Design Region B_2 : Bar 7 + Bar 8

The above approach is extended for all the Standard Patterns.

Shear reinforcement

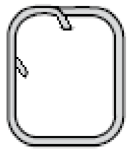
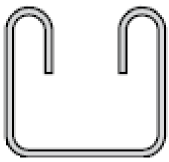
Shear Reinforcement Shapes Library

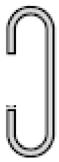
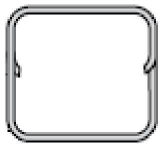
Vertical shear reinforcement is provided in the form of links which can be single or multiple with 1 ( ) or 2 (or) vertical legs.

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

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

Shear Reinforcement Typical Shapes

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

99		Single Leg Link/Stirrup
63		Torsion Link/Stirrup

Shear Reinforcement Patterns Library

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

The Standard Patterns for shear reinforcement are:

Closed

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

Open

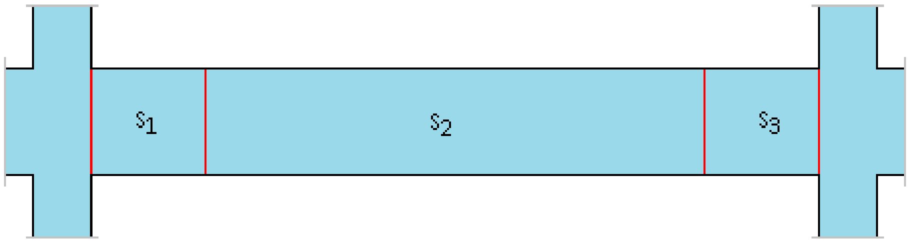
Open links with top closers (shape codes 47 and 21) with additional double leg (shape code 47) or single leg links (shape code 99) if required by the design. The design dictates the number of vertical legs required. You can choose if single leg links are acceptable.

Torsion

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

Shear Reinforcement Regions

When considering shear, the design shear checks are performed in each of 3 regions S_1, S_2 and S_3 as shown below. In each region, the maximum vertical shear from all load combinations and analysis types, V_{zi}, is determined and this maximum value used to determine the shear reinforcement required in that region.



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

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

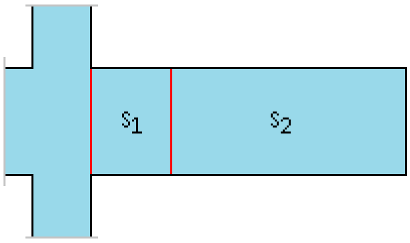
In this case in the central region S_2, shear reinforcement is provided to meet the maximum of the minimum code requirement or minimum user preference whilst in regions S_1 and S_3, designed shear reinforcement is required.
The position and length of region S_2 is determined from considerations of the shear resistance of the concrete cross-section and this then enables the lengths of regions S_1 and S_3 to be determined.

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

Or

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

In cantilevers, the regions are as shown below.



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

Features of concrete column design

Autodesign (concrete column)

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

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

- When **Autodesign** is selected an iterative procedure is used to design both the longitudinal bars and links. This applies the spacing maximisation method which attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link spacing.
- When **Autodesign** is not selected (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient.

Select bars starting from

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

The options are:

- Minima (default)
- Current bars

Selecting **Minima** removes the current arrangement and begins with the minimum allowed bar size from the selection order.

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

Stacks and reinforcement lifts

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

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

A reinforcement "lift" is defined as the column or wall height between two levels anywhere in the building throughout which the cross-section and the reinforcement arrangement is constant. For the cross-section to be constant, all aspects of the shape, dimensions and rotation must be identical. Reinforcement changes in one stack in a reinforcement lift would apply to all stacks in the reinforcement lift.

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

Column design forces

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

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

Features of concrete wall design

Autodesign (concrete wall)

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

- When **Autodesign** is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and links. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link spacing.
- When **Autodesign** is not selected (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient.

Select bars starting from

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

The options are:

- Minima (default)
- Current bars

Selecting **Minima** removes the current arrangement and begins with the minimum allowed bar size from the selection order.

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

Autodesign mode.)

Wall design forces

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

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

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

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

Concrete slab design

Features of concrete slab analysis and design

Slab on beam idealized panels

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

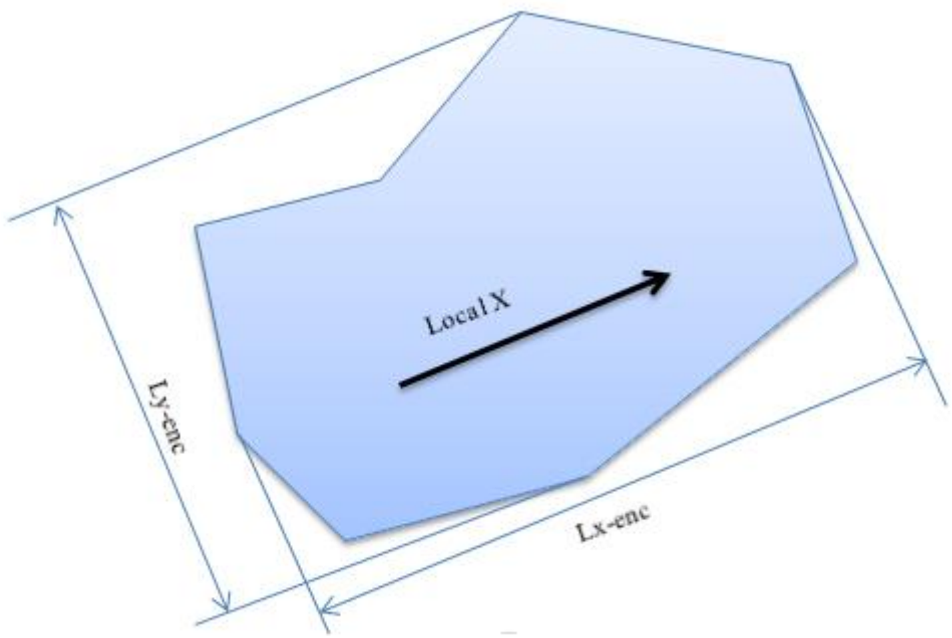
To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel rotation angle):

Lx-enc = maximum overall length of the panel measured parallel to local X

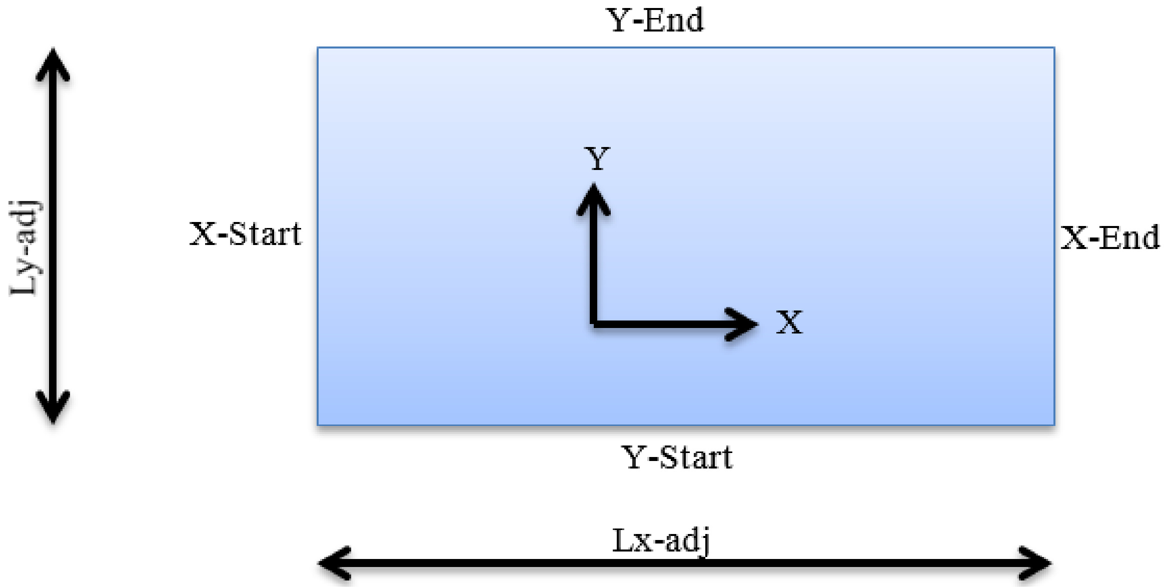
Ly-enc = maximum overall length of the panel measured perpendicular to local X

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)

Typical flat slab design procedure

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

Overall Slab Design Workflow

1. Split/join panels as necessary and set up Pattern Loading

2. Analyse All (or Design All) - to establish analysis results

3. Consider Simple (linear) Deflection

4. Select a Level (or sub-model) to be designed and within that level:

    a. Add Patches
    b. Design Panels
    c. Review/Optimise Panel Design
    d. Design Patches
    e. Review/Optimise Patch Design
    f. Add and Run Punching Checks

5. Rigorous Deflection Check

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

6. Move to next level or sub-model and repeat steps 4 and 5.

7. Create Drawings and Quantity Estimations

8. Print Calculations

Typical slab on beams design procedure

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

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

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

1. Set up Pattern Loading

2. Design All - to establish analysis results

3. Select a Level (or sub-model) to be designed and within that level:

    a. Add Beam and Wall Top Patches
    b. Design Panels
    c. Review/Optimise Panel Design
    d. Design Beam and Wall Patches
    e. Review/Optimise Beam and Wall Patch Design

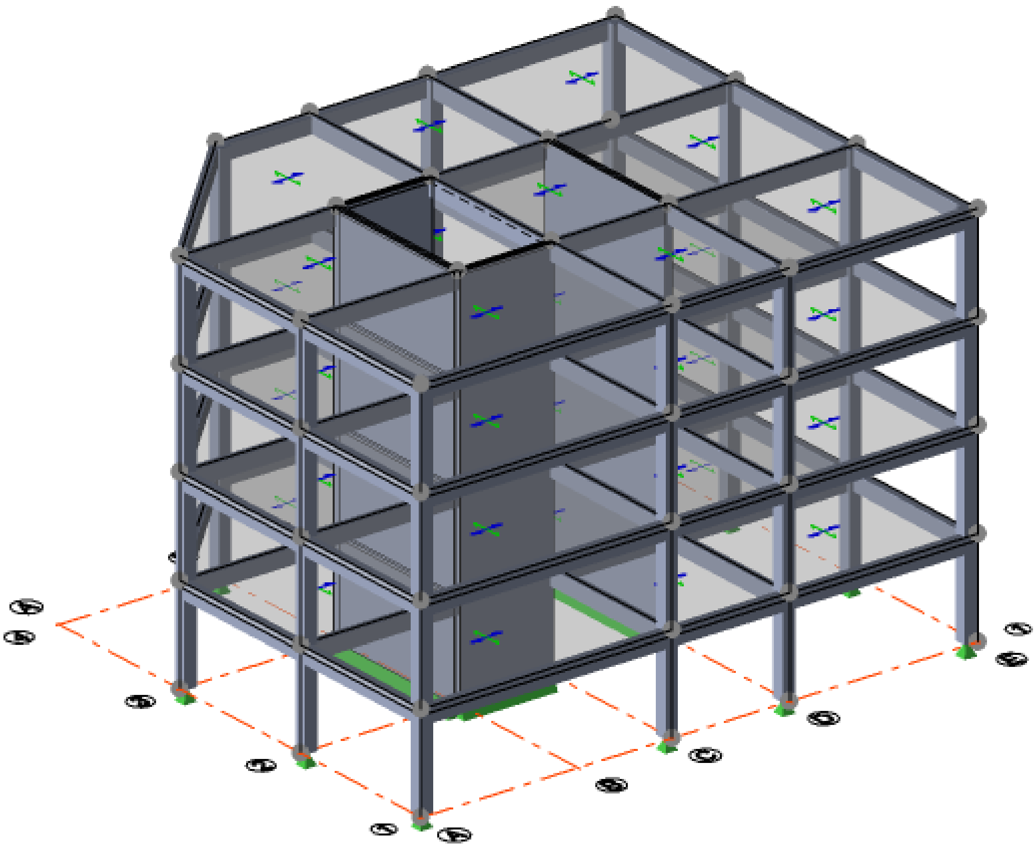
4. Move to next level or sub-model and repeat step 4.

5. Create Drawings and Quantity Estimations

6. Print Calculations

Slab on beam design example

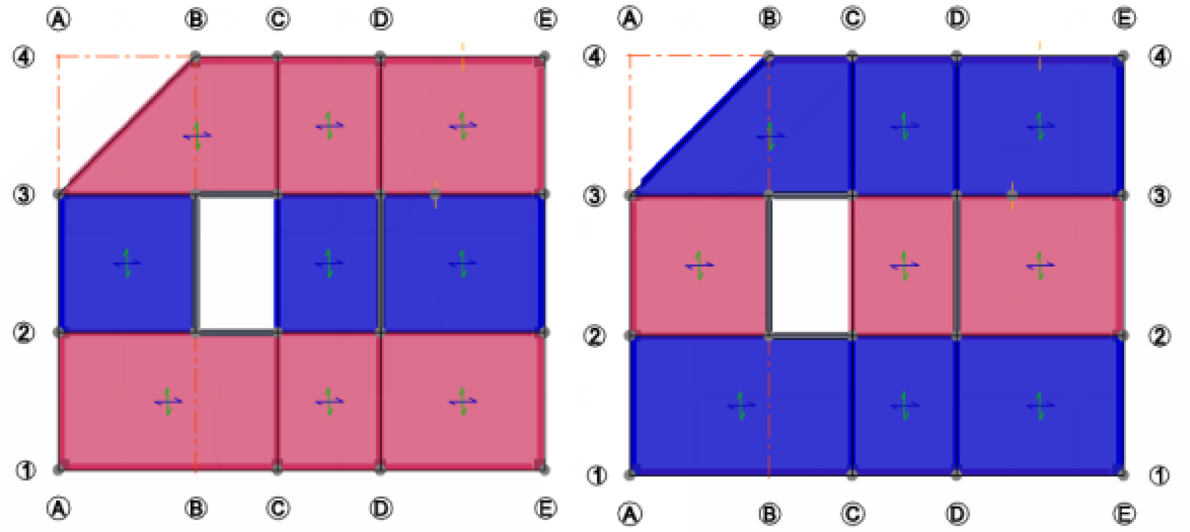
A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.



Note that not all the slab panels are rectangular.

Set up Pattern Loading

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to meshed slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Load Patterns** on the Load ribbon.



Design All

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

Typically these moments are taken from the FE Chasedown model results - as each floor is analysed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however:
- you may choose to mesh them to cater for the possibility of un-braced flat slab design.
- more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Select a Level

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the “3D geometry” of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

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

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

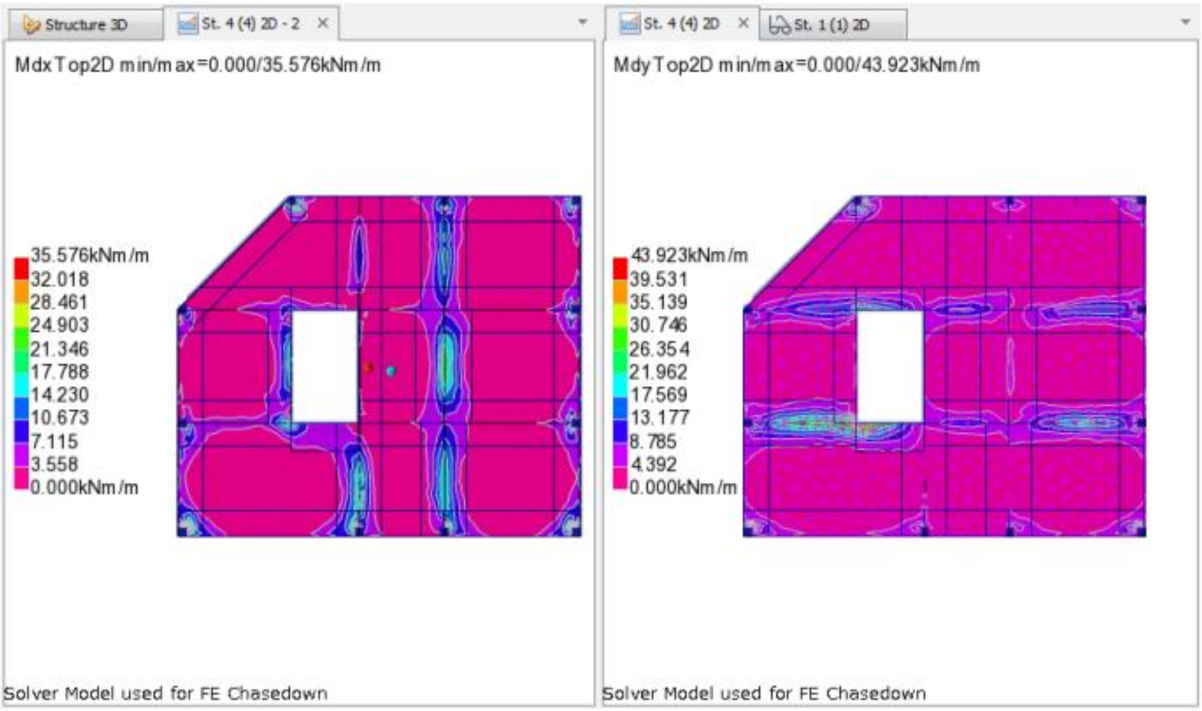
Add Beam and Wall Top Patches

You may optionally want to add beam and wall “top surface” patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to **none** and the panel design should still pass.

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating beam patches as discussed below).

It is suggested that you add patches in the **Results View** while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mdx top moments on in one view on the left and Mdy top moments in a second view on the right, as below:



By doing this, it is possible to see how patches extend over the moment contours.

It is suggested that at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimisation stage.

Design Panels

Slab on beams panel design takes account of any beam or wall patches (by excluding the patch areas from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

1. From the Design ribbon run **Design Slabs** in order to design or check all the panels in the model - by default newly created panels will all be in “auto-design” mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right-click and choose either **Design Slabs** or **Check Slabs**. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

These right-click options operate on the same basis as the options for beams and columns:
- **Design Slabs** *will re-design the slabs (potentially picking new reinforcement) regardless of the current autodesign setting.*
- **Check Slabs** *will check the current reinforcement in the slabs regardless of the current autodesign setting.*

3. If you chose to set the top reinforcement in the slab panels to **none** but the design detects that top reinforcement is required, the affected panels will fail. In this situation you should increase the widths of the adjacent beam or wall patches before checking or designing the slab panels once again.

4. When panels are being designed (as opposed to checked), the design does not currently automatically increase reinforcement to satisfy deflection, in which case the panels might fail. In this situation you could manually increase the reinforcement until deflection is satisfied.

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

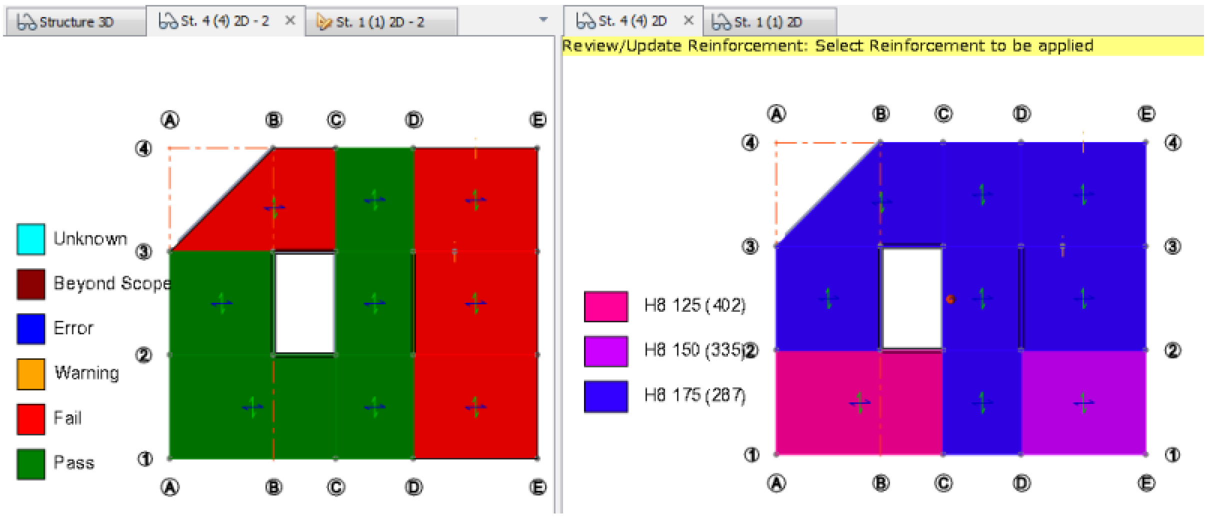
As part of the design process a span-effective depth check is performed in the appropriate span direction. If the span is the same in both directions, the more “continuously supported” direction is checked, or if both are equally supported the direction which is set to be the outer bar layer is checked.

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

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

Review/Optimise Panel Design

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



The view on left shows **Slab Design Status**, (with slab patches turned off in Scene Content to assist clarity), the view on right shows **Slab Reinforcement** in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.).

Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a *"minimum spacing (slab auto design)" = 150mm.*

After using the **Review View** update mode to standardise reinforcement you can then run **Check Slabs** from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between **Status** and **Ratio** views - if utilisations all < 1 but some panels failing then problems are to do with limit checks. The **Ratio** view is better for helping you focus on the critical panels.

Design Beam and Wall Patches

Having established and rationalied the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design any beam or wall patches that you may have defined.

To do this, either:

1. From the Design ribbon run **Design Patches** in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right-click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

These right-click options operate on the same basis as the options for beams and columns:
- **Design Patches** *will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.*
- **Check Patches** *will check the current reinforcement in the patches regardless of the current autodesign setting.*

Review/Optimise Beam and Wall Patch Design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimised)?
- Wall patches - can the width be adjusted (minimised)?
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

- Now click **Slab Reinforcement** in the **Review View** to review and standardise the patch reinforcement.

Create Drawings and Quantity Estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

Print Calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Interactive concrete member design

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.

> *Iterative cracked section analysis is currently only available for the Eurocode and ACI/AISC Head Codes.*

- Slab Deflection Methods
- Rigorous Slab Deflection Workflow
- Slab Deflection Parameters
- Event Sequences in Depth
- Slab Deflection Analysis
- Slab Deflection Types
- Check Lines in Depth
- Slab Deflection Status and Utilization
- Slab Deflection Optimisation
- Slab Deflection Calculations in Depth
- Slab Deflection Example (Eurocode)

Related video

- *Rigorous slab deflection (Eurocode)*

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 behaviour is characterised 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.

Two alternative methods for checking deflections exist. Either:

- Deemed-to-satisfy checks or

- A theoretical rigorous deflection check.

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 and deflection affecting sensitive finishes is expected to be < span / 500

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 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 handbook.

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 recognise 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 various input parameters - this is illustrated in the Sensitivity guidance (Eurocode) topic which shows the impact of the different parameters on the deflection.

Rigorous Slab Deflection Workflow

Rigorous Slab Deflection Analysis in *Tekla Structural Designer* is available on the Slab Deflection toolbar.

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. See: Overall Slab Design Workflow

After the slab reinforcement has been established a suggested workflow for performing Slab Deflection Analysis is as follows:

1. Define the slab loading event sequences.
   See: Event Sequences in Depth

2. Run the slab deflection analysis and review deflections for the final load event.
   See: Slab Deflection Analysis

You can analyse the current or selected level (sub-model) or choose to analyse 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 and place check lines as required.
   See: Check Lines in Depth

4. Graphically review the slab deflection status.
   See: Slab Deflection Status and Utilization

5. Make adjustments as necessary until the slab deflection status passes.
   See: Slab Deflection Optimisation

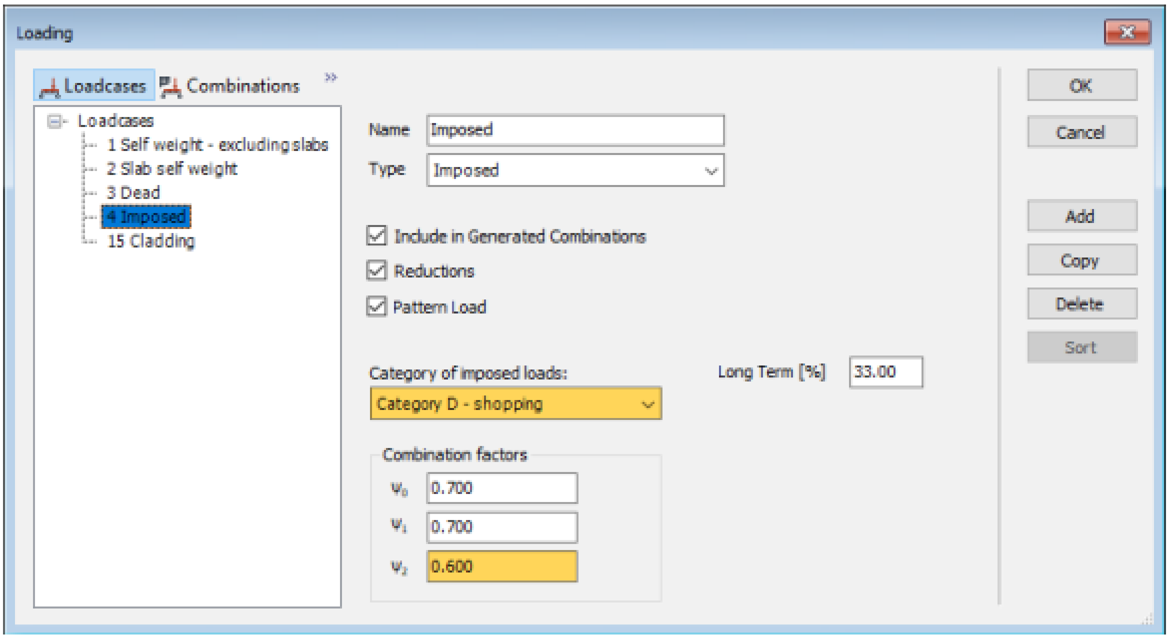
Slab Deflection Parameters

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

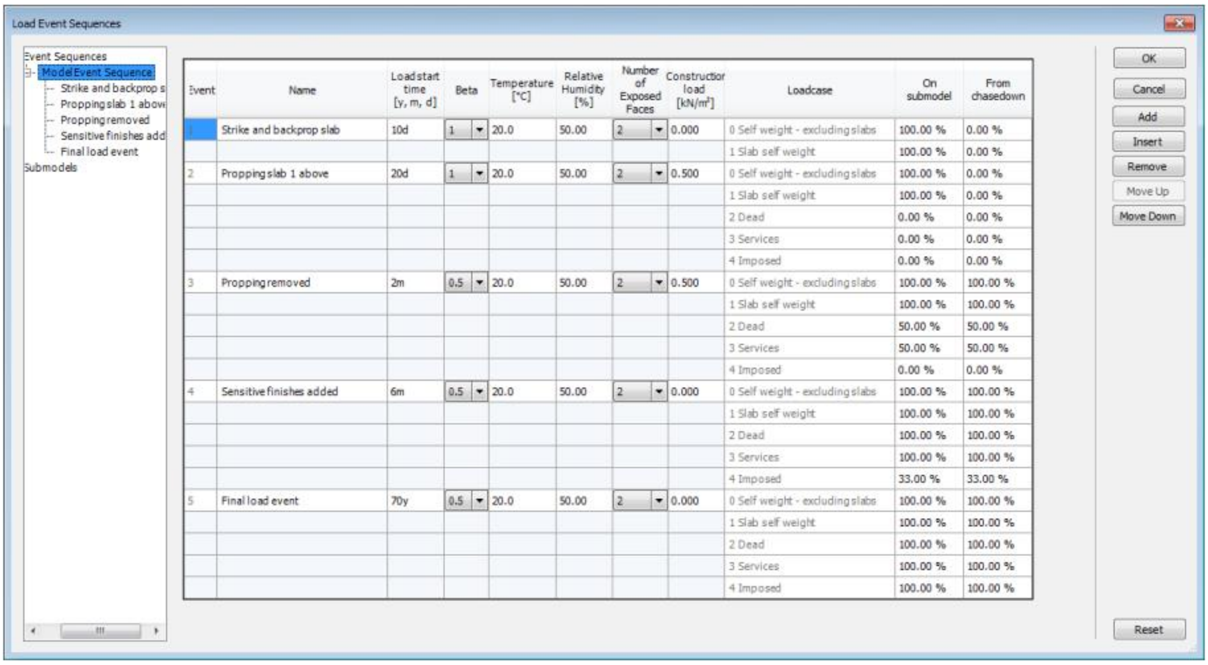
Quasi-permanent load factors

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

This property is set in the Event Sequence dialog.



Beta relates to the duration of the load and tension stiffening effects.

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.

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:

i) - 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.

ii) - 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.

iii) - 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.

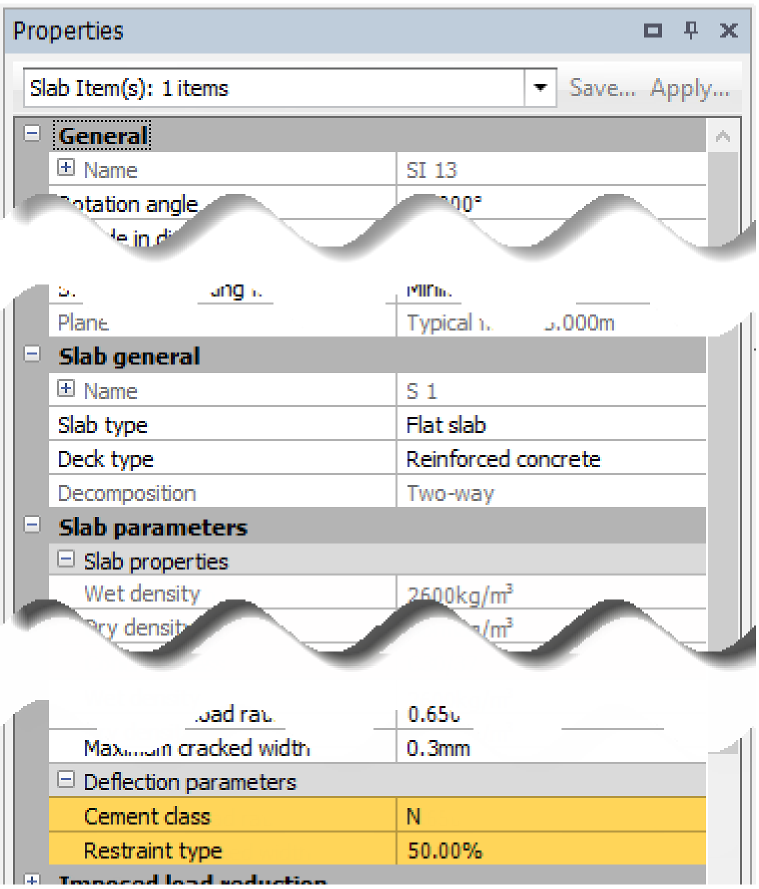
In summary, it is prudent to consider whether cracking is likely to increase in later events for which you wish to set Beta=1.0.

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

This property is located under the Deflection parameters heading in the slab item Properties Window.



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.

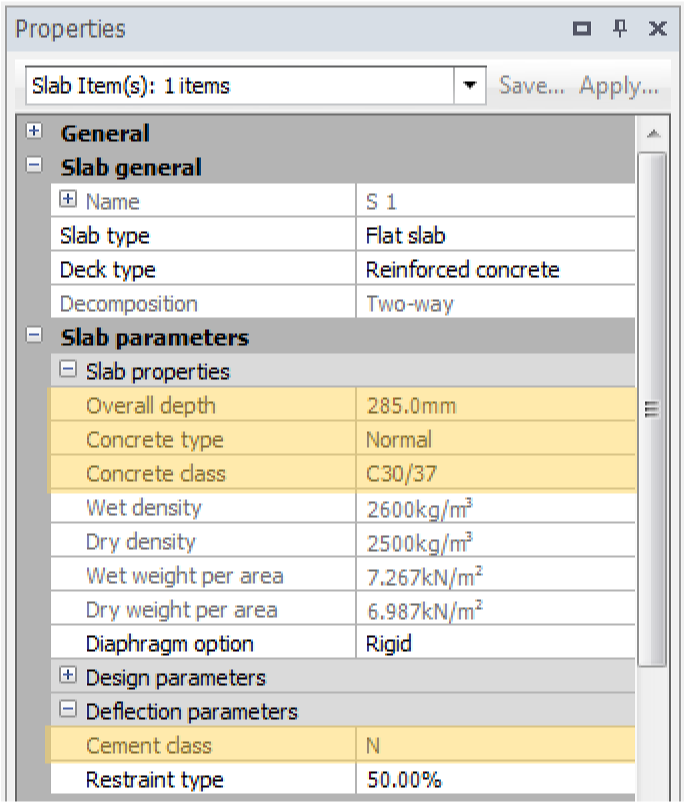
The value to adopt is a matter of judgement.

For guidance on this refer to EC2 cl 7.4.3(4) and TR58

- Affects tensile strength (f_{ct}) and hence cracking moment

Concrete Properties

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 dependant. 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/mm2) 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: Stiffness Adjustments.

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 via the Slab Deflection ribbon > Options > Modification Factors. These are user defined values with assumed defaults.

As mentioned in Concrete Properties (Eurocode), 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/mm2 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 Options** dialog.

We recommend an allowance with an upper limit of 30%. The default is set as 0.25 (25%). For the basis of this factor see: Shrinkage allowance.

Sensitivity guidance (Eurocode)

The impact that each of the input parameters has on the calculated deflections can be tabulated as follows:

Quantity	Increase	Impact on deflection	Decrease	Impact on deflection
Beta	$\beta \rightarrow 1$	δ ↘↘	$\beta \rightarrow 0.5$	δ ↗↗
Temperature	t → 80	δ ↘	t → 0	δ ↗
Relative Humidity	RH → 100%	δ ↘	RH → 0%	δ ↗
Number of Exposed Faces	n → 2	δ ↗	n → 0	δ ↘
Cement Type	C → R {R, N, S}	δ ↘	C → N or S {R, N, S}	δ ↗
Restraint	R → 100%	δ ↗	R → 50%	δ ↘

Event Sequences in Depth

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.

Event Sequences are NOT structure event sequences. They do not describe all the events starting from the first day of the overall construction.

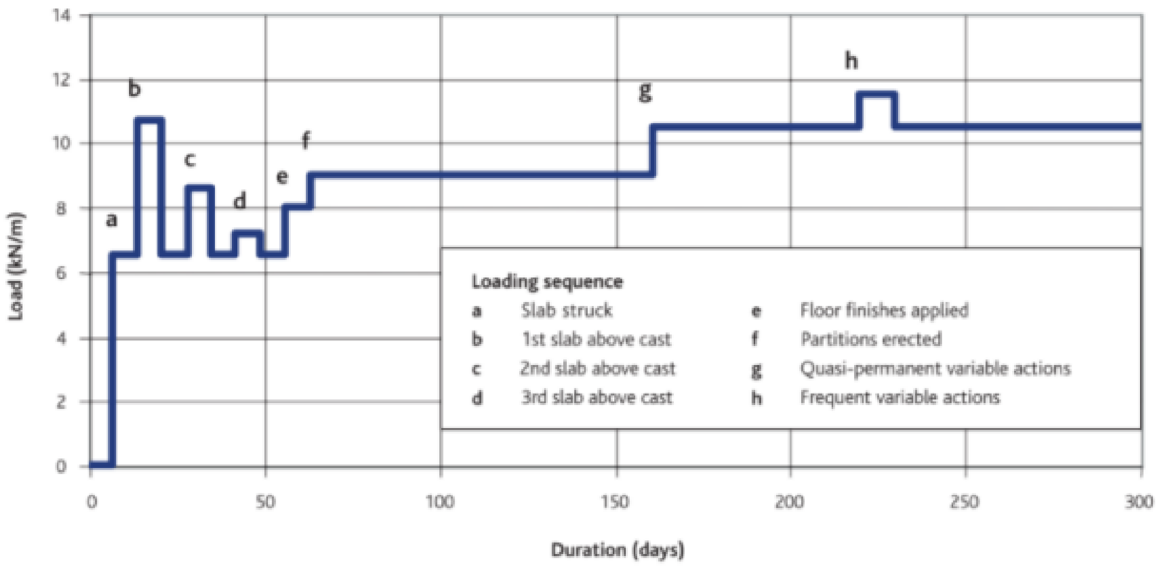
Construction Stage Events

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

   Propping loads from casting slabs above have an impact on already struck slabs - dependant upon the extent of back-propping adopted.



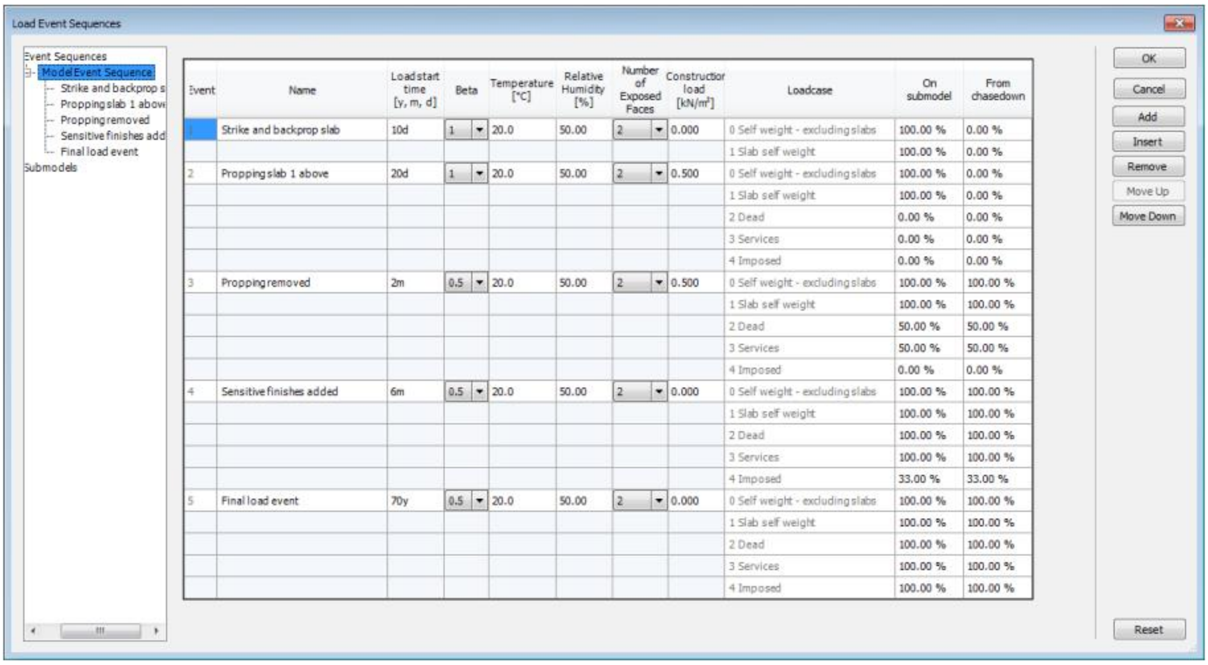
Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

3. Adding Partitions
4. Finishes

Model Event Sequence

The slab **Model Event Sequence** can be as complicated as you want it to be. The provided default Model Event Sequence is relatively simple.

