ProtaStructure 2026

Basic Training Guide

Version 1.0 May 2025

Please get in touch with us for your training and technical support queries

asiasupport@protasoftware.com | globalsupport@protasoftware.com

Publisher

PROTA SOFTWARE

Limitation of Responsibilities

Prota shall not be held responsible for any losses caused by documentation, software, or usage errors.

In addition to Prota License Agreement Terms, it is the responsibility of the user

- to check of results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- make sure that the software is used correctly per the reference manual and documentation,

Intellectual Property

Trademarks

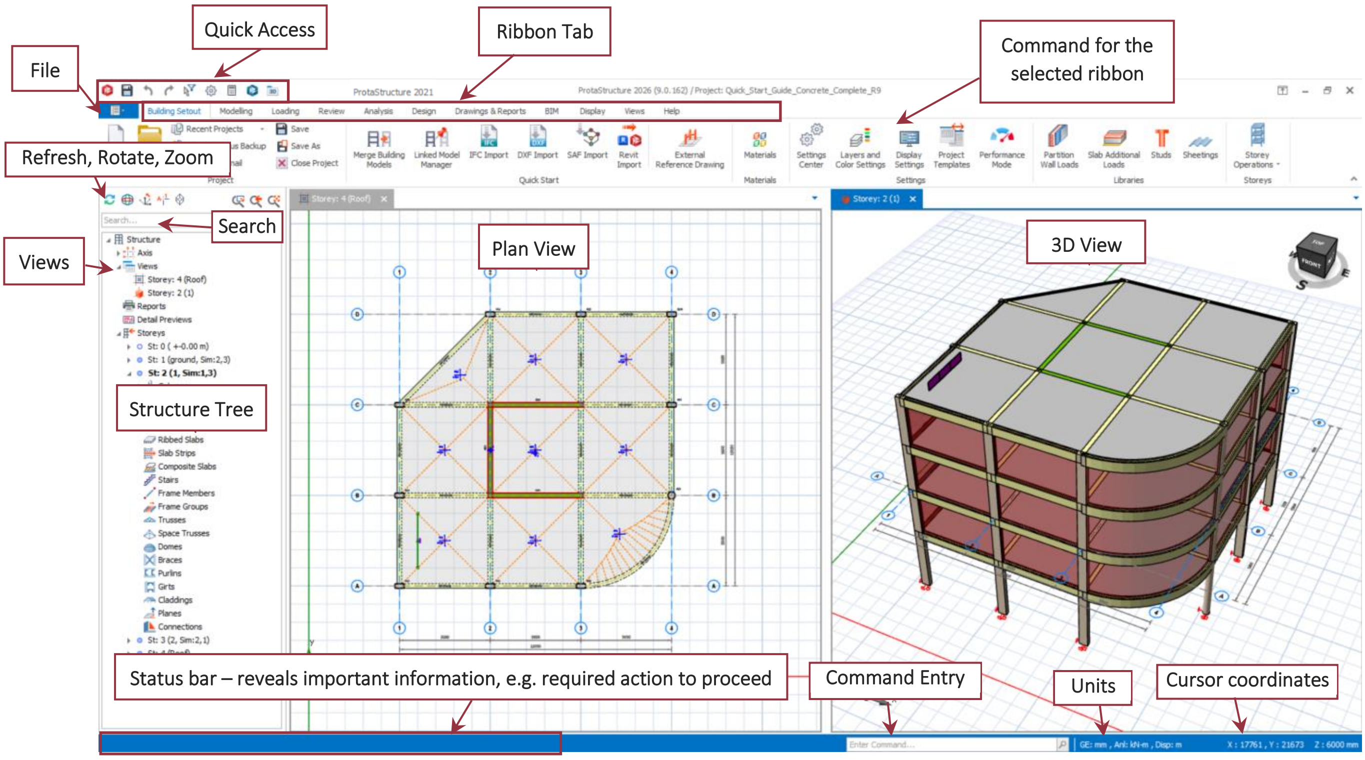
Table of Contents

Introduction

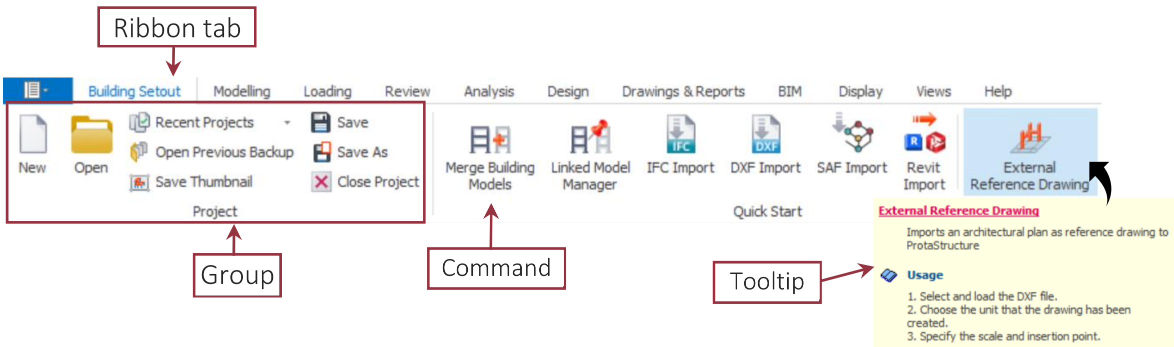
Thank you for choosing ProtaStructure. This Quick Start Guide aims to get you up and running quickly. You should be able to model, analyse and review the results for this simple model in around 1 hour.

User Interface

ProtaStructure 2026 welcomes you with a modern & efficient user interface designed from scratch for ease of use. The various components of the ProtaStructure user interface are as shown below:



Ribbon User Interface

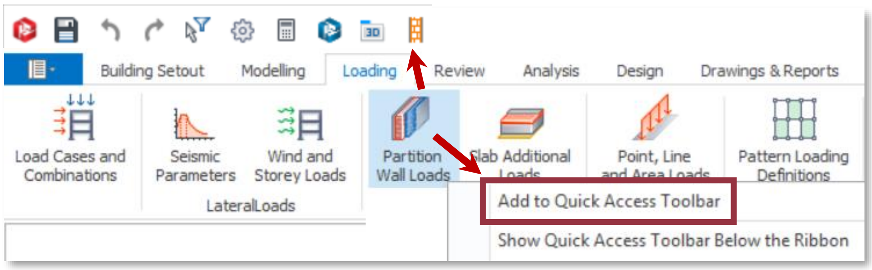


The ribbon consists of several toolbars placed on various tabs. Each toolbar contains related commands organized into logical & functional groups. Placing the mouse cursor over the command will reveal the Tooltip explaining how to use the function.

Generally, you create the model from the left to right ribbon tab. i.e., start with **Building Setout** > **Modelling** > **Loading** > **Review** > **Analysis** > **Design** > **Drawings & Report**.

Quick Access Toolbar

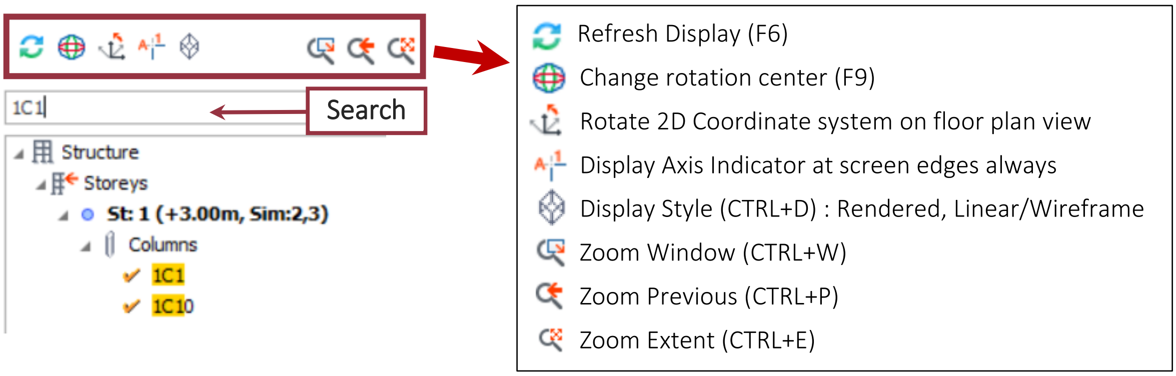
The Quick Access toolbar displays commonly used commands. Any command can be added to the Quick Access toolbar by right-clicking on the command → *Add to Quick Access Toolbar*.



View Operation & Structure Tree Search

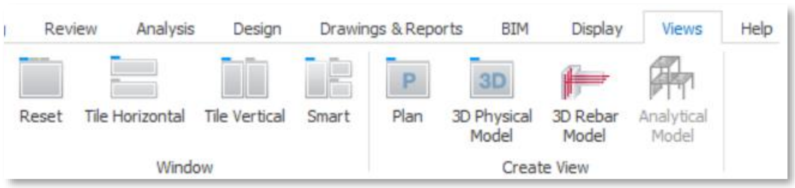
View operation icons enable you to refresh the views, change rotation center & 3D display style, etc.

Place the mouse cursor on the icons to read the tooltip for details.



Are you looking for a specific member or object? Type in the member label in the structure tree's **Search** box and let ProtaStructure find it for you.

Views

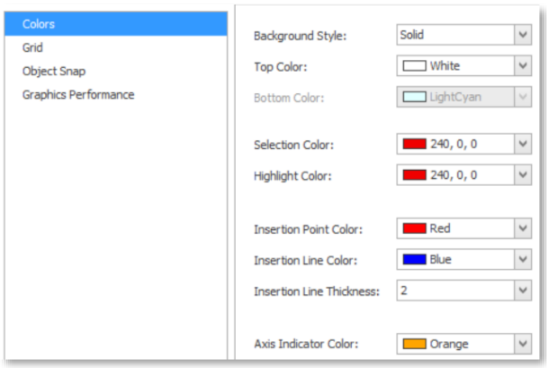


Create as many views as you need, such as a plan, 3D Physical Model, Analytical Model, and 3D rebars. Views can be organized using smart window layout options.

Customize your work area by docking and floating views. The interface is compatible with multiple monitors. If you want to make the most of your screen's cape, just move one of the views to another screen.

Display Settings (under Building Setout tab)

Colours: Choose the background colour and various active modelling objects



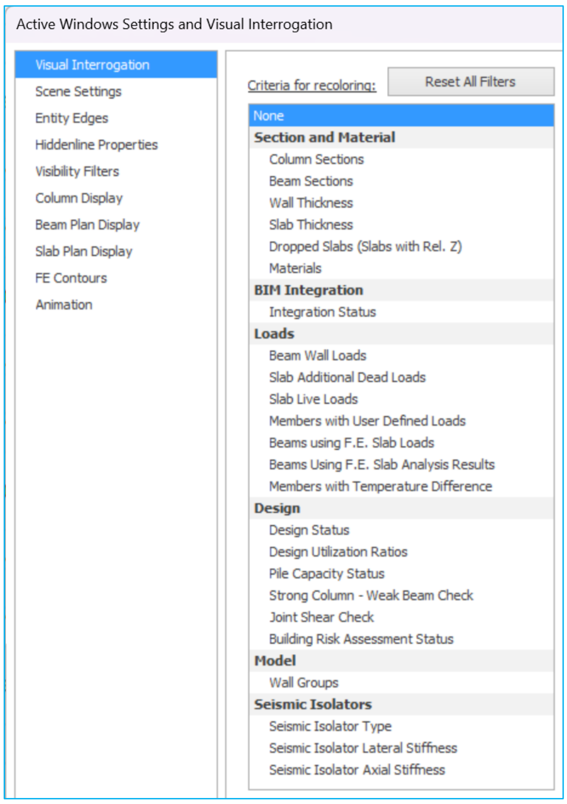
Grid: Sets the spacing of the grid system to allow ease of modelling as objects can snap to the intersection of the grids

Object Snap: Choose the various snap options such as Start/End/Corner, Perpendicular or Orthogonal Grid, etc.

Graphics Performance: Options that affect graphics performance. Defaults will give optimal performance.

Visual Interrogation (under Review tab)

Visual Interrogation: Colour-code members to various criteria such as design status, material types, beam with wall loads. It is a vital visual tool to check and verify the inputs and results of the model.



Scene Settings: Control the views' graphics and switch on/off the guiding Grids & Coordinate Axis, etc.

Visibility Filters: Filter to the specific storey, Axis, or member type.

Column Plan Display: The column wall axial, moment & shear forces to be displayed on the plan view.

Beam Plan Display: Allows brick/partition wall label, load value & height to be color-coded and displayed in the plan view. In addition, you can show the total user-defined load (only after the analysis) & display beam elevation marks.

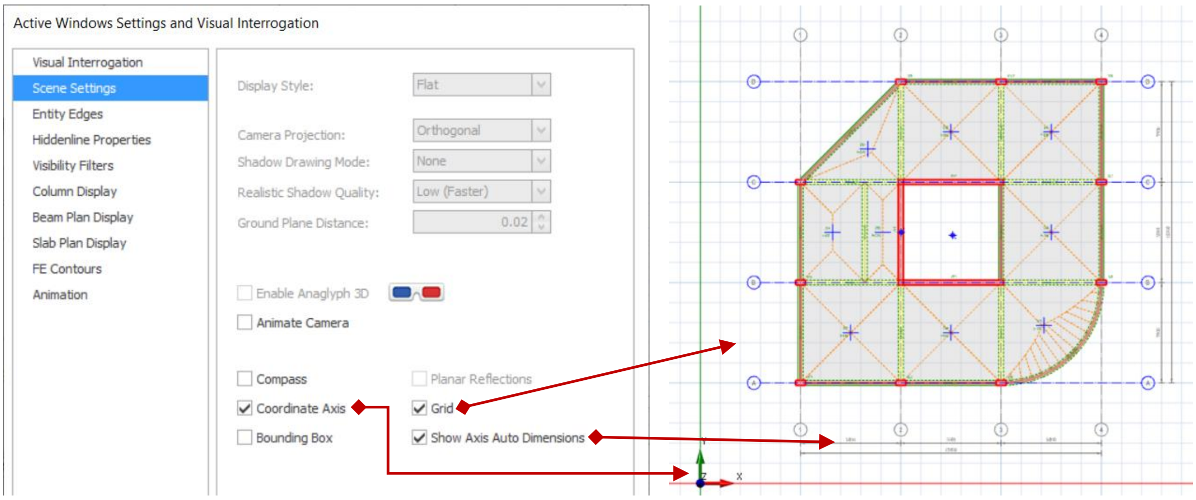
Slab Plan Display: The plan view shows the dead and live load values.

FE Contours: Shows the FE contours exported from the FE Analysis Post-Processing

Animation: Animate by spinning the model in 3D view

Each modelling window can have separate view settings.

For example, you might want to colour the slab live loads on the plan view, and at the same time, colour design status in the 3D view.



In the **Scene Settings** tab, the **grid** allows you to switch on/off the grey rectangular grids in the background. **Coordinate Axis** will enable you to switch on/off the coordinate symbol. **Show Axis Auto Dimensions** enable to switch on/off the auto dimension between axes.

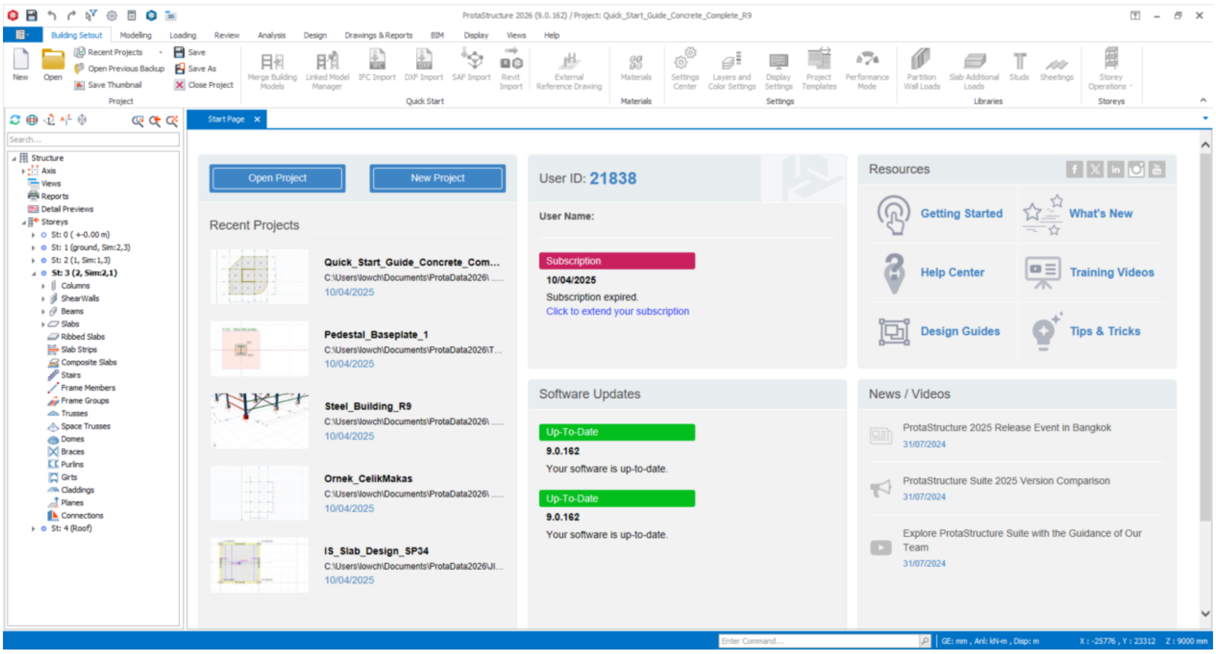
Layer Toolbars (under Display Tab)

Layer and colour settings		Switch on/off layers and modify the name, colour, opacity, line type, line weight, font type, and text height.
Axis Layer Group		Switch on/off the axes layer
Column Layer Group		Switch on/off the column layer
ShearWall Layer Group		Switch on/off the wall layer
Partition Wall Layer		Switch on/off the partition/brick wall layer
RC Beam Layer Group		Switch on/off the beam layer
Slab Layer Group		Switch on/off the slab layer
Ribbed Slab Layer Group		Switch on/off the ribbed slab layer
Slab Load Layer Group		Switch on/off the slab load layer
Slab Strip Layer Group		Switch on/off the slab strip layer
Steel Bars Layer		Switch on/off the slab reinforcement layer
Steel Member Layer		Switch on/off steel members such as frame, truss, brace, purlins, girts, sag rods, etc
Claddings		Switch on/off roof claddings
Composite Slab		Switch on/off composite slab

Ghost Axis Layer Group		Switch on/off the ghost axis layer
Plane Definition Layer		Switch on/off the plane definition layer
Text Layer Group		Switch on/off all the texts
Footing Layer Group		Switch on/off the footing layer
External Reference Drawing		Switch on/off External Reference Drawing

Start Page

When the ProtaStructure is launched, the "Start Page" will appear.