Default Model Event Sequence (Eurocode)

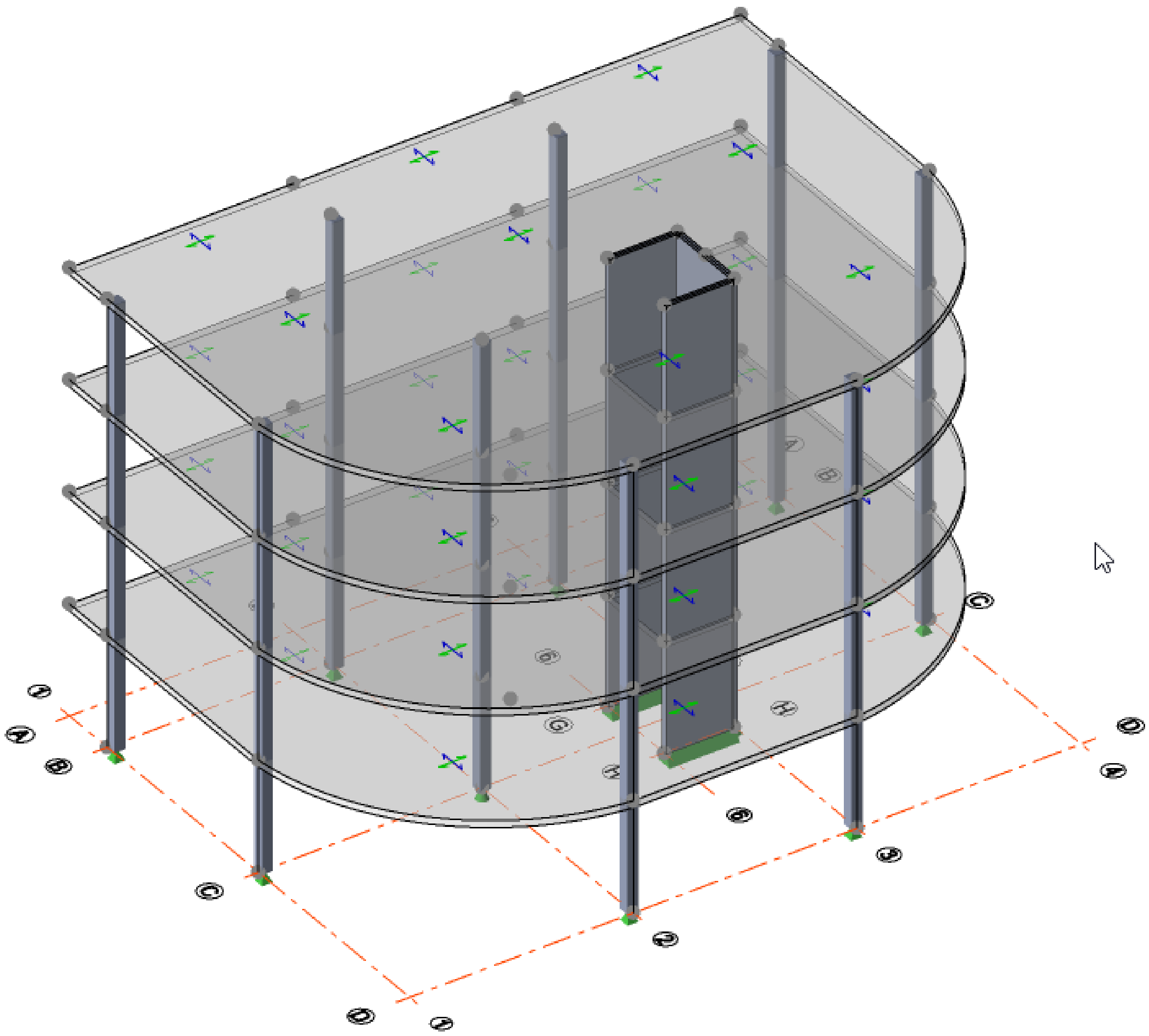


The input parameters required for each event are:

- Event Number - Automatic
- Event Name - A user defined name to help explain the event
- Load Start time - The time at which the event takes place
- Beta - a coefficient to take account of the influence of the duration of loading. (See: Beta coefficient (EC2)).
- Temperature - Used in the calculation of the Composite Modulus. The effective age of concrete is adjusted to account for the defined temperature.
- Relative Humidity - Used in the calculation of creep.
- Number of Exposed Faces - Used for the calculation of creep.
- Construction load - The construction load you wish to allow for at the chosen load event.
- Loadcase - You select the load cases you wish to be included in the event.
- On submodel - You define the % of the loadcase that you wish to consider.
- From chasedown - You define the % of the loadcase that you wish to consider.

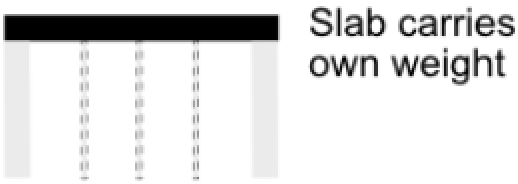
Model Event Sequence in Depth

Let us consider a multi-storey building where the slab layout is the same on each level.

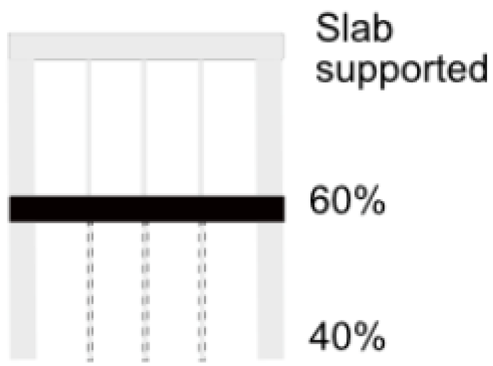


If we think about the slab model event sequence that occurs for the slab at level 1 it could be something like this:

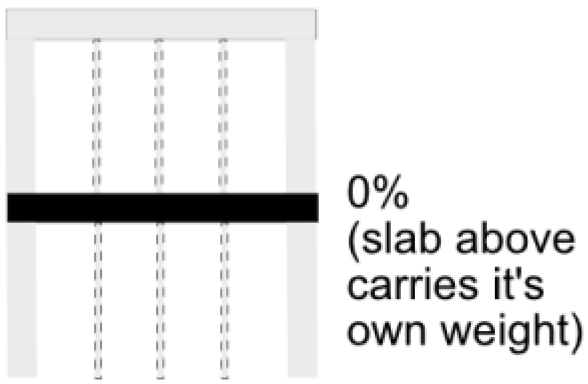
1. Strike and backprop slab (slab carries it's own weight)



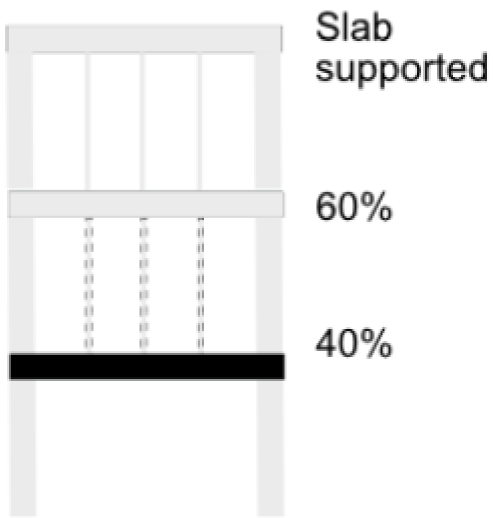
2. Cast Slab above (slab carries a proportion of the weight of slab above - the proportion is dependent on the number of levels of backpropping and there is also some debate about the efficiency of load sharing between the supporting levels. In this example we will assume 2 levels of propping and that the level directly below supports 60% and level 2 below supports 40%.)



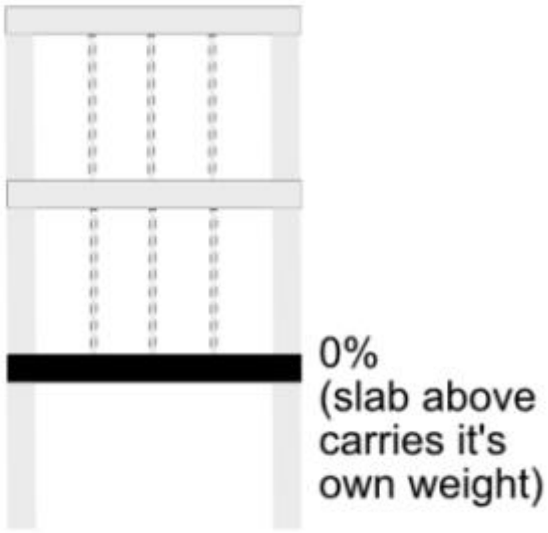
3. Strike and backprop slab above (slab above now carries its own weight)



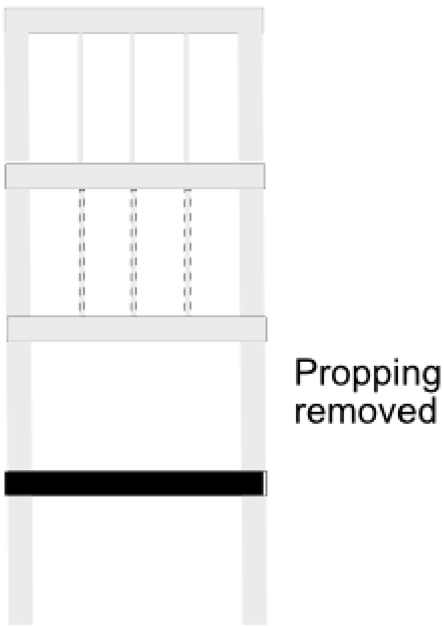
4. Cast slab 2 above (slab carries a proportion of the weight of the slab 2 above)



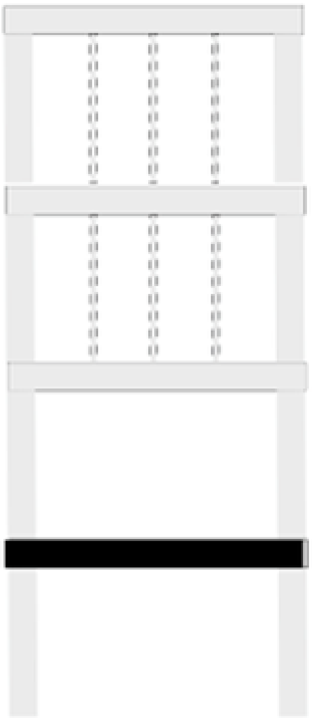
5. Strike and backprop slab 2 above (slab 2 above now carries its own weight)



6. If propping extends through 3 levels then there can be events for casting and striking the slab 3 above. In this example, the propping is removed and used two levels above.



7. Additional load from finishes (in particular from sensitive finishes)

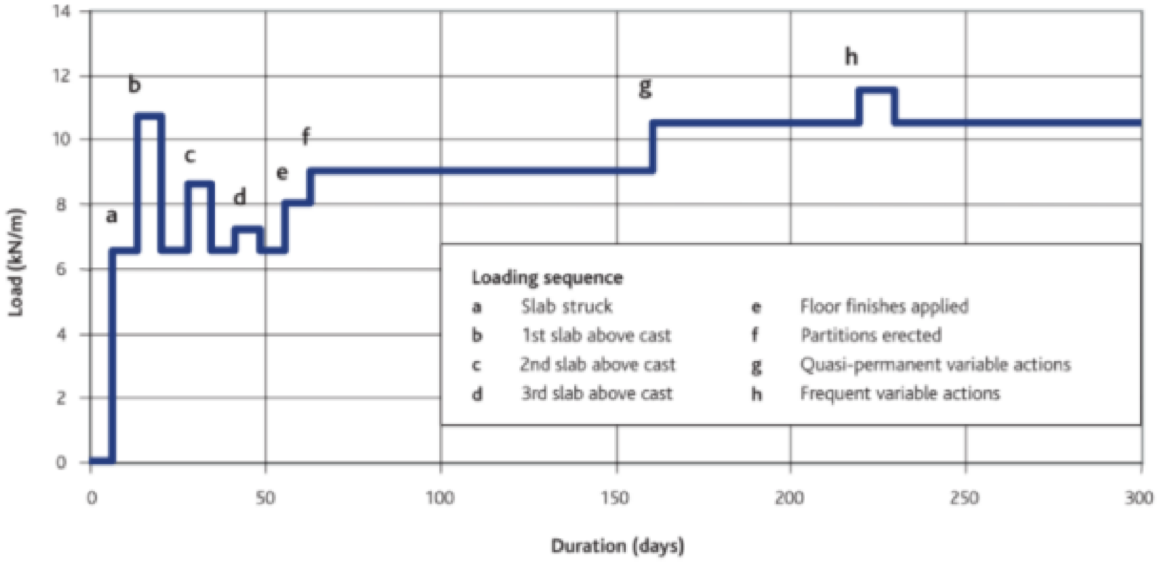


8. Start of occupation.

9. Final Load Event

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

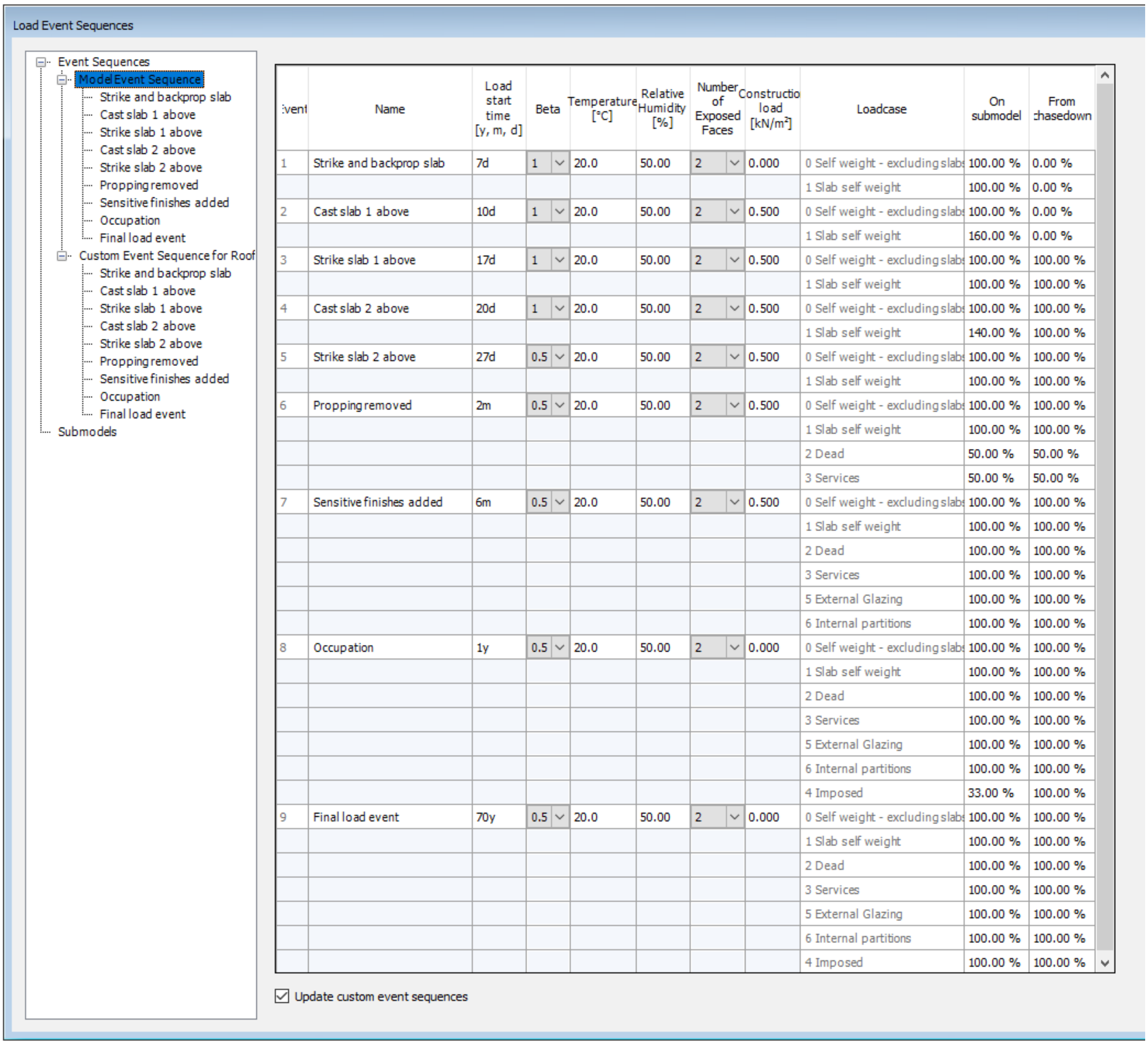
The view shown below is a graphical representation of the above but in this case is recognising 3 levels of backpropping.



Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

This sort of event sequence can be described as shown below within the Event Sequences dialog. 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.

Model Event Sequence:



:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From :hasedown
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 %

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

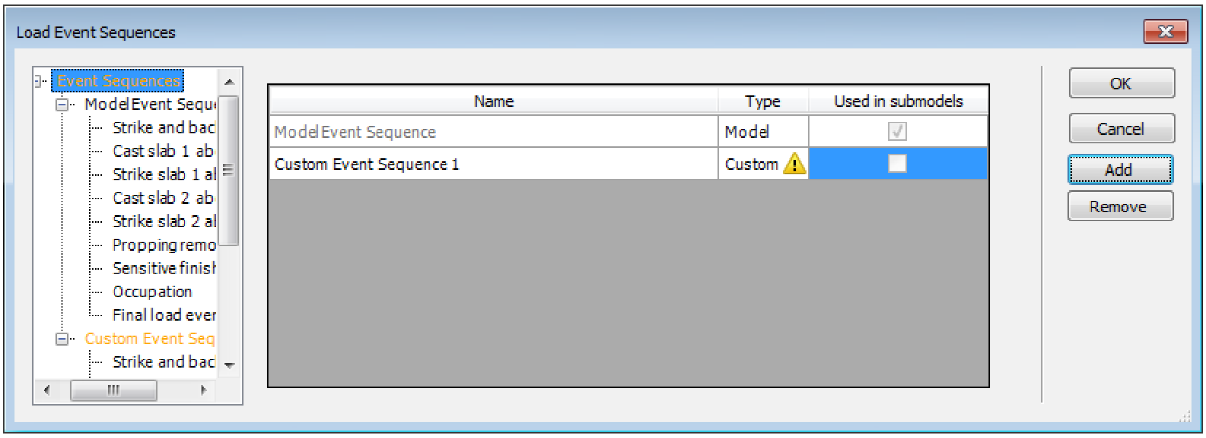
2. Keep the event, but adjust the included slab self weight to allow only for the roof load as the previous event. i.e the event has no change in load to that of the previous load event.

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.

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).

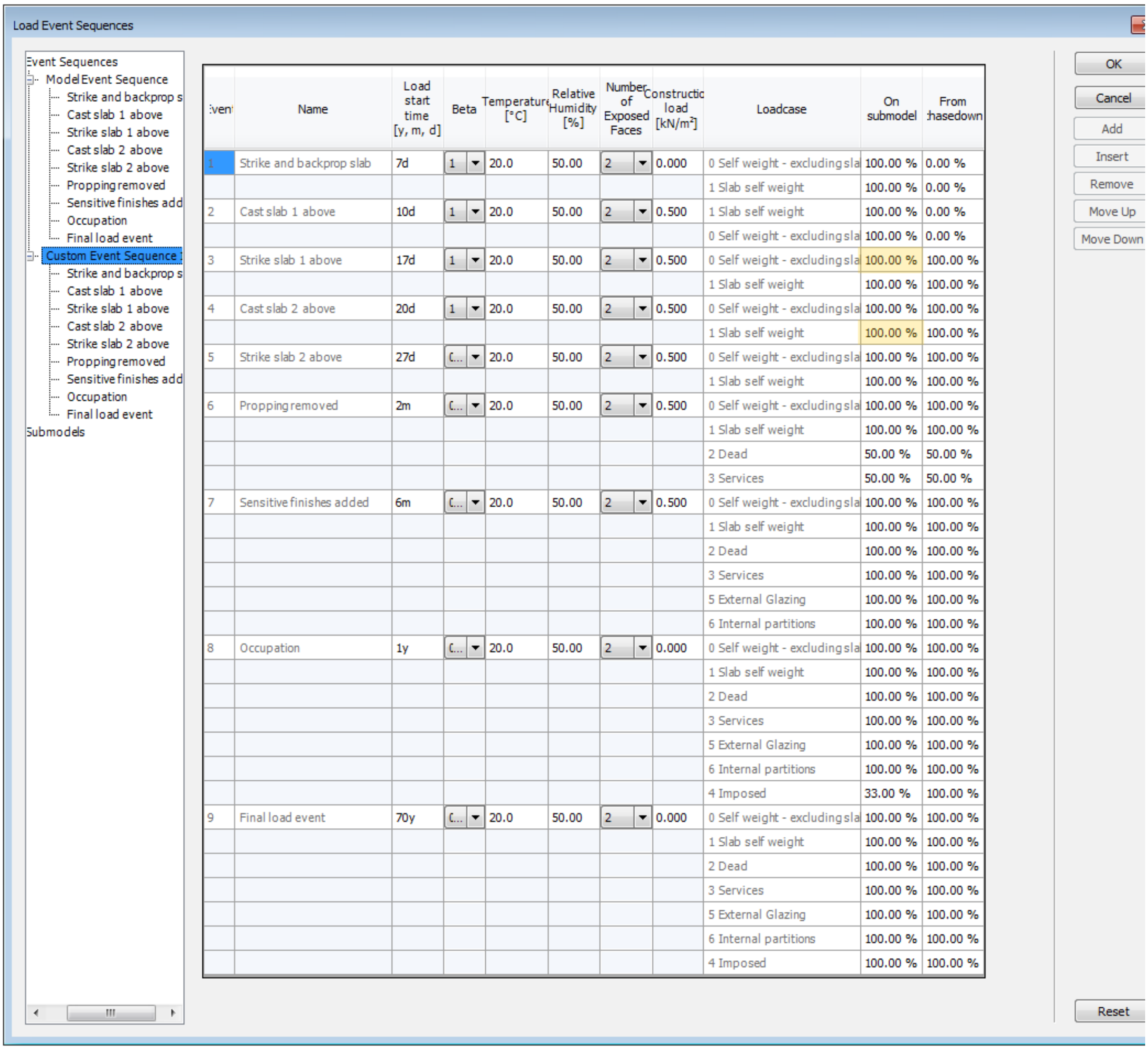


Once it has been added and given a name you can then edit the Custom event sequence by selecting it in the list.

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.



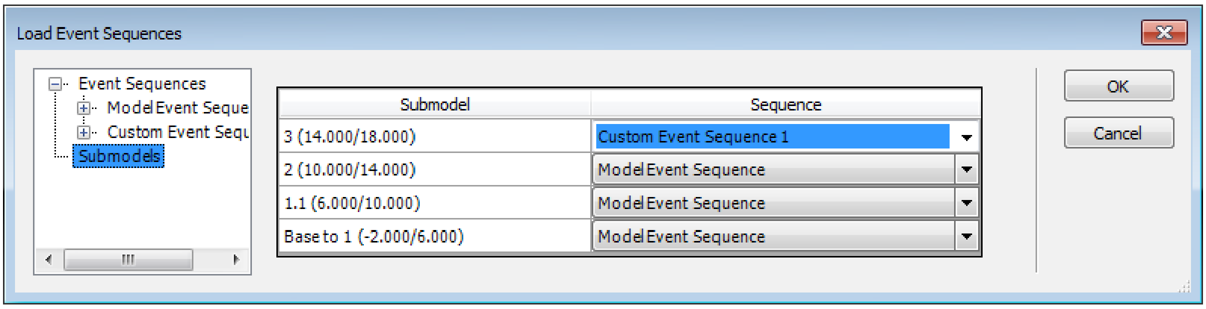
:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructic load [kN/m²]	Loadcase	On submodel	From :hasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding sla	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	1 Slab self weight	100.00 %	0.00 %
								0 Self weight - excluding sla	100.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding sla	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 sla	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	C...	20.0	50.00	2	0.500	0 Self weight - excluding sla	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	C...	20.0	50.00	2	0.500	0 Self weight - excluding sla	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	C...	20.0	50.00	2	0.500	0 Self weight - excluding sla	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	C...	20.0	50.00	2	0.000	0 Self weight - excluding sla	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	C...	20.0	50.00	2	0.000	0 Self weight - excluding sla	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 %

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 include events and loadcases. You can, however, edit the Load start time, Beta, Temperature, Relative Humidity, Number of Exposed faces, Construction loads and the % of load to apply in the combinations. 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

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.

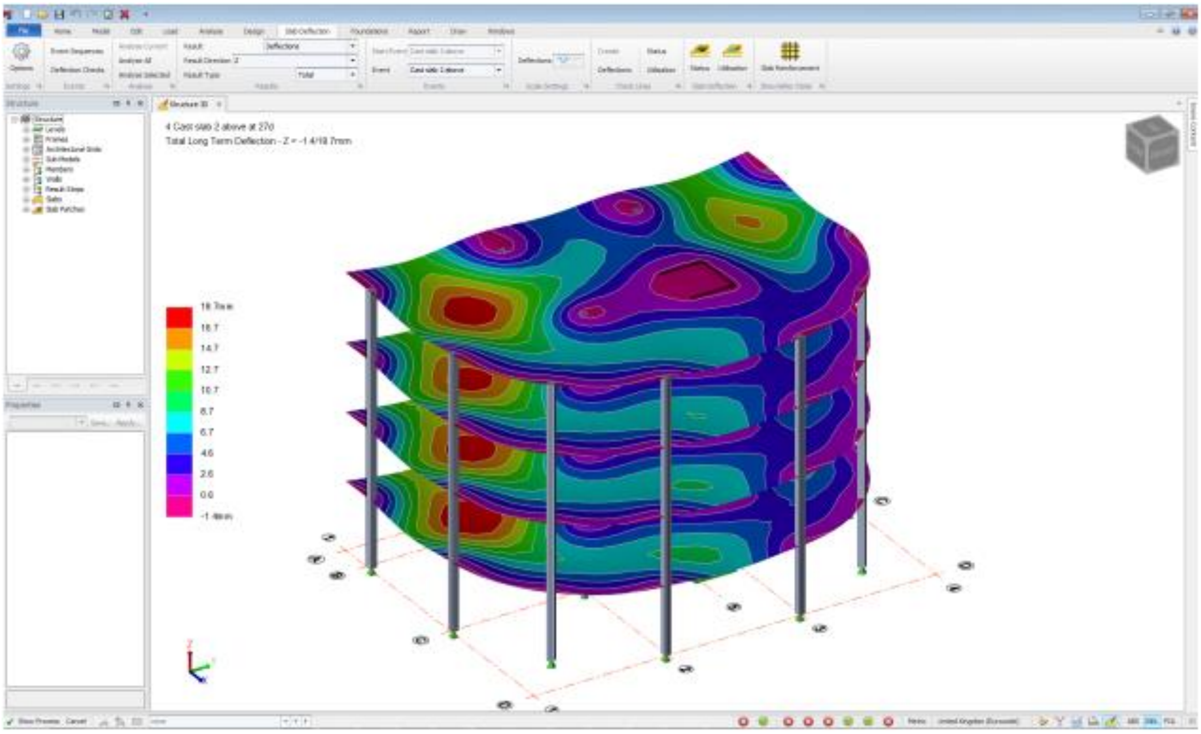
Understanding Event Sequence Deflections

The model described in the Model Event Sequence in Depth topic 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

Irrespective of whether you choose to analyse the current level (sub-model), a selected level, or all slabs in the model, the same basic process is followed.

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.

Slab Deflection Results

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 dependant upon your chosen analysis.

Deflections

Three deflection types are available: Total, Differential, and Instantaneous.

For further details, see: Slab Deflection Types

Extent of Cracking

You can also review contours to display the extent of cracking at any load event.

For further details, see: Extent of Cracking

Relative Stiffness

You can also review the relative stiffness in a particular result direction for any specified event.

For further details, see: Relative Stiffness

Effective Reinforcement

You can also review the area of effective reinforcement for a chosen result direction for each FE element.

For further details, see: Effective Reinforcement

Slab 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.

Check Lines in Depth

Check lines - a unique feature in *Tekla 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 judgement, 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).

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.

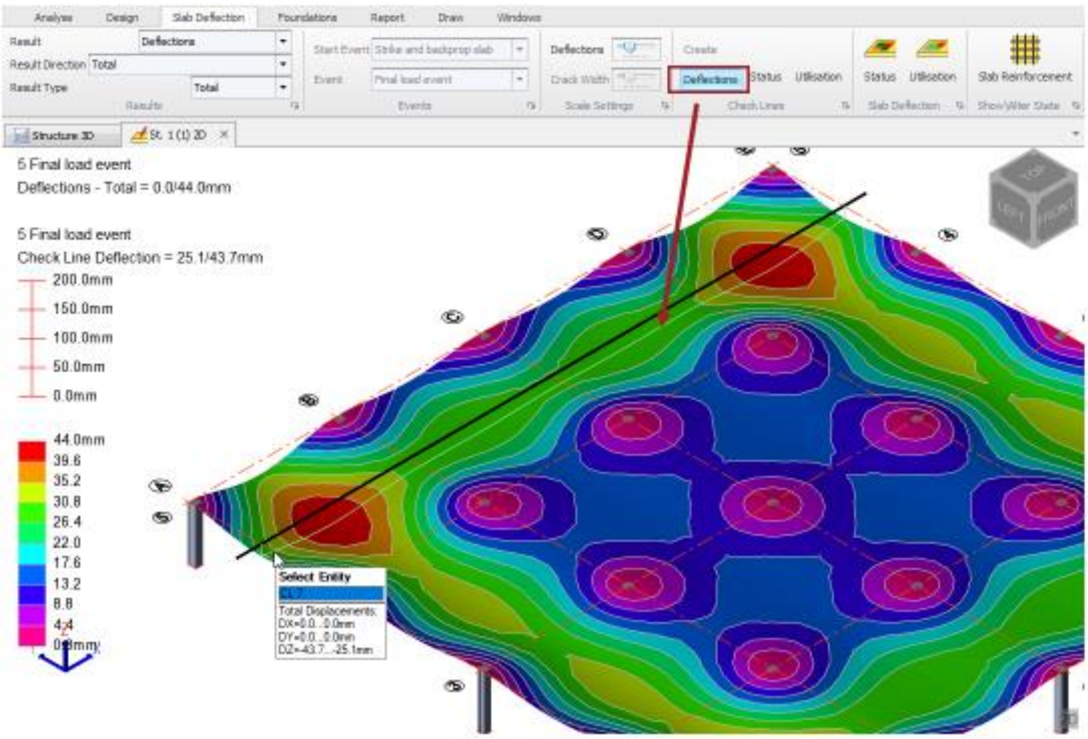
When you click Create, the Properties Window automatically includes those slab deflection checks from the Slab Deflection Check Catalogue where “Use in new Check Lines” was checked. Note that a maximum of six checks can be assigned to a single check line; if you require more than six checks, then multiple 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.

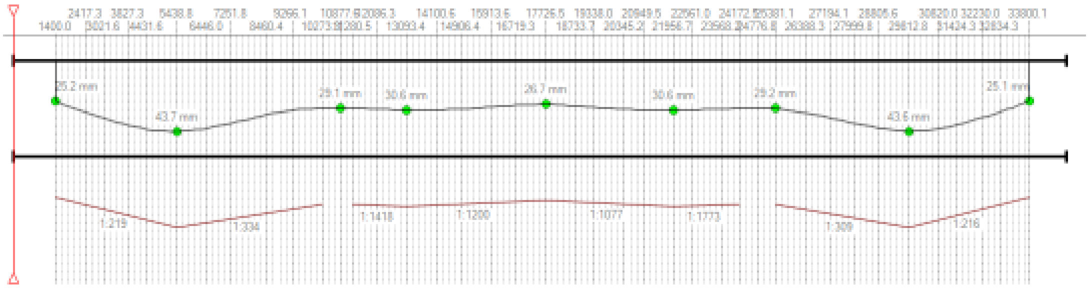
You can also edit the “Use in New Check Lines” option in the catalogue prior to running the Create command to change the default checks automatically assigned to each new check line.

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 visualise 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.

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.

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.

Slab Deflection Status and Utilization

The Status and the Utilization can be graphically displayed for both Check Lines and Slabs.

Check lines

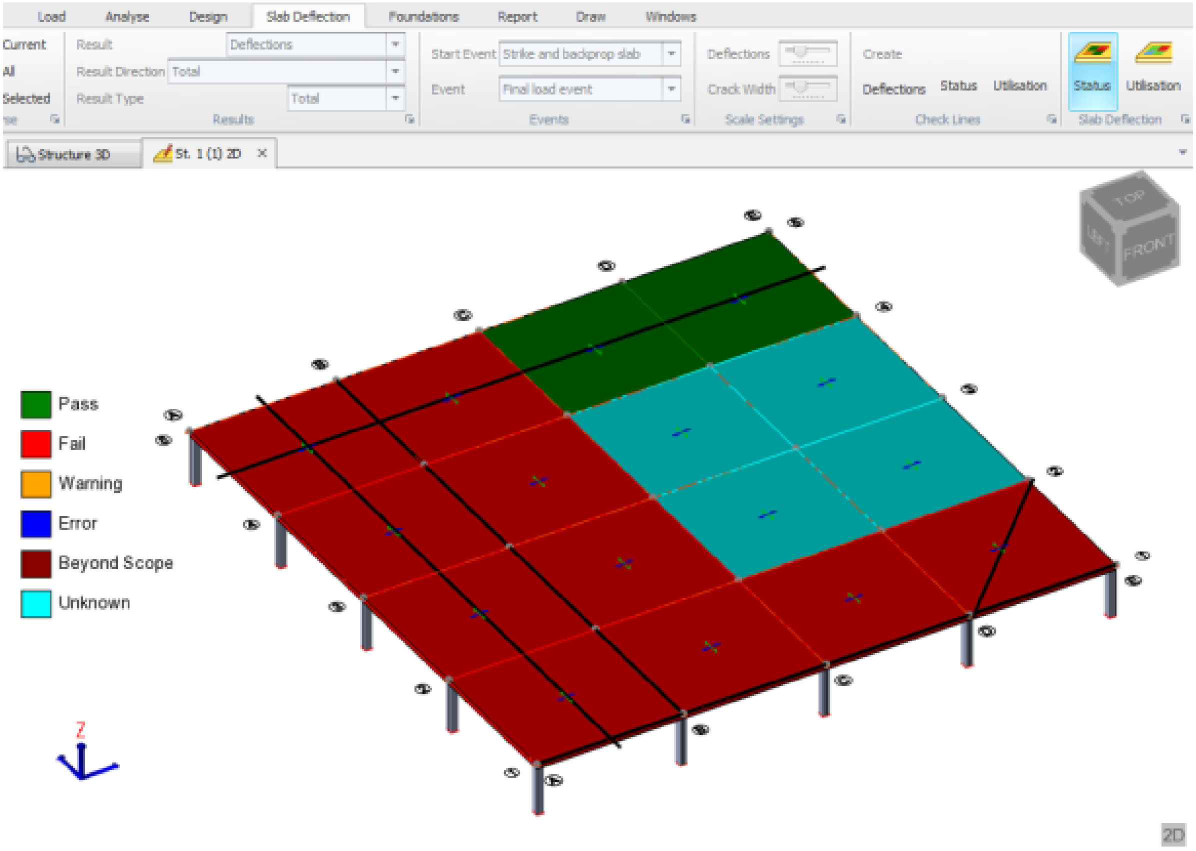
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilisation ratio.

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.

Slabs

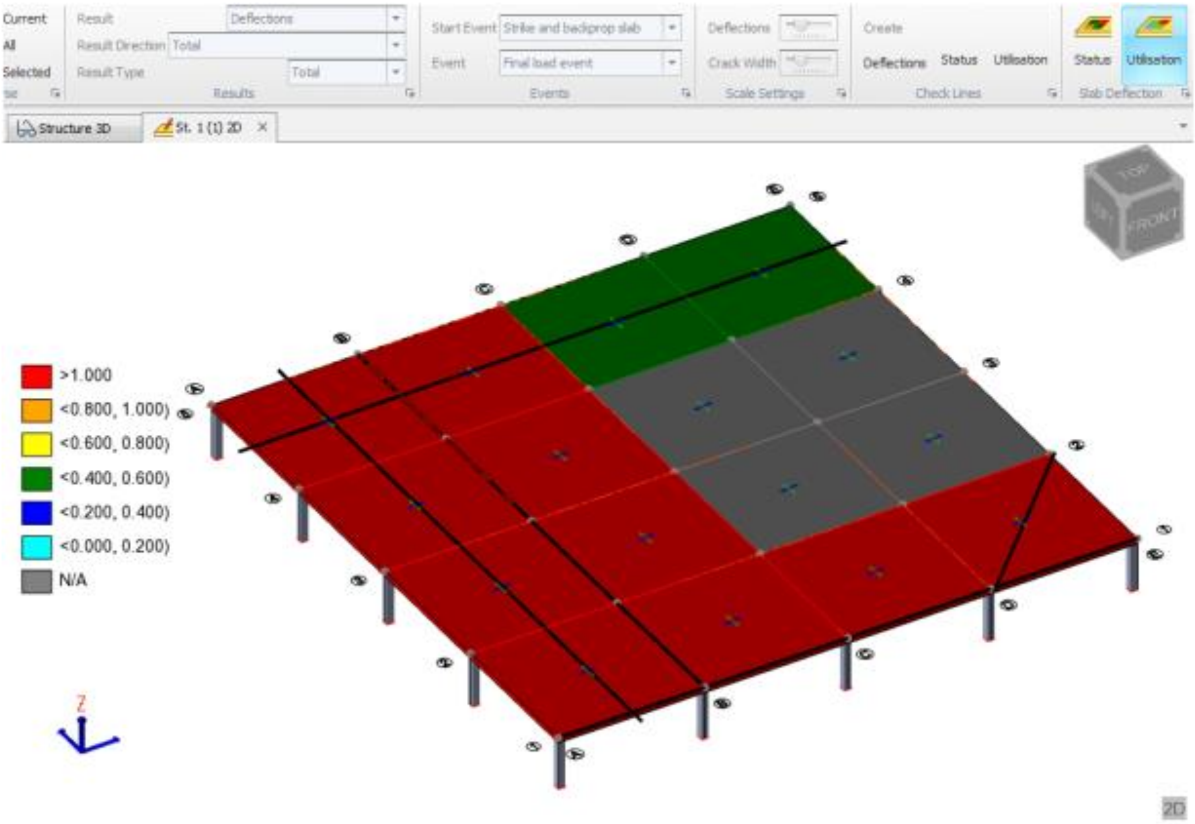
Every check line is associated to at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

In the screenshot below;



- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed.
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

The Utilisation can also be reviewed.



This is the worst utilisation from all associated check lines.

Slab Deflection Optimisation

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 optimisation.

Slab Deflection Calculations in Depth

Topics in this section

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 dependant upon the effective concrete properties, effective reinforcement, and the forces that develop
- So this calculation is unique for each direction of each shell for each event.
- The stiffness of cracked sections is dependent on the degree of cracking - so the procedure is iterative (force -> stiffness -> new force, etc)

- At the converged conclusion of this you can see (and check):
  - the extent of cracking via the Extent of Cracking results view
  - the stiffnesses determined via the Relative Stiffness results view

Composite Creep

Design codes typically provide a way of calculating an effective creep modulus at time, t for a constant load/stress applied at time t_0. Typically this is presented as $\phi(t, t_0)$.

An effective Young's Modulus is then calculated as $E_{c,\,eff}\ (t, t_0) = E_{c,28}\ /\ [\ 1 + \phi(t, t_0)\ ]$

However, codes typically do not give guidance on how to deal with a loading history where loads vary over the time period being considered.

Technical Report 58 introduces guidance on this topic and proposes a method by which the loading history can be taken into account. (Reference Section 8.4.1, equation 8.37 and also the example on page 36):

$$\left(\frac{\Sigma w}{E_{comp}}\right)n = \frac{w_1}{E_{eff,1}} + \frac{w_2}{E_{eff,2}} + \ldots\ldots + \frac{w_i}{E_{eff,i}} + \ldots\ldots + \frac{w_n}{E_{eff,n}}$$

Where:

n = event under consideration

w_i = incremental load in event i (= load in event i - load in event (i-1))
(note that this will be a negative value when load is removed)

$E_{eff,i}$ = $E_{c,\,eff}\ (t_{end,n}, t_i)$ (i.e. covers period from start of event i to end of event n)

The above is logical when you consider a single member subjected to a constant loading arrangement that is increased or decreased at each event. However, when you consider an entire slab with many panels receiving different loading increments in different events it does not seem reasonable to consider all the panels together. Two examples of this are:

1. Why should the addition of cladding loads affect internal panels to the same extent as edge panels?

2. In a transfer slab why should panels that don't support columns be affected to the same extent as those which do?

The aim of TR58 is clear, in that loading on a span/panel is taken as an indication of stress. What this fails to consider is that loading on another span/panel can also induce stress (although in most situations this will be a secondary effect). It is also clear that you do not actually need "loads", you just need "relative loads" or some other measure of the relative work done in each event. With this in mind a more general approach has been developed where the relative "work done" by each panel is determined by considering the strain energy in each event:

1. Calculate total strain energy 'Q_0' for each Slab Item for a unit load case

- 'Q_0' = sum of 2D Element strain energies

2. For each Load Event 'i'

- Calculate incremental strain energy 'Q_i'
  'Q_i' = sum of incremental 2D Element strain energies
- Calculate equivalent incremental load factor 'λ_i'
  $\lambda_i = Q_i / Q_0$

3. E_{comp} can be established from the equation below where "incremental work done" replaces "incremental load" in the TR58 equation.

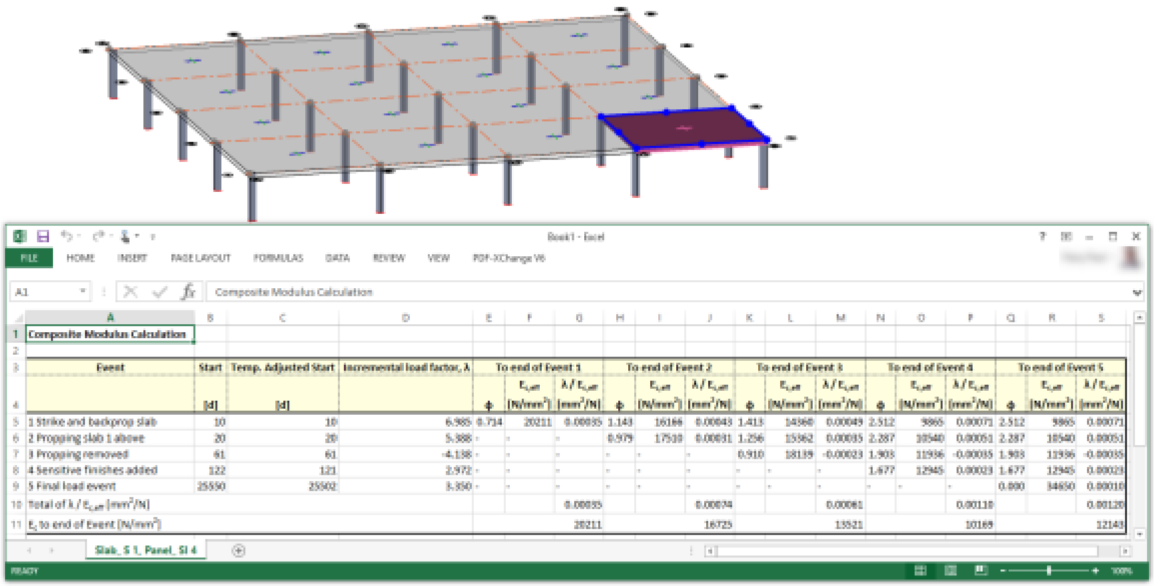
$$\left(\frac{\Sigma\lambda}{E_{comp}}\right)n = \frac{\lambda_1}{E_{eff,1}} + \frac{\lambda_2}{E_{eff,2}} + \ldots\ldots + \frac{\lambda_i}{E_{eff,i}} + \ldots\ldots + \frac{\lambda_n}{E_{eff,n}}$$

There is an array of intermediate values which lie behind the calculation of the composite modulus, E_{comp} for each slab item, for each event. The composite modulus calculation is provided as an excel spreadsheet report.

You can either generate a report for a chosen slab, selected slabs or all slabs, dependant upon your selection method.

- To obtain slab modulus reports for all slab items, right click anywhere in a scene view and choose **Export Eff. Modulus report to Excel > For all slab items**
- To obtain slab modulus reports for selected slab items, select the slabs in the structure view regime, right click anywhere in the scene view and choose **Export Eff. Modulus report to Excel > For selected slab items**
- To obtain slab modulus reports for a chosen slab items, right click a slab panel in the structure view regime and choose **Export Eff. Modulus report to Excel > For current slab items**

Using the single storey multi-bay flat slab model from the Slab Deflection Example (Eurocode), a typical composite modulus report for Slab_S1,Panel_SI 1 is shown below.



Composite Modulus Calculation																		
Event	Start	Temp. Adjusted Start	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		
	[d]	[d]		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	6.985	0.714	20211	0.00035	1.143	16166	0.00043	1.413	14360	0.00049	2.512	9865	0.00071	2.512	9865	0.00071
2 Propping slab 1 above	20	20	5.388	-	-	-	0.979	17510	0.00031	1.256	15362	0.00035	2.287	10540	0.00051	2.287	10540	0.00051
3 Propping removed	61	61	-4.138	-	-	-	-	-	-	0.910	18139	-0.00023	1.903	11936	-0.00035	1.903	11936	-0.00035
4 Sensitive finishes added	122	121	2.972	-	-	-	-	-	-	-	-	-	1.677	12945	0.00023	1.677	12945	0.00023
5 Final load event	25550	25502	3.350	-	-	-	-	-	-	-	-	-	-	-	-	0.000	34650	0.00010
Total of $\lambda / E_{c,eff}$ [mm²/N]						0.00035			0.00074			0.00061			0.00110			0.00120
E_c to end of Event [N/mm²]						20211			16725			13521			10169			12143

By referring to the example in TR58 - page 36 you should be able to closely replicate the information provided in the table. It should be noted that *Tekla Structural Designer* also 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.

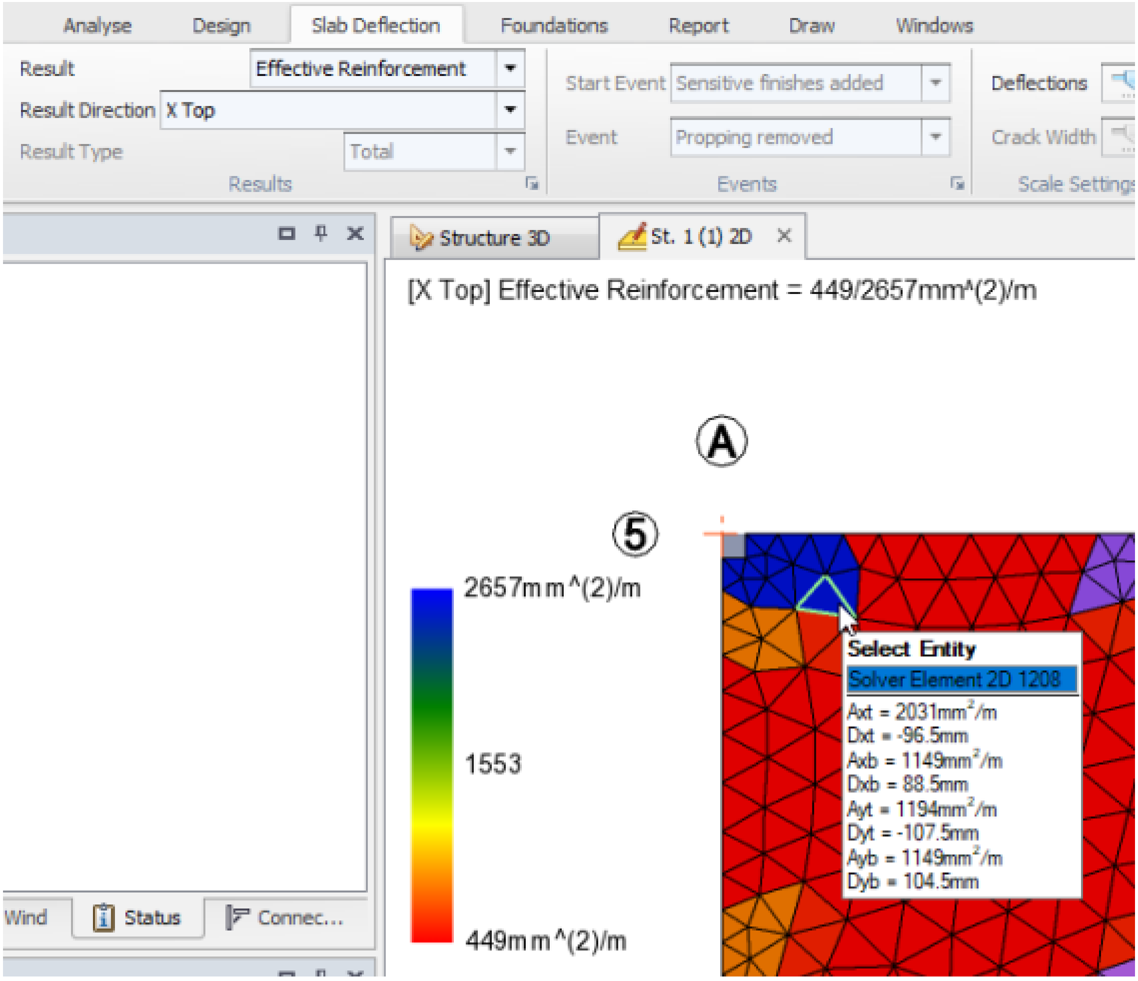
Effective Reinforcement

In addition to the effective modulus discussed in the Composite Creep topic, 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 colour 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 centre of X top reinforcement.
- A_{xb} : X Bottom Effective Reinforcement.
- D_{xb} : Distance from the section centroid to the centre of X bottom reinforcement.
- A_{yt} : Y Top Effective Reinforcement.
- D_{yt} : Distance from the section centroid to the centre of Y top reinforcement.
- A_{yb} : Y Bottom Effective Reinforcement.
- D_{yb} : Distance from the section centroid to the centre of Y bottom reinforcement.



Extent of Cracking

The effective modulus (discussed in the Composite Creep topic), 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 behaviour between the cracked and uncracked states. This expression uses a Distribution factor, ζ that apportions the behaviour between a fully-cracked state (±1.0) and an uncracked state (0) for interpolating the stiffness when a state of partial cracking exists.

$$\zeta = 1 - \beta (M_{cr} / M)2$$

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

Mcr is the hogging (positive) or sagging (negative) cracking moment.

Ma(+|-) is the relevant Wood-Armer moment in the direction for which the display is shown (X or Y).
This is calculated from Mx, My & Mxy 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, Mcr > Ma. If you hover over an FE element the tooltip provides some intermediate calculation results to verify this distribution factor.

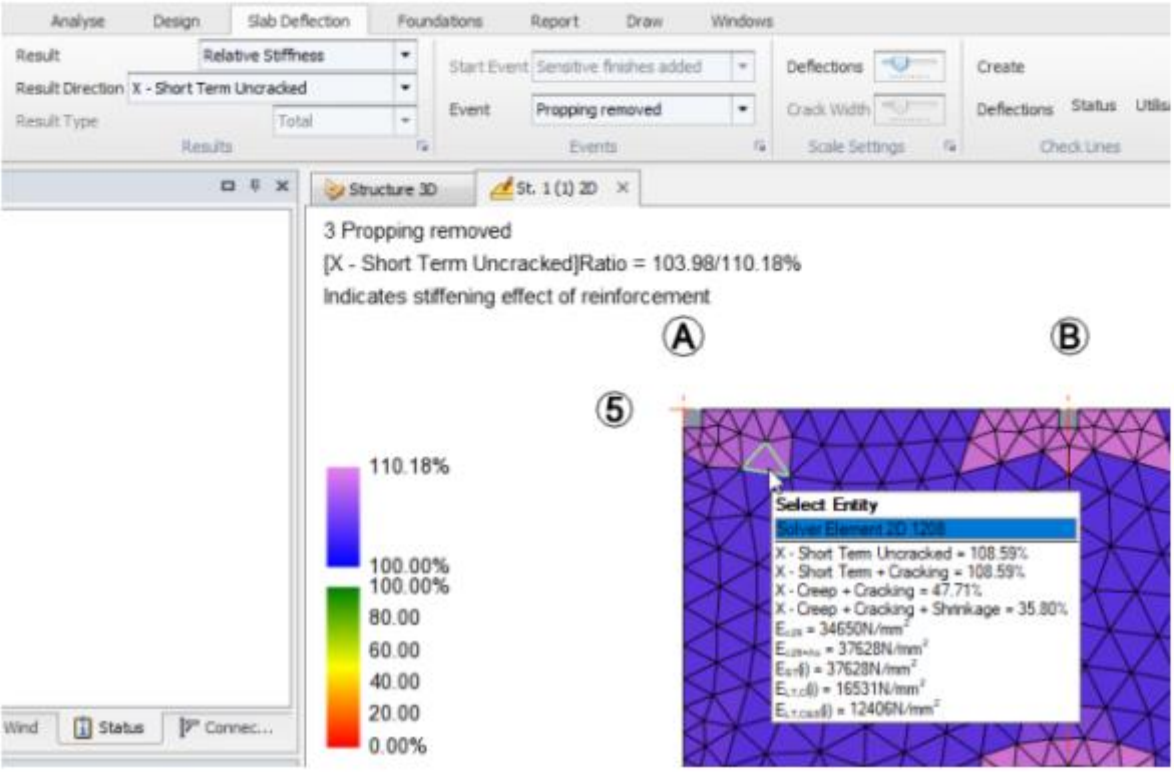
The tooltip shows the applied moment for the Event in question, rather than the worst moment of all Events up to and including the one being looked at - this allows engineers to deduce that a greater level of cracking was caused by an earlier Event.

Relative Stiffness

The tooltip provides detailed information on the relative stiffness calculations.

The tooltip lists:

- Ec28 : Edb*1.05*stiffness adjustment factor. (Where Edb = the short term modulus from the concrete materials database). The stiffness adjustment factor used is determined from the the Slab Deflection ribbon > Options, Modification Factors page. Provided for information - not directly used in any analysis.
- Ec28+As : the short term modulus including for reinforcement. Provided for information - not directly used in any analysis.
- EST(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.
- ELT,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.
- ELT,C&S(i) : ELT,C(i) with further adjustment to allow for effect of shrinkage (= ELT,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 = Ec28+As / Ec28
- Short Term + Cracking = EST / Ec28
- Creep + Cracking = ELT,C / Ec28
- Creep + Cracking + Shrinkage = ELT,C&S / Ec28

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.

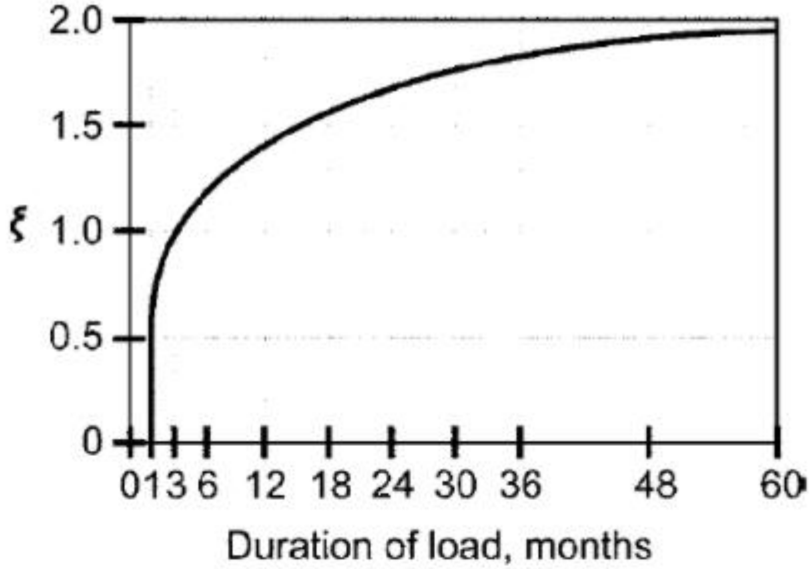


Fig. R9.5.2.5—Multipliers for long-term deflections.

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	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		
	$A_s' = 0$	$A_s' = 0.5A_s$	$A_s' = A_s$
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
1m	0.6	0.30
3m	1	0.50
6m	1.2	0.60
1yr	1.4	0.70
3yr	1.8	0.90

5yr 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

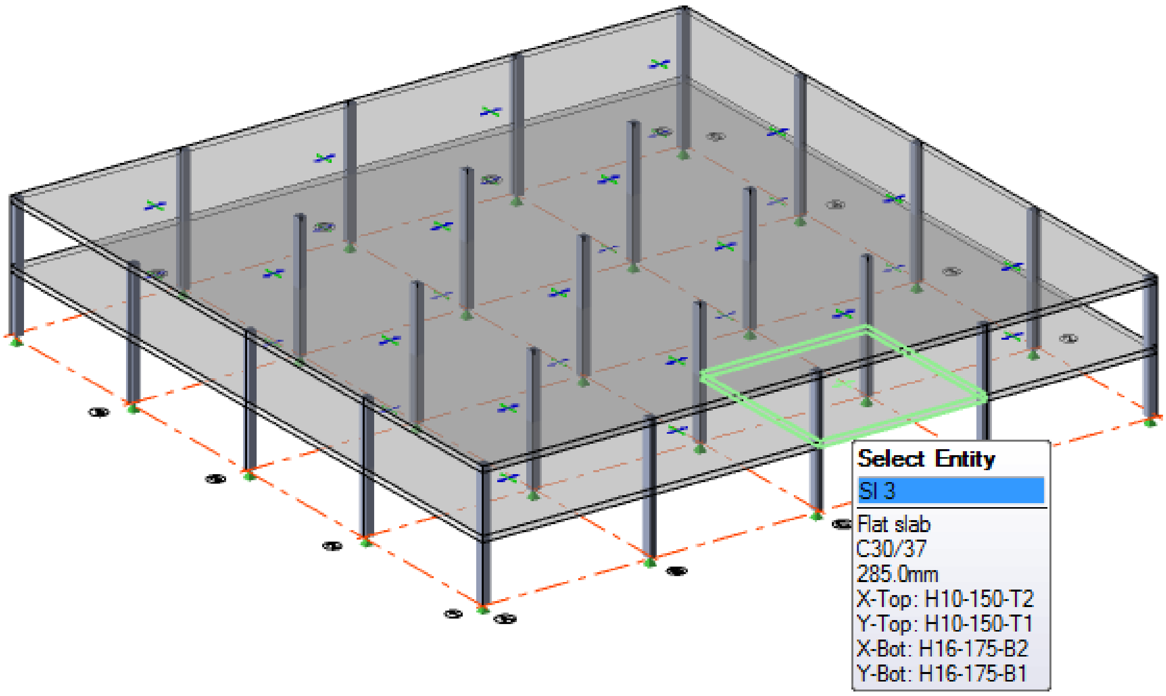
The Shrinkage deflection that occurs at each event is then the total shrinkage 10.8 mm x the shrinkage multiplier calculated above.

- 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.

Discussion of Settings and Result Sensitivity

Introduction

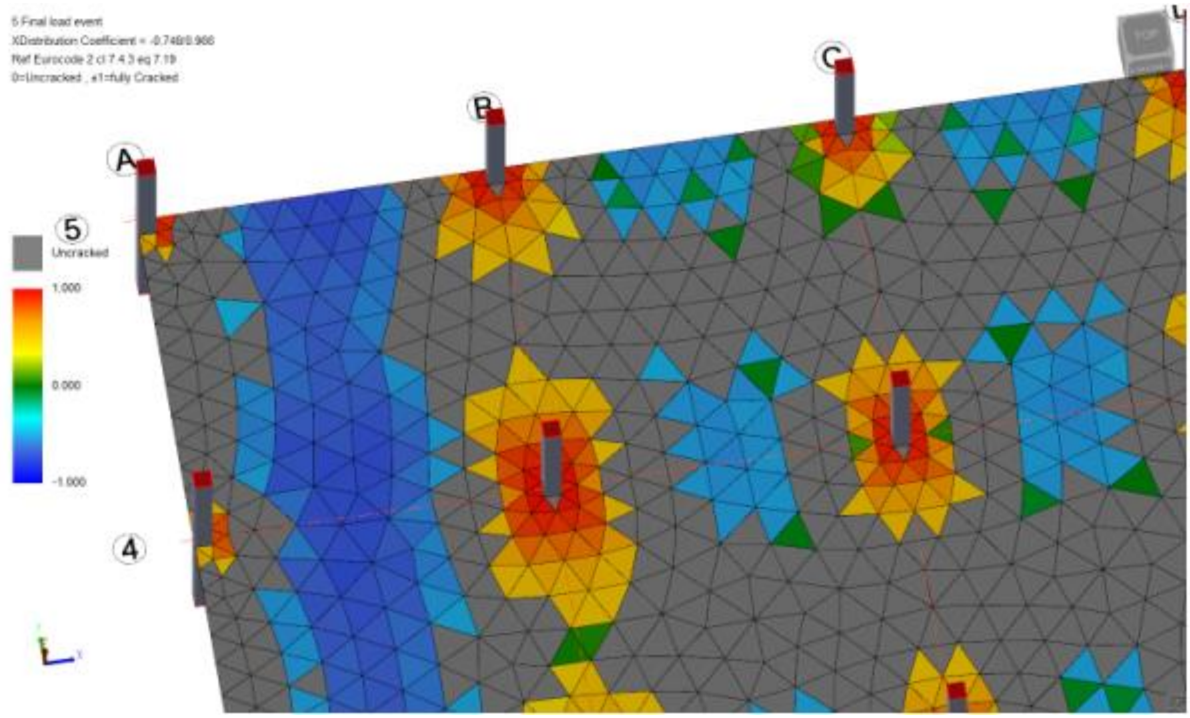


In this discussion we will focus on the first floor of the Slab Deflection Example (Eurocode) model which might be regarded as a typical floor in a taller building. The layout is a simple 8m square grid of columns. The slab has been sized at 285mm thick C30 concrete based on deemed-to-satisfy rules taken from “Economic Concrete Frame Elements to Eurocode 2 - The Concrete Society” for a slab subjected to imposed load of 5.0kN/m2. Additional finished loads and wall perimeter load have been applied to match the loads assumed in that guide.

So this is a slab that would be expected to work. As is shown in Deemed to Satisfy Checks (Eurocode Slab Deflection Example), max deflections need to be around 45mm in order to pass the Span/250 limit.

In the following sections the impact of adjusting various settings will be discussed. It may be noted that the total deflections viewed throughout this exercise almost never drop below 45mm. However, in the final section we will discuss a set of assumptions and minimal changes that can be applied in order to show this slab might be considered acceptable.

Mesh Sensitivity

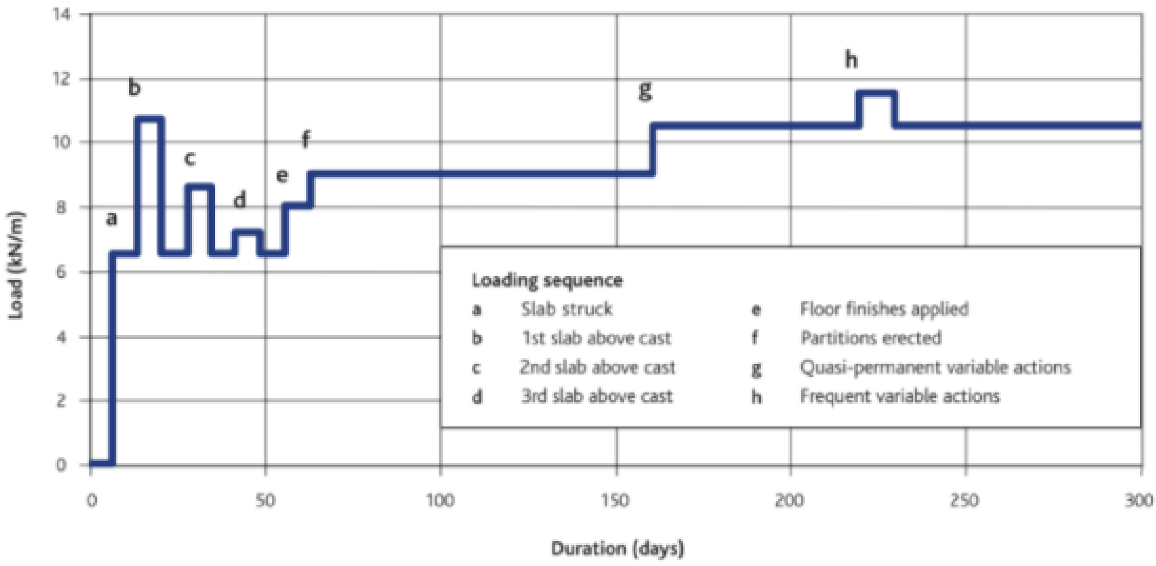


The view above shows the meshing with max shell edge length set to 1.0m - mesh is refined around the columns and relaxes into the spans. There are about 10-12 shell edges in the length between column faces. For design purposes this level of refinement is considered adequate. The table below shows the impact of different mesh refinement on the predicted deflections.

	Deflection (mm)		**% Variation**	
	Total	Differential	On Total	On Differential
1.5m - 2034 shells/floor	54.5	24.8	-0.9%	-0.8%
1m - 3130 shells/floor	55.0	25.0	-	-
0.5m - 9046 shells/floor	55.3	25.1	+0.5%	+0.4%

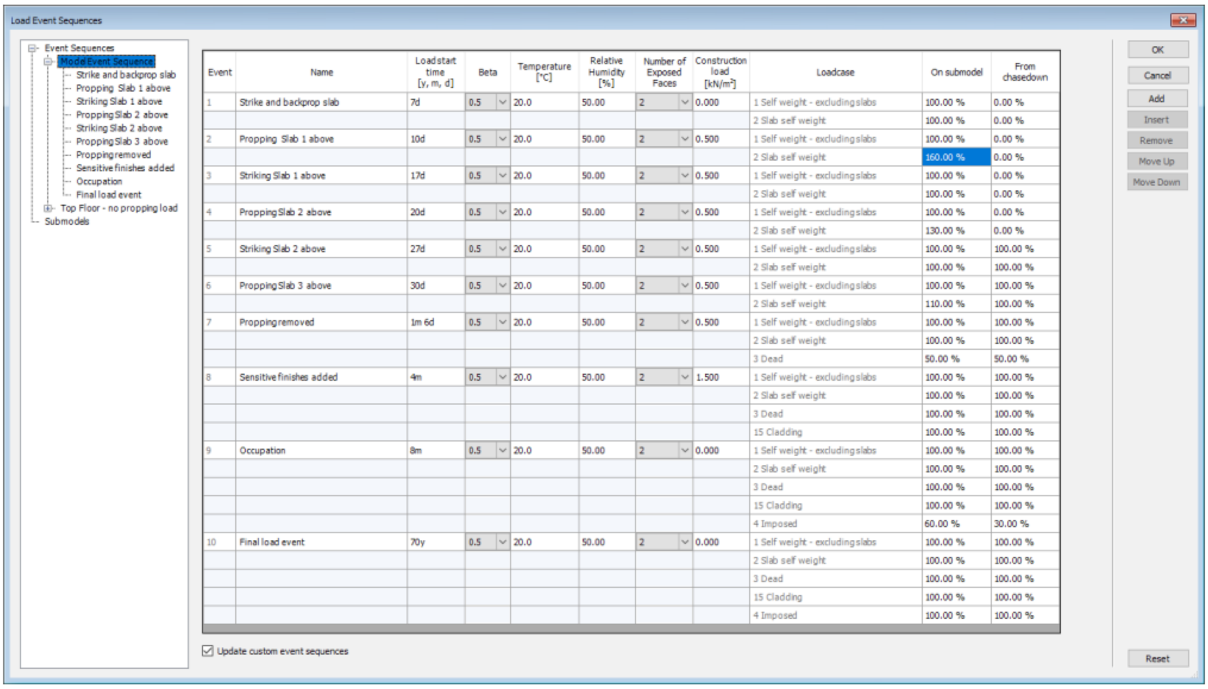
As meshing is refined the deflections increase slightly, however, the result demonstrates very little mesh sensitivity. It can be considered converged at the 1m edge length, tripling the number of shells in the model makes less than 0.5% difference to both total and differential deflections.

Event Sequence Level of Detail



Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

In the above figure, events a to d are a series of early age construction events considering the loads due to propping of the slabs above. These can be described in the Tekla Structural Designer event sequence dialog as shown below.



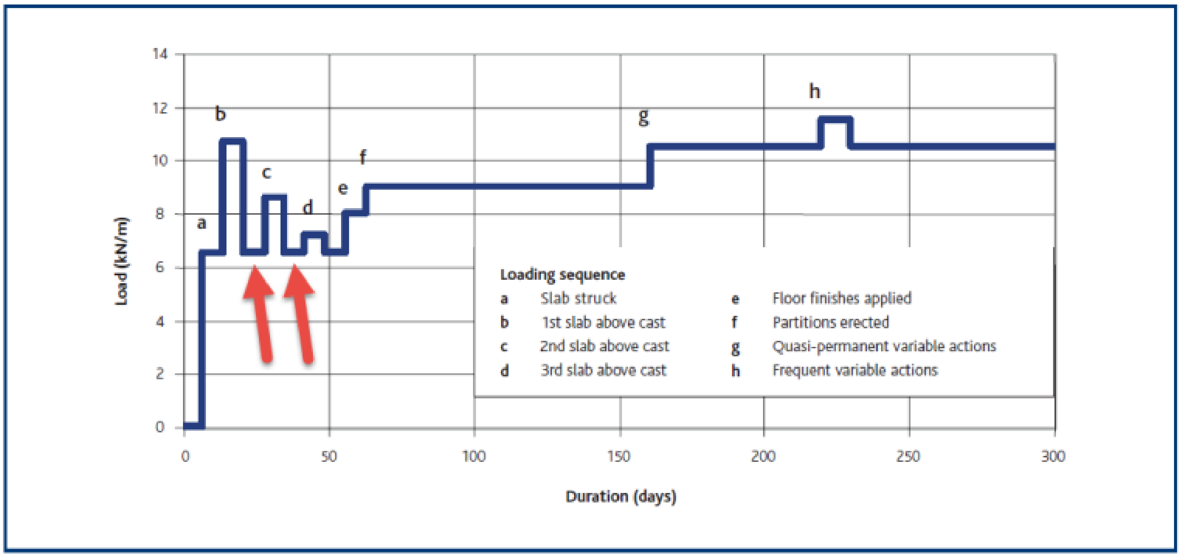
Quite clearly the timing of events a and b, and the magnitude of load at propping event b must be important. The question we will consider in this section is the degree to which the exact timings of subsequent propping events B and C impact on the results. The following table indicates the results of simplifications to the sequence. In lines 1 to 6 a different aspect of sensitivity is assessed and each of these are discussed in more detail in the sections below.

	Variable	Initially Assumed Value	For Minimum Total Deflection			For Maximum Total Deflection			Potential % Impact	
			Value	Total def	Diff. def	Value	Total def	Diff. def	Total	Diff
Event Sequence Detail starting with 10 days/floor, 3 propping events 60%, 30%, 10%:										
1	Time between striking and propping	3 days	0 days	54.7	24.8	3 days	55.0	25.0	0.5%	0.8%
2	Simplified Sequence 1		0 days	54.7	24.8	3 days	55.0	24.9	0.5%	0.4%
3	*Pace of Construction 1a*		*15d/floor*	*54.9*	*24.9*	*5d/floor*	*55.1*	*25.0*	*0.4%*	*0.4%*
4	Pace of Construction 1b		15d/floor	52.1	23.8	5d/floor	59.1	26.6	13.4%	11.8%
5	Simplified Sequence 2		15d/floor	51.6	23.6	5d/floor	58.9	26.5	14.1%	12.3%
6	Propping Event Percentages	60% / 30% / 10%	35/35/30	50.6	23.3	70/20/10	56.2	25.4	11.1%	9.0%
7	Combined speed and propping	10d / 60%	15d/35%	47.2	22.5	5d/70%	60.0	27.0	27.1%	20.0%
8	Beta factor for first propping event	1	0.5	48.6	23.1	1	55.0	24.9	13.2%	7.8%

Line 1 - Time between Striking and Propping

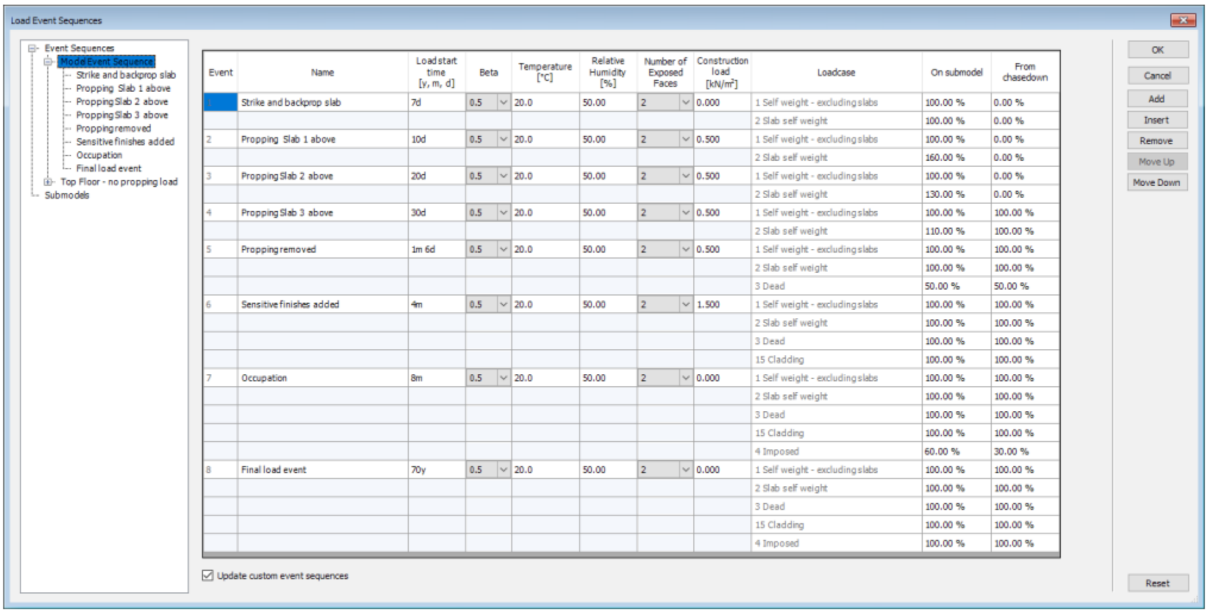
In the detailed sequence shown above there is a 3 day gap between striking a slab and then introducing loads due to propping of the slab above. This is a potentially brief period of reduced loading in-between the propping events highlighted in the figure below.

Figure 2
Loading history for a slab – an example



At the design stage the engineer is unlikely to know this level of detail, and the question is - does it really make much difference? In line 1 you can see the impact of setting this gap to either zero or 3 days, there is less than a 1% impact on total or differential deflections. This means we can look at simplifying the event sequence as discussed below.

Line 2 - Simplified Sequence 1



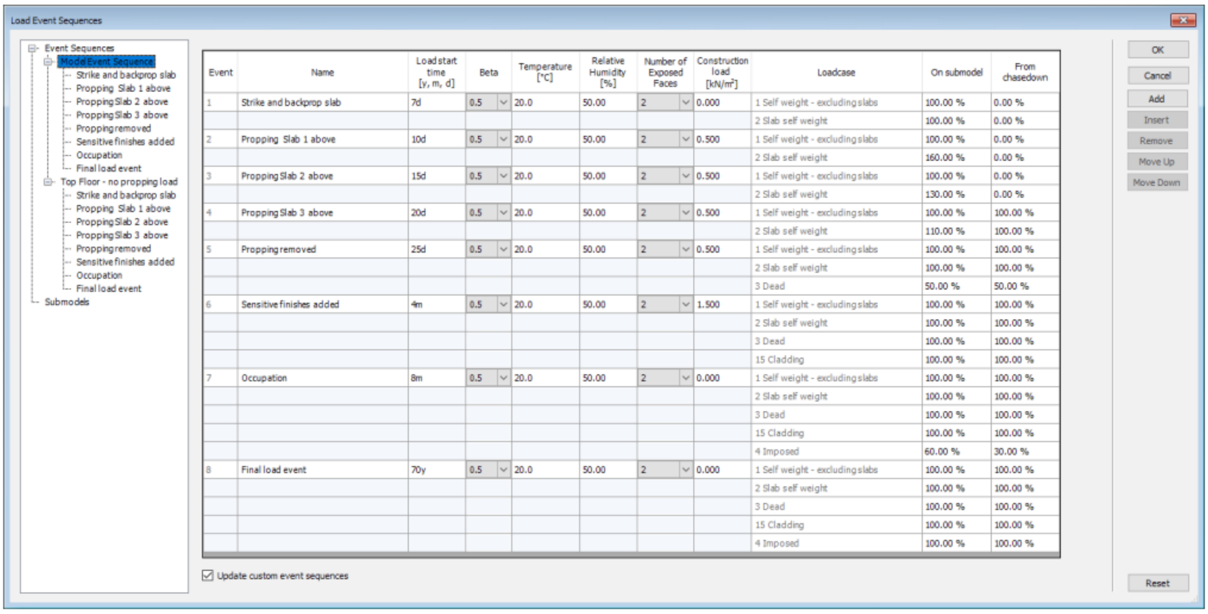
Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping Slab 1 above	10d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping Slab 2 above	20d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	130.00 %	0.00 %
4	Propping Slab 3 above	30d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	110.00 %	100.00 %
5	Propping removed	1m 6d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
6	Sensitive finishes added	4m	0.5	20.0	50.00	2	1.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
7	Occupation	8m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	60.00 %	30.00 %
8	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

In this example the unloading events between b. and d. are completely removed but the gap between a. and b. is retained:

- When the gap is 3 days the result is almost identical to the line 1 above
- When the gap is 0 days the result is identical to the line 1 above

Therefore we see that the small potential impact observed on line 1 is all associated with the timing of events a. and b. This simplified sequence has retained these two events and so there is almost no measurable loss of accuracy in eliminating the subsequent gaps.

Line 3 - Pace of Construction 1a (pace after event b.)