The Start Page contains the following functions:

- Open existing project
- Start a new project
- Read Prota **News** and watch Prota **Videos**
- Access Prota **Resources** such as Help Centre, What's New document & Training Guide
- Read important **Notification** and download new **Software Update**
- View **Subscription** status

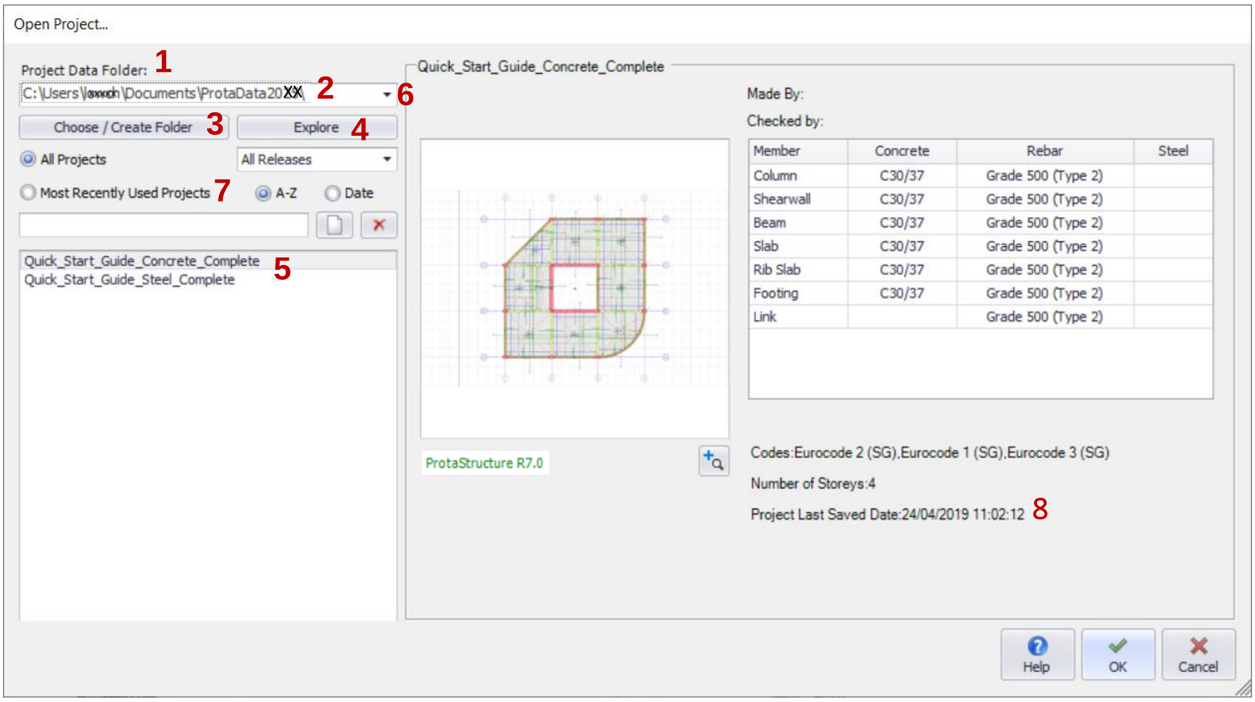
You can always open the **Start Page** by going to the **Help** tab → **Start Page.**

Software update is checked automatically when there is internet connection. If there is a new update, a link will be shown, when clicked will automatically update to the latest version.

Starting a new project

➢ *Click **Open Project** in the **Start Page** & the Open Project dialog will appear.*

Here is the explanation of the functions in this dialog :



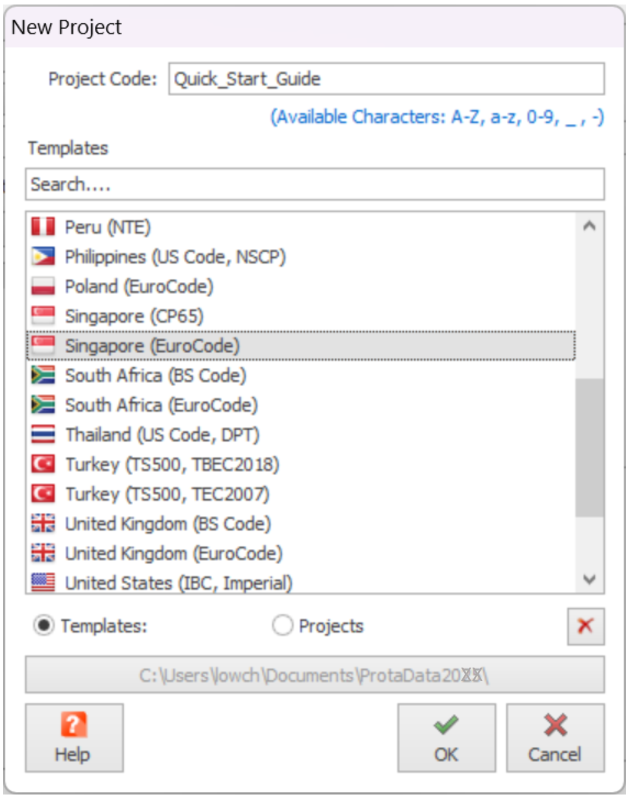
1. The **Project Data Folder** is the main folder where the project models are stored.
   The project model consists of several files stored in a single folder called the **Project folder**. This **Project folder** is created as a subfolder in the **Project Data Folder.** The project folder name will always be created precisely the same as the **project name**.
2. A Data Folder called **ProtaData20XX** would be installed under **My Documents** by default.
3. If required, you can specify a different location by clicking **Choose/Create Folder**.
4. Clicking on **Explore** will open the current Project **Data Folder** in windows explorer.
5. You can open existing projects stored in this **Data Folder** by selecting the project list
   After selecting the project, a preview of the project will be shown on the right-hand side
   Double-clicking on the project name or clicking **OK** will open the project.
6. Click on the **dropdown** list below to expose the most recent accessed Data Folders.
7. Tick **Most Recently Used Projects** to show the recent projects opened quickly.
8. You can also read **Project Last Saved Date** to locate the saved project quickly.

Previous versions of ProtaStructure model can be opened directly in latest version :

- ➢ Firstly, you must choose the correct **Data Folder** by clicking **Choose/Create Folder**.
- ➢ Browse to the desired **Data Folder** (note data folder is the main folder, not the project folder)
- ➢ You will then be able to see the project and open it.
- ➢ You will be prompted to save the model as another name.
- ➢ After converting, the project will open.
- ➢ Newer version projects can't be opened in older versions of ProtaStructure (not backward compatible).

➢ *Click **New Project** and type the project name as shown using the '_' character for spaces.*

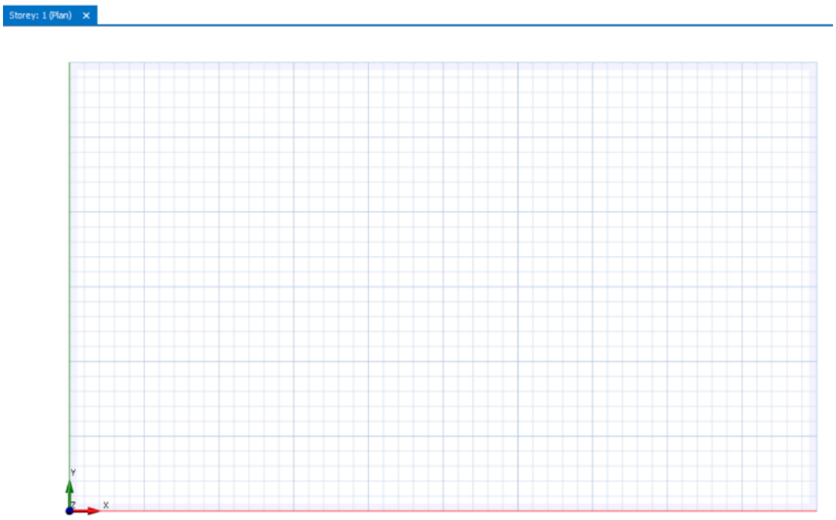
➢ *Select **Singapore (Eurocode)** template*



Templates are used to rapidly establish default model parameters such as design codes, material properties, member design settings, etc. Alternatively, you can choose to duplicate settings from an existing project by picking "**Projects**".

➢ *Click **OK**.*

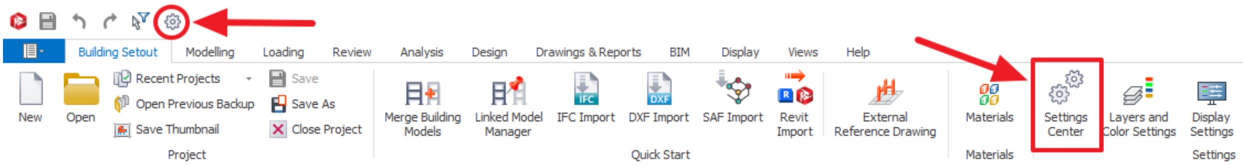
The main modelling area will now show a set of rectangular grids in the background.



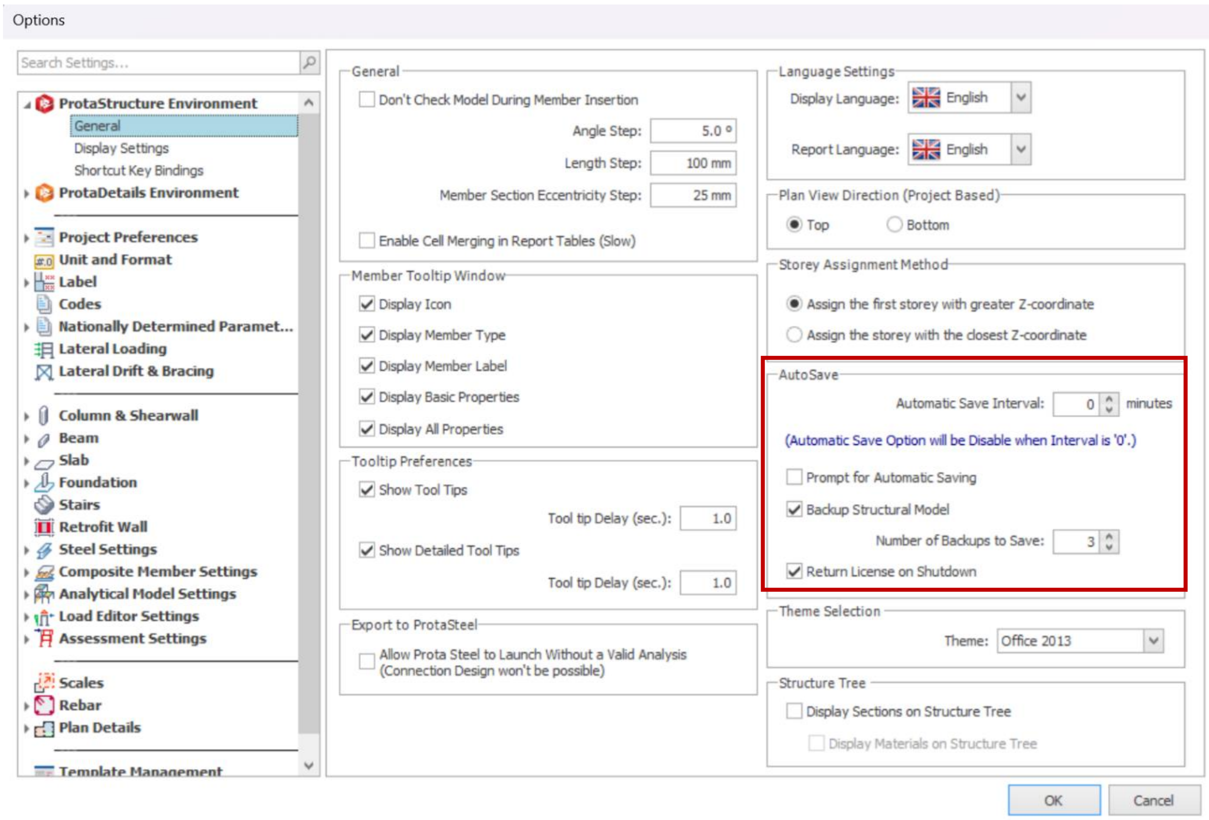
By default, these girds spacing is 1m with a major grid coloured darker every 5m. The defaults can be changed via the **Display Setting** button under **Review** tab.

Settings Center

The **Setting Center** can be accessed by clicking on the **Settings** icon in the **Quick Access Toolbar** or from the **Display tab** (as shown below).



The **Settings Center** centralizes all the program's default settings, including analysis, design and detailing, and Units & format settings.

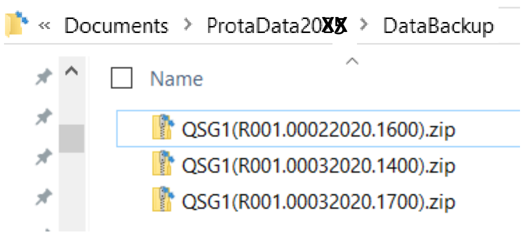


For example, when you initially start a project, it's advisable to review the **Automatic Save Interval** in the **General** settings. By default, "0" means it's disabled.

- *Change the **Automatic Save Interval** to **10 minutes** to ensure the model is auto-saved.*
- *Untick **Prompt for Automatic Saving** to set the auto-save to run silently in the background.*

Backup Structural Model

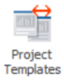
The **Number of Backups to Save** refers to the automatic & silent additional backup created every full hour. For example, the 1st backup file will be created at 9:00 am sharp, 2nd backup at 10:00 & 3rd backup at 11:00. At 12:00 pm, the backup will overwrite the 1st backup, and the cycle continues.
The backup zip files can be found in a folder called "**Backup**" in the same **Project Data Folder**. The files name will have the date and time stamp (example shown below)



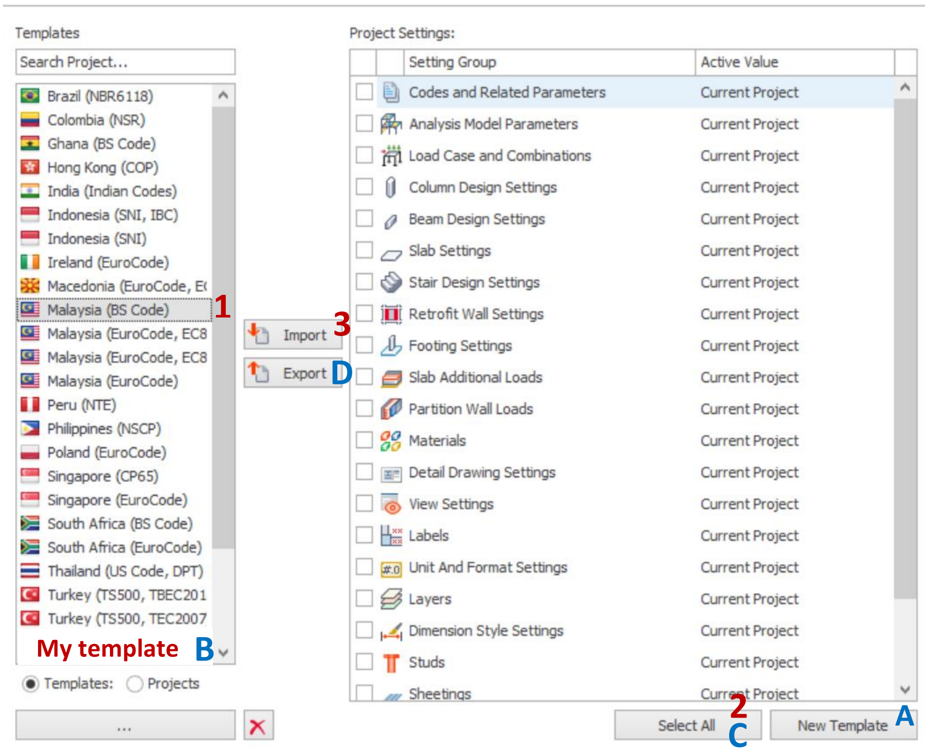
To restore the backup, there are 2 options :

1. If the model can be opened, go to **File** menu > pick **Open Previous Backup**.
2. If the model can't be opened, manually extract the backup zip file and replace all files to the same project folder, over-writing the existing files.

Project Template



The available templates are shown when you start a new project, and you must choose one. You can access these templates via the **Building Setout** tab.



<u>Importing an existing Template</u> → refer to step 1,2,3 above

You can change the template of a currently opened project by selecting a different template and **Importing** it :

➢ Pick a **Template** (1) → Pick **Select All** (2) → **Import** (3)

All the settings of the imported template will apply to the current project.

<u>Exporting a new Template</u> → refer to step A, B, C, D above

You can save the settings of your current project as your template by **Exporting** it :

➢ Pick **New Template** (A) → Give it a name → **OK** → The new template will be created (B)
➢ **Select** the new template (B) → **Select All** (C) → **Export** (D)

The template will be successfully exported & can be re-used the next time you start a new project.

Selection Methods

Selection is performed by placing the mouse cursor on a member or Axis in the modelling view. Although there is no entity to select now, the information is essential as you work through the model.

- *Left-click on the entity to select the entity. Selected entities are also highlighted in the* ***Structure Tree***
- *To select multiple entities, hold down the* ***CTRL*** *key while you pick them*
- *You can select entities directly from the* ***Structure Tree***

You can drag with the mouse to access further selection options:

- *Drag from* ***left to right*** *to create a rectangular box. When you release the mouse button, all entities* ***wholly contained*** *within the box will be selected*
- *Drag from* ***right to left*** *similarly, and all the entities that* ***cross its boundaries*** *will be selected*

If you **right-click** the selected entities, you will see a pop-out **context menu** that allows you to edit and perform other tasks related to that entity, such as Properties, Delete, etc.

Pressing **ESC** will deselect all entities.

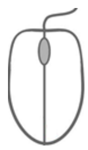
Zoom & Pan Methods

The useful functions for zoom are just above the structure tree :

- Zoom Window → Zoom into the area defined by dragging a rectangle.
- Zoom Previous → Zoom to the previous view.
- Zoom Extents → Zoom to the selected entities. If no entities are selected, it will zoom to show all entities.

You will find it easiest to use the mouse wheel to:

- Zoom in → scroll your mouse wheel **up**
- Zoom out → scroll the mouse wheel **down**
- Pan (move) → **hold down** the middle mouse wheel and **drag**



Modelling Axes

The very first step to building a model is to define axes. Axes intersections then become the nodes at which members are inserted. Hence, axes must be created correctly. There are three ways to model axes:

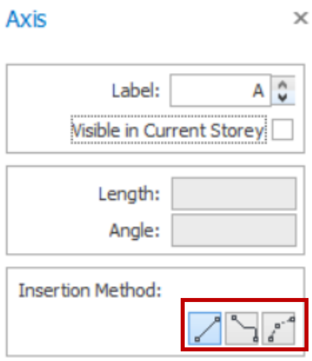
1. **Axis toolbar** to build axes individually
2. **External Reference Drawing** to import all axes from the DXF drawing file.
3. **Orthogonal Axis Generator** to create a system of axes quickly

Axis / Grid Tool

Modelling individual axes is simple using the program's Dynamic Input System (DIS).

➢ *Click on **Grid** icon in the **Modelling** tab.*

There are several insertion methods for the Axis:



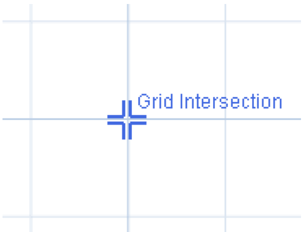
Single Segment allows you to create a straight-line axis by clicking on the start and endpoint of the Axis.