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping Slab 1 above	10d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping Slab 2 above	15d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	130.00 %	0.00 %
4	Propping Slab 3 above	20d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	110.00 %	100.00 %
5	Propping removed	25d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
6	Sensitive finishes added	4m	0.5	20.0	50.00	2	1.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
7	Occupation	8m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	60.00 %	30.00 %
8	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

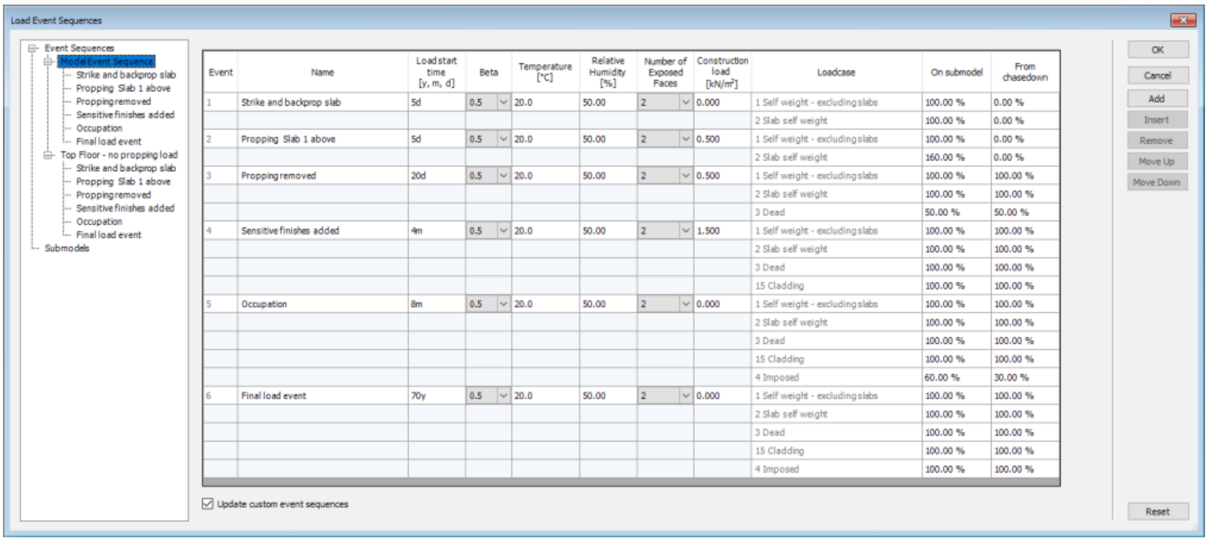
This is a somewhat unrealistic example where we are keeping the timing of events a and b fixed at 7 and 10 days, and then looking at the impact of assuming a faster (5d/floor) or slower (15d/floor) pace of subsequent construction events. We are seeing < 1% impact, so once again the timing of these subsequent events is of relatively small importance.

Line 4 - Pace of Construction 1b

This is a more realistic example where the timing of events a and b are also adjusted for a faster and slower pace of construction. A pace was set to 5 days/floor and then 15 days/floor with events a and b both being on either the 5th or 15th day. This has a more significant impact of around 10 to 15% on both total and differential deflections.

Clearly it is the timing and loading in events a and b that is most important amongst these early construction events. We can take this a stage further and consider eliminating events c. and d.

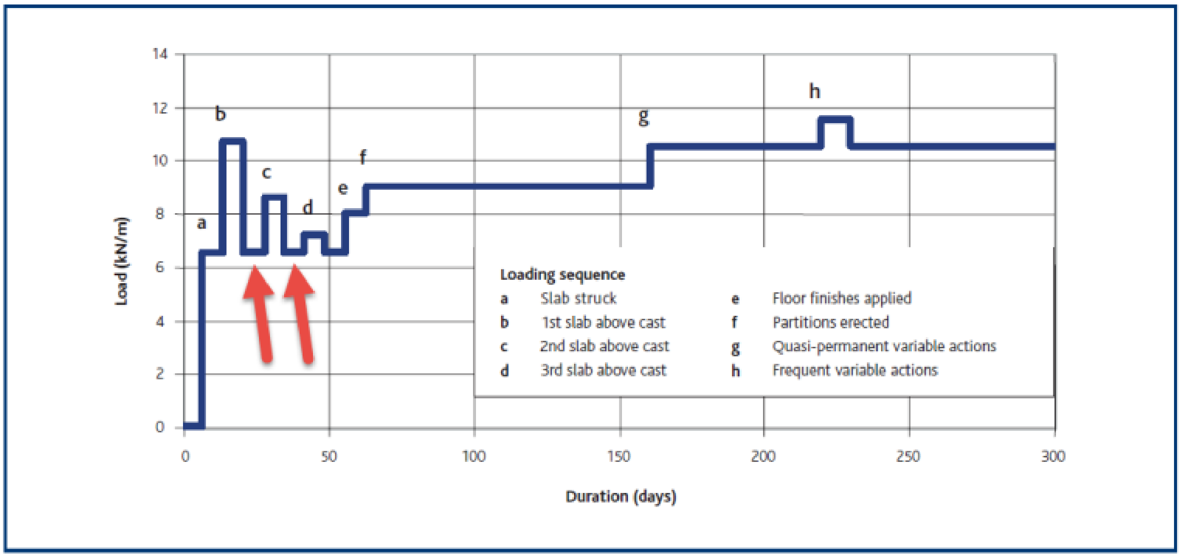
Line 5 - Simplified Sequence 2



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	5d	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping Slab 1 above	5d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	20d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	0.5	20.0	50.00	2	1.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
5	Occupation	8m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	60.00 %	30.00 %
6	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

In essence this is simplifying the sequence so that events c and d are ignored and event b is assumed to exist for a longer period as illustrated below.

Figure 2
Loading history for a slab – an example



This looks like a significant change but comparing lines 4 and 5 you can see it makes very little difference (< 1%) to the total and differential deflections being considered for either the fast or slow paces of construction.

Note that we are not suggesting you should routinely adopt this level of simplification in the sequence, the above example is used merely to demonstrate the insensitivity to events c and d. Accurate prediction of these events should not be a primary concern.

Line 6 - Propping Load

Any search will find a good deal of discussion on the topic of propping loads. A good start on this is the freely available guide from IStructE shown below.

38 | **TheStructuralEngineer** December 2016 | **Technical** Temporary Works Toolkit

Temporary Works Toolkit

Part 4: An introduction to backpropping of flat slabs

The Temporary Works Toolkit is a series of articles aimed primarily at assisting the permanent works designer with temporary works issues. Buildability – sometimes referred to now as "construction method engineering" – is not a new concept and one always recognised as vital to the realisation of one's ideas; it ought to be at the forefront of an engineer's mind.

www.twforum.org.uk

Temporary Works forum
Promoting best practice in the construction industry.

Eur. Ing. Peter Pallett BSc, CEng, FICE, FCS
Above Ground Temporary Works Consultant, Pallett Temporary Works Ltd.

In this various methods are described the simplest of which tabulates maximum loads on the slabs as shown below.

Table 1: Method 1 percentage of load transfer for flat slabs less than 350mm thick

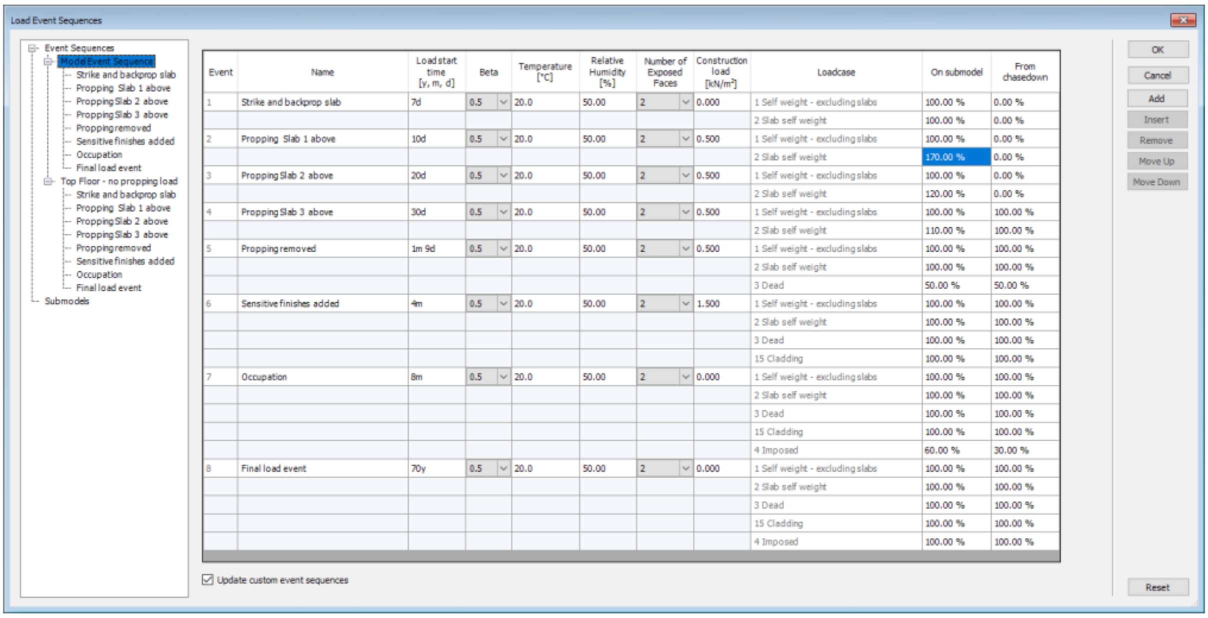
Location	Load	No backprops fitted	One level of backprops		Two levels of backprops	
			On slab	In prop	On slab	In prop
New slab cast on falsework	w_p	100%	100%		100%	
		100%		100%		100%
Supporting slab		100%	70% w_p	–	65% w_p	–
Backprops	w_{b1}	None	–	30% w_p	–	35% w_p
Lower slab (2)		–	30% w_p		23% w_p	–
Backprops	w_{b2}	None		None	–	12% w_p
Lower slab (3)		–	–	–	12% w_p	–

Notes:

1) Assumes all floors are of similar construction and have similar stiffness at time considered
2) Assumes lower and supporting slabs have been struck and have taken up their deflected shape and are carrying their own weight
3) The distribution is that percentage of the applied load onto the supporting slab. Each floor slab will also have to carry its own self weight and any imposed construction loads already on the floor
4) Determination of the characteristic strength of the slabs to carry the applied loads is not considered
5) All floors are suspended floors and Method 1 slabs are flat slabs

Historically, where 2 levels of backpropping were used, a very simple assumption of equal load in the levels below was made (i.e. 33%). Other research on this topic suggests that when pre-loading of props is taken into account the peak values of 65/70% are unlikely to arise in practice. With detailed control over pre-loading it is possible to achieve something closer to the traditional assumption of equal load sharing.

In *Tekla Structural Designer* the allowance for propping load can be made in various ways. The simplest way is to allow for an amplified self weight as indicated below.



On Line 6 the impact of assuming 35%/35%/30% vs 70%/20%/10% is shown. It has around a 10% impact on total and differential deflections.

Line 7 - Combined Speed and Propping

The comparisons above lead to the conclusion that timing of events a and b and the magnitude of the propping load at b have the greatest impact on predicted deflections. In this line the impact of combined extremes of these values are considered together:

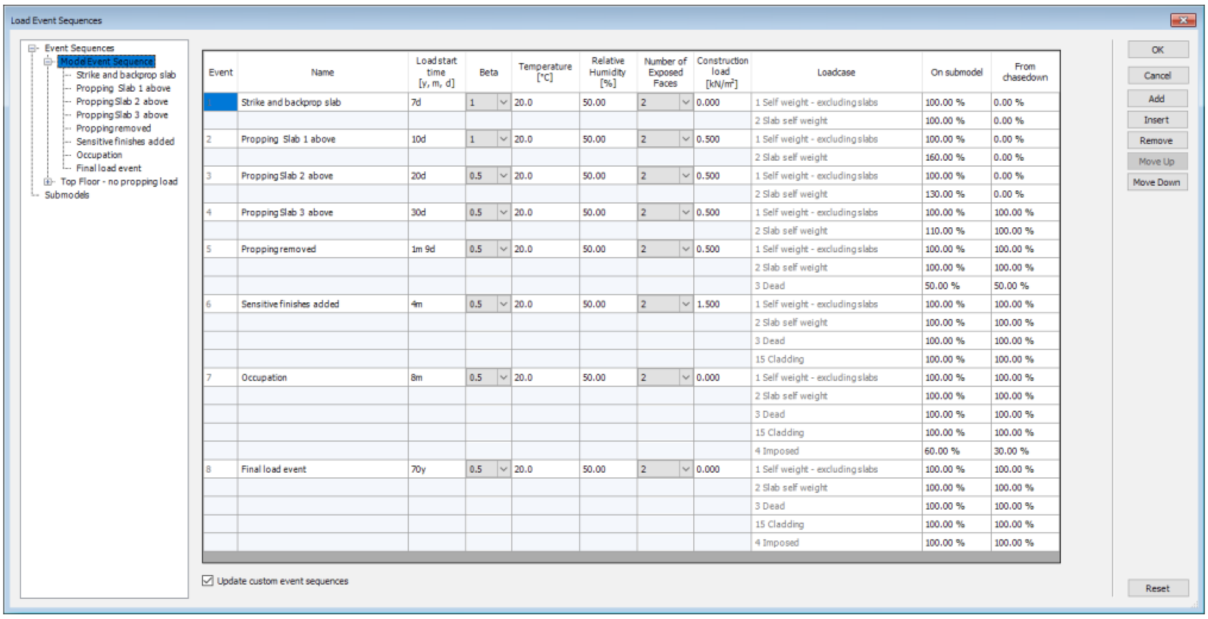
- Slow pace of construction (15 days per floor) and an assumption of just 35% propping load.
- Fast pace of construction (5 days per floor) and an assumption of 70% propping load.

This gives:

- Potential variation in total deflections in the order of 30%
- Potential variation in differential deflections a little lower - perhaps in the order of 20%

Line 8 - Beta coefficient used for construction events

EC2 clause 7.4.3 introduces a Beta coefficient which allows for the influence of duration of loading. A value of 1 may be considered for a single short term loading event. However, it is not completely clear what sort of event and duration falls into this category - TR58 seems to suggest that 0.5 should always be used for propping events. We can look at the impact of assuming a 1.0 value is applicable to the first event as indicated in the sequence below.



Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping Slab 1 above	10d	1	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping Slab 2 above	20d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	130.00 %	0.00 %
4	Propping Slab 3 above	30d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	110.00 %	100.00 %
5	Propping removed	1m 9d	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
6	Sensitive finishes added	4m	0.5	20.0	50.00	2	1.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
7	Occupation	8m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	60.00 %	30.00 %
8	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

The table shows this has the impact of reducing total deflection from 55 to 48.6mm. However, note that this has reduced the deflection below the 50.6mm value determined on line 6 where a more constant propping load of 35% is assumed. It does seem questionable that the beneficial effect of assuming beta = 1.0 for the first event with a high propping load should result in total deflections lower than when a smaller more constant propping load is assumed across a series of events.

Therefore, in the remaining sections of this sensitivity study we will continue to adopt the TR58 guidance that a value of 0.5 should be used.

Summary

The above has demonstrated the construction event history can be greatly simplified without any significant impact on the predicted deflections. The key items which require realistic assessments / allowances are:

- Time of Slab striking.
- Time of introducing load due to propping the slab above.
- The propping load level.

Not examined within the above is the allowance for variable construction loading during the construction period. This was assumed as 0.5kN/m2, assuming a larger value will clearly have some additional impact on deflections.

Finally, it is noted that if higher propping loads such as 70% of the weight of the slab above are assumed it is quite possible these could be the most significant loading events the slabs will ever see. This is especially true if the design loading for the slabs is relatively low (e.g. domestic loading allowance). The slab deflection analysis does not consider strength checks at any of the defined events. If this sort of situation applies then the creation of additional design combinations should be considered.

Estimated Values and Assumptions

Throughout the Event Sequence comparisons above a series of other assumed values have remained constant, we can now look at the potential impact of making different assumptions for these:.

	Variable	Initially Assumed Value	For Minimum Total Deflection			For Maximum Total Deflection			Potential % Impact	
			Value	Total def	Diff. def	Value	Total def	Diff. def	Total	Diff
Estimated Values/Assumptions (Simplified sequence - swt at 7 days, propping loads 60%/30%/10% at 10 day cycle)										
1	Concrete E value	C30 - 33000N/mm2	+20%	52.9	23.9	-30%	59.8	27.2	13.0%	13.8%
2	Temp During Construction	20 deg	20 deg	55.0	24.9	5 deg	59.7	26.9	8.5%	8.0%
3	Restraint type	100%	50%	50.9	23.2	100%	55.0	24.9	8.1%	7.3%
4	Relative Humidity during Construction	50%	80%	53.3	24.5	50%	55.0	24.9	3.2%	1.6%
5	*Relative Humidity - external structure*		*80%*	*52.7*	*23.9*	*50%*	55.0	24.9	*4.4%*	*4.2%*
6	Exposed Faces after construction	2	1	54.8	24.8	2	55.0	24.9	0.4%	0.4%
7	Time of sensitive finishes	4 months	6 months	54.9	23.5	2 months	55.2	27.7	0.5%	17.9%
8	% of Finishes before sensitive finishes	50%	100%	55.1	23.1	10%	54.9	26.4	-0.4%	14.3%
9	**All Extremes at Once**			**47.3**	**18.9**		**65.1**	**34.4**	**37.6%**	**82.0%**

In this case the simplified event sequence is used with striking at 7 days and an assumed propping load of 60% at 10 days, 30% at 20 days, and 10% at 30 days.

Line 1 - Concrete E value

The Eurocode indicates a typical E value for each concrete grade but notes that the actual value will be aggregate dependent and may vary in the range +20 to - 30%. This line demonstrates that this range on the E value has an impact of 10-15% on both total and differential values.

Perhaps this is a lower impact than you might initially expect, this is because the concrete E value has less impact on the stiffness of cracked sections.

Line 2 - Temperature During Construction

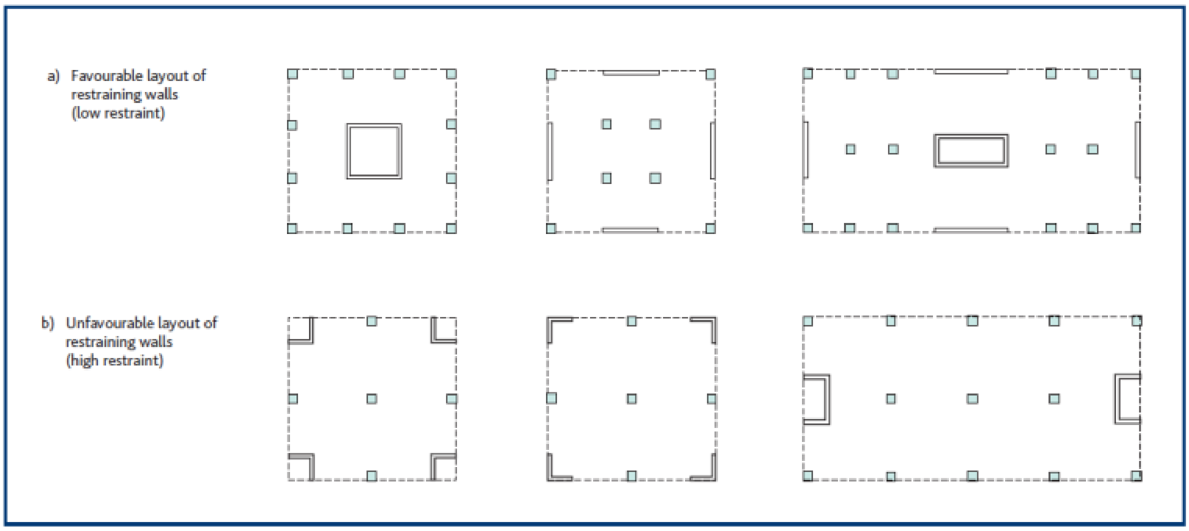
20 degrees is often assumed as an average throughout the life of a structure, but what is it reasonable to assume during the construction phase especially for winter construction? This line demonstrates that assuming 5 degrees for the initial 4 months after casting could have an impact of 5 to 10 % on both total and differential values.

Line 3 - Restraint Type

The degree of restraint resisting shrinkage is a matter of judgement. The code suggests 100% for high restraint and 50% for low restraint. This line demonstrates that this range can have an impact of 5 to 10 % on both total and differential values.

The Eurocode asks you to take the level of restraint into account but gives no guidance on what constitutes "low" or "high" restraint. Some guidance can be found in the figure below.

Figure 1
Typical floor layouts



Extract from Concrete Centre Guide "How to design concrete structures using Eurocode 2 - 8. Deflection Calculations" - Figure 1

Where outlying corner panels are critical to deflection it may be worth considering the restraint level to these rather than using the worst case that might be applicable anywhere on the slab.

Line 4 - Relative Humidity during construction

Normal recommendations for the UK are to assume RH = 50% for internal and 80% for external. 50% is often assumed as an average throughout the life of a structure, but what if this is increased to 80% for some part of the construction period. This line demonstrates that assuming 80% for the initial 4 months after casting would have the beneficial impact of reducing both total and differential deflections by a few %.

Line 5 - Relative Humidity external structure

Normal recommendations for the UK are to assume RH = 50% for internal and 80% for external. This line demonstrates that assuming 80% for the entire life of the structure can have an impact of reducing both total and differential deflections by a few %.

Note that comparing lines 4 and 5, the difference between assuming external conditions during construction and external for the entire life of the structure appears quite minimal.

Line 6 - Exposed Faces after Construction

When the exposed surface area is reduced the creep coefficient is reduced and deflections will reduce. This line demonstrates the effect of assuming only one surface is exposed after finishes are applied. For this example this has minimal impact on deflections.

Line 7 - Time of Sensitive Finishes

This line demonstrates the impact of adjusting the time at which it is assumed sensitive finishes begin to be introduced. Times of 2 months and 6 months are compared. It is shown that this has negligible impact on the total deflections but will have a very big impact (up to around 20%) on the differential deflection occurring after this point in time. This is primarily because more creep and shrinkage will have occurred prior to the later event. Clearly this will be important if you are assessing the deflection impact on specific sensitive finishes.

Line 8 - % of Finishes before Sensitive Finishes

Other finishes loading may be introduced before or after the time at which sensitive finishes begin to be introduced. In the basic sequence this was assumed to be 50% of the added dead loads. This line demonstrates the impact of adjusting this assumption between 10% and 100%. It is shown that this has negligible impact on the total deflections but could have significant impact (very load dependent) on the differential deflection. Clearly this will be important if you are assessing the deflection impact on specific sensitive finishes.

Line 9 - All extremes at once

This line demonstrates the potential impact of using worst case scenario assumptions everywhere. Comparing all the other extreme or best case scenario assumptions we see:

- Over 30 to 40% potential variation in Total Deflection
- Over 80% potential variation in Differential Deflection

Summary

Bear in mind:

- The shrinkage allowance was not considered in the above. It was assumed as 25%, but could perhaps have been considered in a range of something like 15 to 30%.
- This is a slab that is expected to be acceptable based on traditional “deemed to satisfy” approaches.
- Note that even when the best case assumptions are made throughout, it still does not pass the total deflection check when the impact of full live load is considered.
- When the check is made based on quasi permanent load only, it is still borderline (more on this later)

The message here is to be very cautious about using conservative assumptions every step of the way. In normal circumstances it may be more reasonable to consider the range you think is applicable to your project and use average values. In addition it is suggested that:

1. The restraint type relevant to the critical slabs be considered carefully

2. The small beneficial effects of assuming external conditions during construction and possibly assuming only one exposed surface after construction should be taken advantage of.

Controllable Values

What can the engineer change or control that will definitely have an impact on deflection estimates? In this section we will consider sensitivity to changes in a number of area that the engineer can either dictate or potentially influence as summarised in the table below.

	Variable	Initially Assumed Value	For Minimum Total Deflection			For Maximum Total Deflection			Potential % Impact	
			Value	Total def	Diff. def	Value	Total def	Diff. def	Total	Diff
Controllable Values										
1	Increase Concrete Grade	C30 (E=33000)	C35 (E34000)	50.9	23.1	C30	55.0	24.9	8.1%	7.8%
2	Increase Slab Thickness 5%	285 to 300mm	300mm	49.5	22.2	285mm	55.0	24.9	11.1%	12.2%
3	Decrease Slab Thickness 5% (and increase reinf't for strength)	285 to 270mm	285mm	55.0	24.9	270mm	60.7	27.9	10.4%	12.0%
4	Adding reinforcement in key areas		44.0T	49.2	22.4	40.0T	55.0	24.9	11.8%	11.2%
5	Cement Class (R/N/S)	N	R	53.5	24.3	S	57.6	26.0	7.7%	7.0%
6	Adjusting Grid		7.5/8.5	39.5	17.9	8.0/8.0	55.0	24.9	39.2%	39.1%

Line 1 - Increase Concrete Grade

Increasing the concrete grade from C30 to C35 reduces deflections by around 8%. Note that:

- There is only a 3% increase in E value between the concrete grades
- In previous section it was shown that varying the E value +/- 20% only has a +/- 5% impact on deflection.
- Changing the concrete grade has a much greater impact because it changes the tensile strength and hence impacts the extent of cracking.

Lines 2 and 3 - Adjusting the Slab Thickness

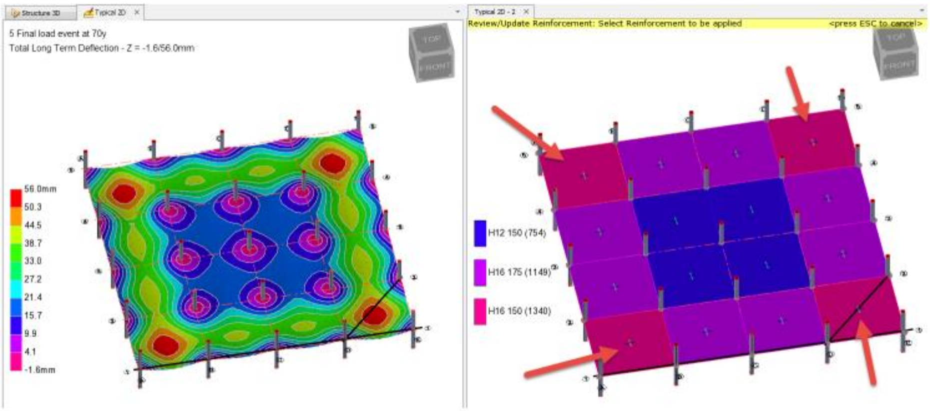
These lines demonstrate that adjusting the slab thickness by 5% will most likely affect slab deflections by something in excess of 10%.

In the case of line 2, where the thickness was increased the reinforcement was not reduced.

In the case of line 3, where the thickness was reduced, the slab was re-designed for strength resulting in approx 5% increase in the weight of reinforcement. For this reason we see marginally less change in deflection when the thickness is reduced.

Line 4 - Adding Reinforcement in Key areas

In this example maximum deflections occur in the four corner panels.



Increasing the reinforcement in these panels will reduce deflections locally. To investigate the impact of this the bottom reinforcement in each of these panels was increased for H16@150 to H20@150 in both directions. This is a 55% increase in reinforcement locally but when averaged over the entire slab area, the total reinforcement content increases by approx 10%.

This targeted 55% local increase in reinforcement delivered just over 10% reduction in deflection in the key panels.

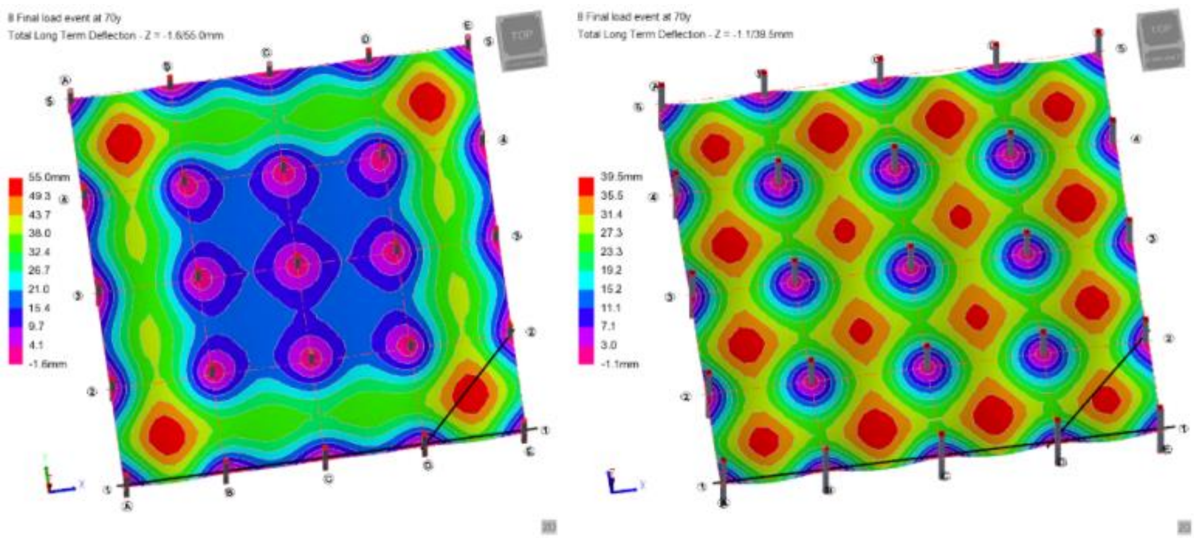
Line 5 - Specified Cement Class

In this example class “N” cement has been assumed. Deflections can be reduced slightly by specifying rapid hardening cement, “R”. In this example the difference between slow, “S” and rapid hardening cement is around 7% impact on deflections, the impact of changing from normal to rapid hardening would be lower than this.

Line 6 - Adjusting Grid

Engineers may generally have very little control over the structural grid, however, if you were in a position to investigate and advise on this there is a clear potential for efficiencies.

In this example, if the edge bays are reduced to 7.5m and the internal bays increased to 8.5m then peak panel deflections become much more even as shown below right.



- Total slab area is unchanged
- The slabs were all re-designed and in this example the total reinforcement increased slightly - so material costs are almost unchanged
- However peak deflections are reduced by around 40%.

There would even be an opportunity for slab thickness reduction if this layout were adopted.

Summary

1. Small changes in slab thickness have a big impact on deflection. The cost implications of trimming an extra 5% off the thickness may be very significant once you deal with the deflection issues this introduces. Advising on this sort of impact is a simple exercise in *Tekla Structural Designer*.
2. If you have a few local issues with peak deflections then local addition of reinforcement is a solution. However, if the problems are more widespread this may not be the most cost effective solution.
3. Increasing the concrete grade may be a more cost effective solution for tackling wide spread deflection issues.
4. If you are in a position to advise on optimising the structural grid then there is a significant opportunity for savings.

Expectation

As noted in the introduction we are looking for deflections in this model to be less than 45mm to meet the normal Span/250 total deflection limit. Throughout the exercises above we never actually recorded a deflection lower than this value.

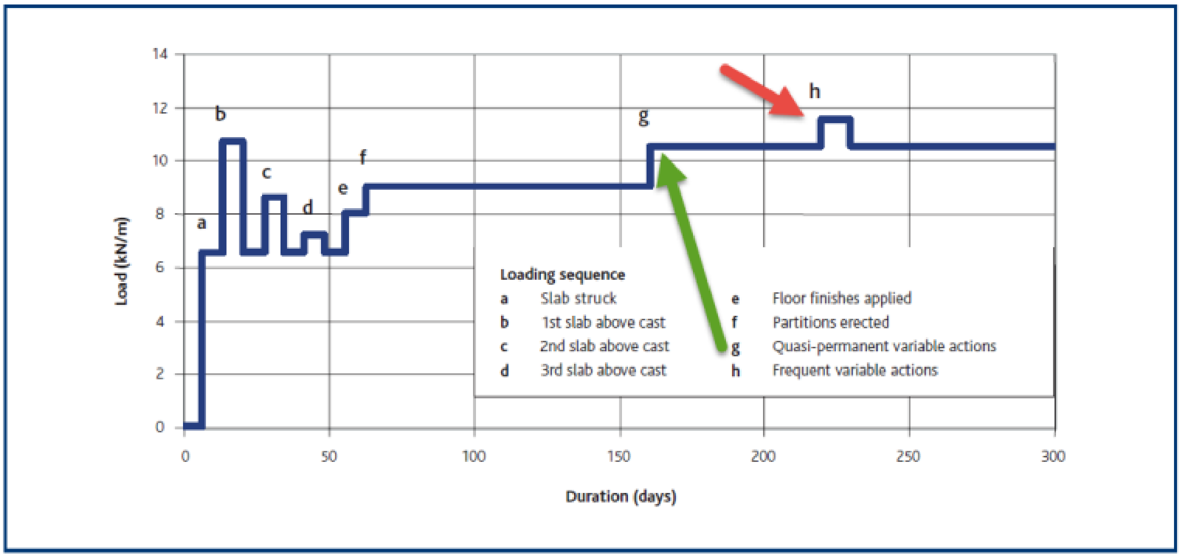
This is to be expected. Rigorous deflection estimation procedures will not readily prove that slabs which have always been assumed to be ok based on "deemed to satisfy" checks meet the assumed deflection limits.

Consider a paper by Dr Robert Vollum of Imperial College *"Comparison of Deflection Calculations and Span to Depth Ratios in BS8110 and EC2"*.

This deals with several aspects of the various methods of deflection checking:

1. It is noted that Span/Depth checks in Eurocode 2 allow thinner slabs than would have been deemed adequate to BS8110.
    - For this example model the older BS8110 version of the economic concrete frame elements guide would have suggested a 300mm slab.
    - This work contributed to limits being applied to the Eurocode guidance in the UK National Annex
2. In the rigorous deflection calculations it is noted that Eurocode 2 only required deflections to be calculated for the quasi permanent condition (load level g on the diagram below).

Figure 2
Loading history for a slab – an example



This may be as little as 30% of the IL and there is no requirement to consider the balance of the IL (event h in diagram)

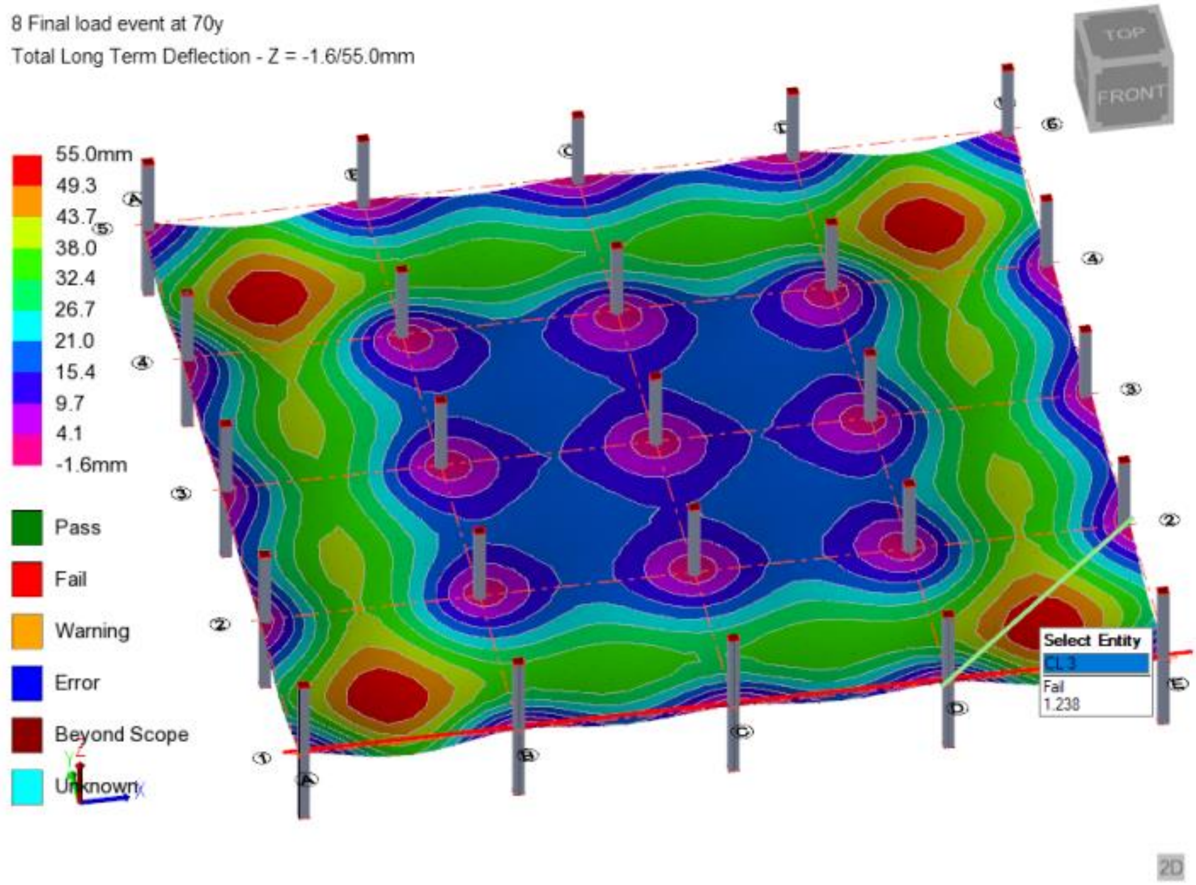
Quite understandably the document questions the logic of this and recommends the instantaneous application of the frequent variable action be considered.

3. The study finds that rigorous deflection estimates will not readily prove that slabs sized on the EC2 span/depth rules meet the EC2 deflection limits. This leads to a suggestion that the assumed deflection limits be re-considered and harmonized across materials:

- Total deflection – span/250 should be span/200 ?
- Deflection affecting sensitive finishes – span/500 should be span/360 ?

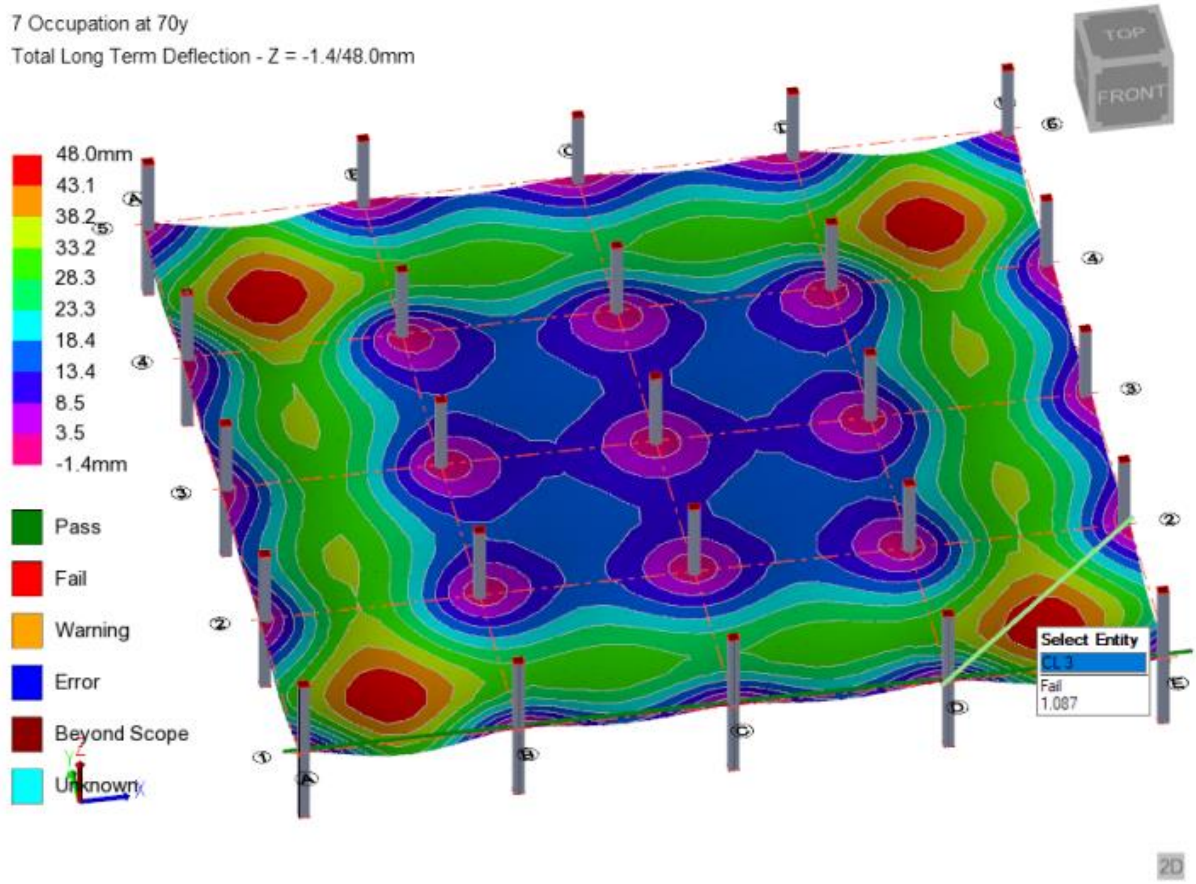
Settings that would Work

So is there some reasonable set of settings that will show this slab to be borderline acceptable when rigorous deflection estimation is used? To achieve this you would probably have to stick to the stated EC requirement about considering only quasi-permanent loading.