Multi-segment axis enables you to create a single axis with multiple segments of any shape

Curve Axis can be created by specifying the radius

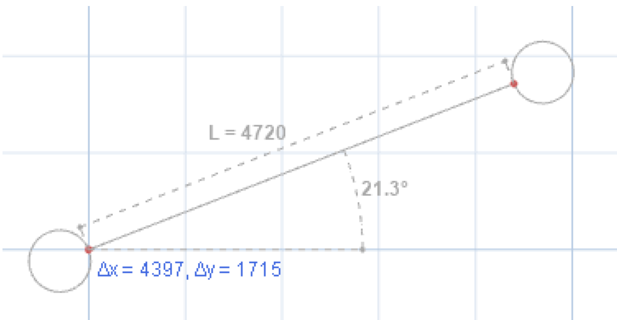
➢ *Ensure **Single Segment** insertion is selected.*

➢ *Put the mouse cursor on any of the grey grid intersections & the "Grid Intersection" snap is shown.*



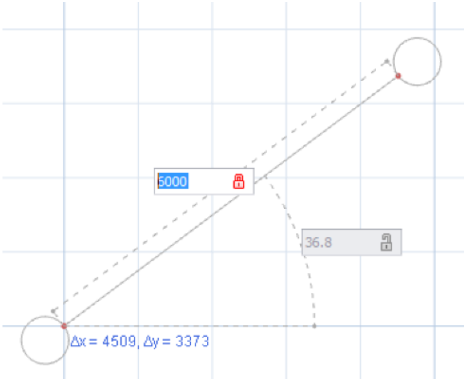
➢ ***Left-click** (& release) to confirm the start point .*

A "rubber band" will appear as you move your mouse cursor to specify the endpoint.



The length (L) and local angle will be displayed during the rubber band operation. In addition, the relative distance **Δx** & **Δy** to the local UCS will also be shown.

➢ *Press **F2** to enable input of the length (L) of Axis via Dynamic Input System (DIS)*



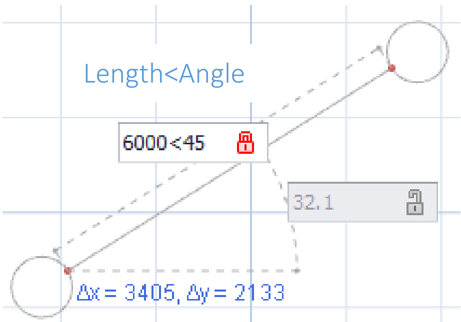
➢ *Press **TAB** to cycle to the following input of angle*

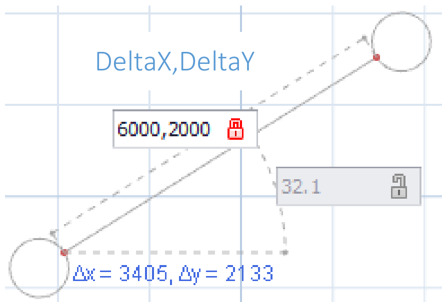
The related parameter will be locked if a value is entered in the textbox. You can unlock by pressing ***ESC*** or pressing the lock icon at the right of the text box.

If a textbox is locked and ***F2*** or ***ENTER*** is pressed, DIS will be deactivated. However, rubber band operation now continues with the locked parameter for ease of use.

The left figure shows the length **L** is locked, so you can freely rotate the Axis without changing the length.

- If both text boxes are locked, pressing **ENTER** will accept the operation, and the candidate point will automatically be picked.
- In either of the text boxes, you can use shortcut notation **Length<Angle** or **Delta-X, Delta-Y** (without the need to switch by **TAB**.)





- Holding down the **CTRL** key when the rubber band appears will lock the length & angle increment as defined in **Settings Centre →ProtaStructure Environment → General → Angle Step** & **Length Step** input box.

➢ *After specifying the length or angle, pressing **ENTER** or left-click will accept the endpoint.*

➢ *Try inserting a **multi-segment** axis*  *by defining points continuously.*

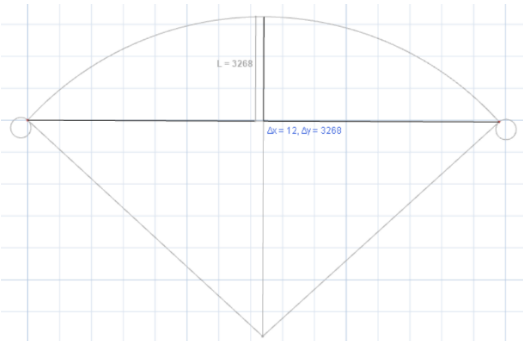
- Use **F2**, **TAB**, **ENTER** to define destination points as mentioned earlier.
- Pressing **ESC** will cause a smart roll-back. If parameters are locked, the DIS is unlocked, then the operation is cancelled. If a pick-continuous command is active, **ESC** will unpick points.

➢ ***Right-click*** *to end the operation and insert the axis.*

- You are in axes creation mode when the axis property dialog is shown. **Close** it, if you want to end the creation of an axis. The same step applies to all member properties dialog.

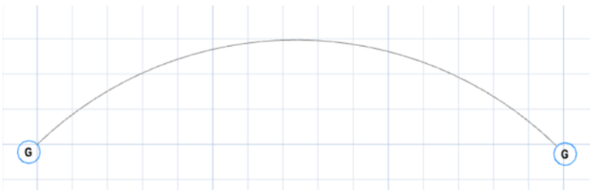
➢ *Insert a curve axis by clicking on the curve axis icon.* 

➢ *Click on the 1st point and then the 2nd point.*



➢ *Move the mouse cursor to the 3rd point that will specify the offset length of the curve.*

➢ *Left-click to confirm the 3rd point → Curve axis will be created*



- Alternatively, hit **F2** to specify the offset length manually.

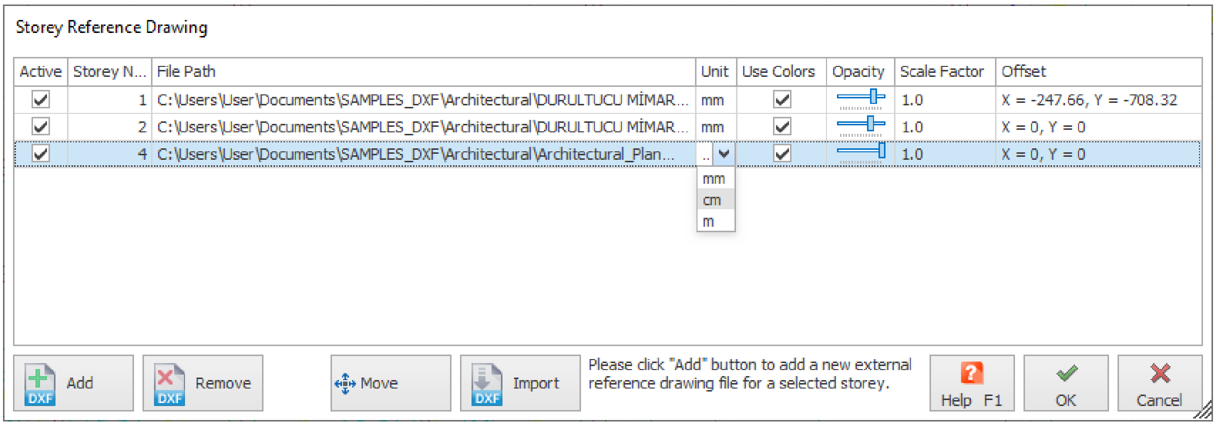
External Reference Drawing

Note: This section is for information. A DXF drawing is required if you want to try out this feature.

ProtaStructure lets you load external DXF drawings and show them as ghost reference layers under your model. ***The External Reference Drawing*** button can be found under the ***Quick Start*** *menu*.

This tool allows you to assign DXF drawings to different stories. Properties like unit, opacity, scale factor, and offset can be set individually for each imported file. The whole state will be saved along with the project data and restored; the next time the project gets opened.

Important! The referenced DXF files should be intact to restore the external drawing layer next time the project gets loaded.



Add

Click ***Add*** to select and load a DXF file. The file will be converted into 2D drawing entities inside ProtaStructure directly upon loading. However, it will not be visible until the *Active* box is checked.

Active

This checkbox controls the visibility status of the external drawing.

Unit

Please select the correct unit of the DXF file from the table's unit column right after the file's loading. The drawing will be scaled immediately.

Storey No

The imported file will be assigned to the active story initially. You can use the 'Storey No.' column to set it to any other level. Only one drawing can be attached to a specific storey.

Use Colours

If this field is checked, colours defined in the file will be used; a grayscale drawing is displayed if unchecked.

Opacity

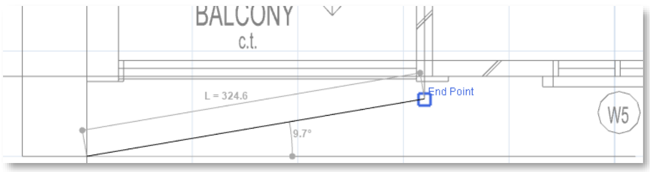
This parameter controls the opacity of the colours. This field applies only if drawing colours are selected to be used.

Scale Factor

This factor scales the whole drawing.

Move & Offset

Click the ***Move*** button and pick two points on the plan view to move the external drawing. Dynamic Input functionality (hit **F2**) can be used here as well. The offset value between the two picked points will be shown in the ***Offset*** column of the table.



Import

The selected reference drawing can be imported into the assigned story. In this case, the ***Import DXF*** interface will be loaded with a pre-defined level and file unit values. The drawing can be imported to the existing model in this mode.

A complete model can be developed from scratch in a controlled environment using the ***External Reference Drawing*** interface and ***Import*** option.

For more guidance, e.g. 3D Physical Model & 3D Analytical Model import, kindly refer to Prota Help Centre : [IFC and DXF Import Guideline]

Orthogonal Axis Generator

Let us now start the new model by creating the required model axes. We will define multiple axes in one go using the **Orthogonal Axis Generator**.

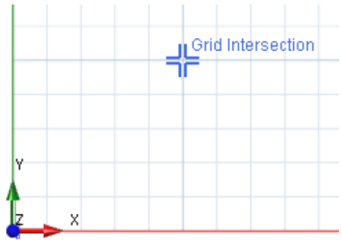
- *Select all existing axes that you have created → press **Delete** (or right-click → **Delete**)*

- *Go to the **Modelling** tab > Select **Orthogonal Axis Generator*** 

  Refer to the bottom status bar; the displayed text tells you how to proceed.

  Pick the Reference Point (Lower/Left) of the Axis Group...

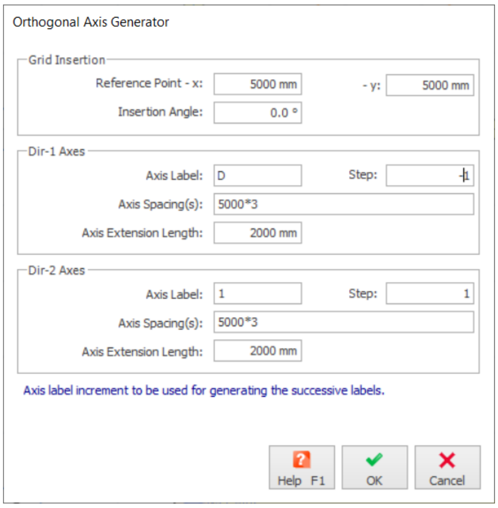
- *Pick the **intersection** of any two primary grids near the origin (as shown below)*

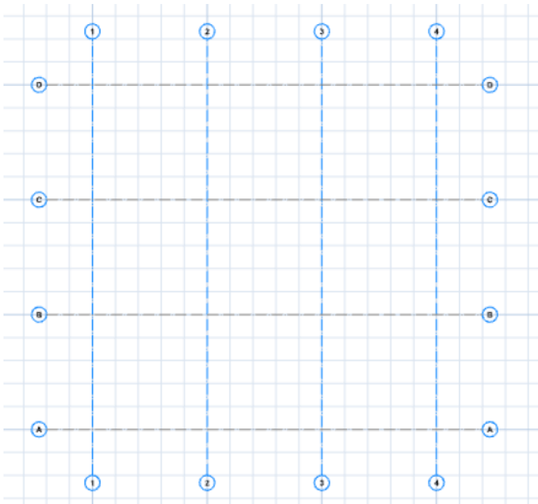


Direction 1 axes are placed horizontally with alphabetic labels (incremented from bottom to top).

Direction 2 axes are aligned vertically with numeric labels (incremented from left to right).

- *Accept all the **default** values in the inputs and click **OK***



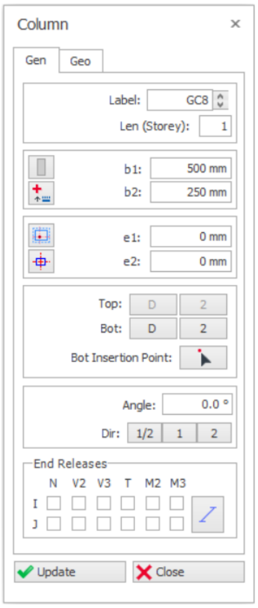


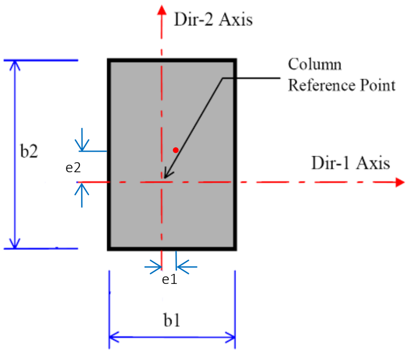
Four horizontal and vertical axes with a 5 m spacing will be created.

If a reversed axis label is desired, then enter "**Step**" = **-1**. For example, for Dir-1 Axes, if you want Axis A to start from the top instead of bottom, enter "Axis Label" = **D** (last label) and "Step" = **-1**

Columns Creation

- *Pick **RC Column** icon*
- *Use the default size **b1** & **b2** and eccentricity **e2** & **e2** is as shown below*

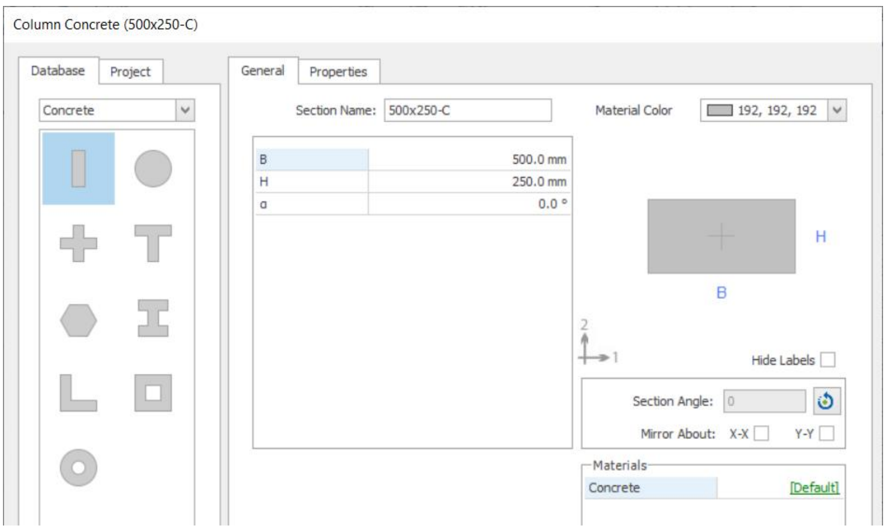




e1 & e2 is measure from the centroid of the column

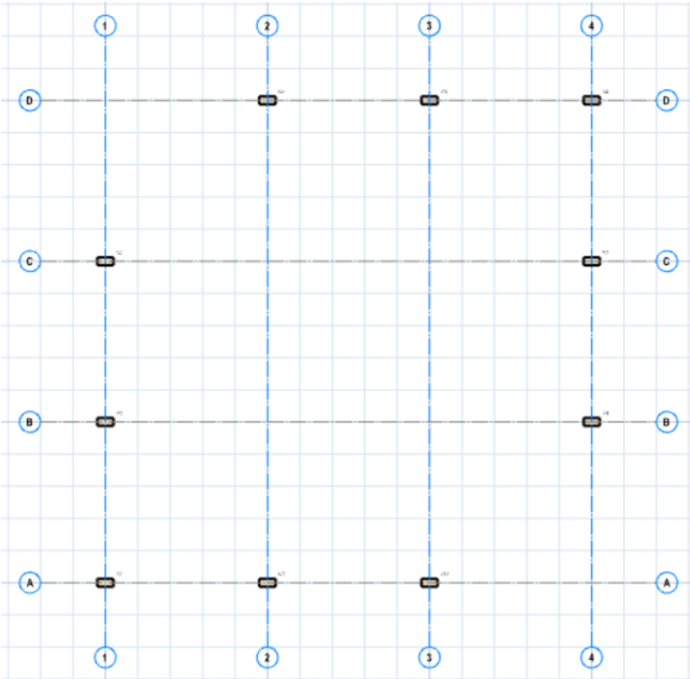
Section Manager : Allows access other section types such as **circular**, “L”, “T”.

Column End Condition : Columns ends are fixed by default. You can apply hinges to top and/or bottom by clicking successively on this icon.



- *Insert columns by clicking on the **intersection of axes***
- *Multiple columns can be inserted by **dragging a box** around the intersection of axes*

Using the two methods, create **ten** columns at positions shown below.



- *Close the column properties.* You finish member creation.

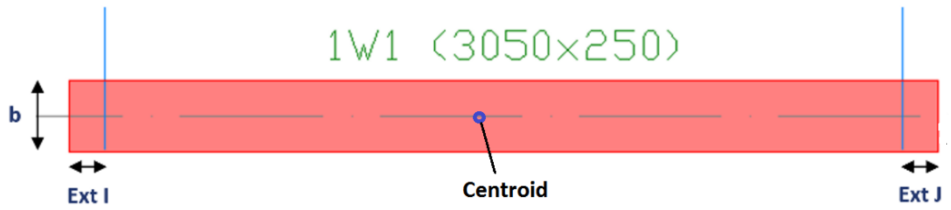
Always close member properties when you finish member creation, otherwise you will be unable to proceed to other functions, such as member selection.

Walls Creation

➢ *Click on the **Wall** icon & the wall properties will appear*

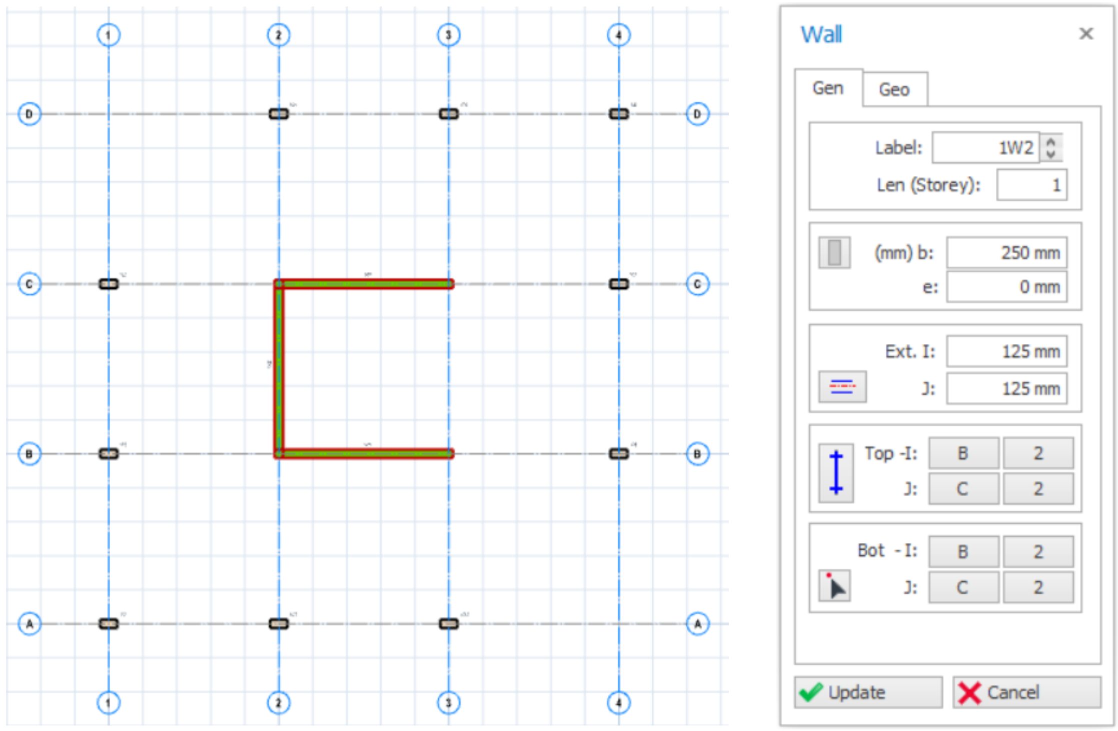
➢ *Use the default wall thickness **b = 250 mm** & **e = 0 mm***

The parameters are explained in the diagram below



e is measured from the centreline of the wall to its centroid. e = 0 means that the centreline of the wall coincides with the centroid of the wall.