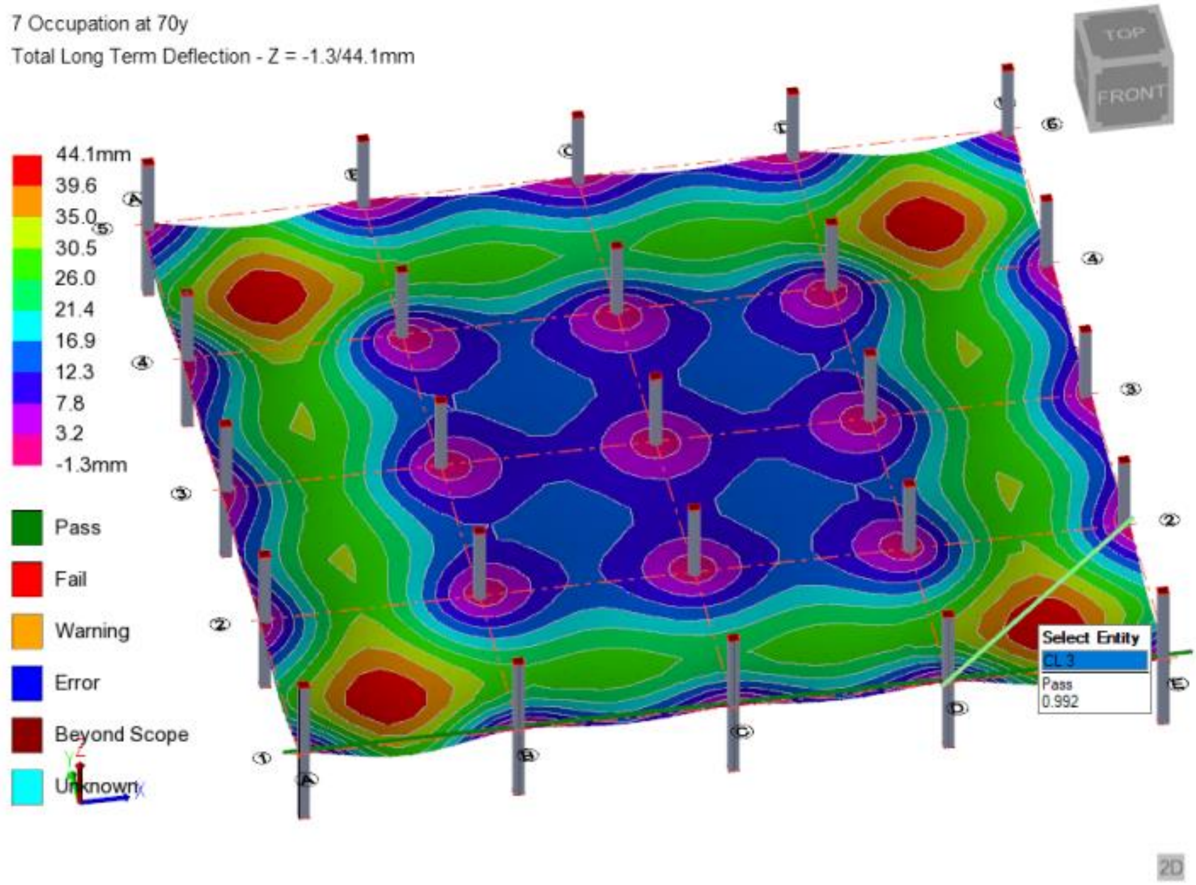
View above shows total deflections before and view below shows the result after making some changes:

1. Set restraint = 50% (clearly ok in this simple example situation)
2. Set propping loads to 55%, 35%, 10%
3. Set temperature of 10 degrees and RH=80% during construction.
4. Set the deflection checks to consider the quasi-permanent load condition (i.e. ignore the final event)



If you are happy to ignore the balance of imposed load then the above gets us very close to something that works (corner panel fails by 9%). If you wished you could also add checks relating to the frequent loading condition where the lower deflection limits get applied.

To control the deflection a little more you could either add reinforcement in the corner panels or increase the concrete grade. Increasing from C30/37 to C35/45 has the additional impact shown below.



Summary

A lot of theory and input parameters has been discussed in this section, however, we should never lose sight that it is not an exact science. Remember it is a rigorous deflection estimate based on a variable material property and input parameters and event timings that are perhaps not accurately known until a contractor is appointed? TR58 recommends +15% / -30% accuracy be considered, but is this more applicable when realistic/average assumptions are made rather than if worst case assumptions were made throughout?

We should not expect to get better answers than "deemed to satisfy" rules. We may find that ordinary slabs that could be deemed adequate would now fail. However, it should be borne in mind that deemed satisfy methods are:

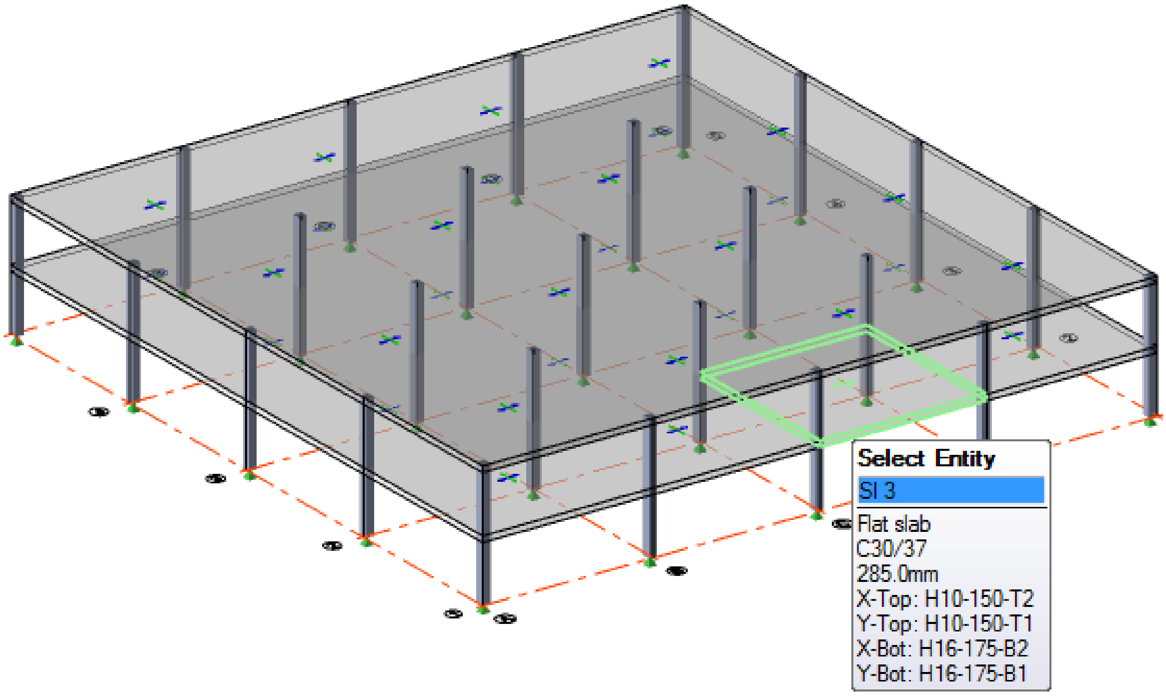
- Time consuming at best of times
- Awkward if you have to allow for irregular geometry or loading.
  - In such circumstances conservative idealisations start to be used (more conservative than a rigorous approach?)
- Only applicable if "Normal Circumstances" apply:
  - If a "normal" pace of construction applies
  - If normal deflection limits apply… (Curtain Walling seems to be the most common exception to this)

Our focus for the slab deflection estimation feature was to make it easy to use and hence to improve productivity. Displacement contours are available for manual review, however, check lines with Pass/Fail status and Utilisations make for a much more productive working environment.

Slab Deflection Example (Eurocode)

Model Details

The example is a simple multi-bay flat slab model 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".



- 8m grid
- 285 thick C30/37 slab
- 400 square columns
- Loading:
  - Finishes - 1.5kN/m2
  - Imposed - 5.0kN/m2
  - Perimeter Cladding - 10.0kN/m
- Slab panel and patch design has already been performed to establish reinforcement requirements.

The above Eurocode slab deflection example model can be downloaded by opening the following address in your default browser:
https://teklastructuraldesigner.support.tekla.com/support-article/2816529

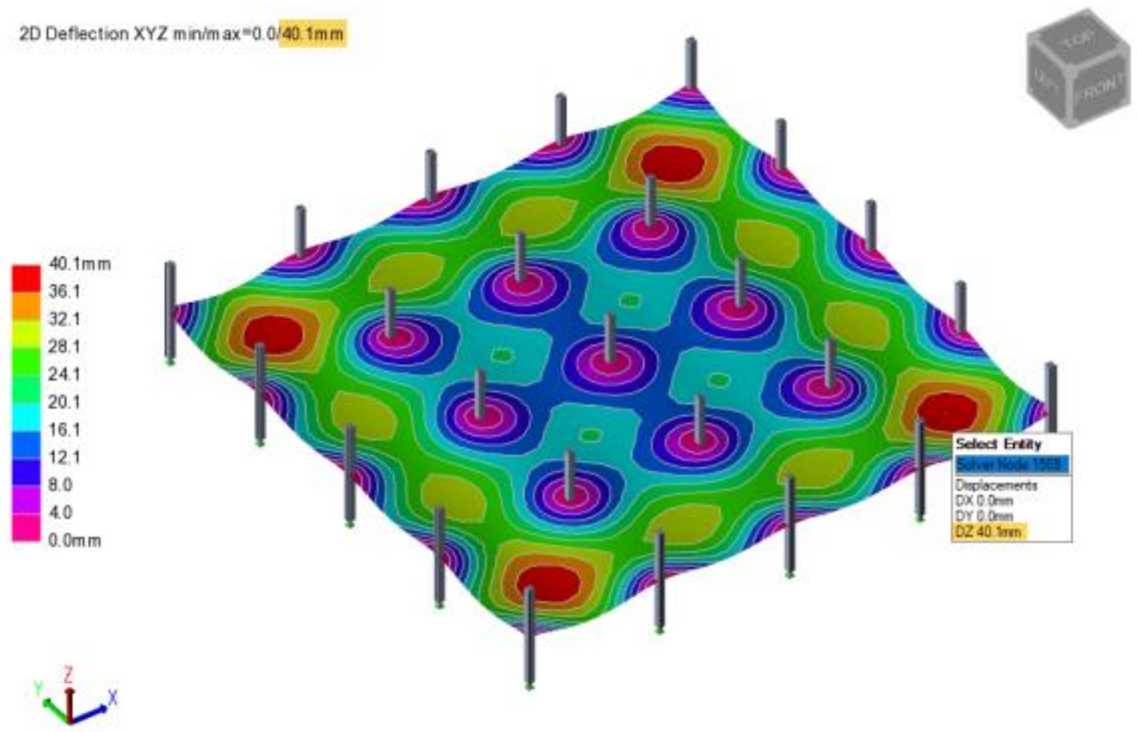
Deemed to Satisfy Checks

First let us consider the deflection using the simplest approach that is available in *Tekla Structural Designer* - linear analysis using adjusted analysis properties.

This is achieved as follows:

1. From the **Analyze** toolbar, run **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 chasedown analysis for the load combination 1, service load results

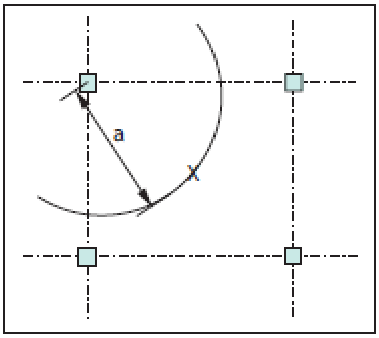
The resulting deflections are as shown below:



We can see that the maximum reported deflection is 40.1mm.

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 $2a/n$.

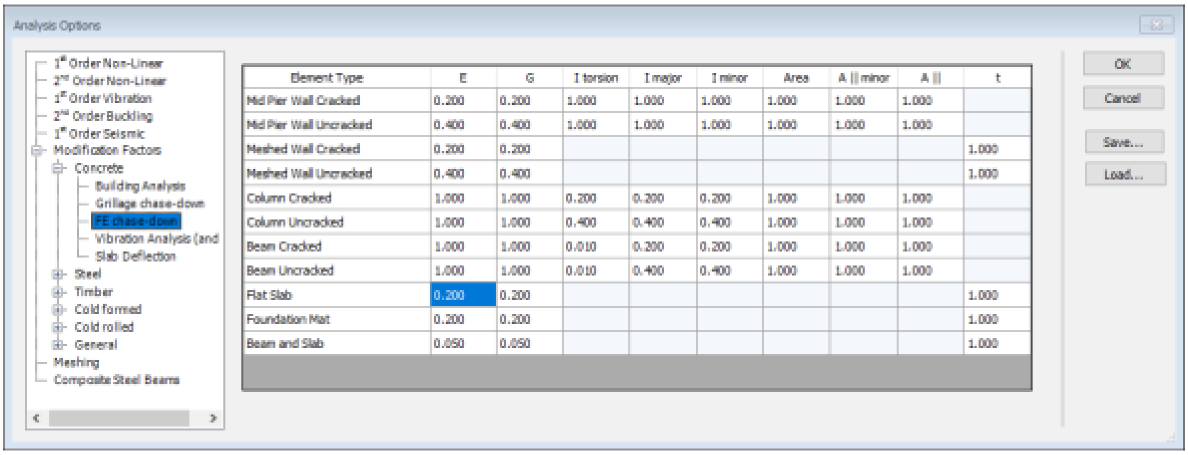
(Maximum deflection on gridlines may be more critical.)

Taking the diagonal dimension across the columns, the deemed-to-satisfy span / 250 rule provides a deflection of [√(8000^2+8000^2)] / 250 = 45.3mm.

This compares favourably. The analysis result is, however, dependant upon the concrete E properties used in the analysis which are adjusted by a factor, 0.2xE to consider such things as creep, cracking and shrinkage.

The Analyse ribbon, Analysis Options - Modifications factor page controls the value of E used in the analysis. This is a User defined value with a default of 0.2

Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 250.



Element Type	E	G	I torsion	I major	I minor	Area	A || minor	A ||	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 Approach (Eurocode Slab Deflection Example)

Rigorous Approach

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: [Slab Deflection Parameters]

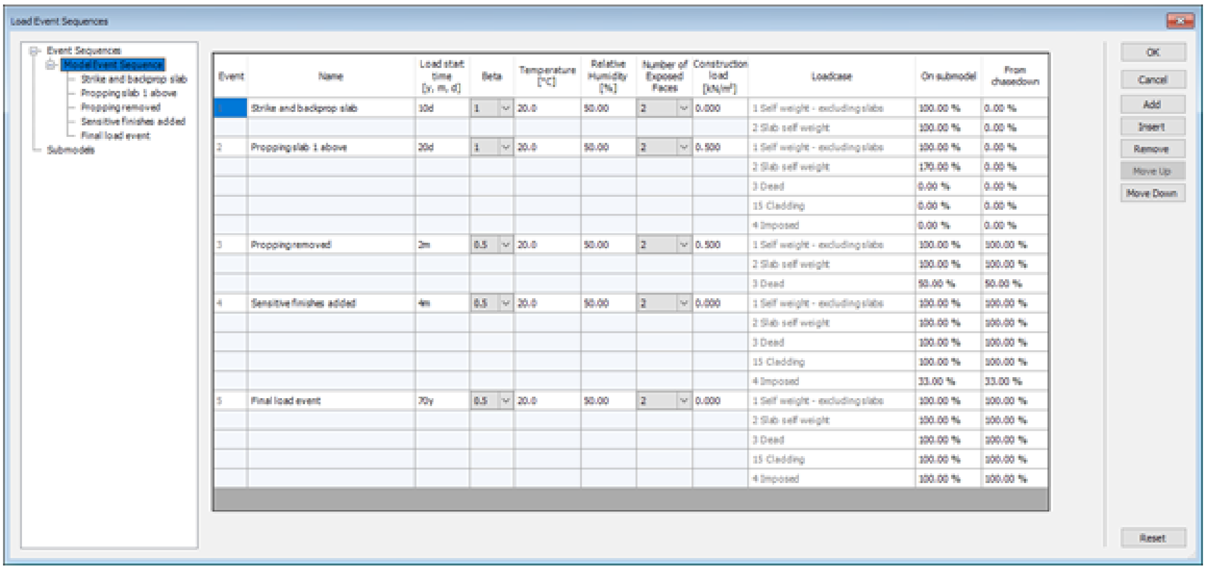
For this exercise, initially it will be assumed that the default settings have already been reviewed and set as required.

1. Review the Model Event Sequence

To review the model event sequence:

1. Click **Slab Deflection > Event Sequences**
2. Click **Model Event Sequence**

This has already been setup for this example as shown below:



- Deflections to the end of each of the above event periods will be calculated by the analysis.
- Lots of input parameters on the dialog will affect these deflections. The sensitivity of the analysis to each of these parameters is tabulated in: [Sensitivity guidance (Eurocode)].

Additional 70% allowance for propping load from slab above too high?

2. Perform Iterative Slab Deflection Analysis

To establish some initial results (with all parameters left as default values):

1. From the Slab Deflection toolbar, click **Analyse All**

In a real model rather than analysing all levels at once, you might instead decide 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.

After analysis the current view automatically switches into the Slab Deflections View regime.

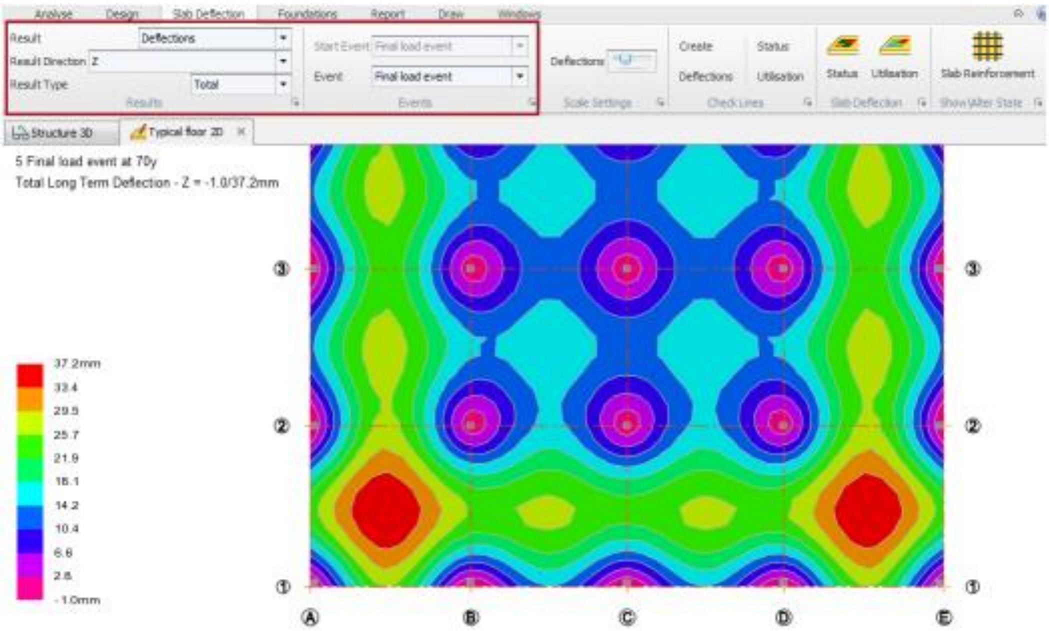
3. Review Deflections for Events

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.



As a comparison with the simple approach earlier (40.1mm), the Total deflection at the final load event for the chosen location is 37.2mm.

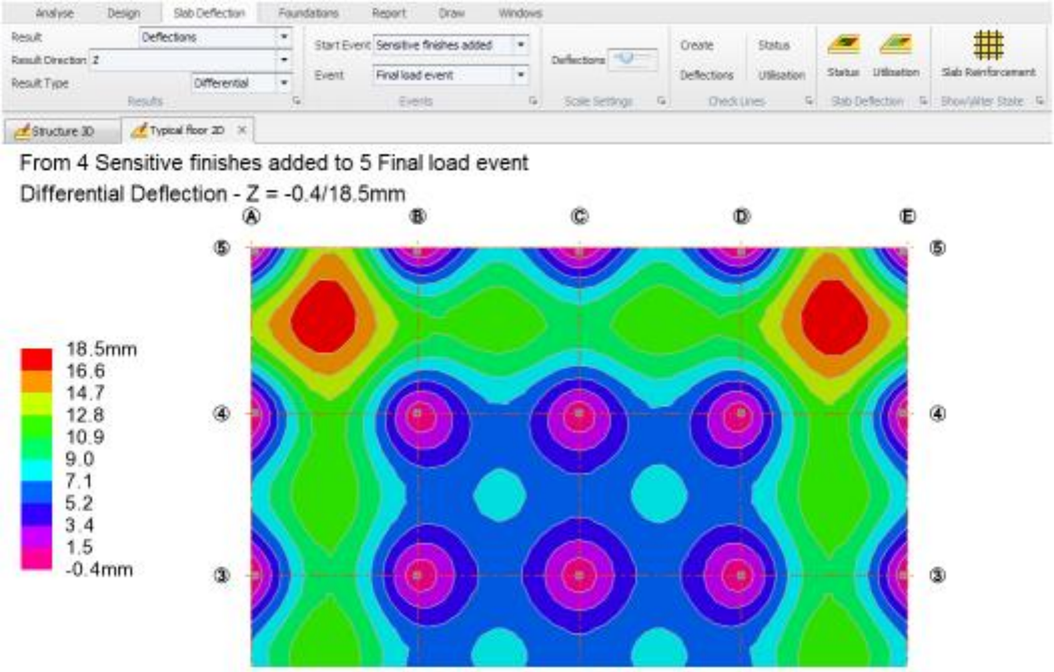
In the above contour plot the deflections are not exactly symmetrical - this is because reinforcement is in the outer layer in the Y direction making the slab stiffer in that direction.

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.

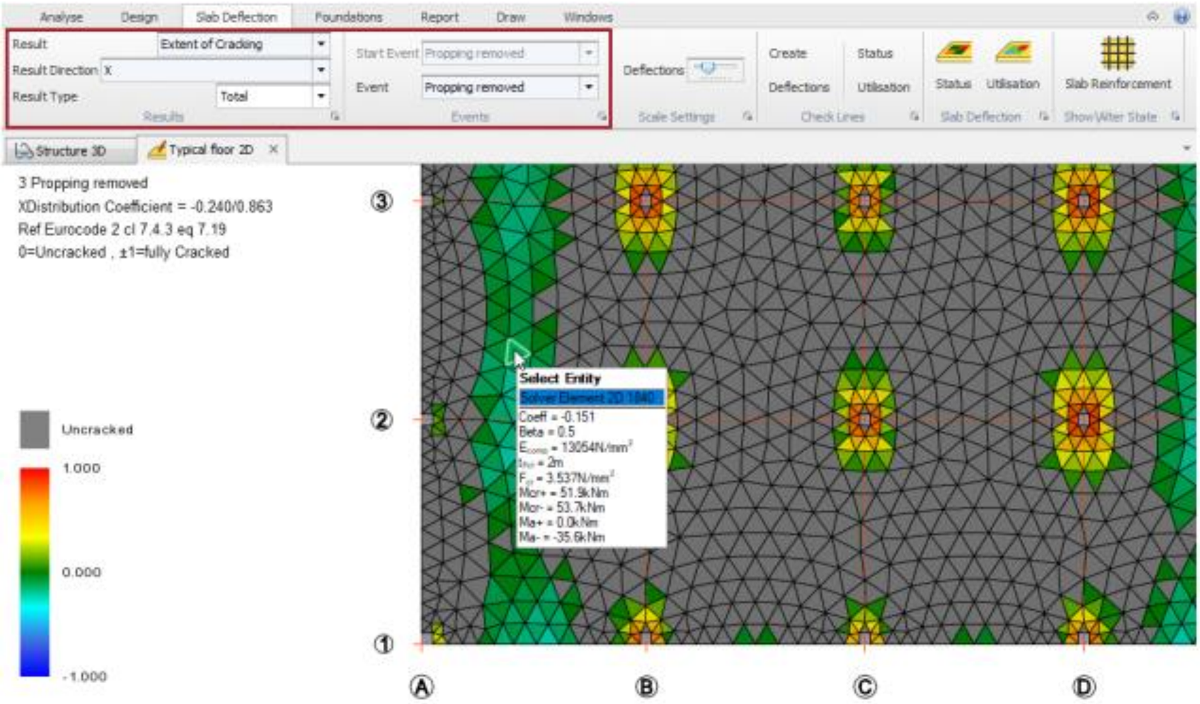
1. From the Slab Deflection toolbar, change the Result Type to **Differential**
2. Select the Start Event as **Sensitive Finishes added**
3. Select the Event as **Final load event**

The 2D view now displays the differential deflections between "Sensitive Finishes added" and the final event.

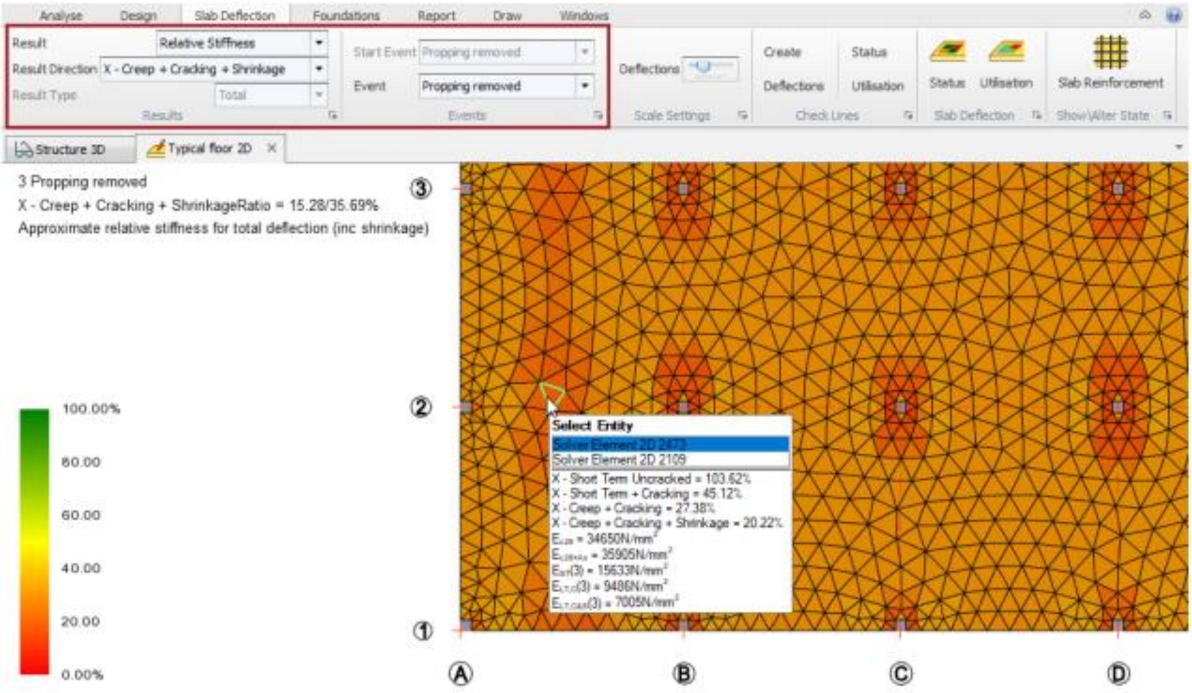


4. Review Other Results

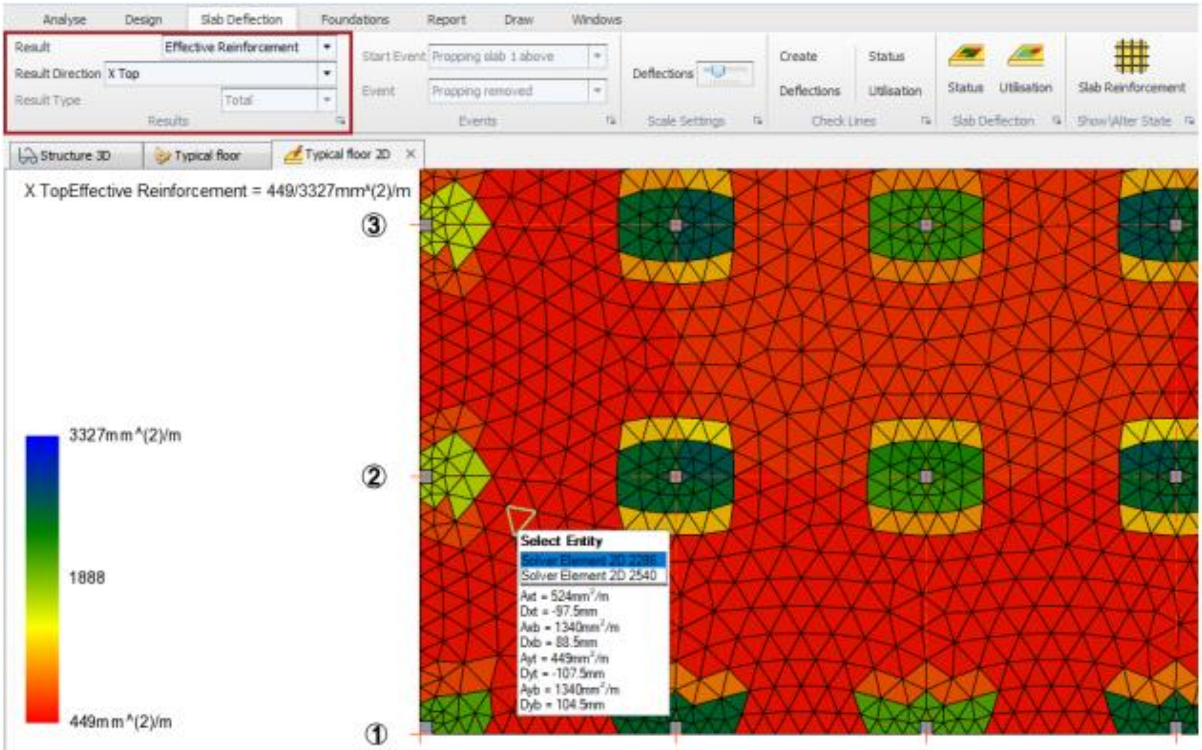
In addition to the deflections you can display the extent of cracking at any load event.



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.



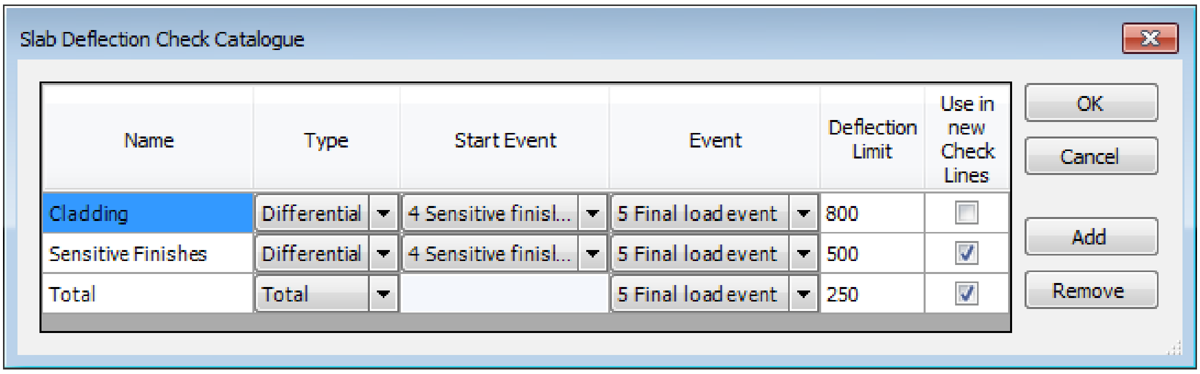
When you hover over any FE element in the slab deflection view regime - values are provided within the tooltip.

5. Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgement.

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.



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

6. Place Check Lines

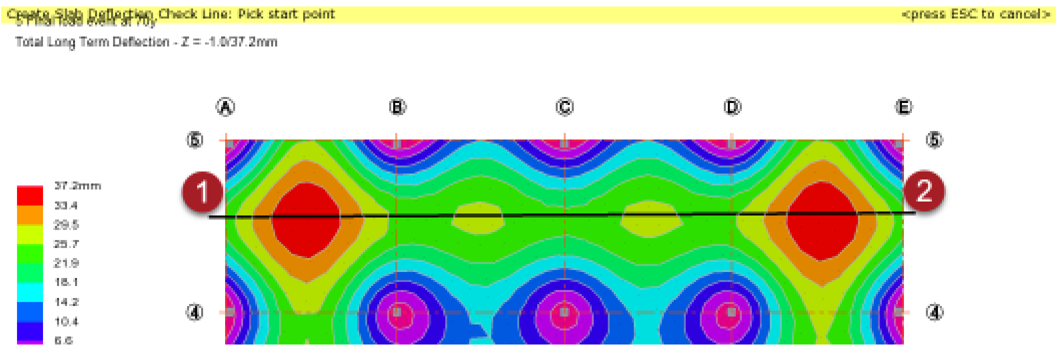
We will define several check lines in this example:

1. Ensure you are in the **Typical floor** 2D plan view and then click **Create**

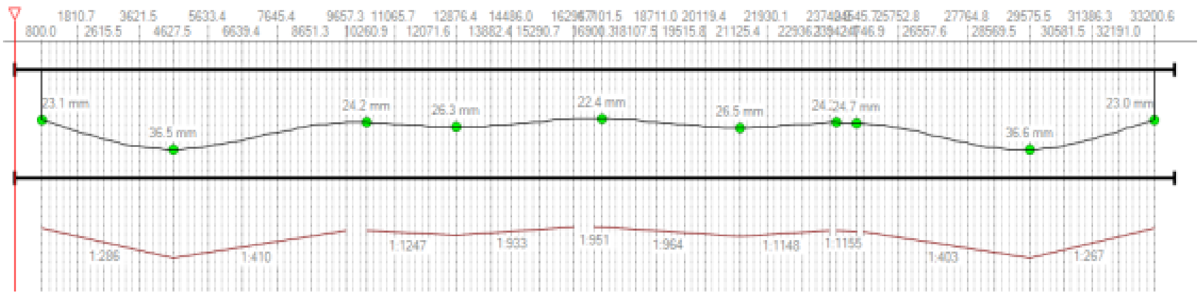
Check lines can only be created in a 2D view. If the Create command is greyed out ensure you switch to a 2D view.

When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue where "Use in new Check Lines" was checked.

2. Place the check line at approximately mid-span between gridlines 4-5 from gridline A to E as shown below.



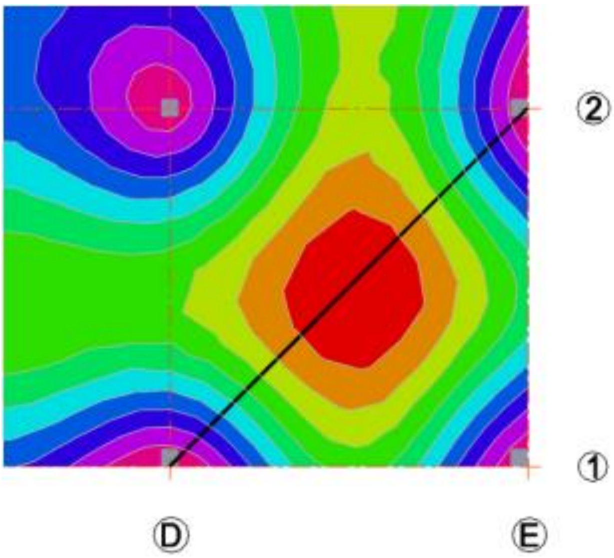
3. Right click on the check line and choose **Open deflections check view** from the context menu to see the deflection results along its length.



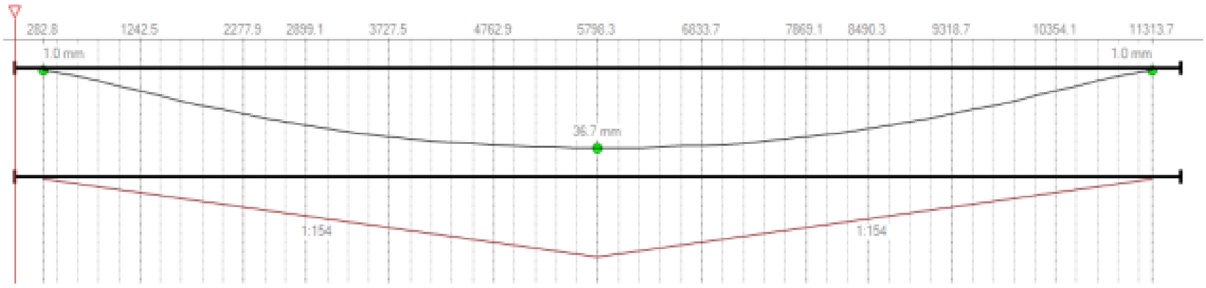
The ribbon allows you to specify the total (as shown above), or differential or instantaneous results for the selected events. *Tekla Structural Designer* then draws average slopes between maximum and minimum points.

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.

4. Create a check line running diagonally between columns in bottom right corner panel where the peak deflection occurs (between gridlines 1-2 / C-D).



5. 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.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

7. 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**

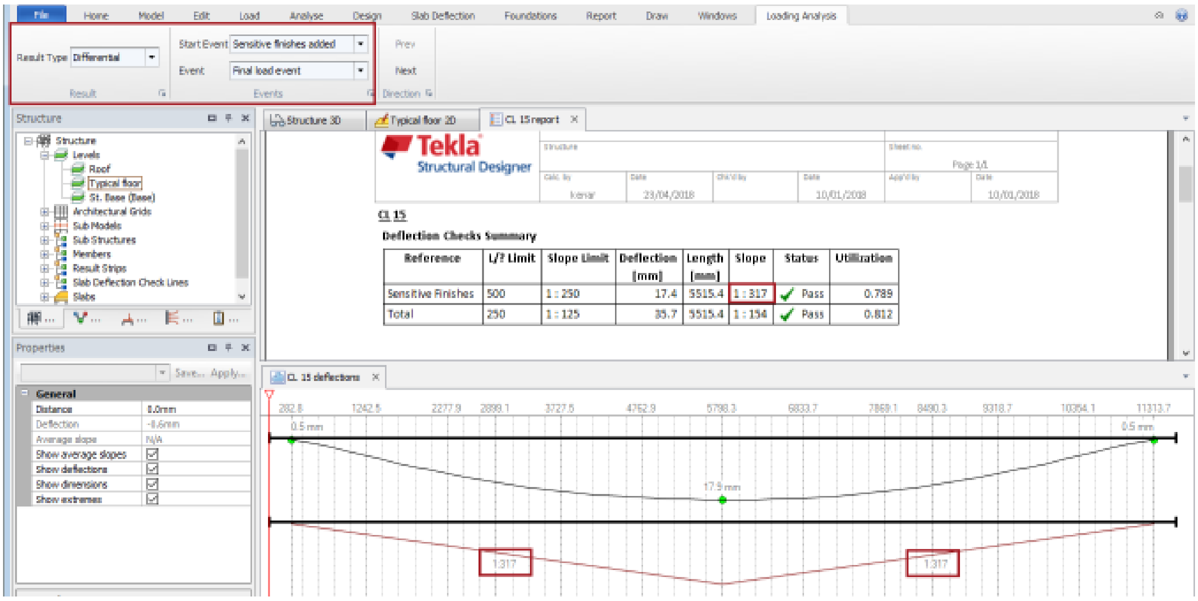
Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	17.4	5515.4	1 : 317	✓ Pass	0.789
Total	250	1 : 125	35.7	5515.4	1 : 154	✓ Pass	0.812

The slope above is reported as 1:154 which is not less than the allowable slope limit of 1:125 and hence a Pass.