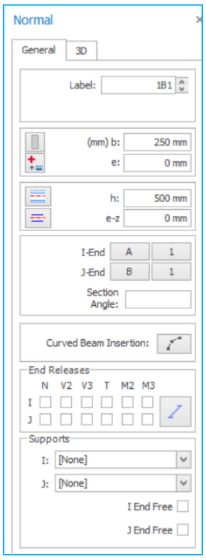
➢ *Insert three walls by simply clicking on the shear wall's start and end node.*

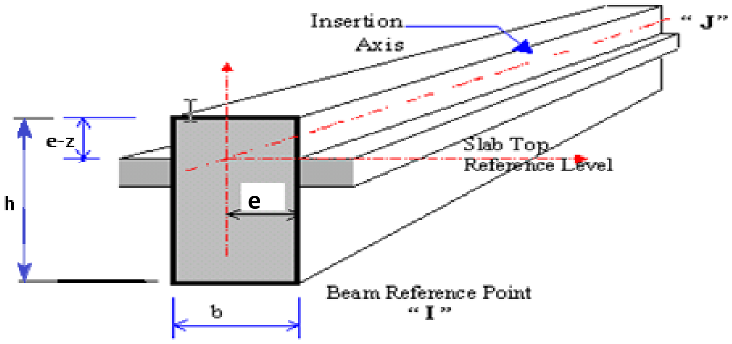


Ext I & J : Slight extension to lengthen the wall should be kept small; less than its width as the extension cannot support any members, such as beams.

Beams Creation

➢ *Click on the **Beam** icon → Pick **Normal** → the beam properties will appear.*





Beam Parameters

e is measured from the beam's centreline to its sectional area centroid. e = 0 means that the beam's centreline coincides with its area centroid.

e-z: +ve value raise the beam from the story level, -ve lowers the beam

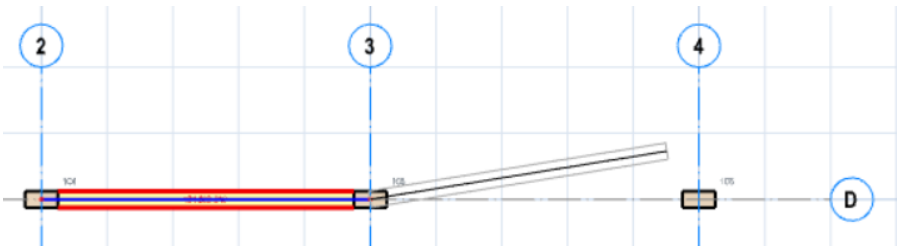
End Releases: Beam ends are fixed by default. To release forces (N,V,T,M), check the box(s).

Click successively on this icon to hinge & release all moments to the left or right end of beam.

➢ *In the beam properties, use the defaults values **b = 250 mm** & **h = 500** mm (as shown above)*

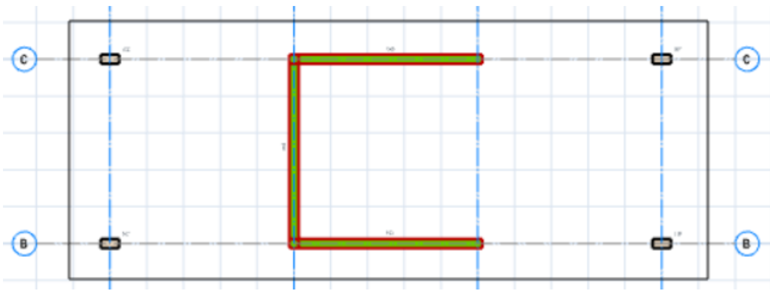
➢ *To create a beam, pick the start and the endpoints by clicking on the **intersection of axes***

Notice that you can continuously create the beams from the previous beam.

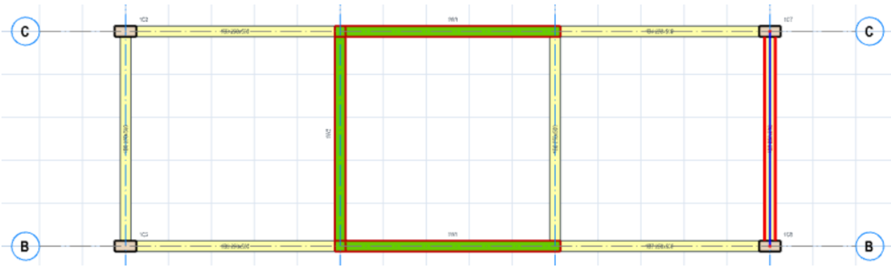


➢ ***Right-click** to end the beam insertion after inserting the two beams as shown above*

Multiple beams can also be inserted by dragging a box enclosing the area you want to insert beams. Beams will automatically be created between columns and walls.

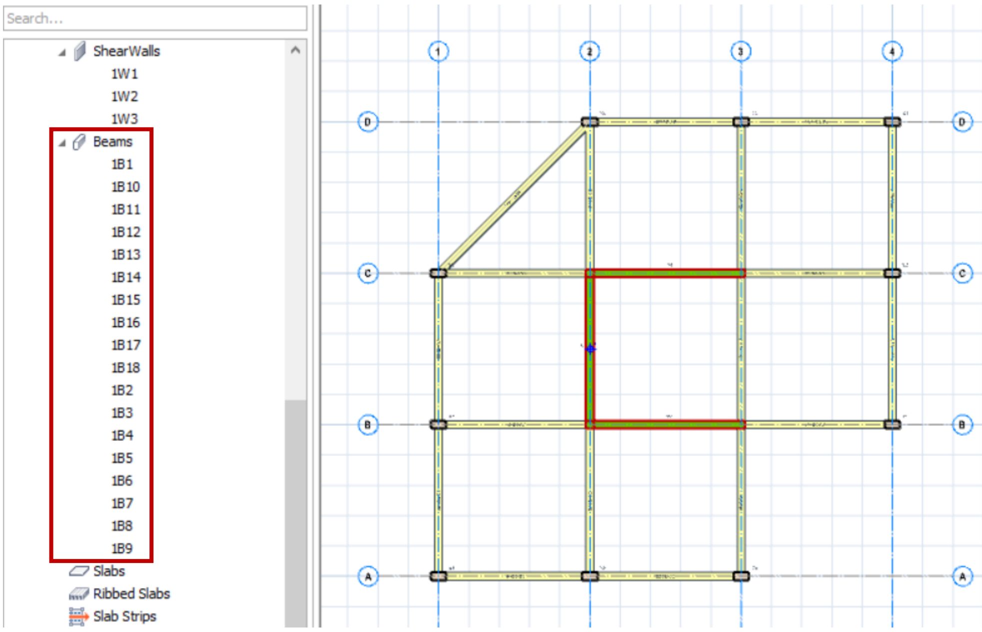


➢ *Click and drag box enclosing the columns & walls shown above.*



Seven (7) new beams will be created.

➢ *Insert the rest of the beams by any method as shown below*

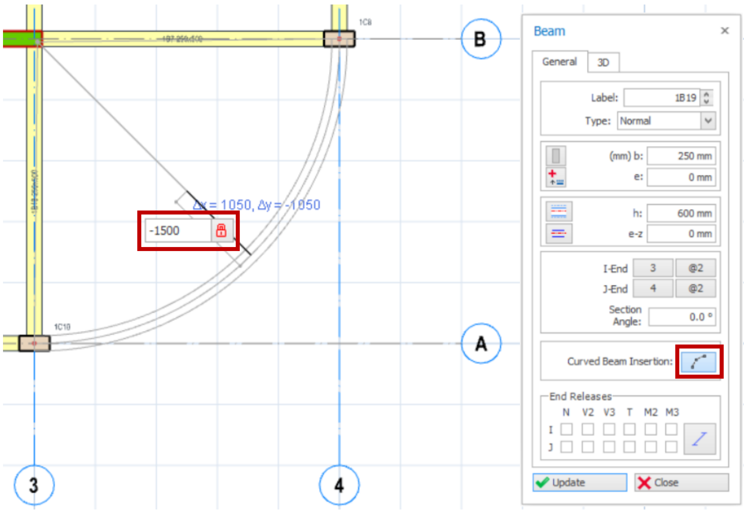


Check the folder **Beams** in the **Structure Tree** to ensure you have inserted a total of **18** beams.

We will now insert a **curve beam** in the lower right corner of the model.

➢ *Click on the **curve beam** insertion icon*  *in the Beam Properties dialog*

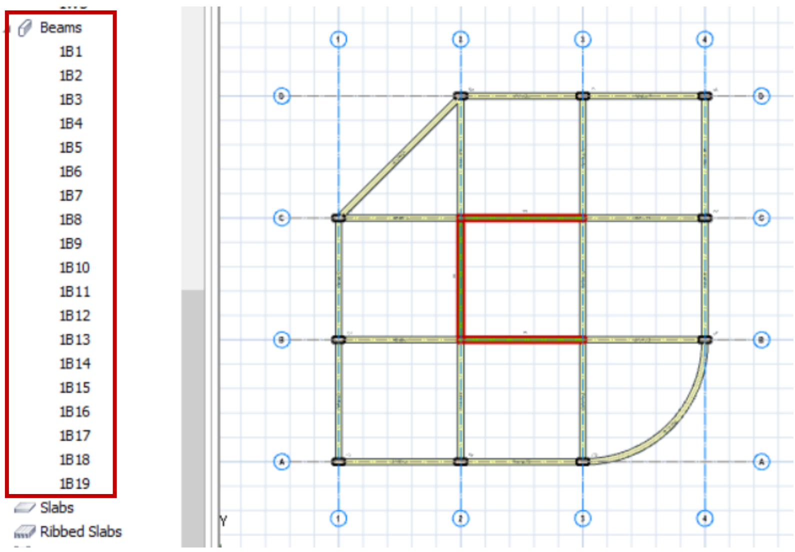
➢ *Click on the intersection of **axes A/3** and then the intersection of **axes B/4** (i.e., start & end of the beam)*



Define the apex distance by simply moving the mouse cursor, and the preview of the curve beam in grey will show automatically.

- *Press **F2** to define the apex exactly as **-1500** mm & press **ENTER***

A curve beam will be inserted (segments are automatically created). Check the Structure Tree that you have modelled for a total of ***19 beams***.

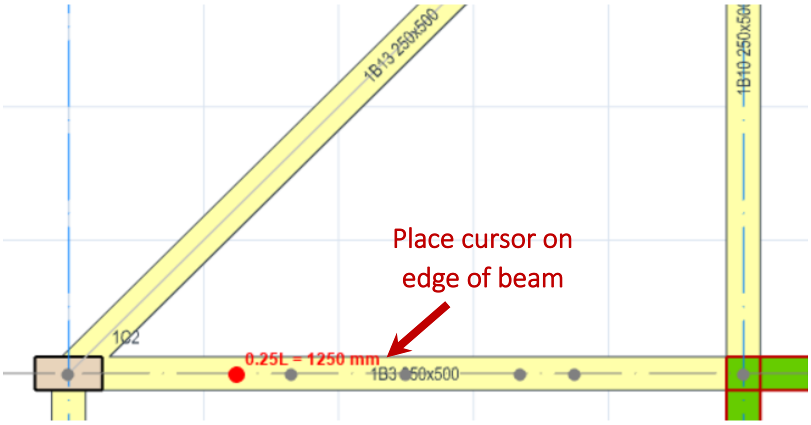


Beams Creation using dynamic snap points

Note: This section is optional and is not part of the final model, but you might want to try it anyway.

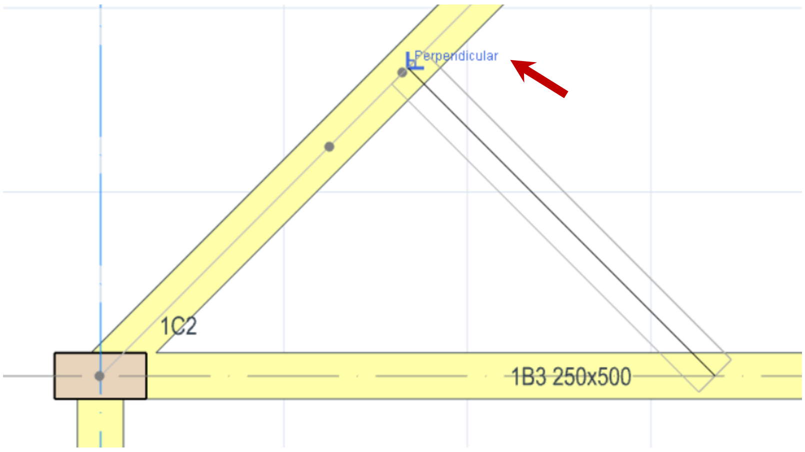
Secondary beams can easily be inserted using dynamic snap points (without creating axes).

- *Click the **Beam** icon & place the cursor on the **edge** of the primary beam (avoid the Axis)*



Notice that snap points on **0.25L**, **0.33L**, **0.5L**, **0.67L**, **0.75L** will be shown when the cursor is placed on the beam.

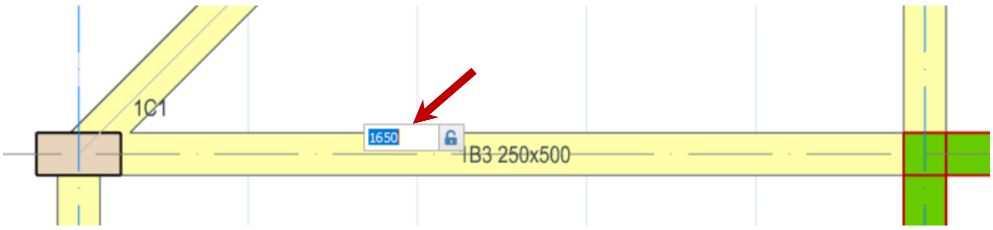
- *Click on **0.67L** as the start point of the secondary beam.*
- *Place the cursor on the destination beam*



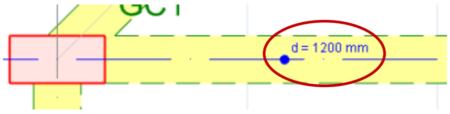
Notice that **0.25L**, **0.33L**, **0.5L**, **0.67L**, **0.75L** & **perpendicular** point will appear

- *Select the desired point as the endpoint of the secondary beam & a new beam will be created*

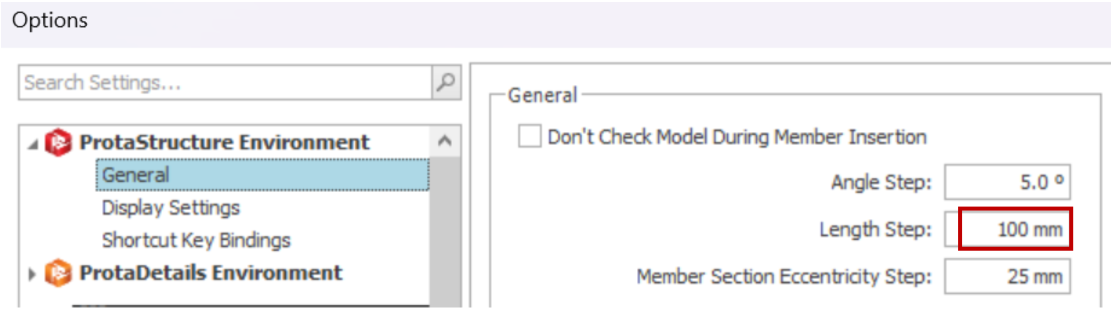
Pressing *F2* while the cursor is at any of the snap points will allow input of the exact distance from the start of the beam.



Pressing ***the CTRL*** key while the cursor is placed on the edge line of the primary beam will expose a distance increment of 100 mm. Left-click to select the desired distance (while still holding down the CTRL key).

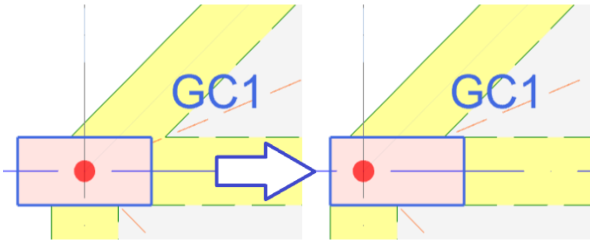


The step increment can be set ***Building Setout*** → ***Settings Center*** → ***ProtaStructure Environment*** → ***General*** → ***Length Step***.



Handy Tip to adjust the position of columns and beams

You can change the position (eccentricity) of the column and beam by simply selecting it & then pressing keyboard arrow keys to move in the direction you want.



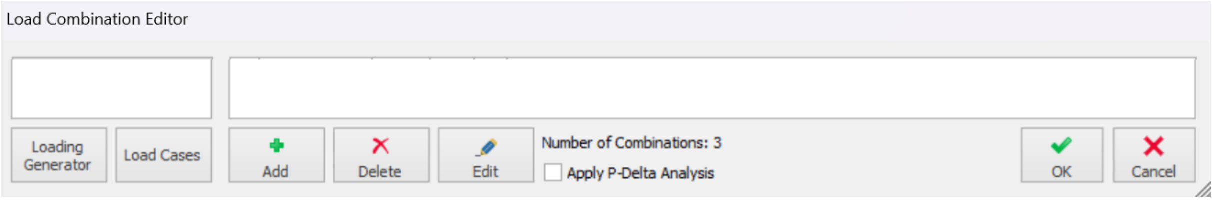
- *Select any column*
- *Press right arrow key → to move the column to the right*

The step by which the member is moved can be set in ***Building Setout*** → ***Settings Center*** → ***ProtaStructure Environment*** → ***Member Section Eccentricity Step*** (by default 25mm).