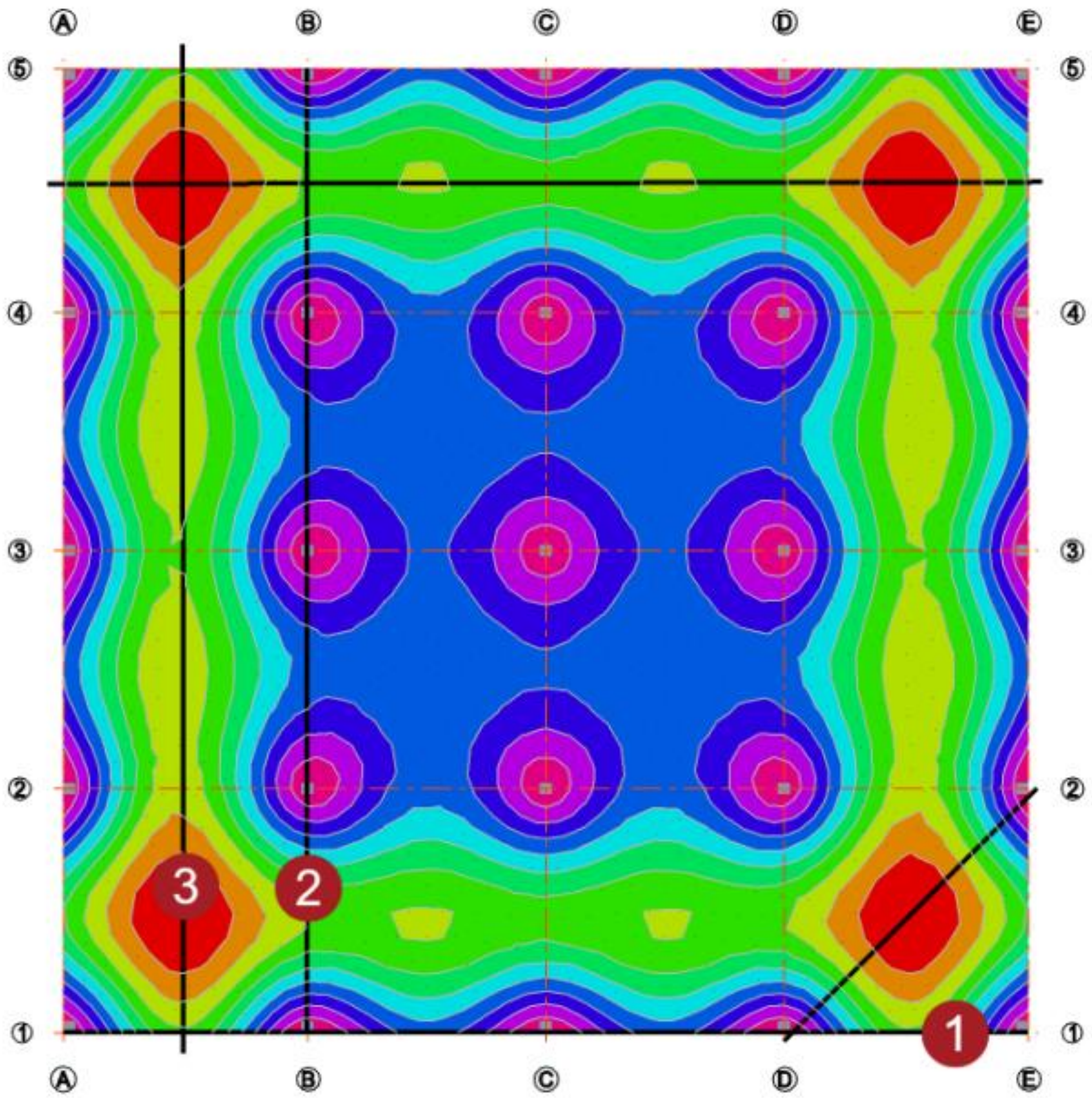
If you click within the Load Analysis Deflection view, the ribbon changes to allow you to display deflection results and slopes for the Result Type - Total or Instantaneous for a chosen event, or Differential between chosen events.

2. Click within the Deflection load analysis view and 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.

1. Add further check lines along grid line 1 and B and between grid line A-B using the default deflection checks in the catalogue as shown below, then 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.

1. Select the check along grid line A/1-E/1 and ensure that it also has a **Check #3** defined as **Cladding**.

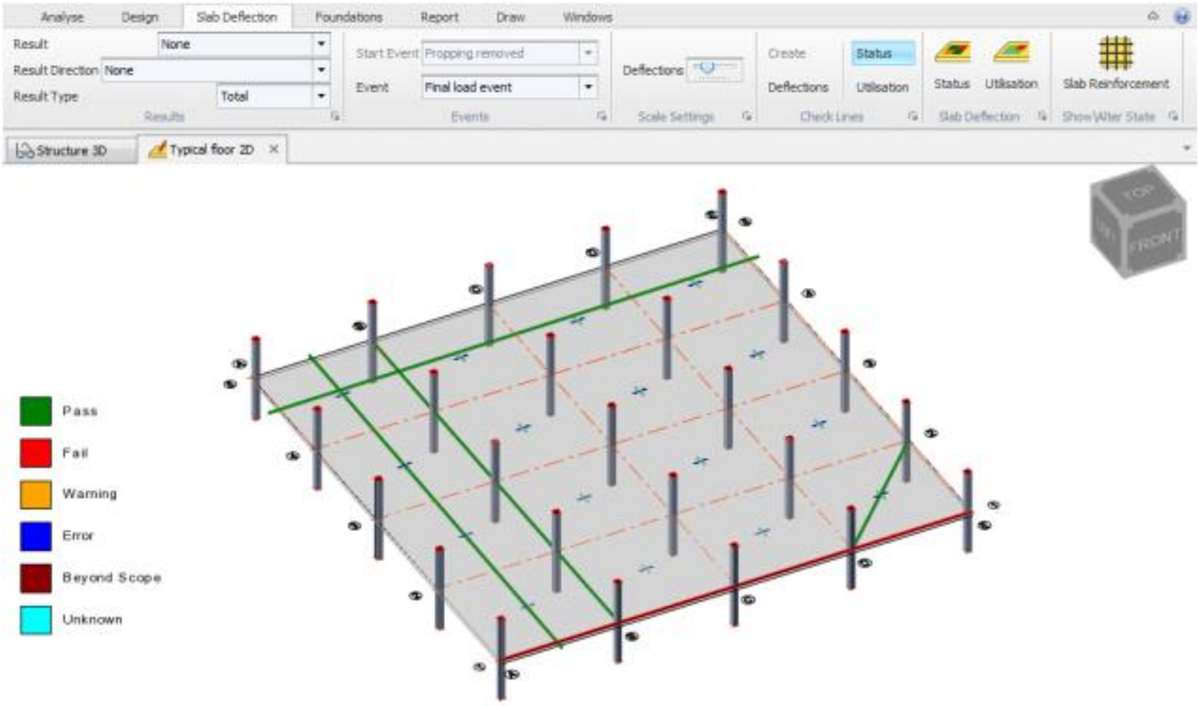
8. Review Check Line Status and Utilisation

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 Utilisation ratio.

1. Click on the Typical floor 2D view to make it active.

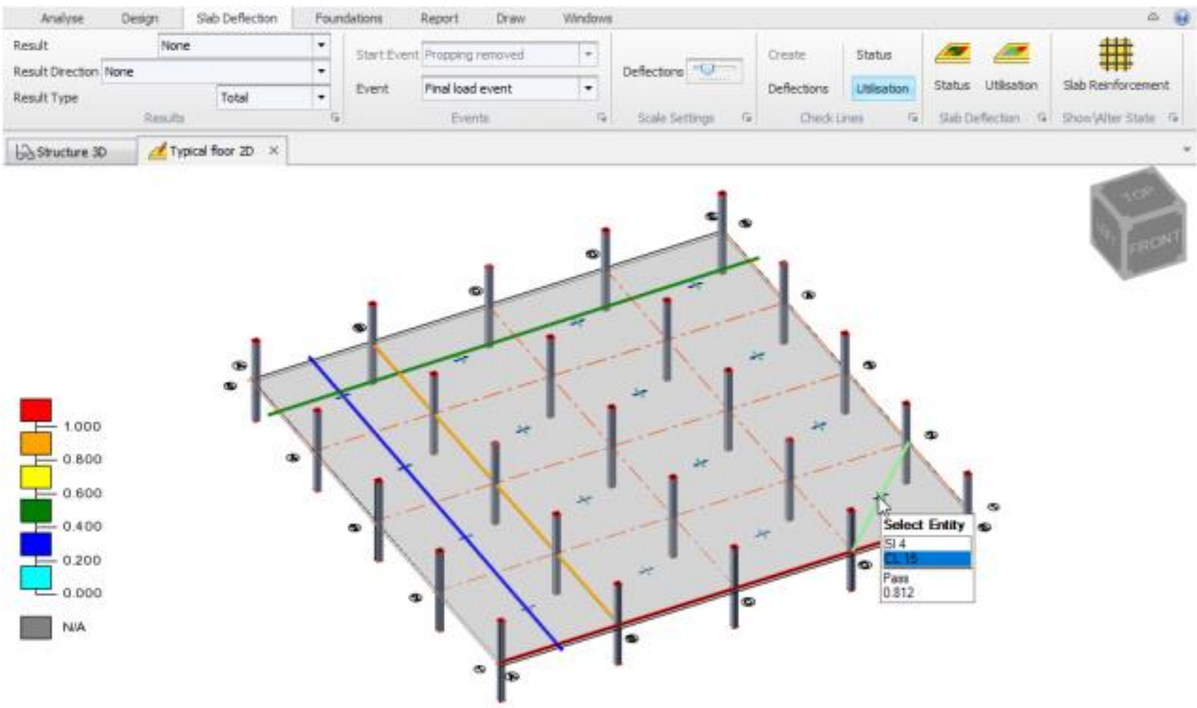
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.

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 utilisation and pass/fall status.*

4. Click **Utilisation** in the Check Lines group to show the critical utilisation for each check line and investigate the tooltip results.

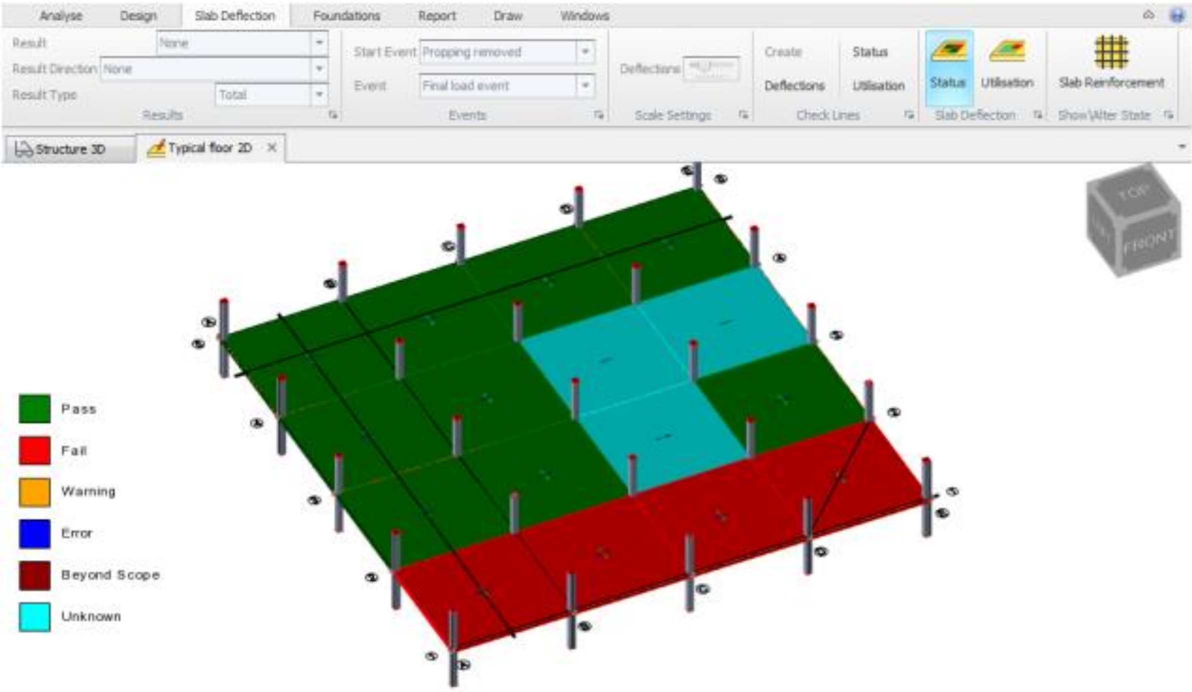


Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilisation

9. Review Slab Status and Utilisation

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.

1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/fail status of each slab.

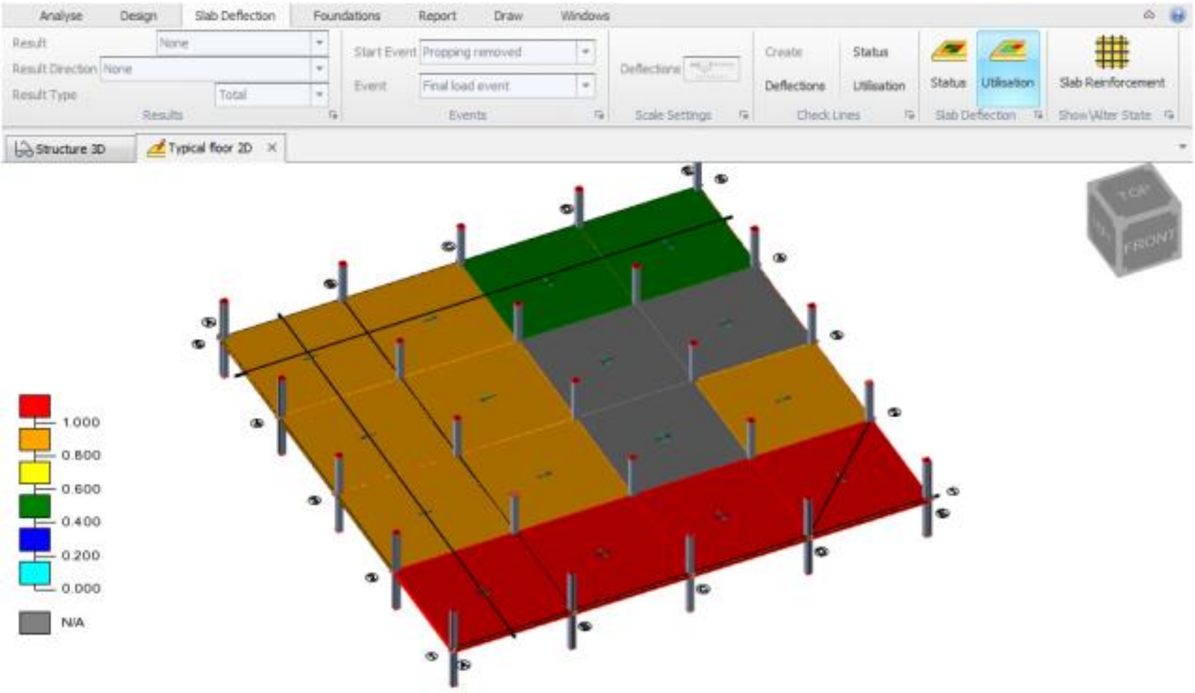


- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed.
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the slab items Fail where the check lines runs along A/1-E/1.

The Utilisation can also be reviewed.

1. Click **Utilisation** in the Slab Deflections group to show the Utilisation of each slab item.

   This is the worst utilisation from all associated check lines.



10. Optimisation

If you find that a slab either fails deflection checks (or passes with a low utilisation) 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 optimisation.

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 utilisation.

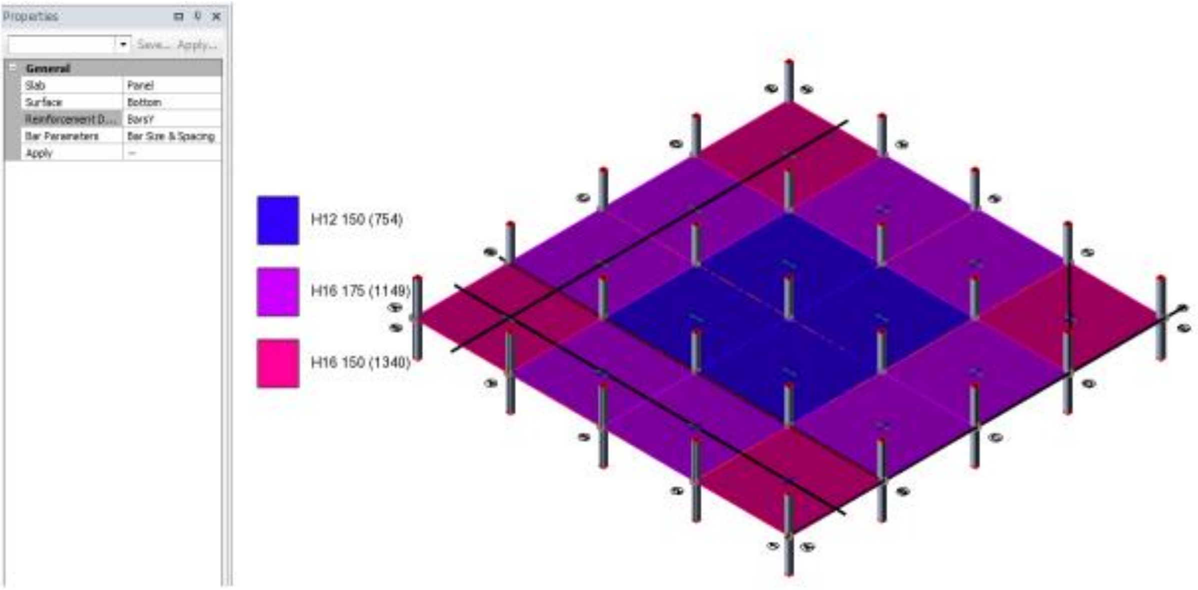
TIP!

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.

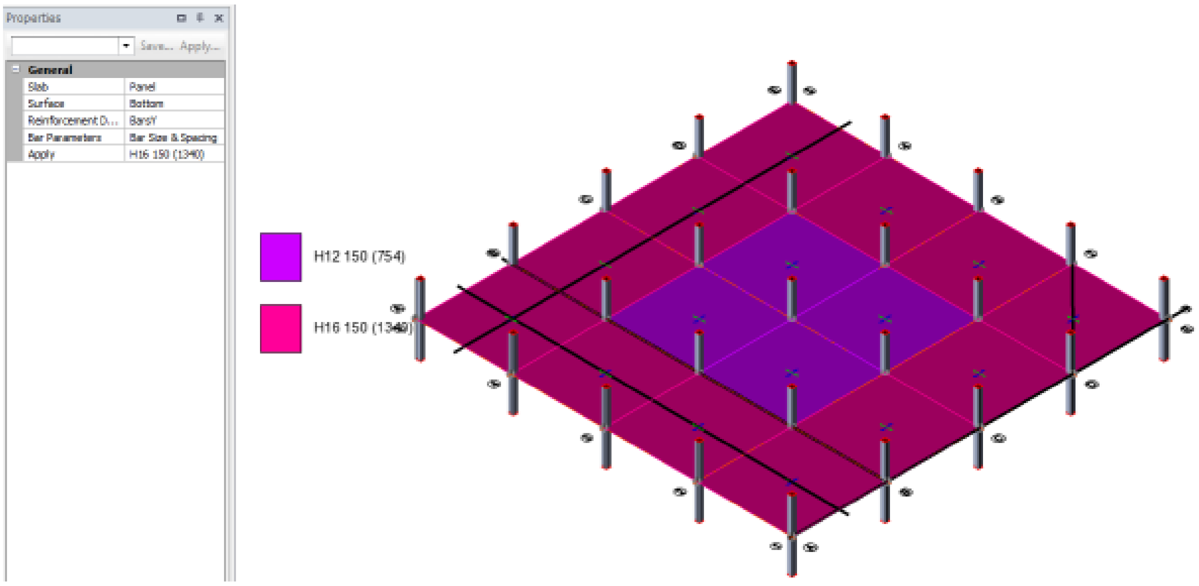
3. In the Properties Window:

- Leave the Slab Reinforcement to modify as **Panel**
- Leave the Slab Layer to modify as **Bottom**
- 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. Click **Analyse Current** to update the results

7. Click **Utilisation** in the Check Lines group to show the critical utilisation for each check line once again.

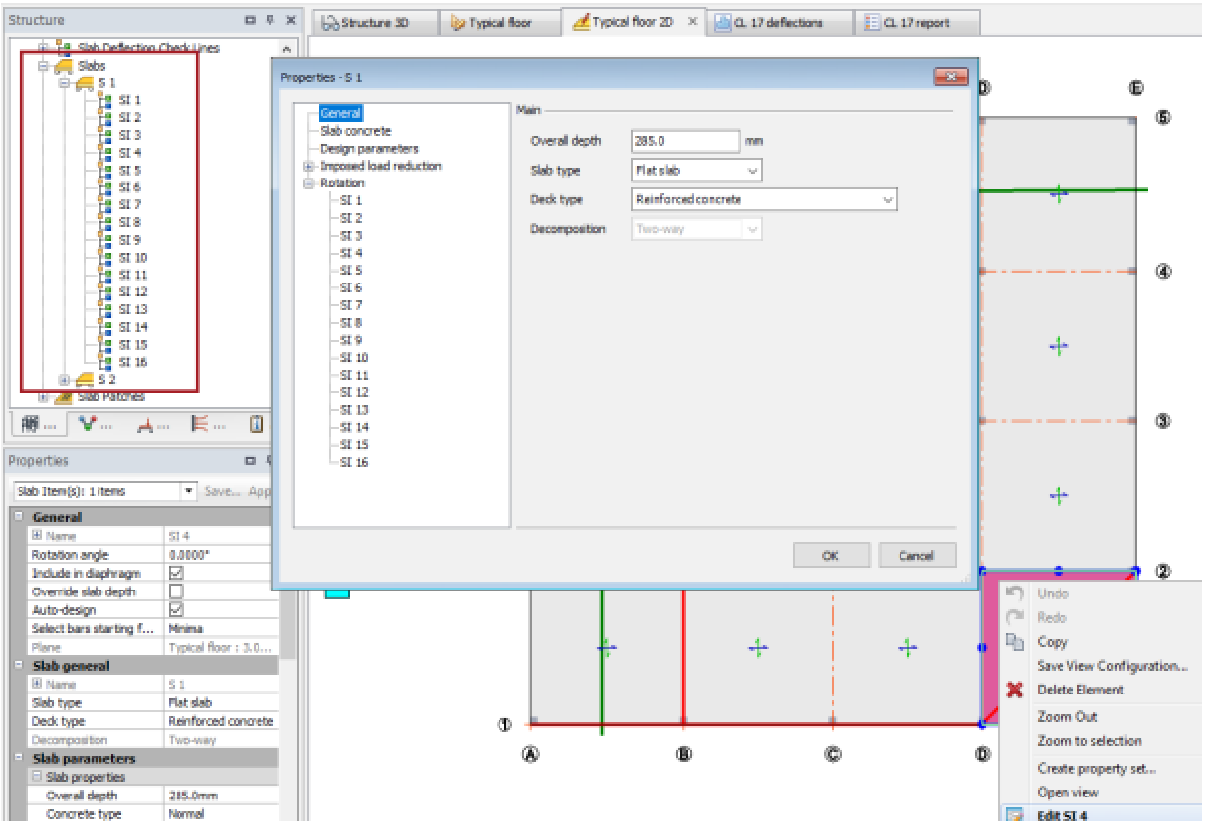
8. Investigate the tooltip results

You should find that although some utilisations 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.

9. Right click slab **Sl 4** between D-E/1-2 on the typical floor level and choose **Edit....** from the context menu.

Note that we right clicked a slab item 4, Sl4, however, the Properties are for S 1. This is the slab group and contains all the slab items identified under the rotation page.

You can also determine the slab items contained in group 1 via the Project Workspace Structure window under the Slabs branch.



10. On the **Slab concrete** page of the dialog, change the Concrete Strength to **C35/45** and click OK

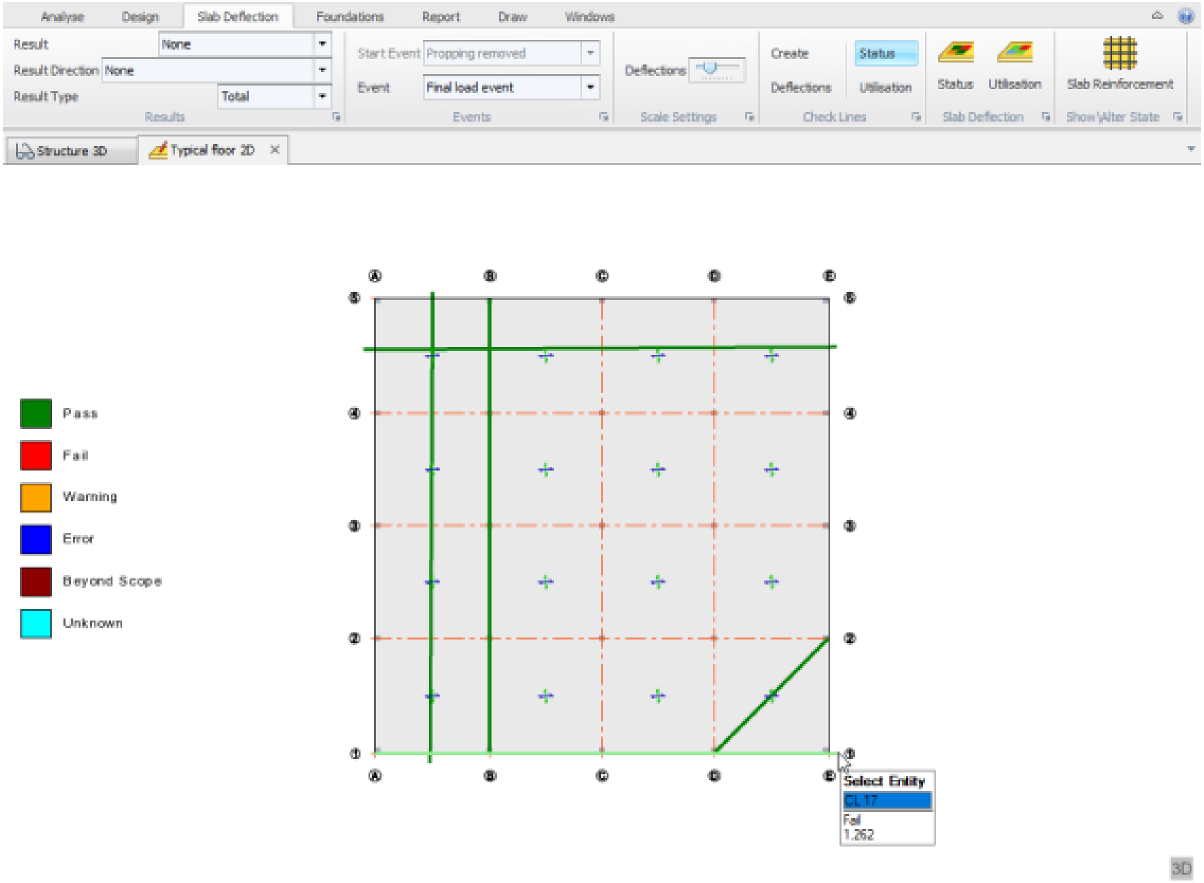
To update the check line results we need to re-run the analysis. A chasedown analysis is automatically performed as part of the slab deflection analysis, however, it should be borne in mind that some edits could affect the element design i.e. reducing the concrete slab thickness would result in an increase to the required reinforcement and hence a Design Concrete (Static), Slab or patch design may be required again.

11. Click **Analyse Current** again to update the results.

12. 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.

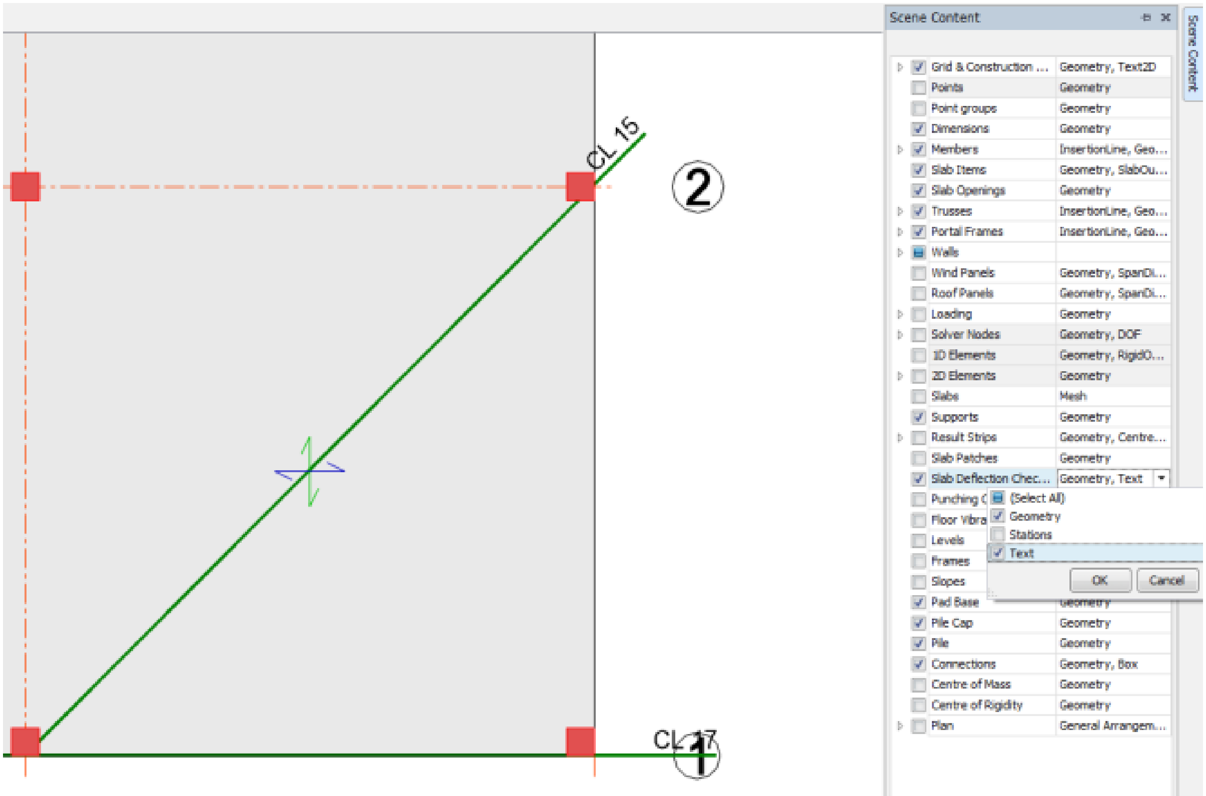
In fact, to obtain a Pass we would have needed to increase the concrete grade to C45/55. Note that this cladding check is not a code requirement.



11. 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.

   We will now generate the report structure that includes a view and the check lines that have been placed on the typical floor level.

3. On the Report ribbon, click **Model Report...**

4. Click **Add** and provide a Name “Check Line Report” for the report.

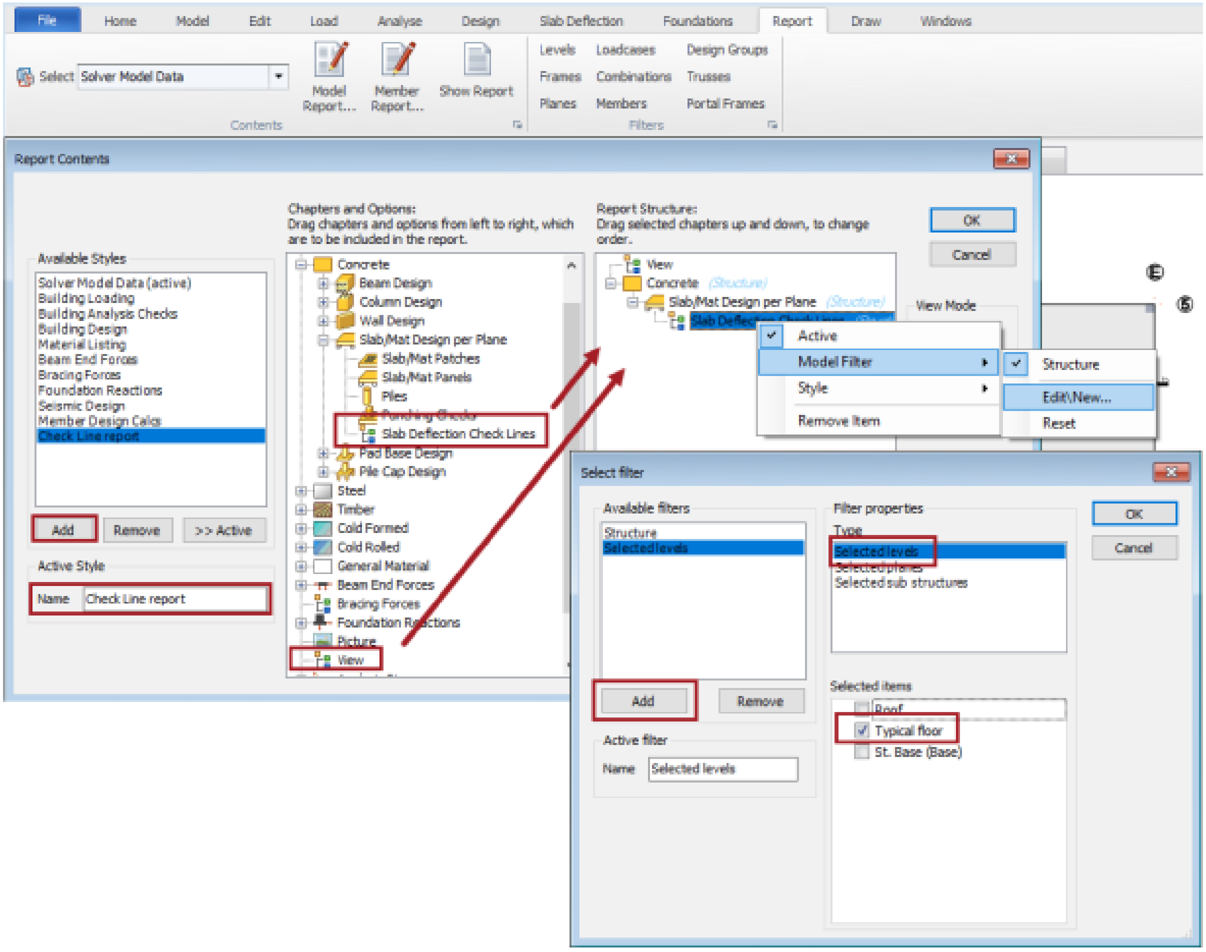
5. In Chapters and Options, drag **View** to the Report Structure area

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 C**oncrete> 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.

To display the report.

1. Use the Select drop list in the ribbon to select "Check Line Report"

2. Click the Show Report command to open the report.

Foundation Design Handbook

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

The following isolated foundations can be designed:

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

In addition, the following mat foundations can be designed:

- **Mat foundation** - a foundation supporting multiple columns and walls on ground springs
- Piled mat foundation - a foundation supporting multiple columns and walls on pile supports.

Isolated foundation design

- Pile cap design procedures

Overview of the isolated foundation analysis model

Association with member supports

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

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

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

Analysis types

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

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

Design forces and checks

Forces acting on supports

The following forces and moments on the supports are determined from the analysis of the active load combinations:

- Vertical force in direction Z
- Horizontal forces in directions Y and X
- Moments around X and Y axis

A torsional moment around the Z axis is also determined, but the base/pile cap is not designed for this in the current release.

Foundation self weight

The foundation self-weight is automatically calculated and applied as an added load, F_{swt}

Soil self weight

The surcharge depth and soil unit weight that have been specified in the base/pile cap properties are used to determine the soil self weight. This is applied as an added load, F_{soil}

In the current release, horizontal pressure caused by soil is not considered.

Additional surcharge loads

For isolated foundations user can apply additional surcharge loads: acting in the global Z direction.

- Permanent (dead) surcharge load
- Variable (live) surcharge load

Design Forces

To obtain the design forces, the loads due to foundation self weight, soil self weight and additional surcharge are added to the forces acting on the supports.

These design forces (axial load and bi-axial shear and moment) are then applied to the base/pile cap at the foundation level.

Design Checks

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

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

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

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

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

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

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

Pad base and strip base design procedures

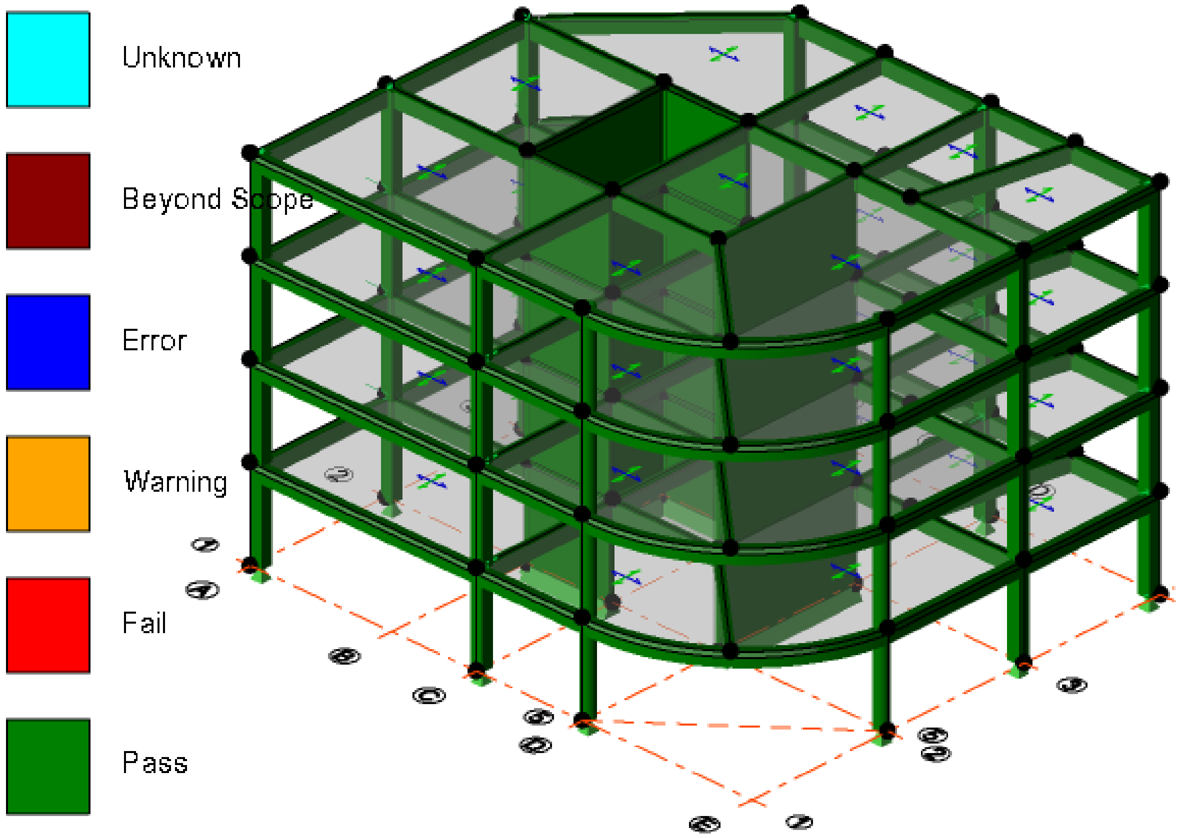
The overall procedure is demonstrated in the following Pad base design example (but is basically the same for strip bases also).

The typical steps required are as follows:

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

Pad base design example

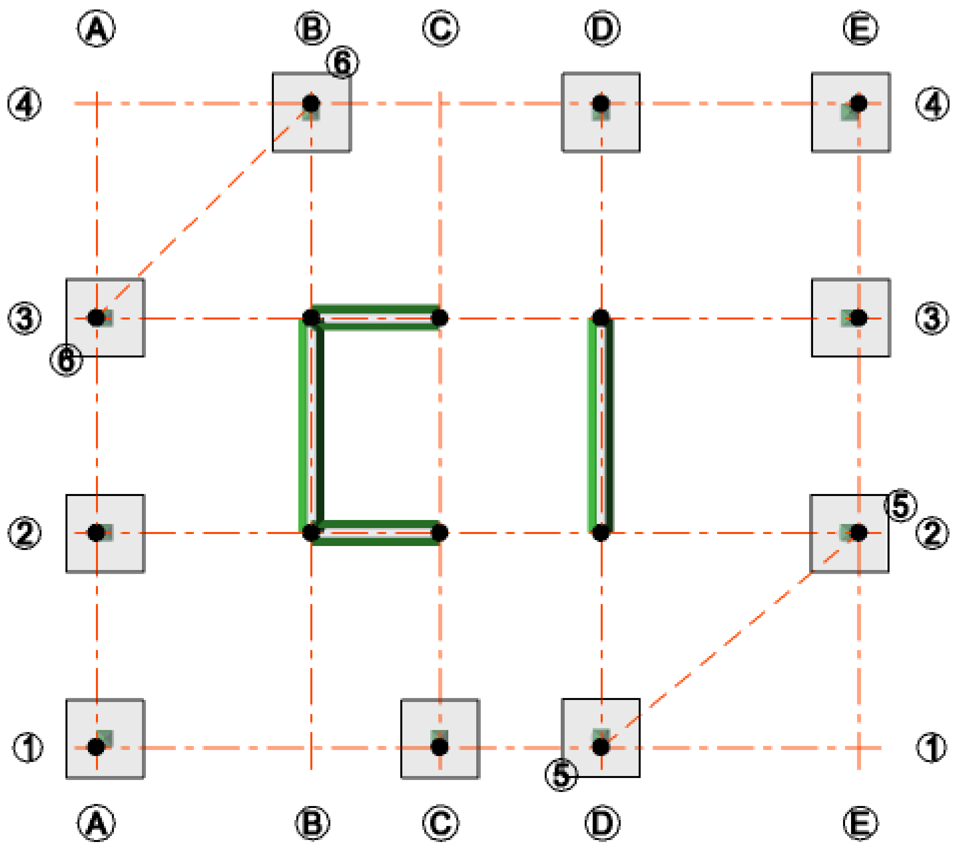
The small concrete building model shown below will be used to demonstrate the base design process.



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

Apply bases under supported columns

At this stage, as you are not aware of the individual base size and depth requirements; you can simply choose to place the bases where required, accepting the default size/depth offered.



Auto-size bases individually for loads carried

To obtain an idea of the range of potential sizes, bases should initially be designed individually for their respective loads, as follows:

1. Access Design Options to ensure that group design is turned off for Isolated Foundations.

2. Select the bases to be auto-sized and in the Properties Window and choose to auto-design both the size and depth; In this way the program establishes suitable base dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

The overall auto-design procedure is summarised as follows:

1. Bearing Design: *- increase size*

2. Bending Design*: - increase reinforcement*
- If max allowable reinforcement is reached then increase depth, set reinforcement back to starting point, and go back to step 1.

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

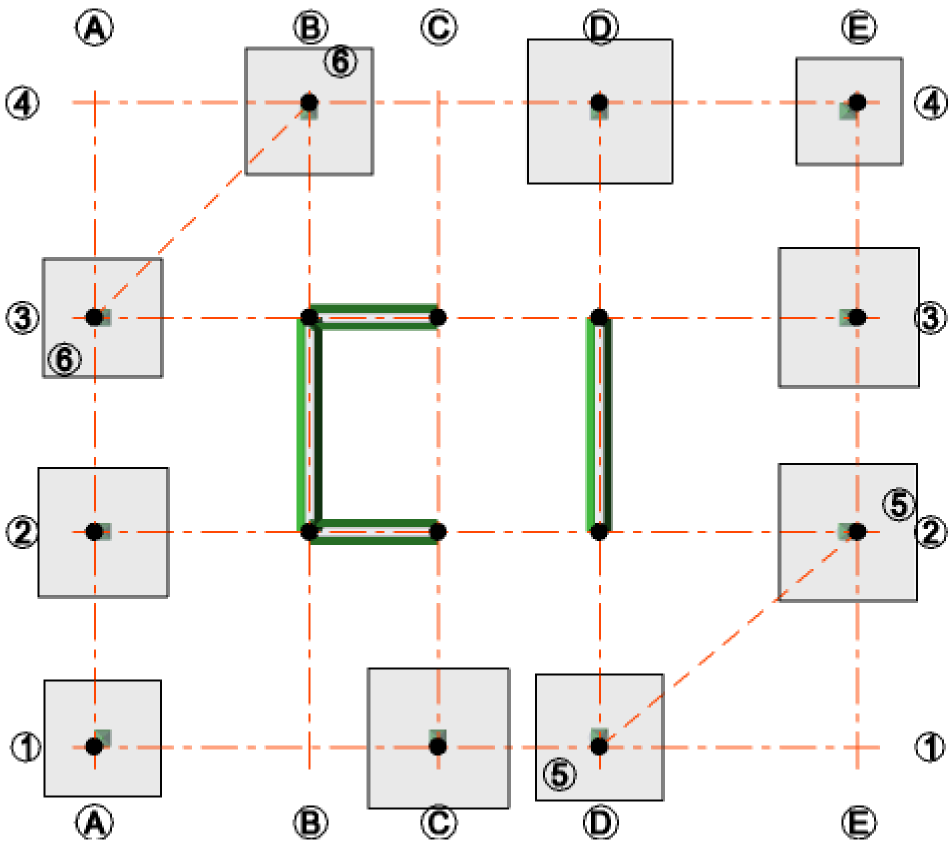
4. **Punching Shear Design** *- increase depth, set reinforcement back to starting point, and go back to step 1*

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

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

3. From the Foundations ribbon click **Design Pad Bases.**

Each base will be sized accordingly (any that are not in auto-design mode will simply be checked).



4. With the auto-design options cleared, you can then adjust individual base dimensions and re-check if required (by right-clicking the base that has been edited and choosing **Check Member**).

   The site boundary may impose restrictions on the positioning of an isolated foundation relative to the column/wall it supports. This restriction may result in a requirement for an offset base, this can be achieved by specifying the eccentricity required in the base properties.

Apply grouping to rationalize pad base sizes

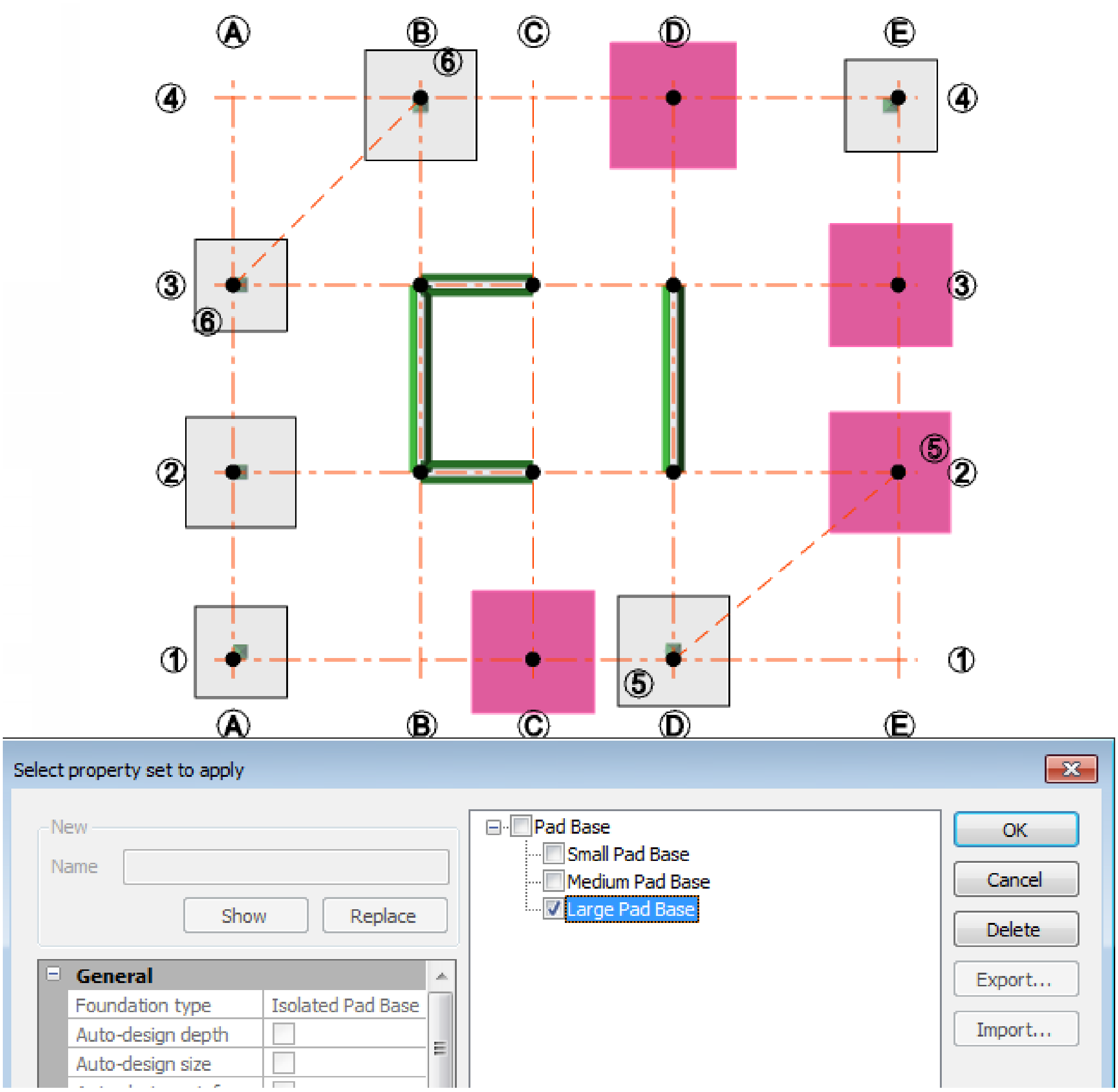
Grouping can only be applied to pad bases - not to strip bases.

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

This is done as follows:

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

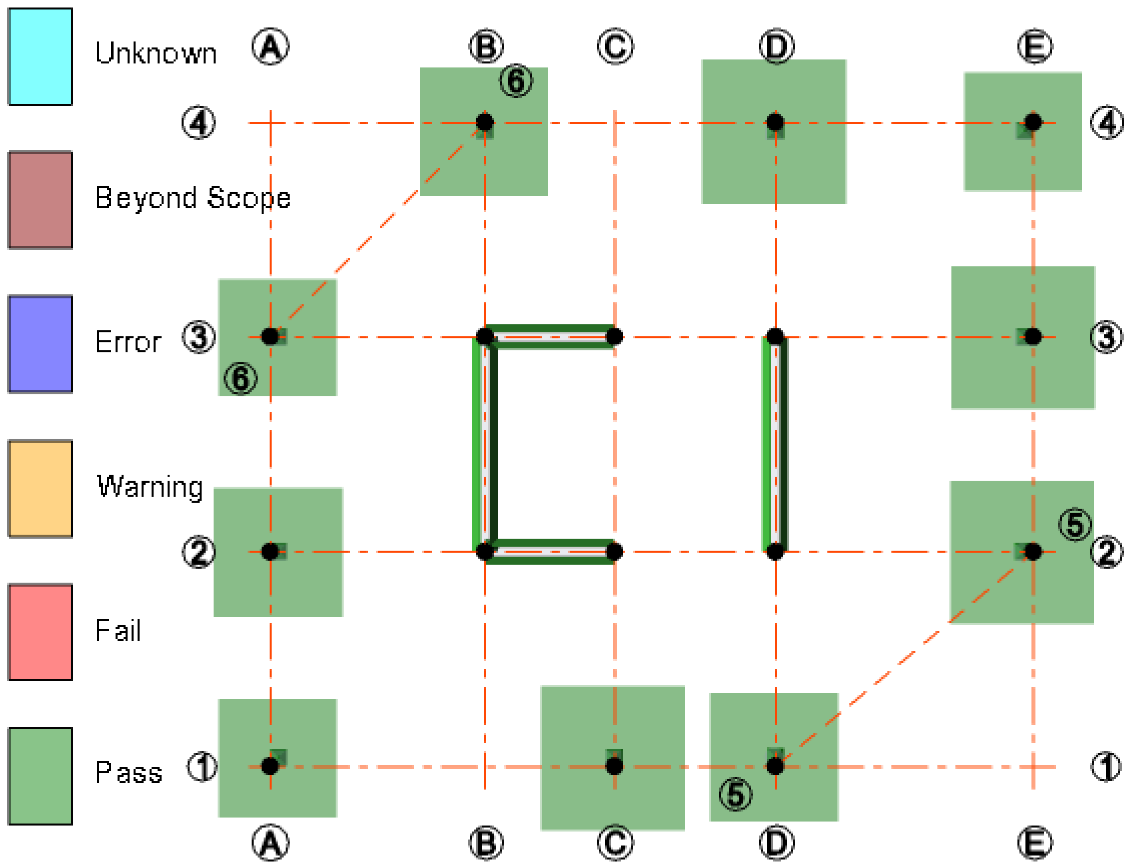
Ensure you click **Apply** *from the* **Properties Window and not from the right-click menu***, otherwise the properties will only be applied to the last base selected.*



6. From the Groups page of the Project Workspace, right-click **Pad Bases** (under the Design branch) and choose **Regroup Members** - this will put those bases that share similar properties into the same group.

7. Open the Design Options dialog, and from the Design Groups page select the option to design isolated foundations using groups.

8. Click **Design Pad Bases** - the results obtained will reflect the grouping that has been applied.



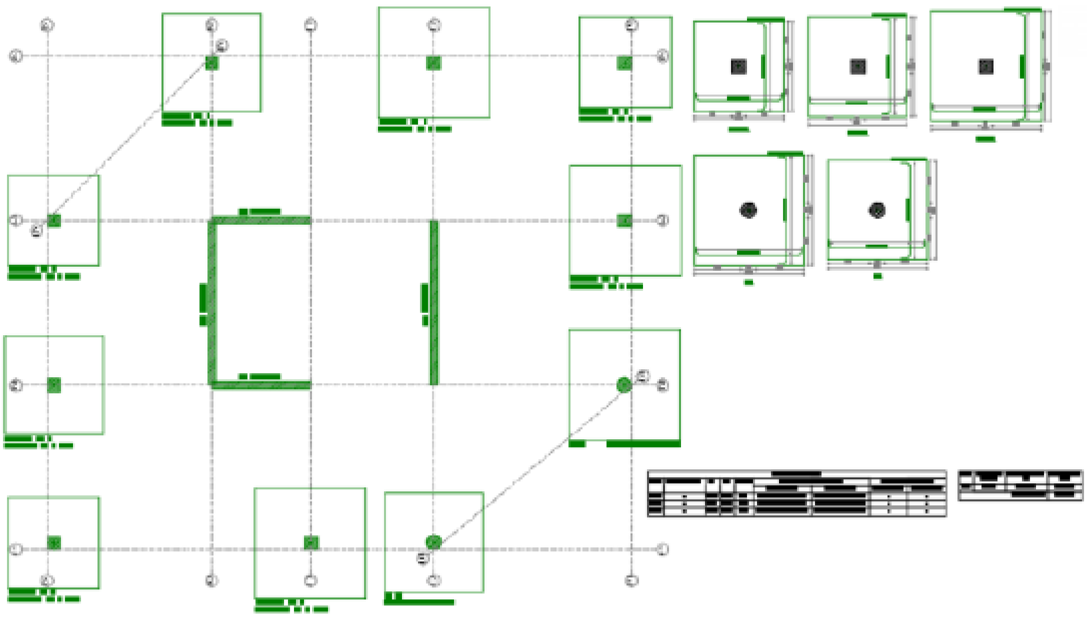
Review/Optimise Base Design

In the **Review View** you can examine the design efficiency by switching from **Foundations Status** to **Foundations Ratio**. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in **Design Options> Concrete> Foundations**.

Create Drawings and Quantity Estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.



Print Calculations

Create a model report that includes the concrete pad base design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Pile cap design procedures

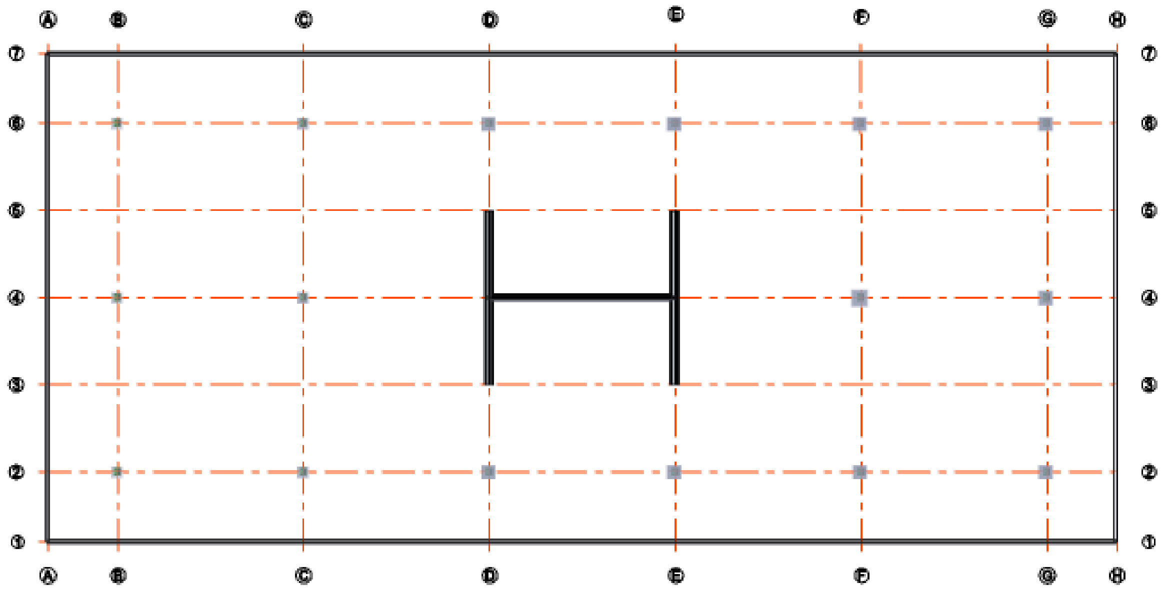
The overall procedure is demonstrated in the following Pile cap design example.

The typical steps required are as follows:

1. Apply pile caps under supported columns
2. Auto-size pile caps individually for loads carried
3. Apply grouping to rationalize pile cap sizes
4. Review/Optimise Pile Cap Design
5. Create Drawings and Quantity Estimations
6. Print Calculations

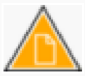
Pile cap design example

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



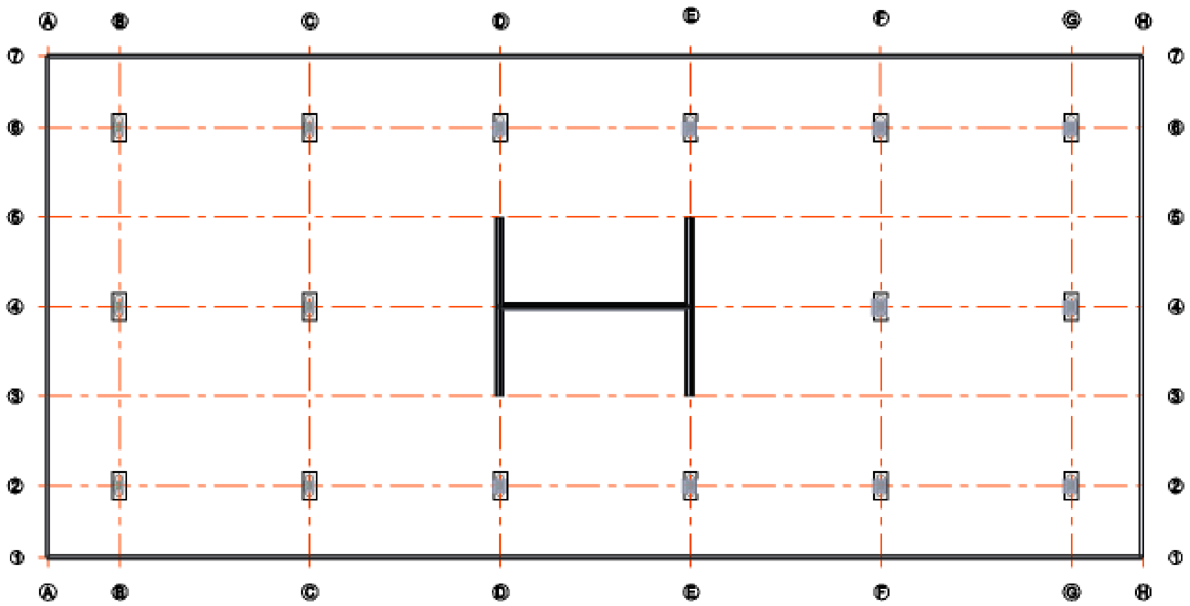
The model has already been designed prior to placing the pile caps.

Apply pile caps under supported columns



Before a pile cap can be placed, the Pile Type Catalogue must contain at least one pile type.

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:

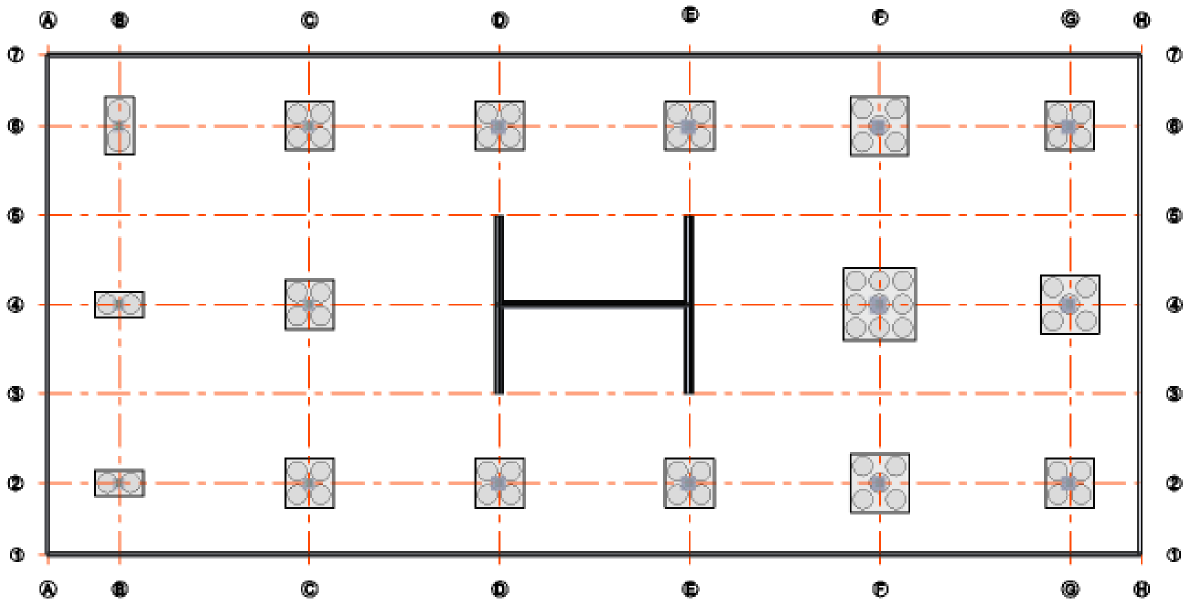
1. Access **Design Options> Concrete> Foundations> Isolated Foundations > Piles** to choose the pile auto-design method required: (minimise pile capacity, or minimise number of piles).

2. Still in the Design Options, 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 ribbon 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).



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.

Apply grouping to rationalize pile cap sizes

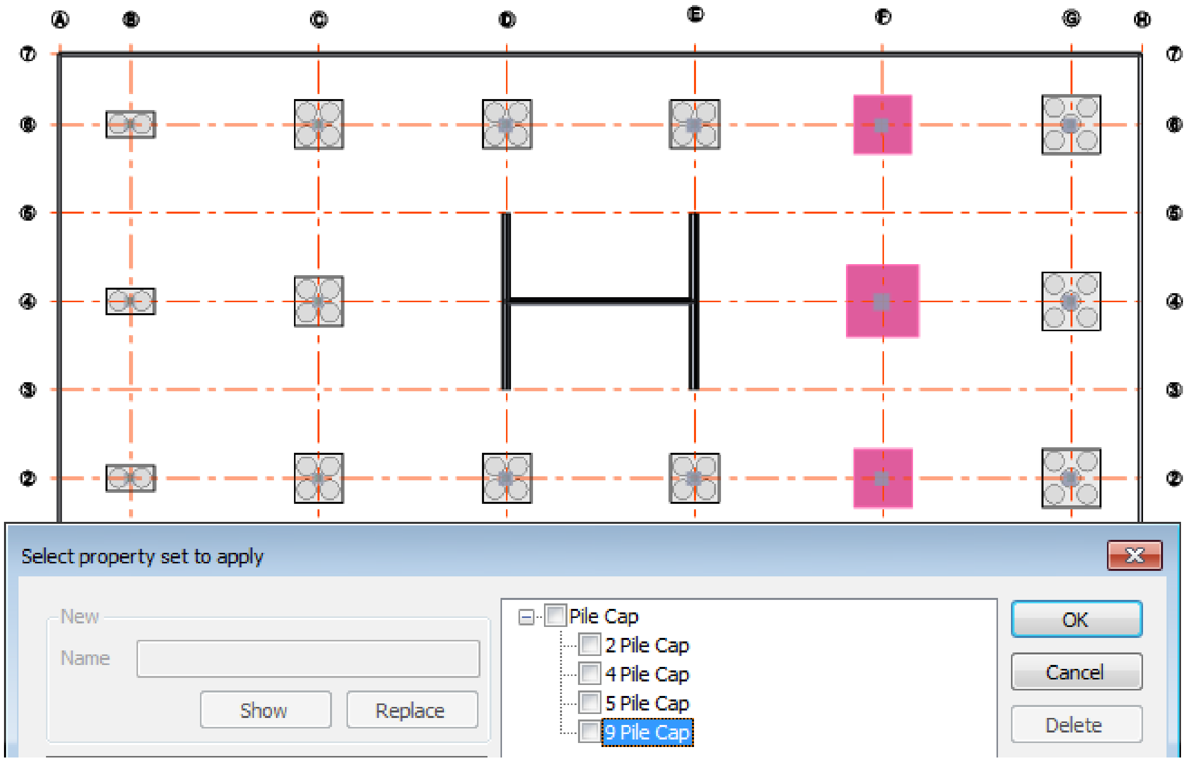
Once pile caps have been sized individually, the designs can be rationalised by activating grouping, in order to obtain one design per group sufficient for all pile caps within the group.

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.

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.

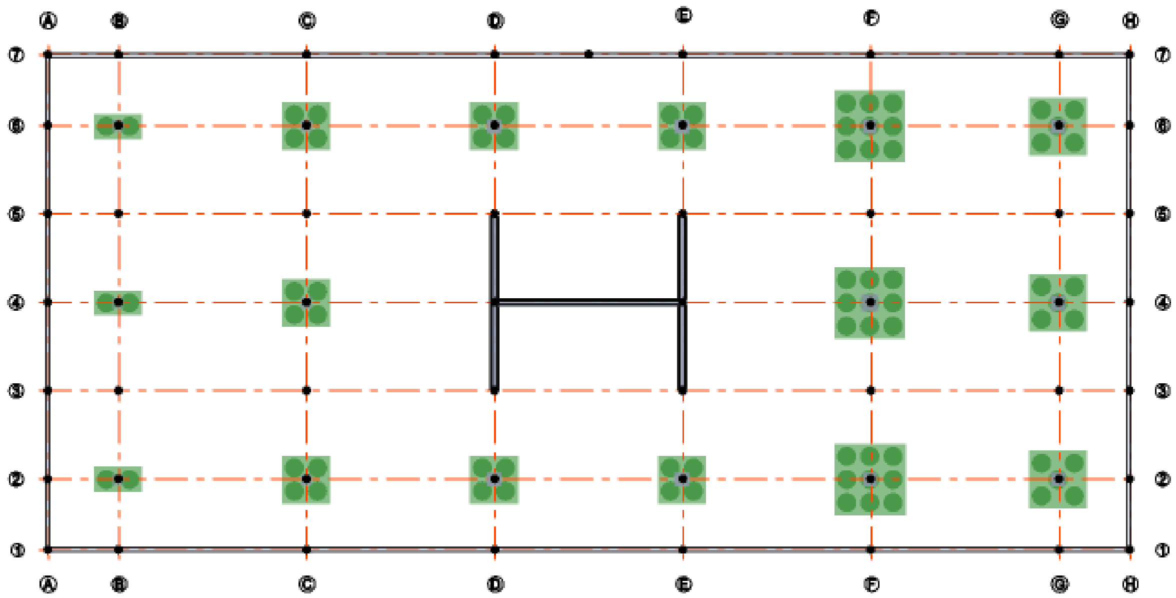


Ensure you click **Apply** *from the* **Properties Window and not from the right-click menu**, *otherwise the properties will only be applied to the last pile cap selected.*

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 Options dialog, 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.



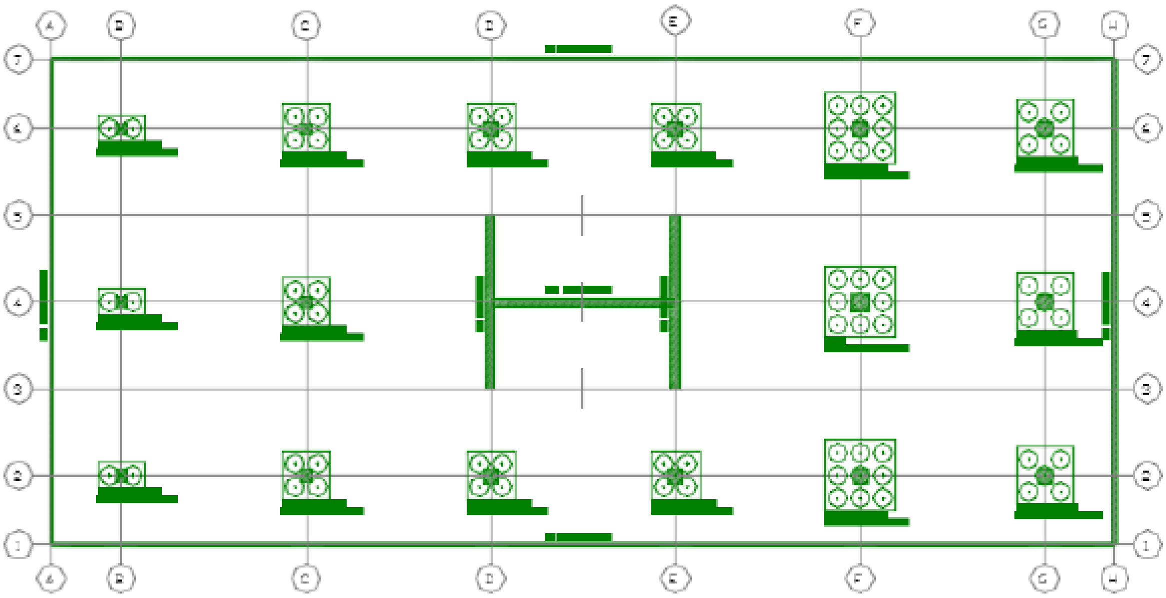
Review/Optimise Pile Cap Design

In the **Review View** you can examine the design efficiency by switching from **Foundations Status** to **Foundations Ratio**. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in **Design Options> Concrete> Foundations**.

Create Drawings and Quantity Estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.



Print Calculations

Create a model report that includes the concrete 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).

Mat foundation design

- Typical piled mat foundation design procedure

Features of the mat foundation analysis model

Analysis Types

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

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

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

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

Soil Structure Interaction

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

When a mat or piled mat is introduced these supports must be removed - as the mat now supports the whole building (either on ground bearing springs, or pile

springs). Consequently adding a mat means re-analysis and hence re-design of the whole building.

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Soil Parameters

Ground Bearing Springs

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

Allowable Bearing Pressure

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground Stiffness - Type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear spring are applied:

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

When non-linear springs are applied:

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

Stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table 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<800kPa)	24,000	48,000
Clayey Soil (qa>800kPa)	48,000	200,000

Horizontal Support

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

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

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

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

Pile Springs

In a piled mat, a spring is inserted into the solver model at the top of each pile. The spring stiffness is a user defined property that is specified in the Pile Catalogue.

Vertical

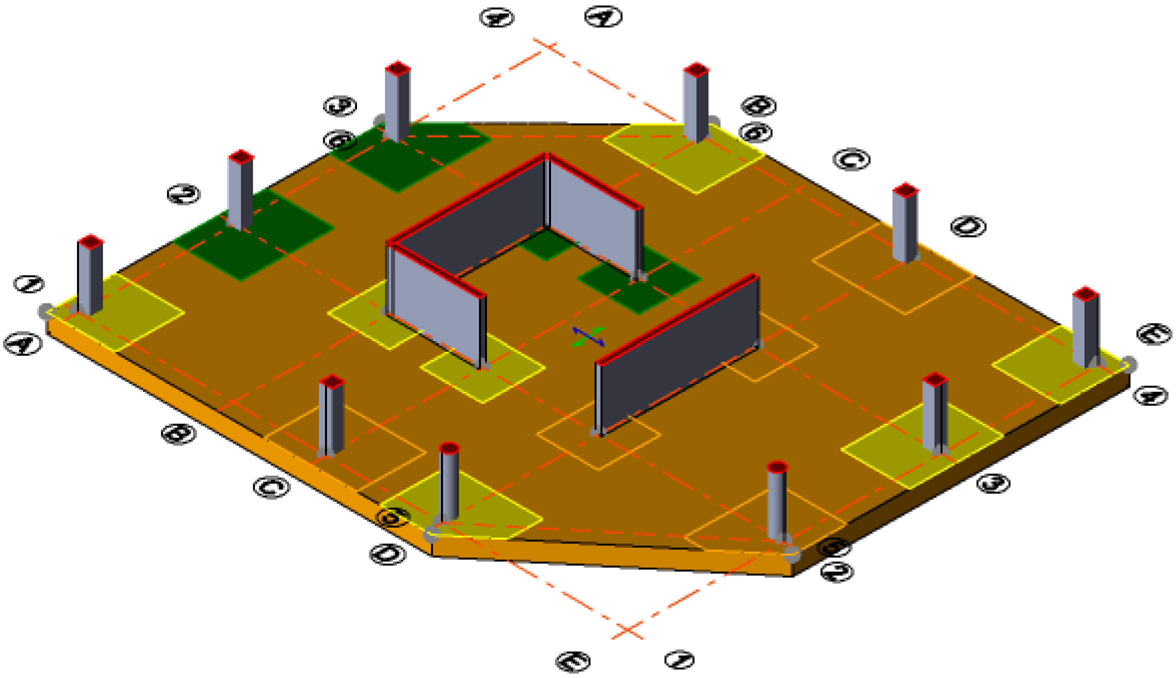
Either a linear, or a non-linear spring stiffness can be defined.

Horizontal

Horizontally you can either specify full fixity, or a linear spring.

Typical mat foundation design procedure

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

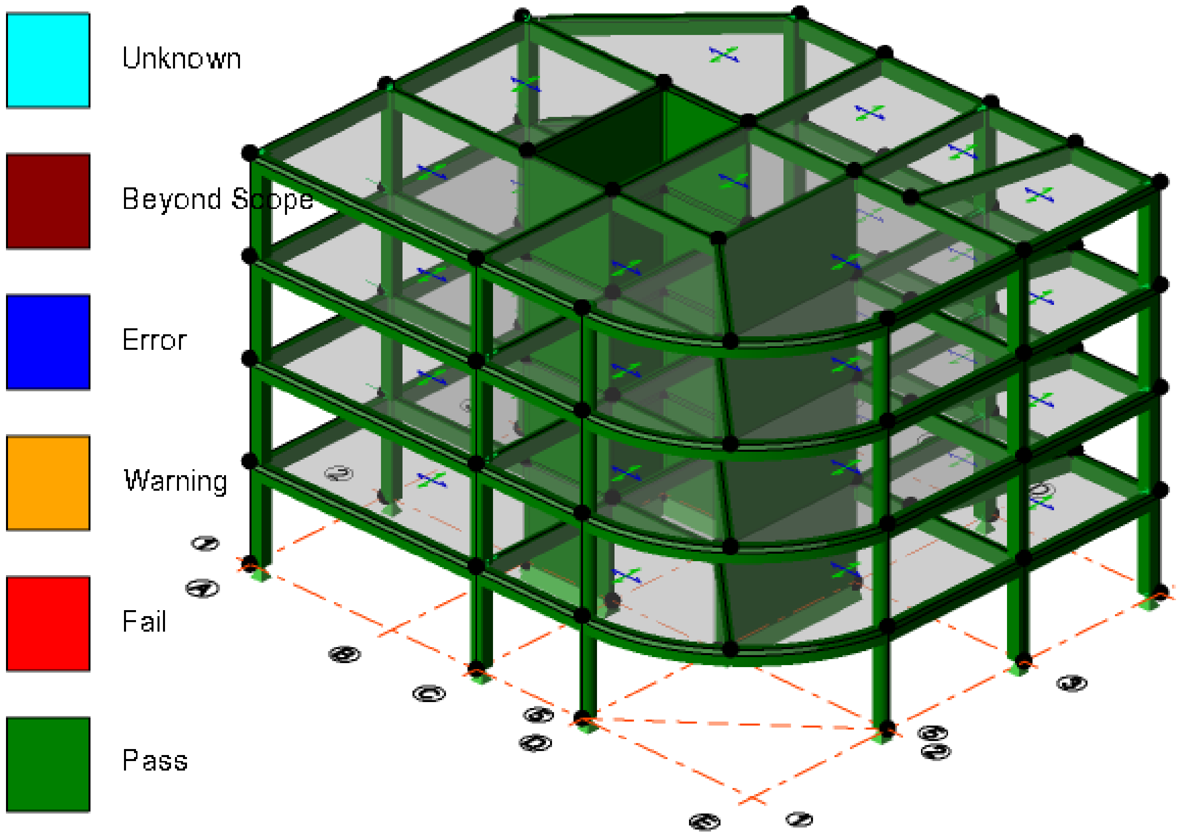


For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

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

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

Design the structure before supporting it on the mat



The model should already be designed and member sizing issues resolved prior to placing the mat foundation.

In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to “check and increase” the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from **current**.)