Load Combinations

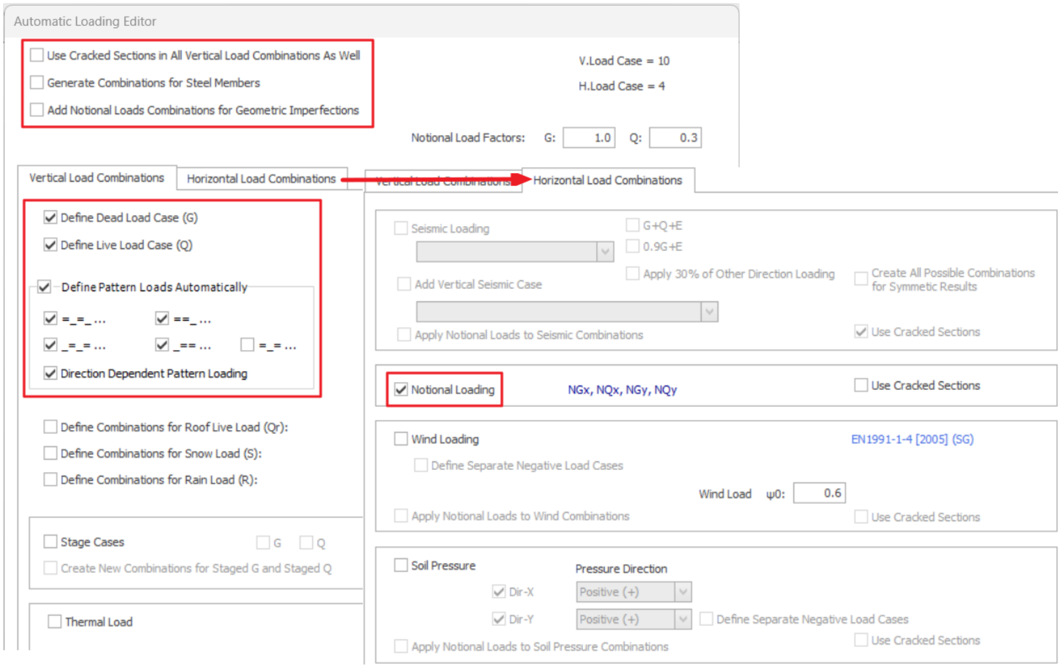
Before creating slabs, we need to define the load cases & combinations.

➢ *Go to the **Loading** tab, click **Load Cases and Combinations**.*



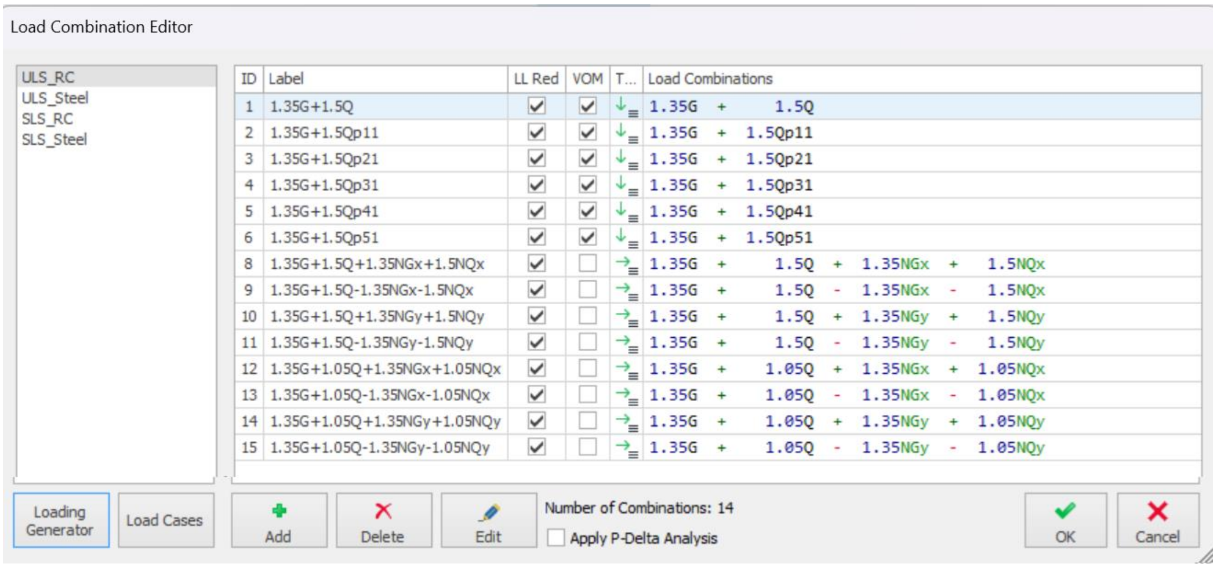
You can add new load cases manually and then load combinations. However, it's easiest to use the Loading Generator to set up load cases and combinations automatically.

➢ *Pick **Loading Generator** & use the options as shown below and click **OK***



You can choose which combination is used for concrete & steel design separately.

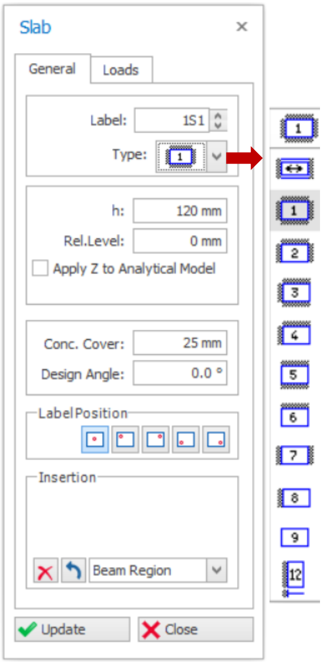
If "**Use Cracked Sections**" is checked, section stiffness factors, as defined via **Material and Section Effective Stiffness Factors** menu, will be applied for that load case.
As shown below, all the load cases and combinations will automatically be generated.



➢ *Pick **OK** to save & close the Load Combination Editor.*

Slab Creation

➢ *Click on the **Slab** icon and the slab properties dialog will appear*



➢ *Click on the **Type** box*

All the possible Slab Types will appear in pop up menu.

The slab type 1 to 12 relates the slab edge continuity and is used in the design of the slab reinforced based on the coefficient / yield-line method.

Type 1 to 12 does not affect the slab load calculation on the supporting beams, only the design of slab using coefficient method.

One-way slab. The slab load will only to be transferred to the 2 supporting beams in the direction of the span.

The direction of one-way span must be specified in the **Design Angle** input box : Angle: 0.0 °

Rel. Level adjusts the top level of slab. Default zero means top of slab is at top of storey level. Positive value raises the slab, vice versa.

Apply Z to Analytical Model : If checked, the adjustment of Rel. Level above will be considered in analysis, i.e. the slab level will also be changed analytically.

Label Position icons activate the slab label and control the slab label's position.

Insertion method: By default, "**Beam Region**" is selected. This means the slab will be inserted within the region bounded by the beams.

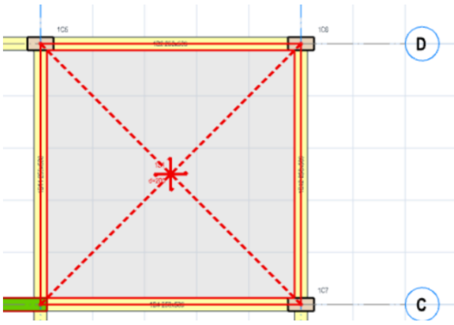
- *Select **Slab Type** as **1***

  For ease in creating this model, we will initially leave Slab Type as 1 for all slabs. Once they have been created & when we are ready to design the slab, there is a function to automatically set the correct type for each slab.

- *Enter the slab thickness **h = 200 mm** and **concrete cover = 30 mm***
- *In the **Loads** tab, enter **Service Dead Load = 1.2 kN/m2** and **Imposed Load = 3 kN/m2***

Different types of service dead load can be defined via **Building Setout → Libraries → Slab Additional Loads**

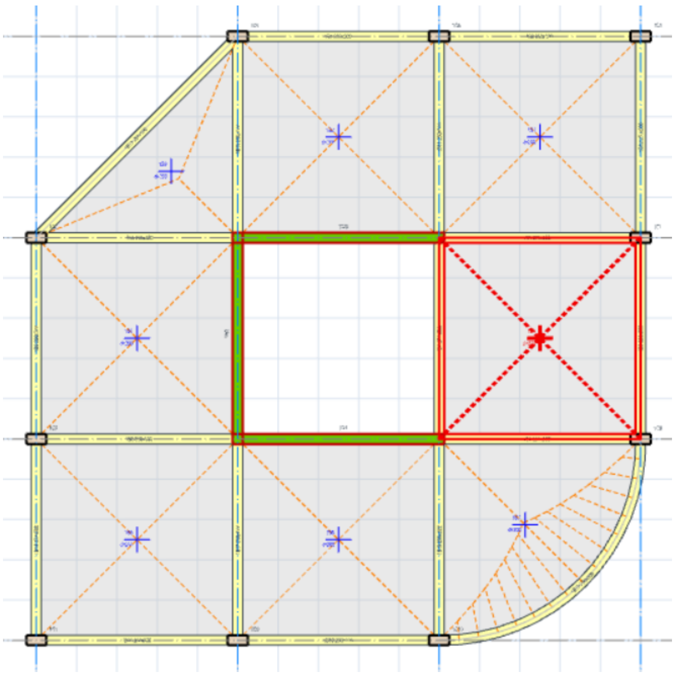
- *Position the cursor in the region bounded by beams @**GL 3, 4, C, D**, and left-click to create a slab.*



The first slab will be inserted.

The yield line shows the tributary area load this is automatically calculated onto the supporting beams. By default, slab load calculation is done automatically onto supporting beam using yield-line method.

- *Create **seven more slabs** to give the layout shown below*

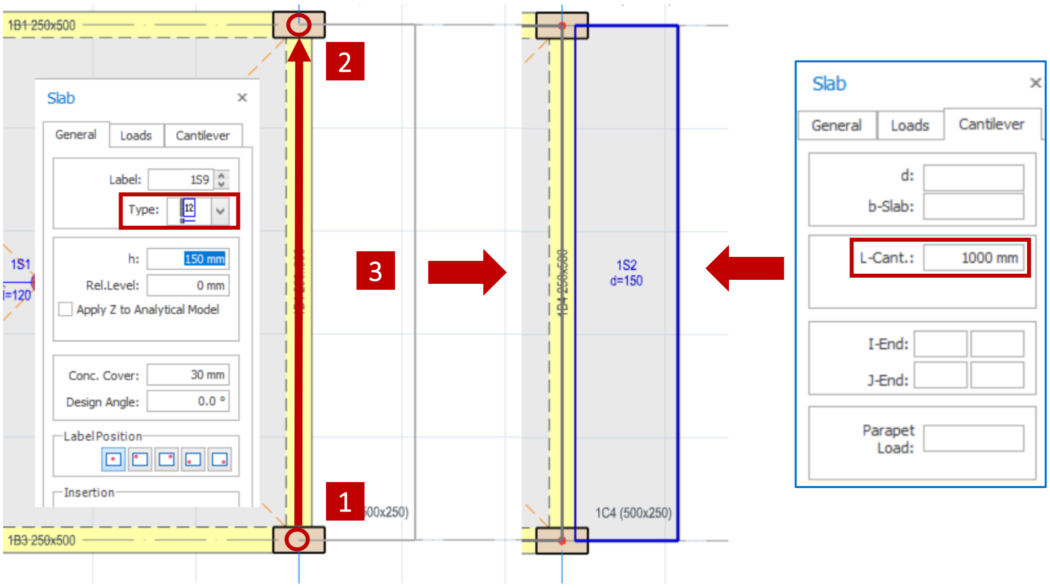


Inserting Cantilever Slabs (Type 12)

Note: This section is optional and is not part of the final model.

Cantilever slabs can be inserted by setting Type = 12 in the slab property. They are inserted by three clicks - clockwise or counter-clockwise direction does not matter. Cantilever slabs can span more than one beam or wall.

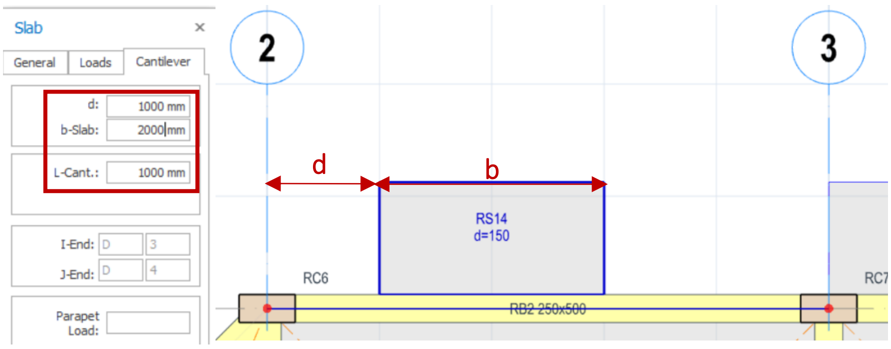
- *Zoom to the top-right corner of the model*
- *Click on* ***Slab*** *icon & choose* ***Type 12*** *& enter* ***h = 150mm***
- *Go to* ***Cantilever*** *tab → Enter* ***L-Cant*** *=* ***1000 mm*** *(cantilever length)*



- ***Pick the first*** *intersection of the Axis to define the start of the cantilever*
- ***Pick a second*** *point to define the end of the cantilever*
- ***Move the cursor*** *to the side where it is to be inserted → a grey preview will be shown*
- ***Pick the third*** *point to confirm & insert the cantilever*

To insert a cantilever slab that does not span the entire length of the beam, we must enter values of ***d*** & ***b-slab*** in the cantilever slab properties :

- *In Slab Properties, enter* ***d = 1000 mm*** *&* ***b-Slab = 2000 mm***



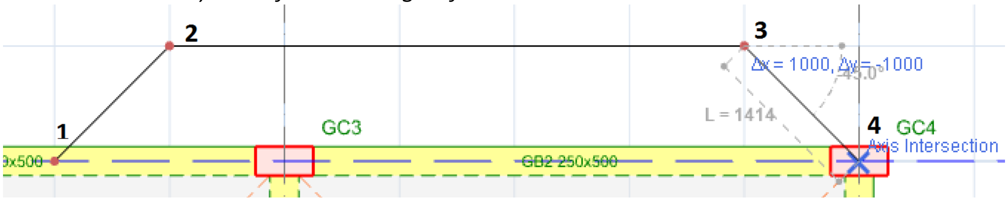
- ***Pick the first*** *intersection of the Axis to define the start of the cantilever*
- ***Pick a second*** *point to determine the end of the cantilever*
- ***Move the cursor*** *to the side where it is to be inserted → a grey preview will be shown*
- ***Pick the third*** *point to confirm & insert the cantilever*

Polyline Slab/Column Edge

Note: This section is optional and is not part of the final model.

The slab edge line can be used to create irregular slab shapes

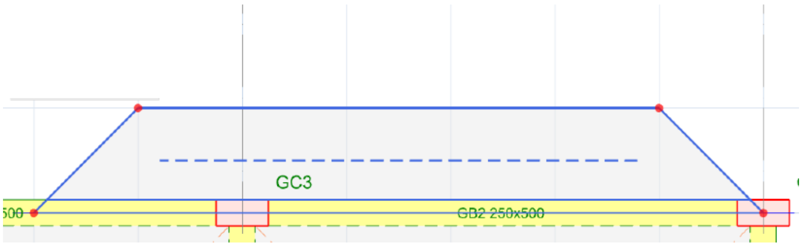
- *Click **Polyline Slab/Column Edge** under **Modelling***
- *Click continuously to define the edge of the cantilever slab.*



The method to draw the edge lines is similar to a multi-segment axis.

- *At the last point, **right-click** & pick **Finish** to end the insertion. The slab edge lines will be drawn*

- *Click on the **Slab** icon → Ensure **Type = 1** is selected*
- *In the **Insertion**, ensure that **Beam Region** is selected.*
- *Click anywhere within the slab edge line, and a new slab will be created.*

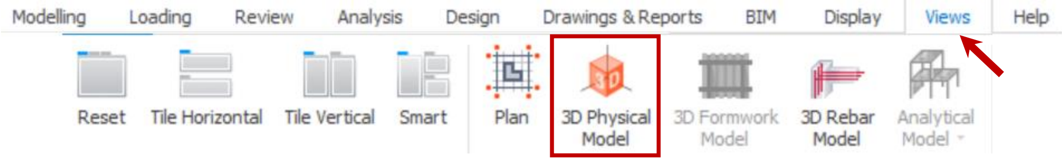


For regular cantilever slab, it is recommended to use **Type 12** in slab properties (instead of slab edge lines).

The slab/column edge line also includes **Circle**, **Arc** & **Rectangle** and the ability to convert to a column or slab hole or slab drop. After drawing a closed shape line → Select it → Right-click → Convert to Column or Slab Hole/drop.

Views Creation

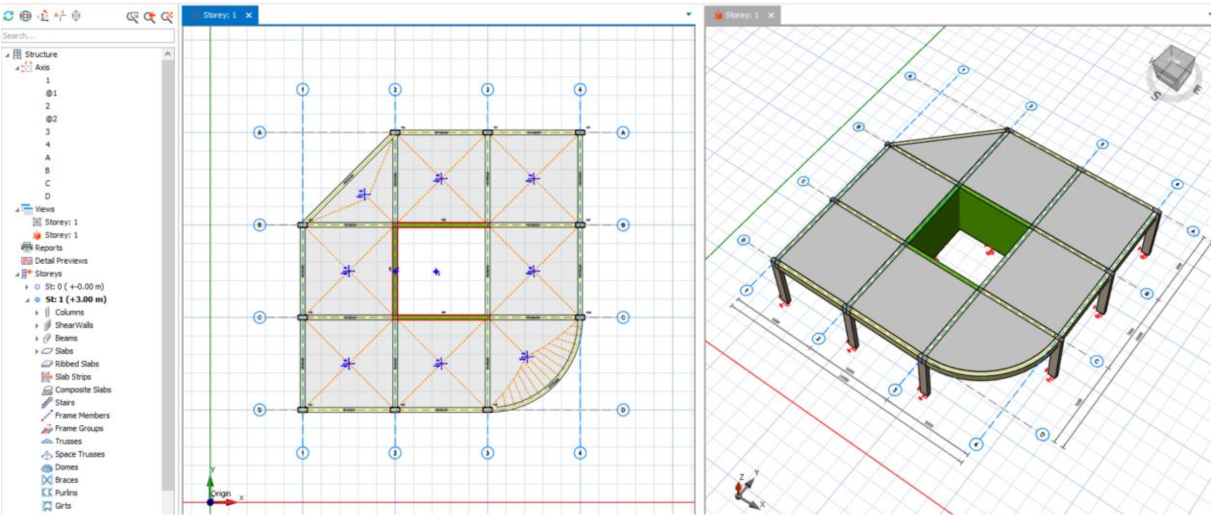
The graphical editor supports multiple windows; this allows you to create different views in separate windows. The **Views** tab commands can be used to create new views and to arrange the views.



- *Go to* ***Views*** *tab → Click* ***3D Physical Model***

  This command will create a 3D physical model view.

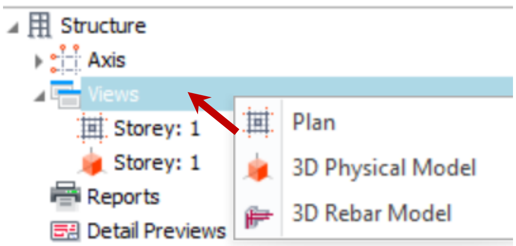
- *Click* ***Tile Vertically*** *→ This will tile the plan and 3D view side by side*

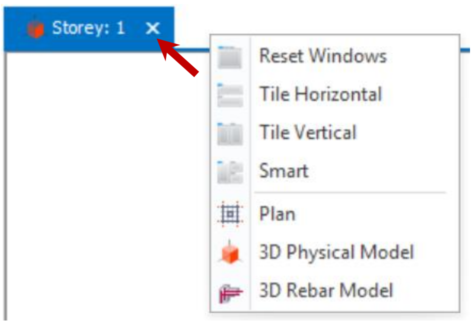


You can select & modify existing members or insert new members in the 3D view the same way as the plan view.

To isolate the active storey in 3D view, right click anywhere on 3D view and click Isolate/Unisolate Storey

Note: Alternatively, you can right-click on **Views** in the Structure Tree or header field of the existing view tabs to create new views.