Create the mat, (either with ground springs, or discreet supports)

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

You are required to manually specify the **Reduce** imposed **loads by** mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

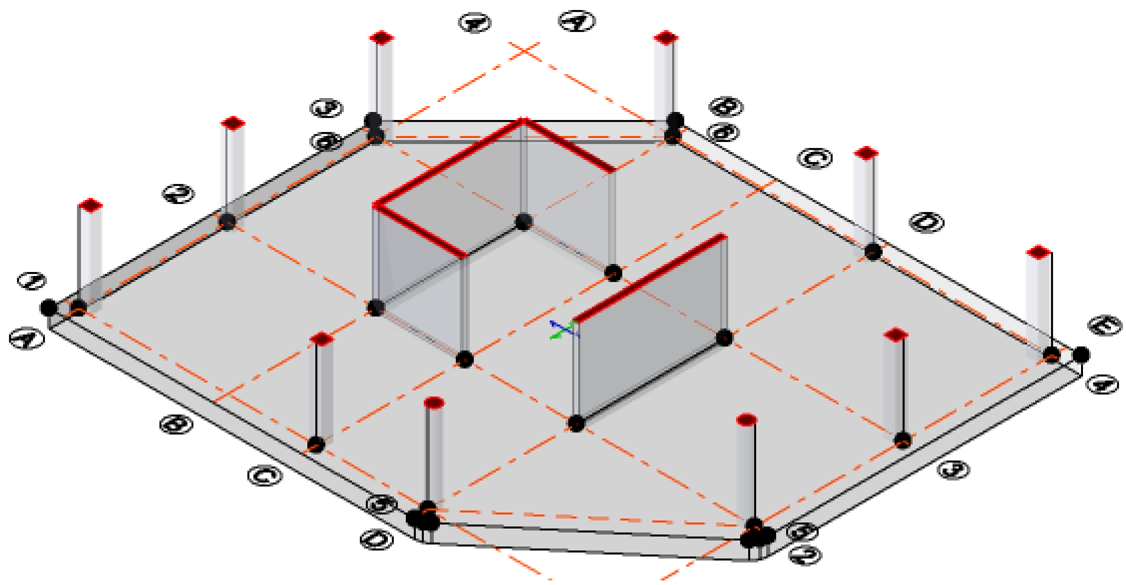
If using an “area” method of mat creation you will also need to specify the amount of slab overhang.

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

The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

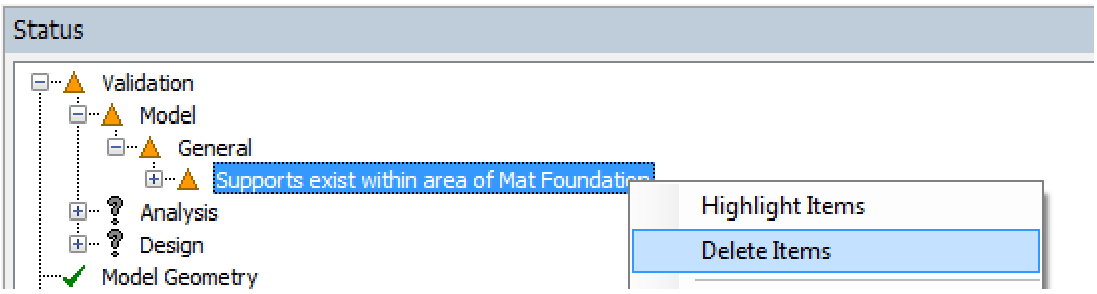
- Imposed loads reduced by 30%
- Default overhang
- mat thickness 600mm
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties



Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing **Delete Items**).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design**.**

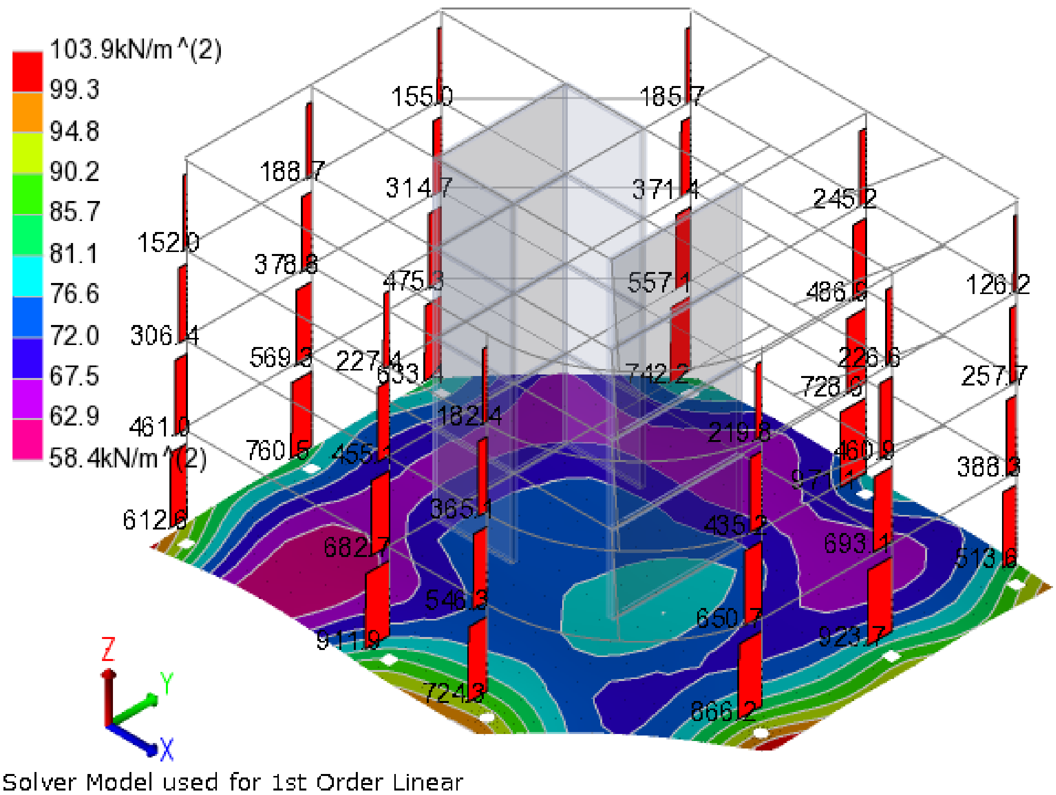
Gravity, lateral and seismic combinations can be analysed by running **Design All (Static)**, and any seismic RSA combinations by running **Design All (RSA)**.

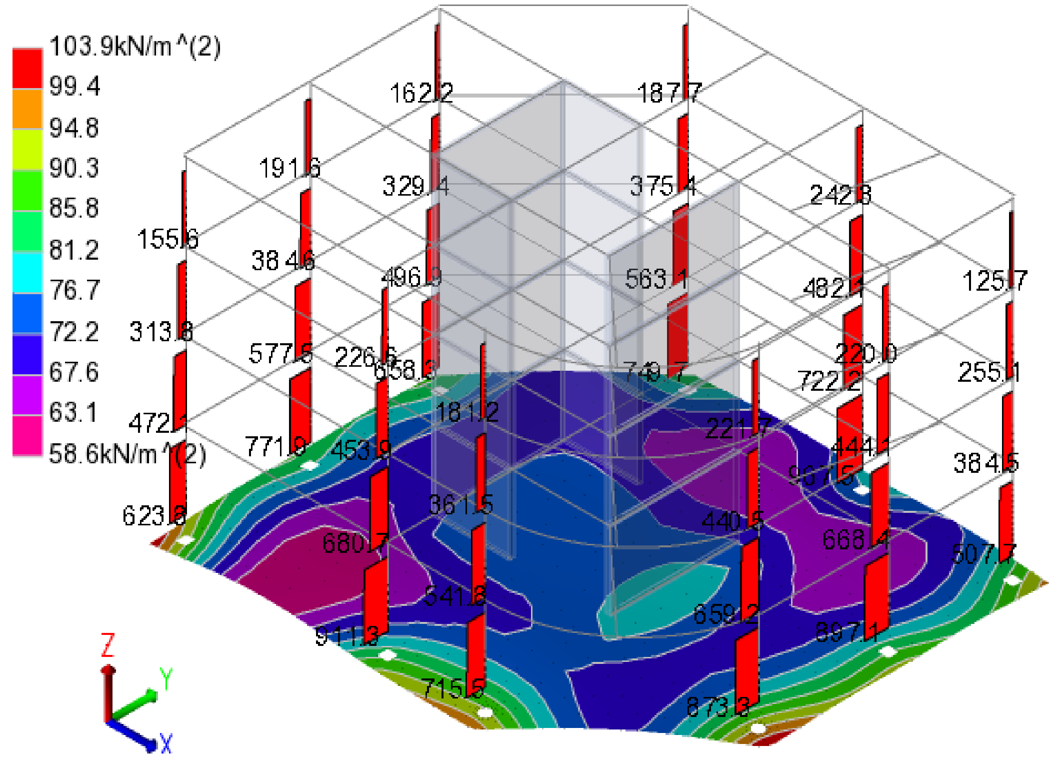
The above processes will also recheck all the member designs taking account of the effects of soil structure interaction.

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

Check foundation Bearing Pressure and Deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.





Solver Model used for 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.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be “Valid” or “Out of Date”.

Unless the status is “Valid” you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking **Design All (Static)** from the Design toolbar.

> *Similarly if an RSA design has previously been performed, but is now out of date* **Design All (RSA)** *should be re-run.*

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring **Select bars starting from** is set to **Current** rather than **Minima**) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

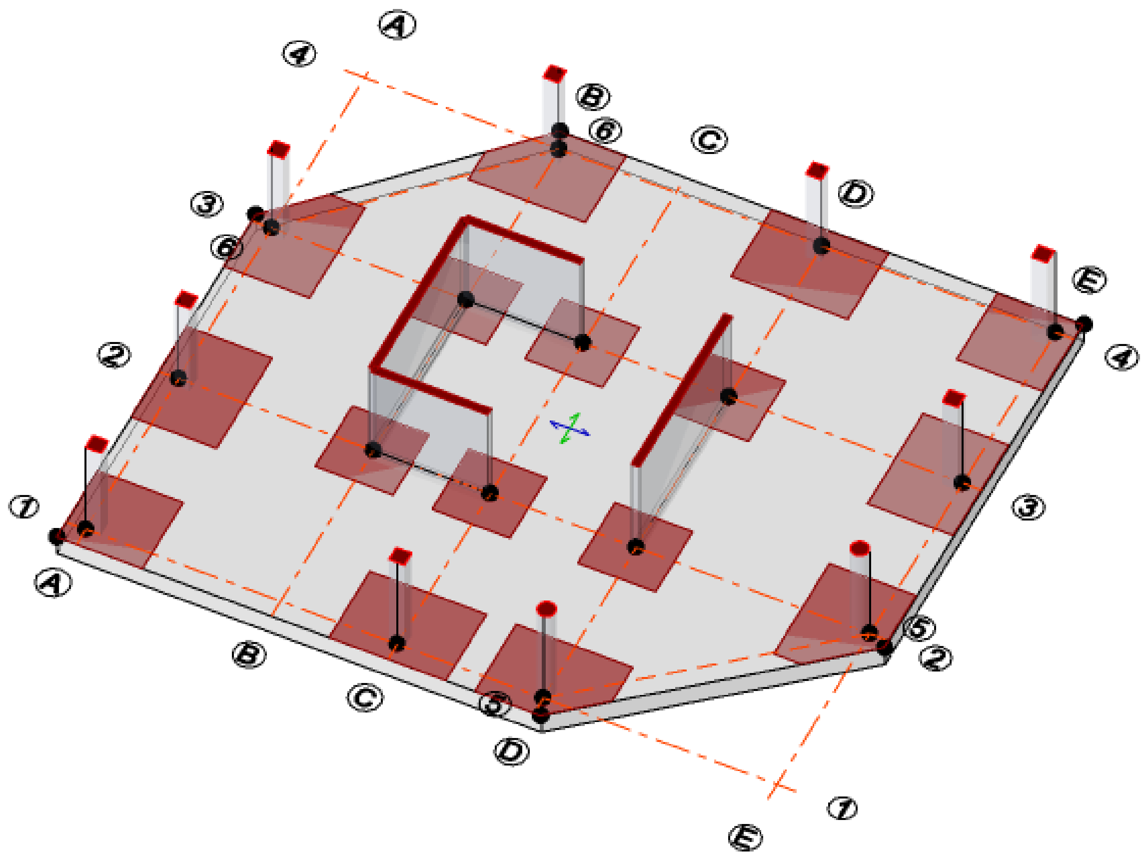
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the “3D geometry” of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered.
Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add Patches

This is an interactive process - requiring a certain amount of engineering judgement..



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimisation stage.

Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design Mats

Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

1. From the Foundations ribbon run **Design Mats** in order to design or check all the mats in the model (each according to their own autodesign setting), or,

2. If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either **Design Slabs** or **Check Slabs**.

These right-click options operate on the same basis as the options for beams and columns:
- **Design Slabs** *will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.*
- **Check Slabs** *will check the current reinforcement in slabs and mats regardless of the current autodesign setting.*

Review/Optimise Mat Design

It is suggested that you use split **Review Views** to examine the results. You could arrange one view to show **Mat Design Status**, and then a second view to show **Slab Reinforcement** in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the **Review View** update mode to standardise reinforcement you can then run **Check Slabs** from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between **Status** and **Ratio** views - if utilisations all < 1 but some panels failing then problems are to do with limit checks. The **Ratio** view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design Patches

Having established and rationalised the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

1. From the Foundations ribbon run **Design Patches** in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,

2. In the 2D View of the level which you want to design right-click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

These right-click options operate on the same basis as the options for beams and columns:
- **Design Patches** *will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.*
- **Check Patches** *will check the current reinforcement in the patches regardless of the current autodesign setting.*

Review/Optimise Patch Design

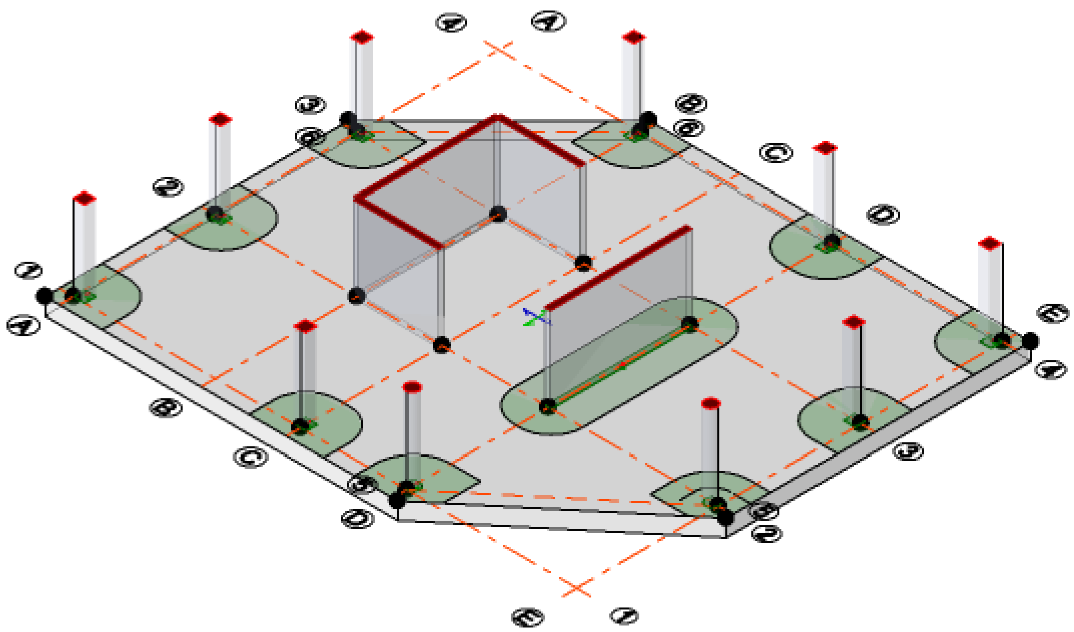
At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimised)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click **Slab Reinforcement** in the **Review View** to review and standardise the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

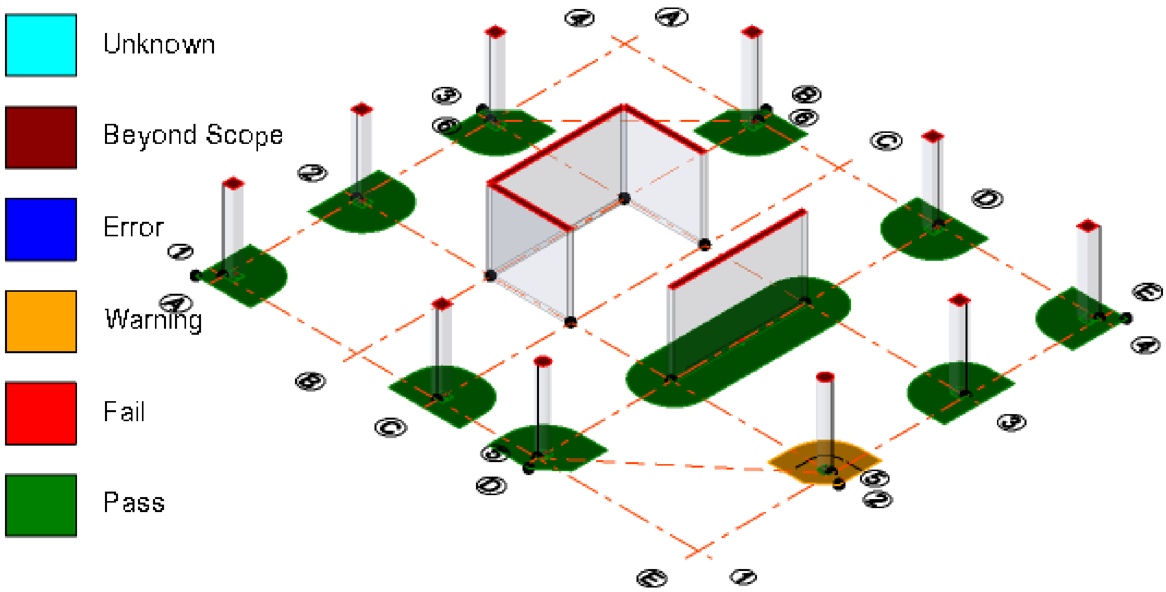
Add and Run Punching Checks

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

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



Once added click **Design Punching Shear**

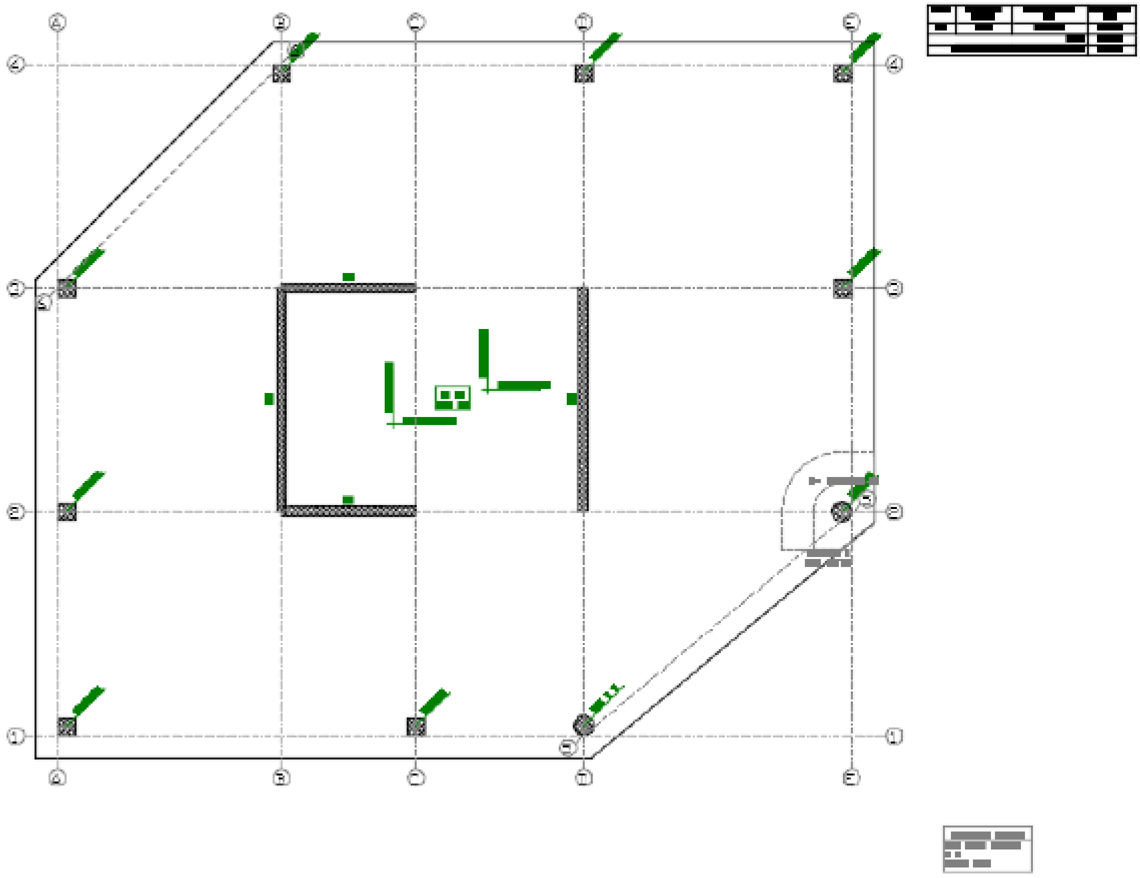


The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create Drawings and Quantity Estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

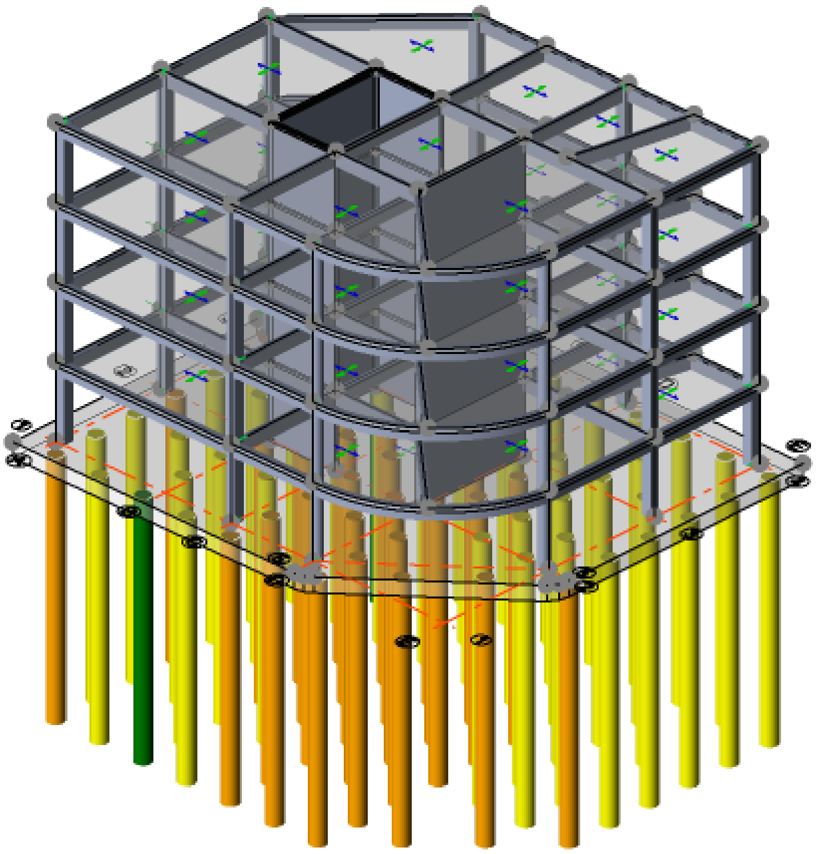


Print Calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Typical piled mat foundation design procedure

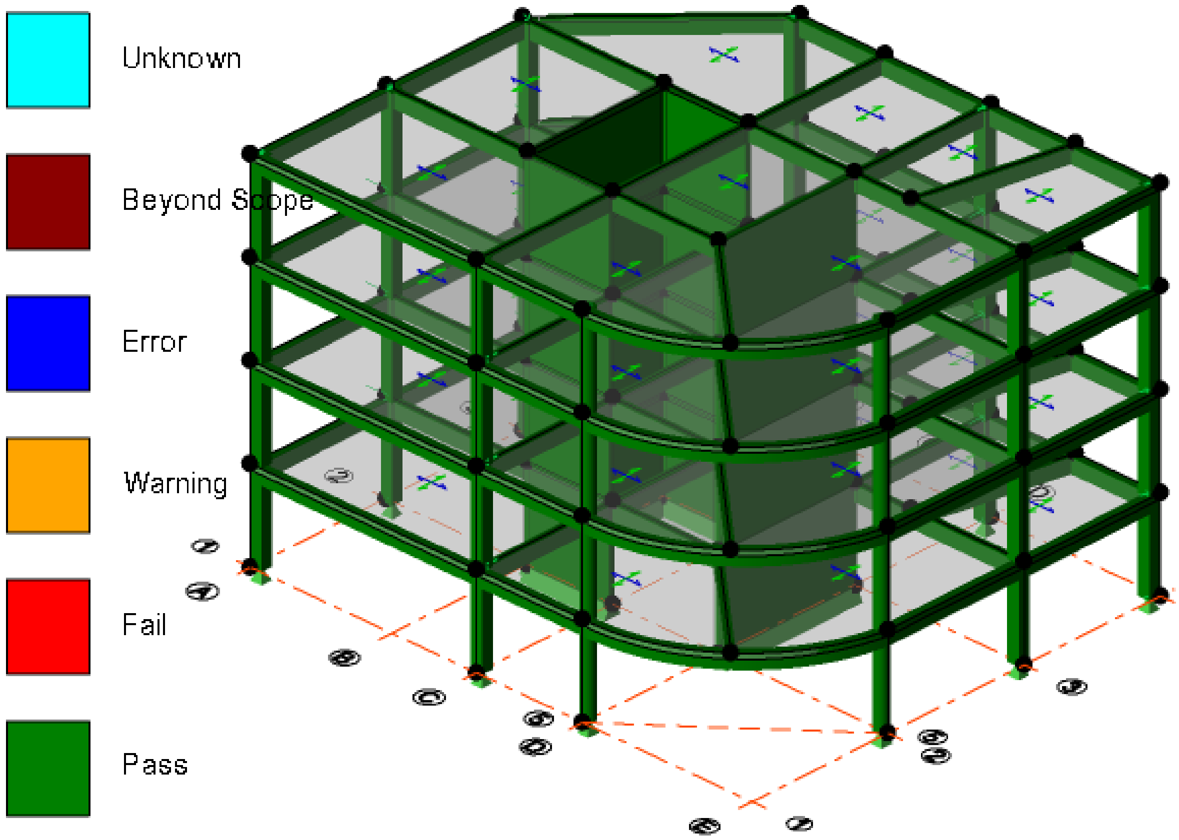
The following example illustrates the typical process to model and design piles in a piled mat foundation.



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

1. Design the structure before supporting it on the mat
2. Create the mat
3. Define the pile catalogue
4. Add piles to the mat
5. Model validation
6. Perform the model analysis
7. Perform the pile design
8. Review the pile design status and ratios
9. Perform the mat design

Design the structure before supporting it on the mat



The model should already be designed and member sizing issues resolved prior to placing the piled 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**.)

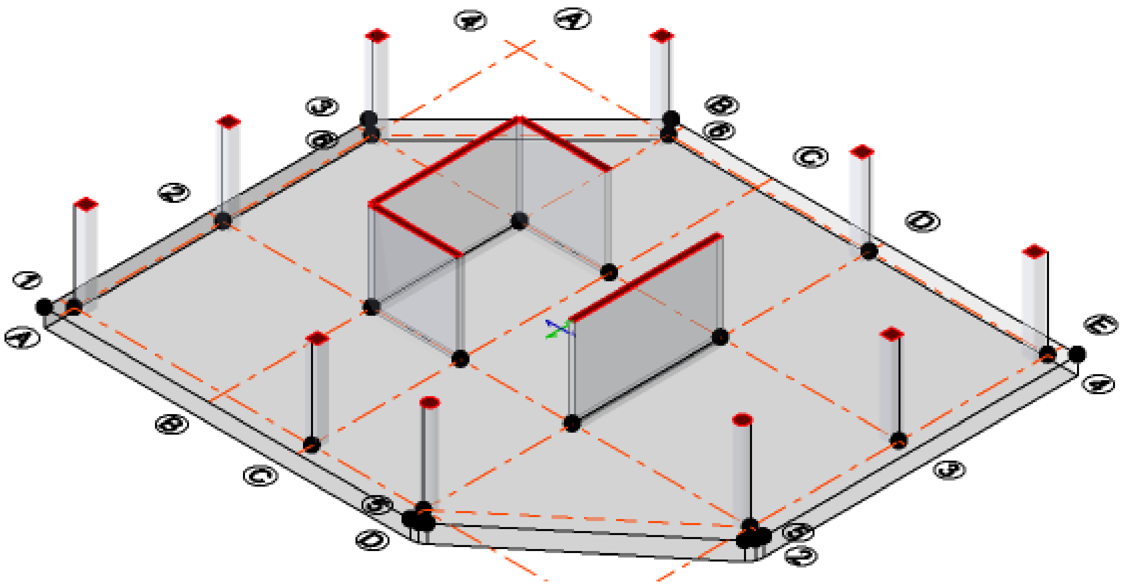
Create the mat

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.)

The “Mesh 2-way slabs in 3D Analysis” option gets activated automatically for the level in which the mat is created.

In this example the minimum area method is used to create a mat with:

- An overhang of 1.0m
- A mat thickness of 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**.

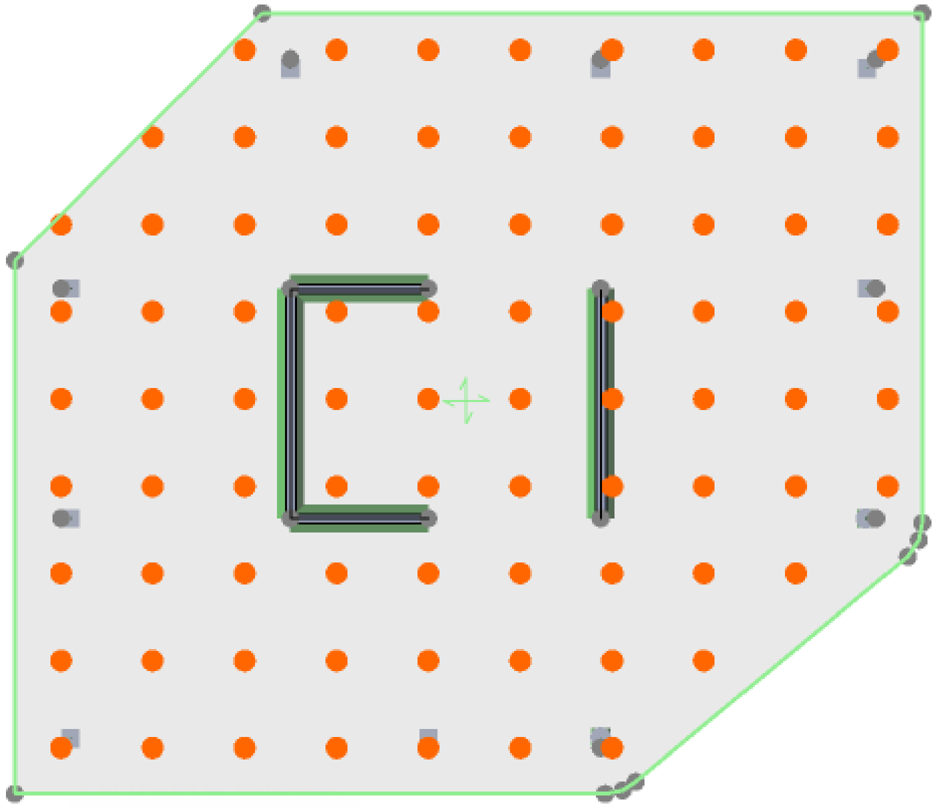
In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

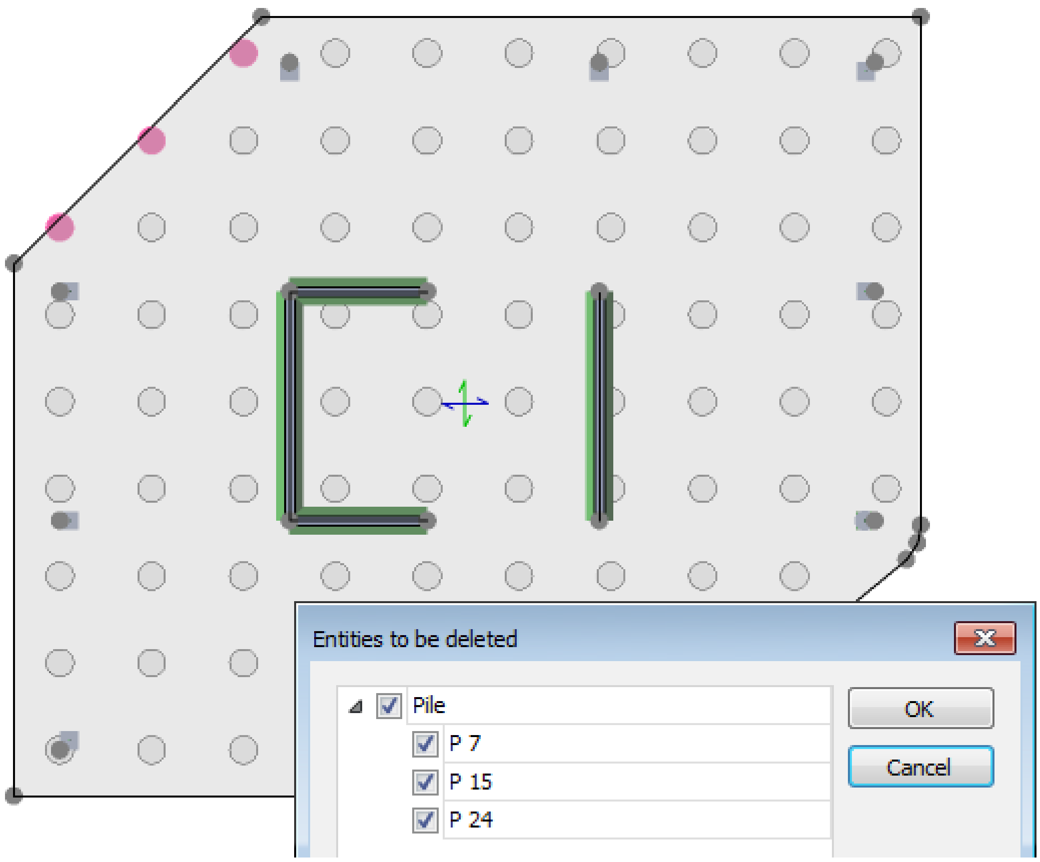
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

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.

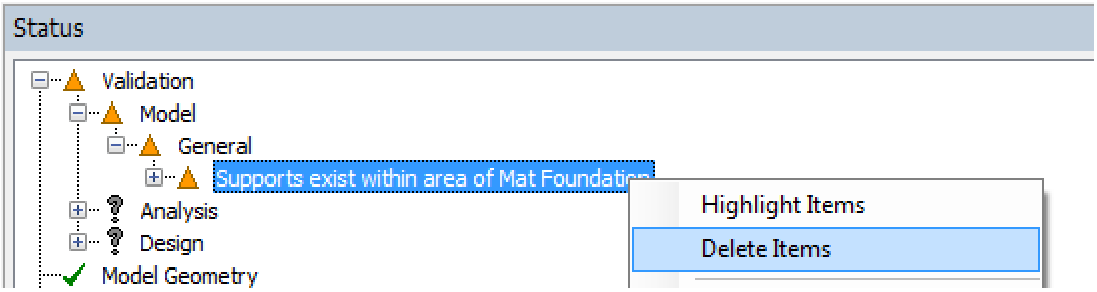


Model validation

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 such member supports exist. The conflict 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 analysed by running **Analyse All (Static)**, and any seismic RSA combinations by running **1st** or **2nd Order RSA Seismic**.

Analyse 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.*

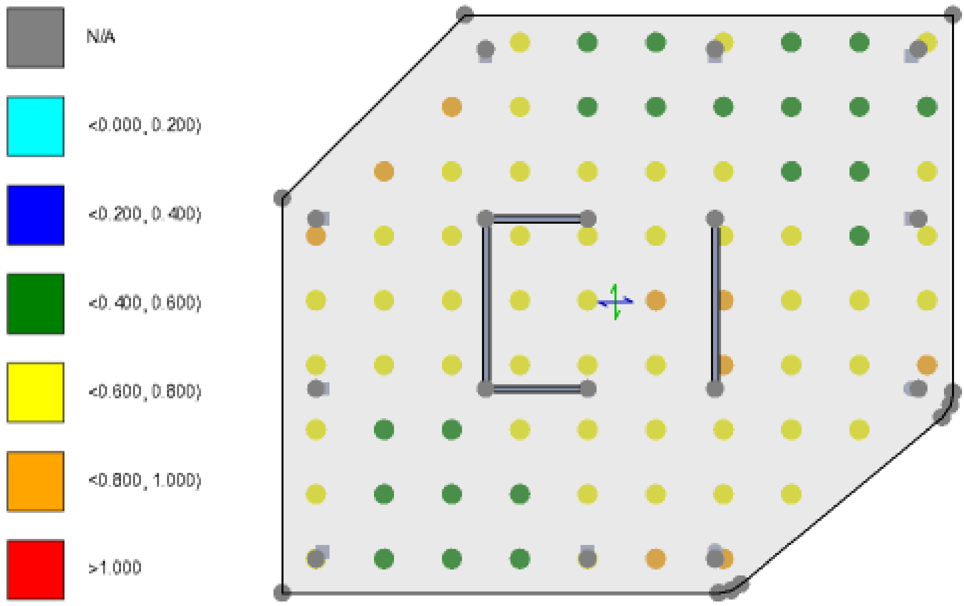
Perform the pile design

The piles are checked (and the mat is designed) by running **Design Mats** from the Foundations ribbon.

The pile types/sizes are not changed during this process.

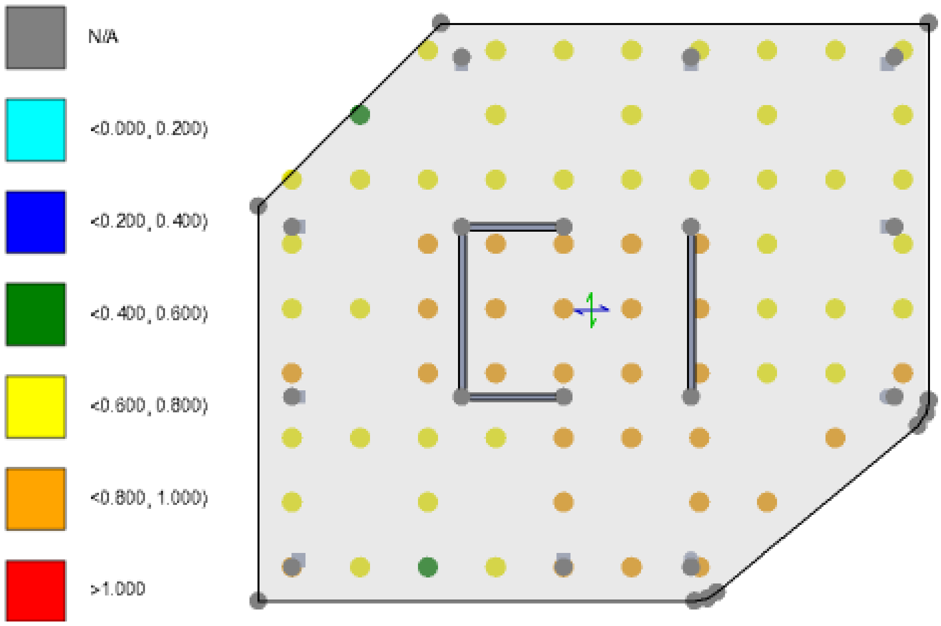
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the **Review View** in order to determine if any remodelling 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.

Perform the mat design

The Design of the mat itself is described elsewhere in the Typical mat foundation design procedure