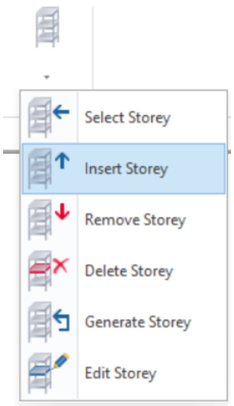


- ***Select*** *the curve beam in the 3D view. Notice the same member will be selected in the plan view.*
- ***Right-click*** *(to access the context menu) →* ***Properties***
- *Change the depth of the beam* ***h-bot*** *to* ***600 mm*** *(as curve beam longer than rest of the beams)*
- *Click* ***Update*** *and* ***Close*** *the beam properties*

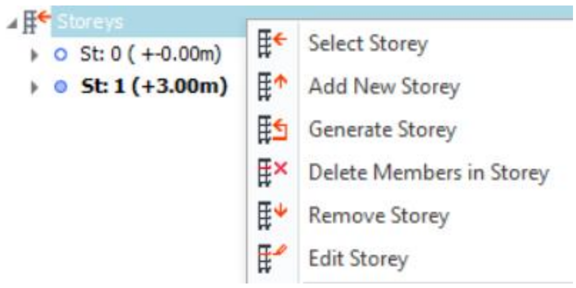
Inserting Storeys & Defining Building Parameters

The final model will be a four-story building. We will now insert the other floors.

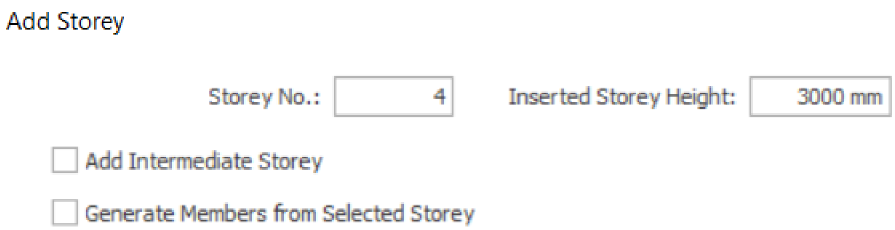
- *Go to **Building Setout** → **Storeys Operations** dropdown → **Insert Storey**.*



- *Alternatively, right-click on **Storey** and pick **Insert Storey***



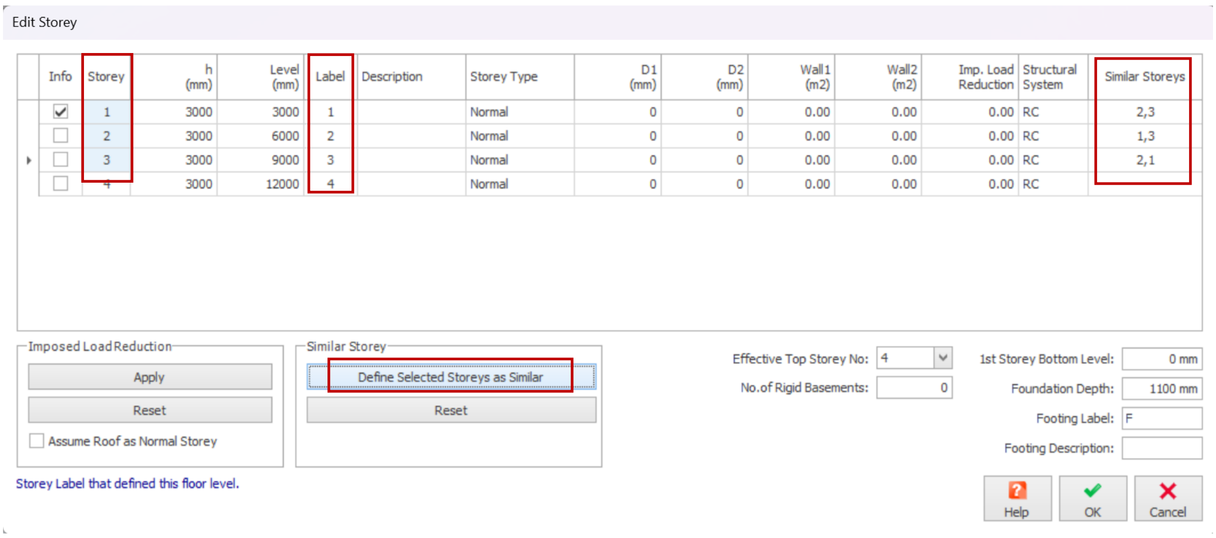
- *Input **Storey No : = 4** → **OK***



- *When prompted to confirm → Pick **Yes***

This automatically inserts stories 2, 3 & 4 with default height 3m. We will now edit the information of the stories.

- *Pick **Edit Storey** from the same **Storey** dropdown menu to enter the Edit Storey dialog*
- *Hold **CTRL** key and multiple select Storey 1, 2 & 3. Then click "**Define selected stories as similar.**"*



Info	Storey	h (mm)	Level (mm)	Label	Description	Storey Type	D1 (mm)	D2 (mm)	Wall1 (m2)	Wall2 (m2)	Imp. Load Reduction	Structural System	Similar Storeys
☑	1	3000	3000	1		Normal	0	0	0.00	0.00	0.00	RC	2,3
☐	2	3000	6000	2		Normal	0	0	0.00	0.00	0.00	RC	1,3
☐	3	3000	9000	3		Normal	0	0	0.00	0.00	0.00	RC	2,1
☐	4	3000	12000	4		Normal	0	0	0.00	0.00	0.00	RC	

Storeys 1, 2 & 3 will now be identical. Since we have already inserted members in story 1, these members will be automatically copied to stories 2 and 3. In addition, changes to a particular similar level will be applied automatically to all similar stories.

For storeys to be identical, they must also have the **same storey height**. In an actual project, ST01 is the ground floor & will have a shorter storey height & hence should not be made similar.

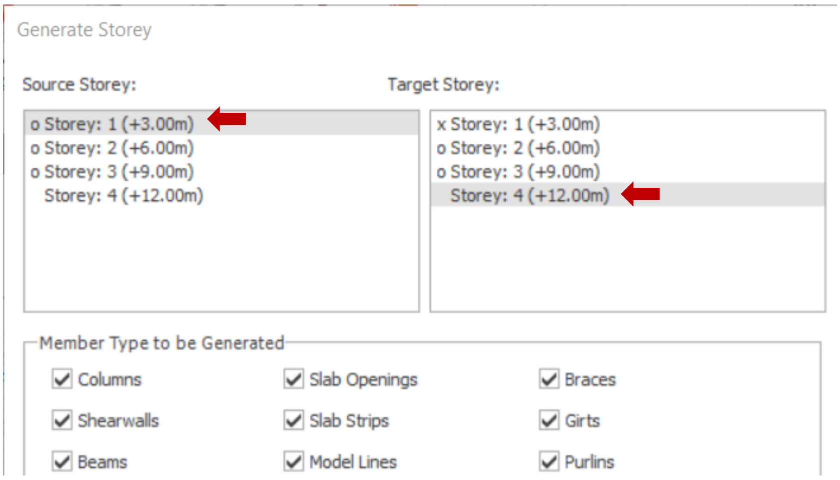
➢ *Change the member label under the Labels column as shown above, e.g., "G" for ST01 and "**R**" for ST04.*

ST01 members will be labelled GB1, GC1, etc.

➢ *Click **OK** to exit and notice that the 3D view shows storey 1, 2 & 3 with identical members.*

Storey 4 (Roof) has no members, so that we will generate (copy) members to this storey.

➢ *Pick **Generate Storey** from the same **Storey** dropdown*



➢ *Under **Source Storey,** pick **Storey 1** & then select **Storey 4** as **Target Storey***

➢ *Click **OK,** and members will be copied from the ground floor to the roof*

➢ *Click **Close** and check that members are indeed created in storey 4*

➢ *Switch to show **storey 4** in the plan view by **double-clicking** on storey 4 in the structure tree*

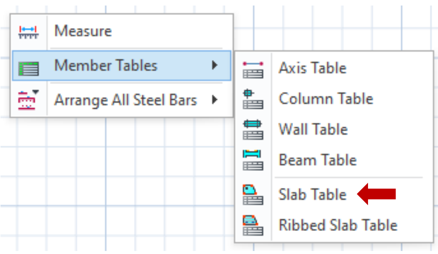
➢ *Create a **new slab** with the same properties over the lift core since this is the roof level*

➢ *Press **ESC** to deselect all previously selected members*

➢ ***Right-click** anywhere on the plan view to expose the context menu*

➢ Select ***Member Tables** → **Slab Table***

The slab table allows you to view & change the parameters of the slabs in a group

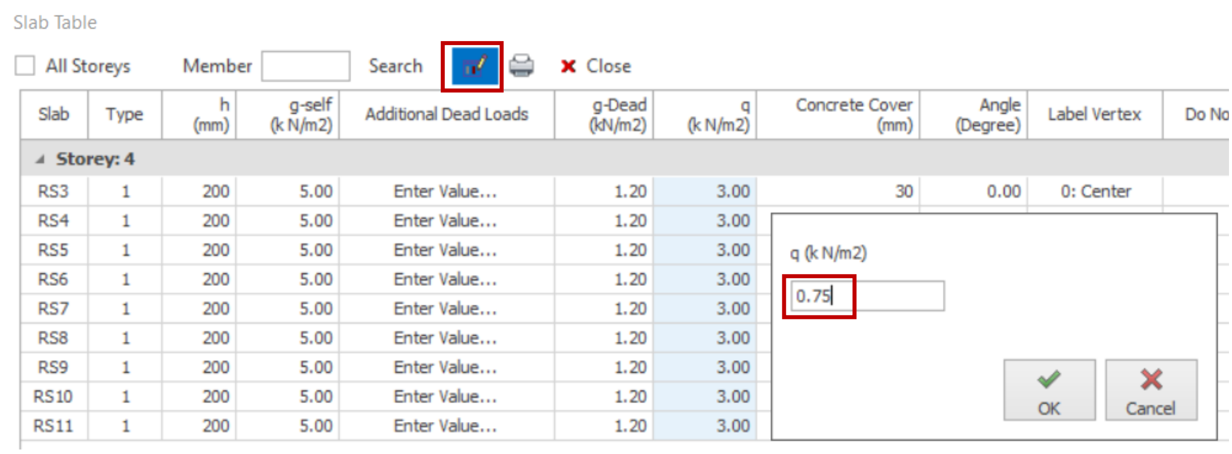


➢ *Click on the **header q** (live load), and the entire columns will be highlighted*

➢ *Click the **Column-wise Edit** icon at the top*

➢ *Enter **0.75** kN/m² → press **ENTER** (all the slabs live load values will be changed)*

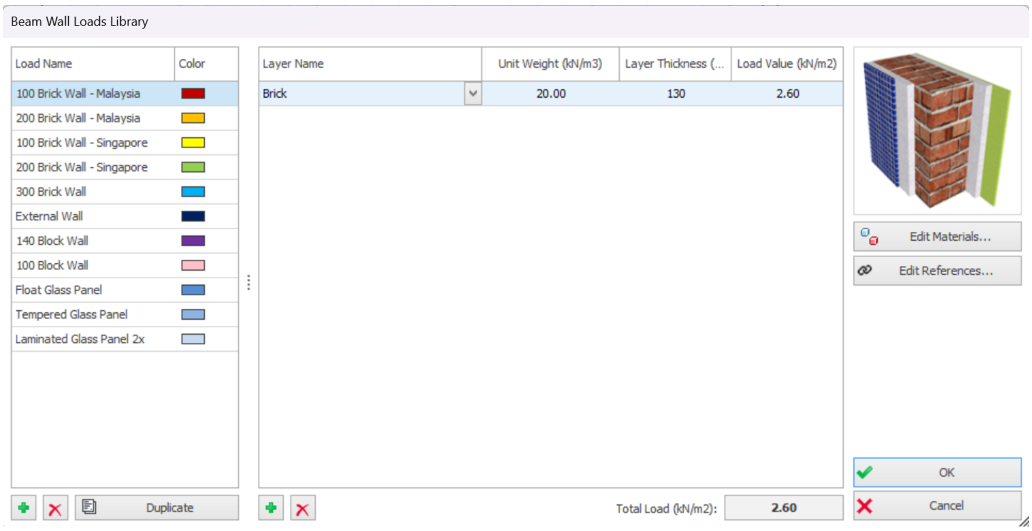
➢ ***Close** the Slab Table*



Wall Loads Library

You can set up pre-defined wall loads and then apply them using a dropdown list when adding the wall loads.

➢ *Go to **Building Setout** → **Libraries** → **Partition Wall Loads***



Defaults wall types are automatically set up, but you can add any new user-defined wall type.

➢ *Choose **Cancel** to exit*

We will now insert brick wall loading on the beams in ST01.

➢ *Make the plan view active by clicking on it*

➢ *Double click on **ST01** in the structure tree to make it active*

➢ *Select the bottom-left most perimeter **beam along with Grid A / 1-2***

➢ *Click **Edit Loads** in the **Beam** tab that appears.*

Member Load Editor

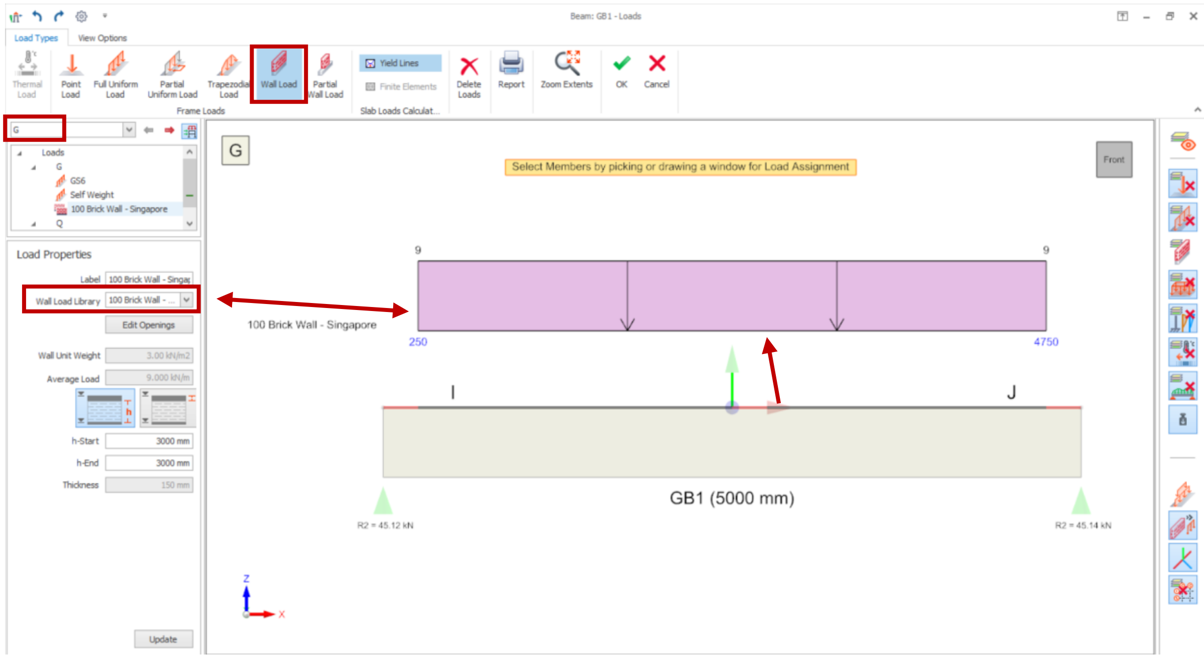
The load editor of the beam will appear once the load cases and combinations are generated.

1. Load Types
    - Click to choose the load type to insert, e.g., point load, full uniform load.
    - Self-weight & decomposed slab loads are auto-calculated & cannot be edited.
2. Load Case & Loads Folder & Load Properties
    - Choose the load case to insert the loads. Existing loads will be listed for the selected load cases.
    - Click on the load name to select it > Right-click will expose menu options to Edit, Cut, Copy or Remove the load.
3. Load Properties
    - Enter Label, Direction & Magnitude values for the load to be created.
    - Select it in the Loads Folder or Loads diagram > Revise values > Click Update to update an existing load.
4. Loads Diagram
    - It displays the graphical representation of the loads, including self-weight & slab loads calculated on beams.
    - Existing loads can be selected by left-clicking on the diagram > Right-clicking will expose menu options to Edit, Cut, Copy or Remove the load.
    - Click on the beam diagram to insert a load after choosing the **Load Type** & input values in the **Load Properties.**
    - The load coordinate system is shown with X, Y, Z arrows. The beam & loads diagram can be rotated by right-clicking & drag.
5. Load Type filter
    - Click the Load type icon to toggle on and off the associated diagrams.

6. View Options & Shortcuts

- These settings control the rendering of the member, loads graphics, text height, scaling & frame labels.
- Show Loads as merged : Loads under the same load case will be combined in a single diagram. This option must be **switched off** to edit or remove the load.
- Show Infill Wall as line Load : If "on" infill wall will be shown as line usual line loads. The infill wall will be rendered with actual height and width if the setting is turned off.

Let us insert a 100 mm preset brick wall load on this beam :



➢ *Pick **Wall Load** under **Load Types** ribbon.*

Notice only the wall load type layer will be active, and all other load types will be switched off

➢ *Ensure **G load case** is selected in the Loads Folder at the left.*

➢ *In the Load Properties, Wall Load Library, pick **100 Brick Wall - Singapore**. Note wall height is preset to **3000** mm = storey height.*

➢ *Place the mouse cursor on the beam & left click to create the load.*

The load will be created and added in the diagram & loads folder.

➢ *Press the **ESC** key to finish the wall load insertion process. Click **OK** to save & exit.*

The wall load is accurately applied to the face of the column, not at the ends of the beam.

Wall Opening, e.g. of windows & doors can be inserted by clicking "***Edit Openings***".

Partial wall can be entered by clicking on **Partial Wall Load** icon

The brick wall load will be rendered in 2D and 3D views. Notice wall load is also copied to a similar storey as expected.

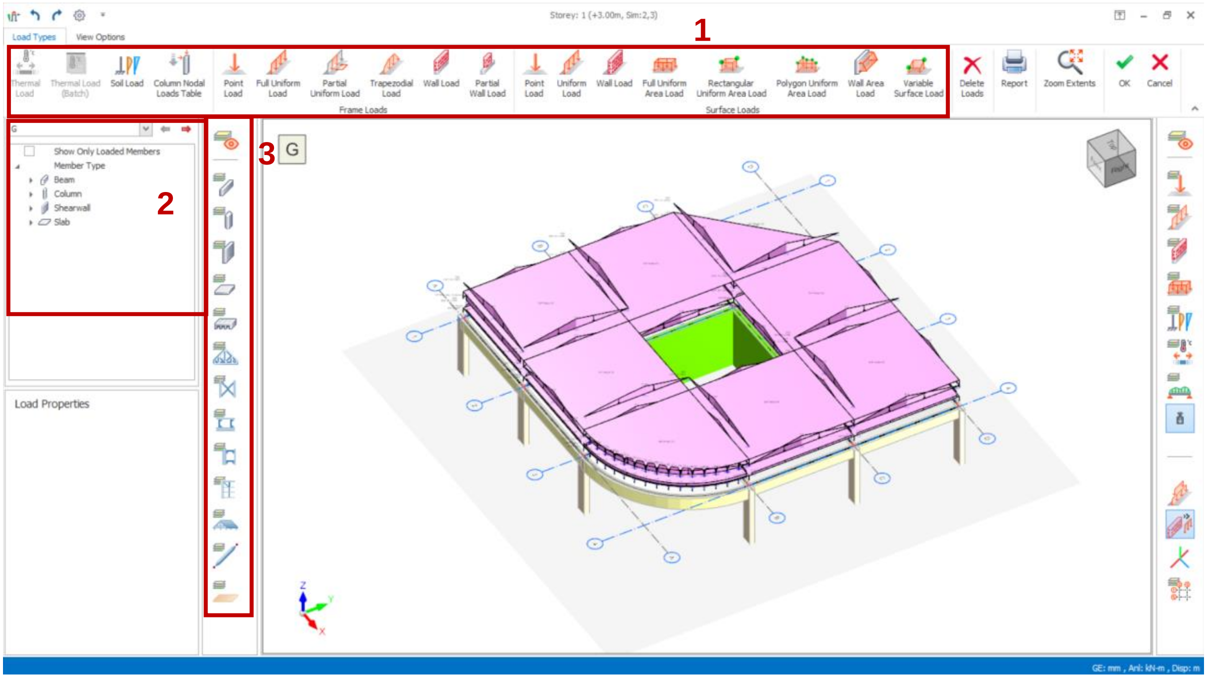
Storey Load Editor

Loads can also be entered for the entire storey, instead of a single member :

➢ *Ensure **ST01** is active & no members are selected (press **ESC** key)*

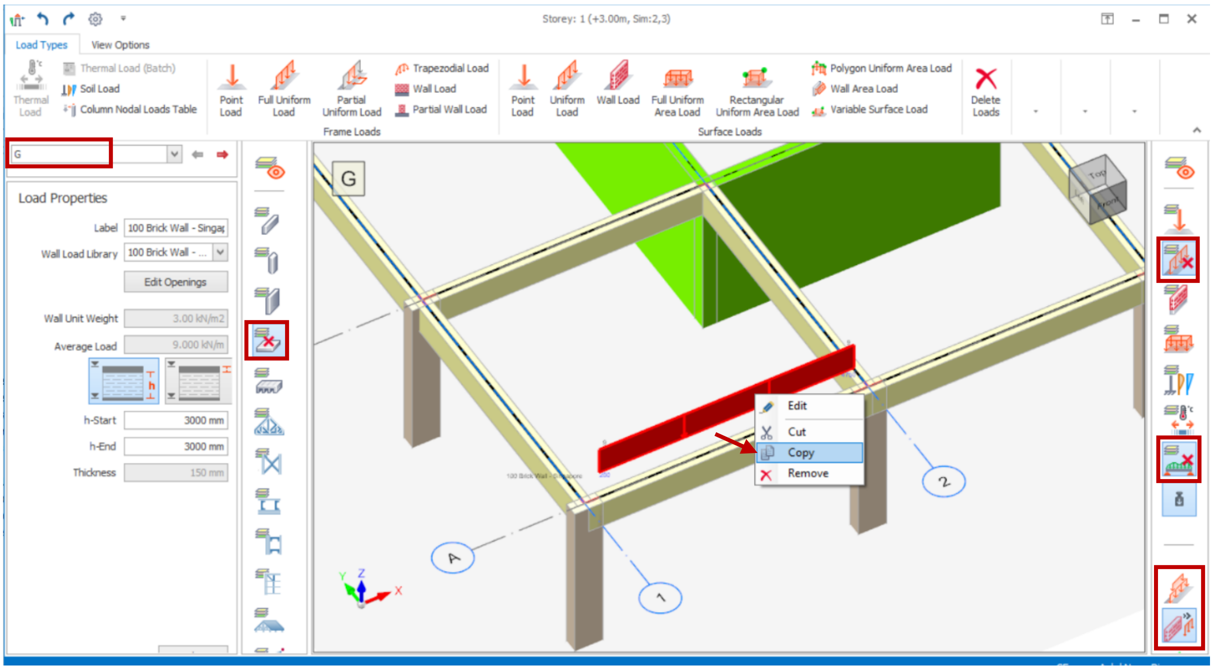
➢ *Go to **Loading** tab > Pick **Load Editor Storey: 1***

The storey load editor is similar to the member's load editor, except that all the members & loads on the active storey are shown. The differences are described below.



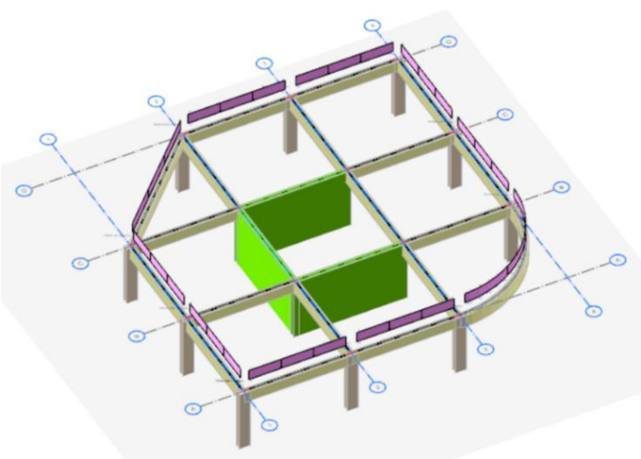
1. Load Types
   - Surface Loads are loads are area loads that can be applied to slabs and walls
   - Self-weight is auto-calculated & cannot be edited.
2. Load Case & Loads Folder
   - Choose the load case to insert the loads. Existing loads will be listed for the selected load case.
   - **Loads folder** resembles the **Structure Tree,** which organizes loads under member type
   - **Show only Loaded Members**: If checked, will omit any members without any loads from listed.
   - Expand the folder > Click on the load name to select it > Right-click will expose menu options to Edit, Cut, Copy or Remove the load.
3. Member Filter
   - Toggle visibility of member type in the diagram. For example, if the Beam layer is off, then all beams & loads applied to beams will be hidden.

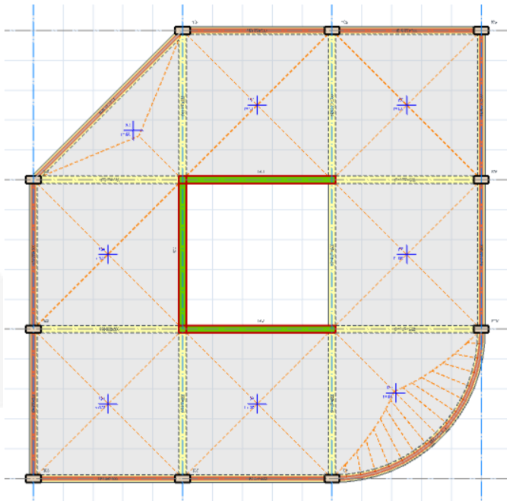
We will now copy the brick wall load to all the perimeter beams.



In View Options : Show Loads as merged is by default deactivated, so that the wall loads are shown separately. Show Infill Wall load as Line Load is by default activated, so wall load is shown as line load, instead of drawn to actual wall height.

- *Choose **G load case** in Loads Folder.*
- *In **Member Filter**, toggle off the **Slabs** layer, so the slab and slab area loads are turned off.*
- *In **Load Type Filter**, toggle off **Line Loads** & **Decomposed Line Loads**, to isolate the wall loads*
- ***Select** the partition wall load in the diagram > **Right-click** > **Copy***
- ***Multiple select** all the perimeter beams by holding down the **CTRL** while selecting the beams.*
- ***Right-click** > **Paste >** Check the same wall load is pasted correctly.*
- *Go to **Load Types** tab > Click **OK** to exit the dialog.*



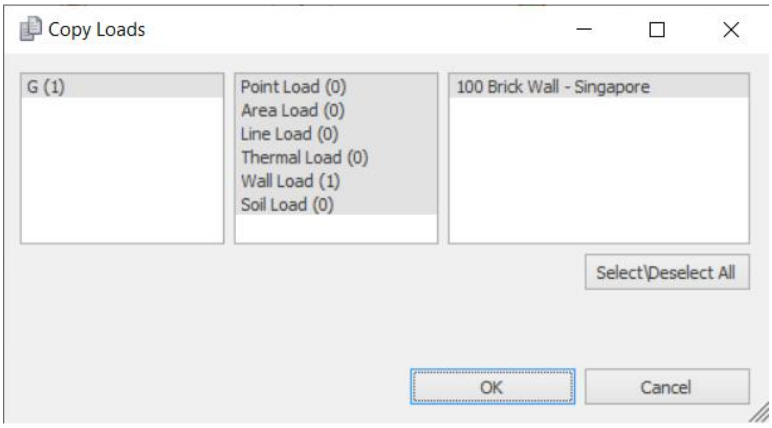


- *Examine the 2D & 3D views to ensure all wall loads are inserted correctly.*

Alternatively, wall loads can also be copied in the plan view by the following steps :

➢ *Select beam in plan view with wall load inserted*

➢ *Right-click > **Copy Loads** > Select the brick wall load > **OK***



The load case & load type panes act as filters for selecting the load name on the right pane.

➢ *Multiple select the perimeter beams where loads are to be pasted (holding **CTRL** key)*

➢ *Right-click > **Paste Loads***

Similar steps can be used to delete loads by selecting beams > Right-click > **Delete Loads**.

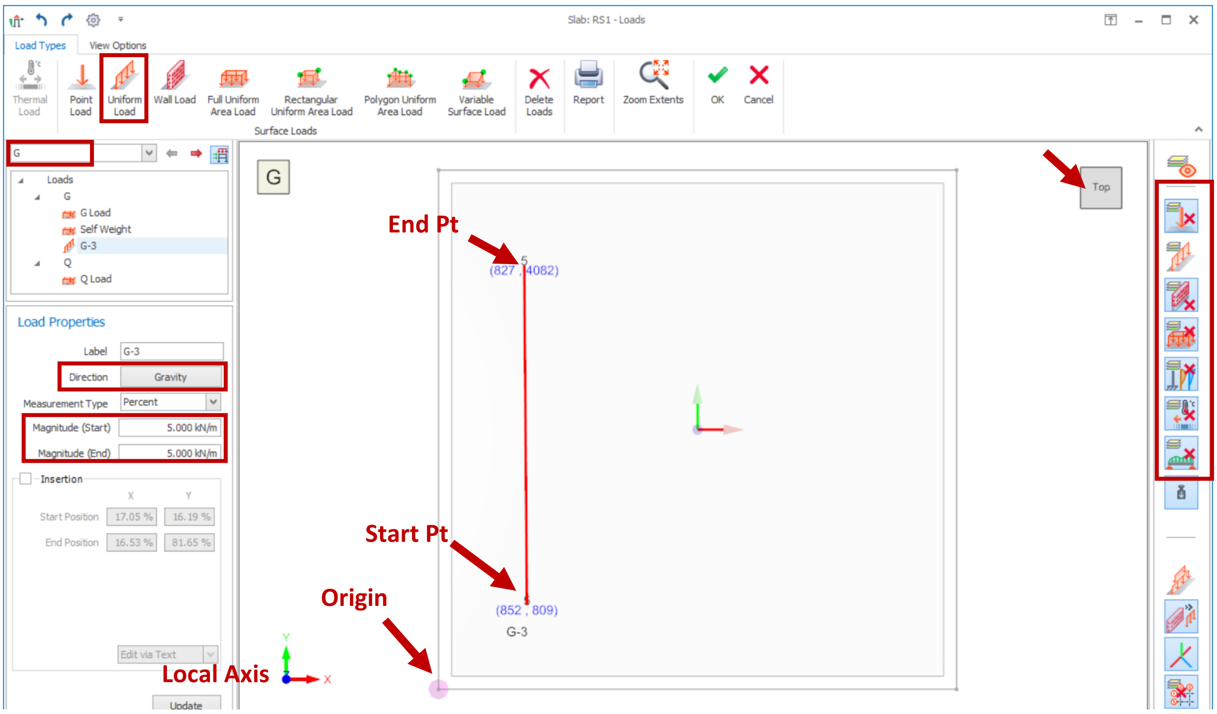
Defining Slab Load via Load Editor

You will learn how to insert slab line load.

➢ *Double Click on the **Storey 4** in the structure tree (to make Storey 4 active).*

➢ *Select the bottom left-most slab, **RS1** > Click **Edit Loads***

A slab load Editor will appear. It is similar to beam load editor but with additional surface loads type.

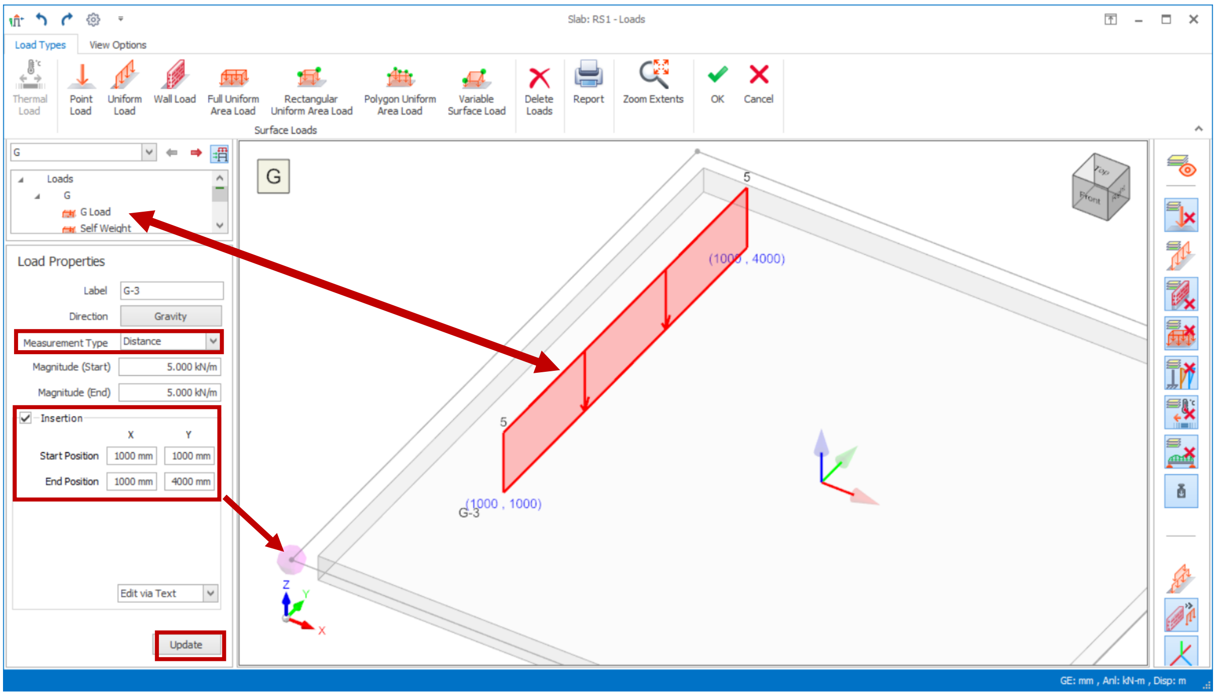


➢ *Go to **Loading** Types tab > Click **Uniform load***

Note all other load type layers will automatically switch off for clarity.

- *In the 3D orientation cube, click on **Top*** *> view direction will change to top*
- *Ensure **G load case** is selected*
- *For **Magnitude (Start)** & **(End)** > Enter **5 kN/m***
- *Pick the **Start point** on the load > Pick the **End point** (approximately vertical)*

The line load will be created with a default magnitude. To edit the load :

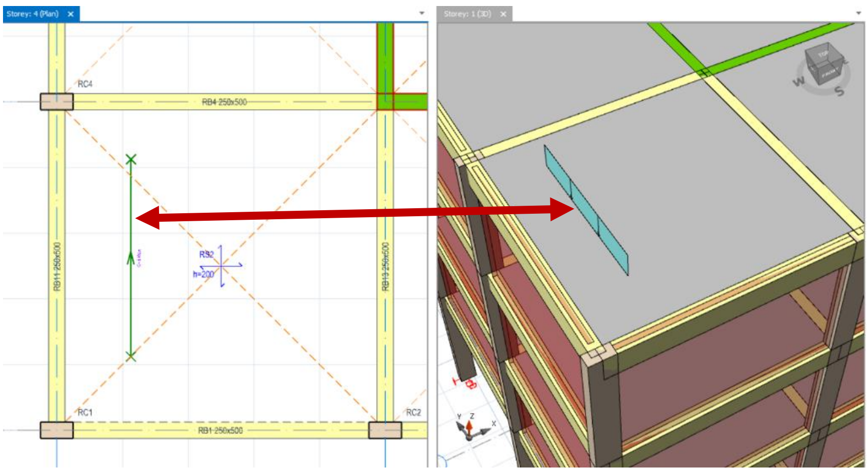


- *Press the **ESC** key to end the line load creation mode.*
- *Select the line **load name** in the load folder or the 3D diagram (rotate the view to suit).*
- *For **Measurement Type** > Choose **Distance***

The Insertion inputs will show the location & magnitude of the load in table. The X & Y inputs are with reference to the slab origin marked by a big dot.

- ***Change** the values as shown above **>** Click **Update >** **OK** to exit the dialog*

The line load will also be shown in the model plan & 3D view.



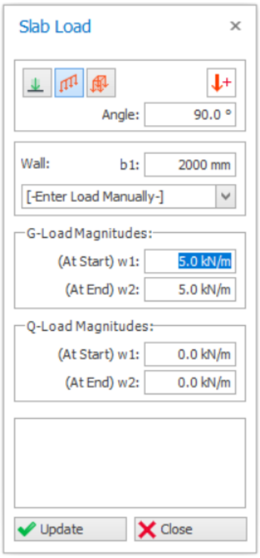
To edit the slab load, you can select the load in plan or 3D view, Right-click > **Properties**.

Defining Slab Load (via plan view)

Slab loads can also be defined on the plan view of modelling.

➢ *Go to **Loading** tab > Click the **Point, Line, and Area Loads** icon*

The slab load properties will appear as shown below.



➢ *Select the **Line Load** icon*

➢ *Define **Angle = 90 degree**, length of line load, **b1 = 2000 mm** and Load values at start and end, **w1 = w2 = 5 kN/m.***

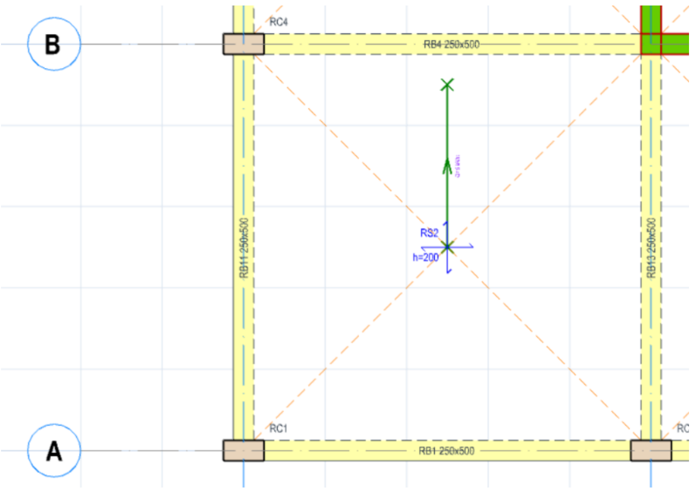
➢ *Pick the **intersection of axes A/1** (reference point)*

➢ *Press **F2** to define the relative coordinates. Enter “**2500,2500**” which means **ΔX = 2500 mm** & **ΔY = 2500 mm** as shown below*

Alternatively, you click on 2 points to define start & end of the line load (without specifying Angle & b1)

You can insert a partition wall load by clicking on the dropdown menu under “**Wall**” and select from the preset wall types library

➢ *Press **ENTER** & the slab line load will be created.*



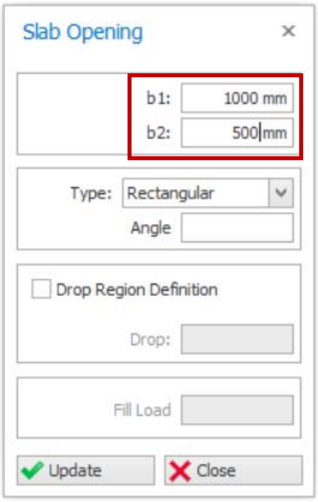
➢ ***Delete** this line load by selecting it > press the **DELETE** key*

You may use the external reference drawing to display the architectural drawing at the back of your model as a guide for creation of members or slab load.

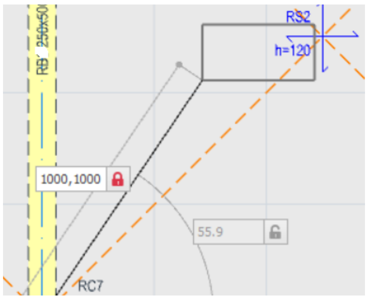
Slab Opening Creation

You will learn how to define the coordinates and size of the Slab Opening.

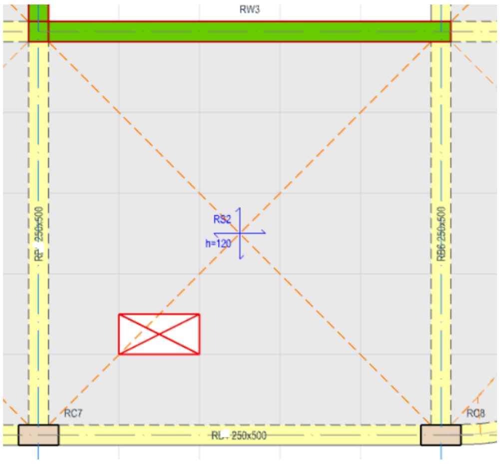
- ➢ *In ST04, zoom to slab between GL A-B/2-3.*
- ➢ *Click the **Slab Opening/Drop** icon*  *in the **Modelling** tab, and the slab opening properties dialog box will appear as the figure below.*



  - ➢ *Define the slab opening horizontal length, **b1 = 1000 mm** & vertical width, **b2 = 500 mm.***
  - ➢ *Pick the **intersection of axes A/2** as the reference point of insertion*
  - ➢ *Press **F2** to define the relative coordinates*
  - ➢ *Enter **1000,1000** which means **ΔX = 1000** mm & **ΔY=1000** mm in the textbox as shown below*



- ➢ *Press **ENTER** & the slab opening will be created*

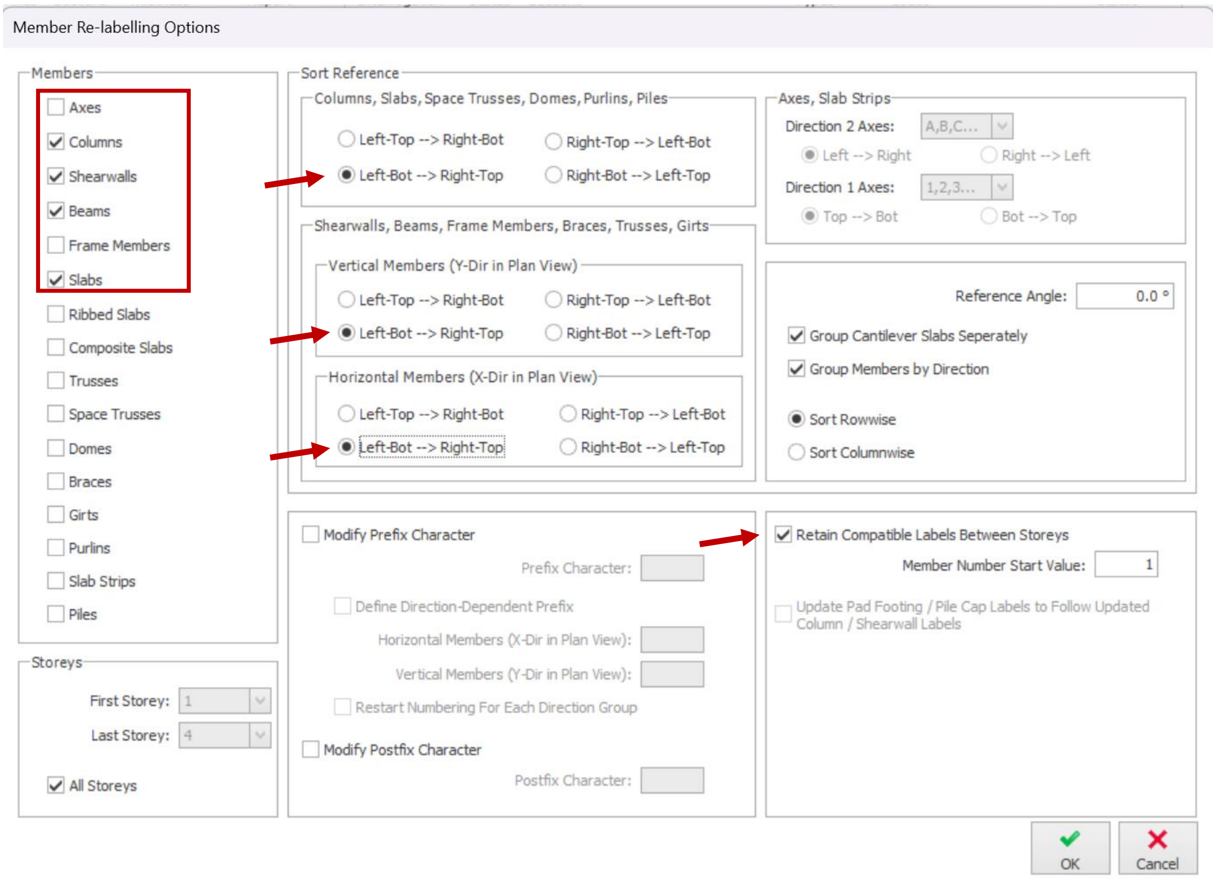


- ➢ *Click **Close** in the Slab Opening Properties dialog box*

Member Re-labelling for Entire Building

The members are labelled automatically in the sequence they are created.

➢ *To re-label the members systematically, go to* ***Review Tab*** *>* ***Re-label Members***



➢ *Select all* ***Members*** *type*

➢ *Change all Sort Reference to* ***Let-Bot --> Right-Top.***

This means that the left-bottommost member will start with number 1.

➢ *Tick Retain* ***Compatible Labels Between Storeys****.*

This setting forces the members at the same position to have the same label across all storeys (e.g., 1C1, 2C1, 3C1 will be at the exact location).

➢ *Click* ***OK****.*

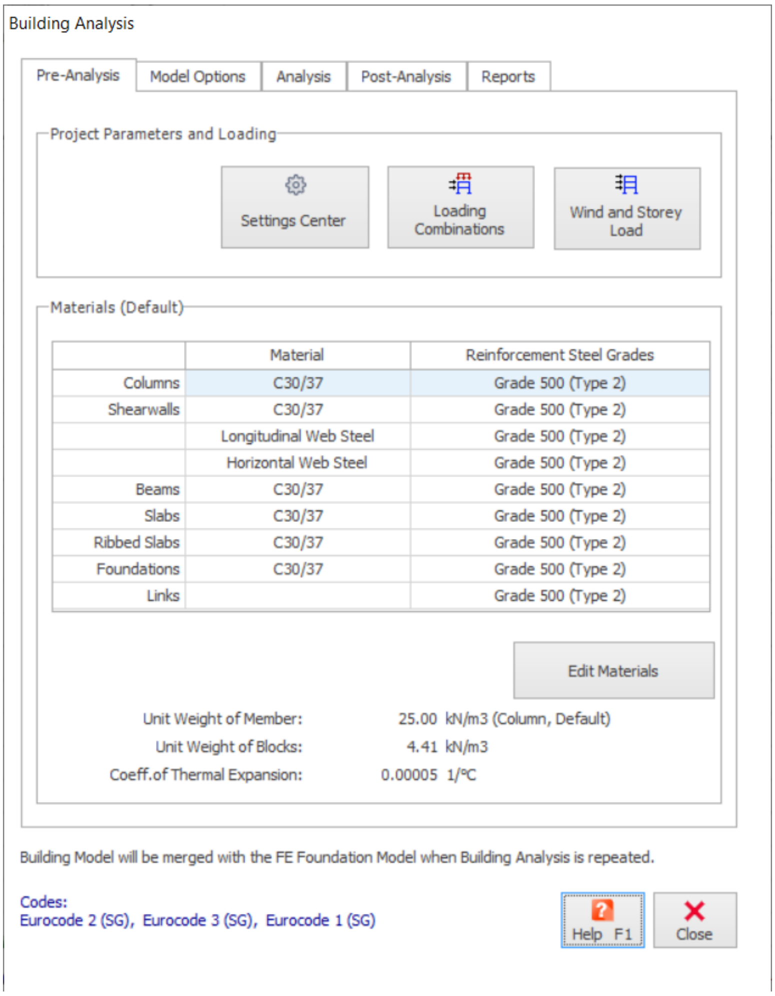
The members will now be re-labelled systematically with label 1, starting with the left bottom-most member.

It is common practice to have vertical member (column & wall) retain the same labels between storeys, whereas this may not be preferred for horizontal member (beam & slab). To achieve this, you can run the re-labelling twice for different member types with **Retain Compatible Labels** checked & then unchecked.

The model is now completed, and we are ready to run the analysis.

Building Analysis

➢ *Go to the* ***Analysis*** *tab →click* ***Building Analysis*** *→* ***Pre-Analysis*** *tab*



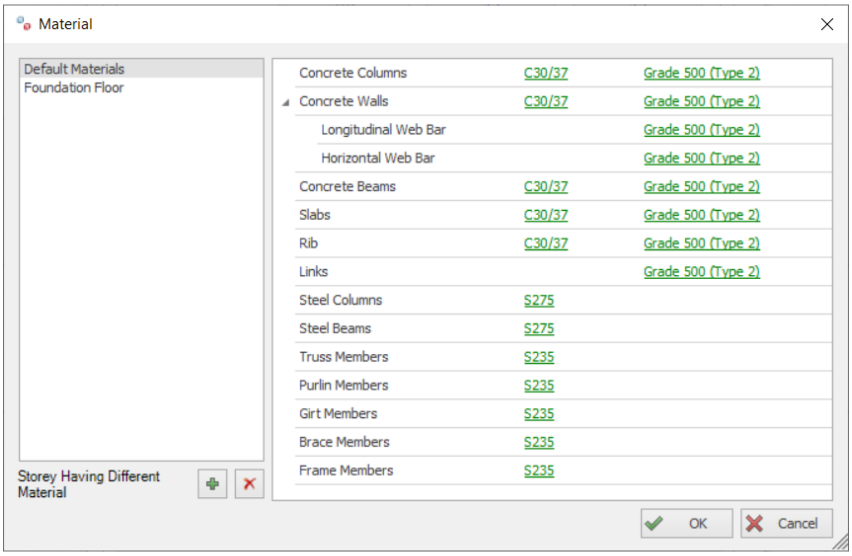
The Pre-Analysis setting is where the main assumptions of the analysis can be defined :

- Settings Center: Access settings center to review & modify analysis parameters such as the design code and notional horizontal load.
- Load Combinations: generate, add or modify any load case or load combination sets.
- Wind and Storey Load: automatically generate wind loads, review & input any lateral load. Automatically generated lateral loads such as the notional horizontal loads will only be calculated and shown after the analysis.
- Edit Materials: review, add, or change the concrete and steel material of the building.

Materials

We will now define the materials for this project via Building analysis> **Pre-Analysis** dialog.

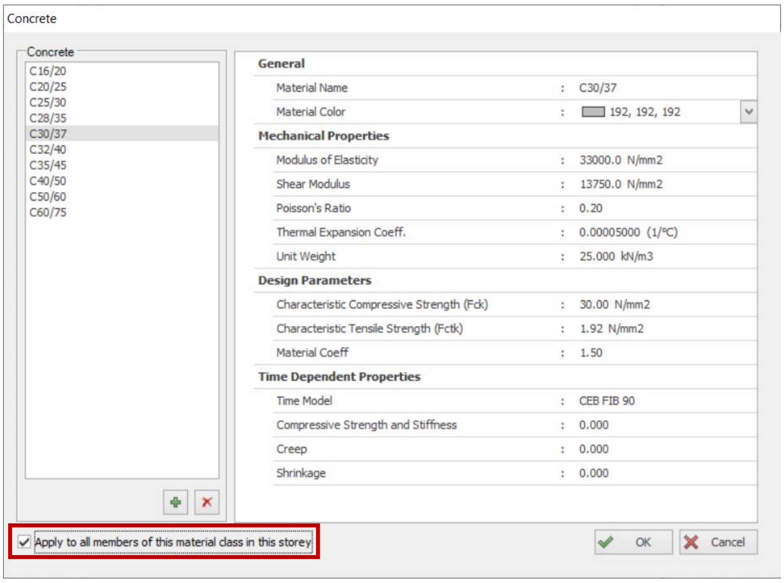
➢ *Pick **Edit Materials***



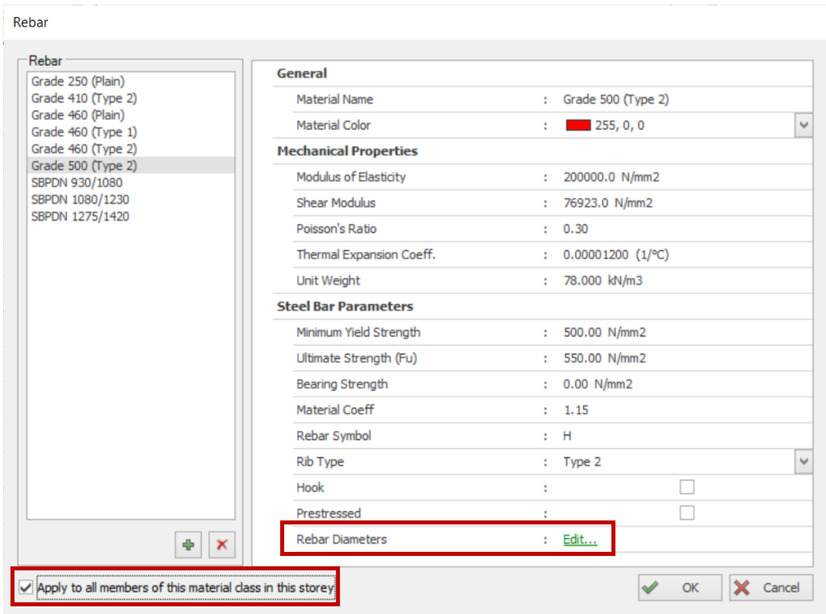
The materials and reinforcement steel grades are shown for all member types. ***Default Material*** is used for super-structure floors. A separate ***Foundation Floor*** category is automatically created for foundation members only.

If a different material is required for a particular storey, a separate material set can be added by picking the icon. Steel reinforcement grades and diameters are also defined here.

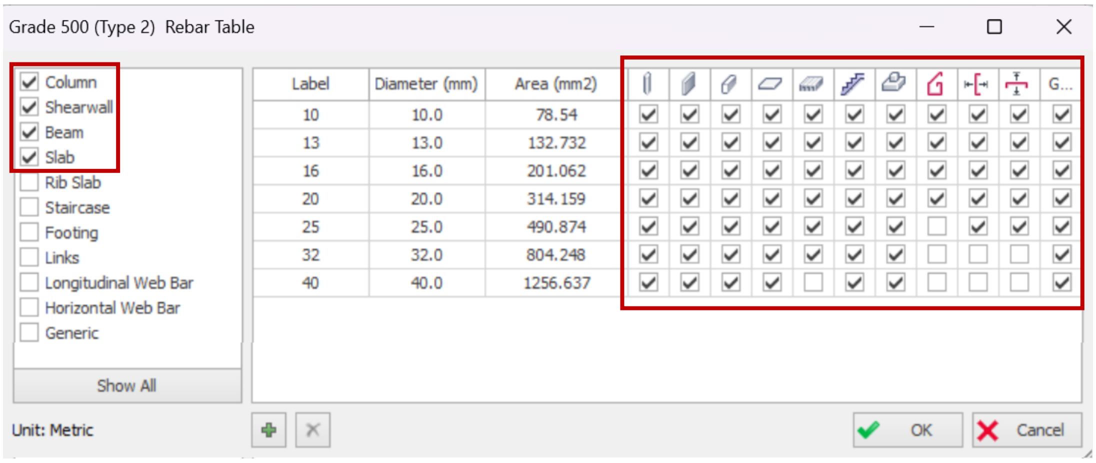
➢ *Pick **Column Concrete Grade** → select **C30/37** → **Apply to all members in this storey** → **OK.***



➢ *Pick **Steel Grade** and ensure that **Grade 500 (Type 2)** is selected and applied to all member types.*



- *Click "**Edit…**" next to **Rebar Diameter &** the rebar table will appear*



- *In left hand **Filter** pane, tick **Column, Wall, Beam & Slab** to hide diameters of other members*
- *Tick the **diameters of bars** to be used in the design of those members as shown above. **Generic** is for all other elements, at least 1 rebar must be ticked.*
- *Click **OK** to close Rebar Table > Click **OK** to close Rebar dialog > **OK** to close Materials dialog.*

Load Combinations

We have already generated the cases and load combinations previously, as shown below.

ULS_RC
ULS_Steel
SLS_RC
SLS_Steel

ID	Label	LL Red	VOM	T...	Load Combinations
1	1.35G+1.5Q	☑	☑	↓	1.35G + 1.5Q
2	1.35G+1.5Qp11	☑	☑	↓	1.35G + 1.5Qp11
3	1.35G+1.5Qp21	☑	☑	↓	1.35G + 1.5Qp21
4	1.35G+1.5Qp31	☑	☑	↓	1.35G + 1.5Qp31
5	1.35G+1.5Qp41	☑	☑	↓	1.35G + 1.5Qp41
6	1.35G+1.5Qp51	☑	☑	↓	1.35G + 1.5Qp51
8	1.35G+1.5Q+1.35NGx+1.5NQx	☑	☐	→	1.35G + 1.5Q + 1.35NGx + 1.5NQx
9	1.35G+1.5Q-1.35NGx-1.5NQx	☑	☐	→	1.35G + 1.5Q - 1.35NGx - 1.5NQx
10	1.35G+1.5Q+1.35NGy+1.5NQy	☑	☐	→	1.35G + 1.5Q + 1.35NGy + 1.5NQy
11	1.35G+1.5Q-1.35NGy-1.5NQy	☑	☐	→	1.35G + 1.5Q - 1.35NGy - 1.5NQy
12	1.35G+1.05Q+1.35NGx+1.05NQx	☑	☐	→	1.35G + 1.05Q + 1.35NGx + 1.05NQx
13	1.35G+1.05Q-1.35NGx-1.05NQx	☑	☐	→	1.35G + 1.05Q - 1.35NGx - 1.05NQx
14	1.35G+1.05Q+1.35NGy+1.05NQy	☑	☐	→	1.35G + 1.05Q + 1.35NGy + 1.05NQy
15	1.35G+1.05Q-1.35NGy-1.05NQy	☑	☐	→	1.35G + 1.05Q - 1.35NGy - 1.05NQy

- *Review & click **OK** to close the dialog.*

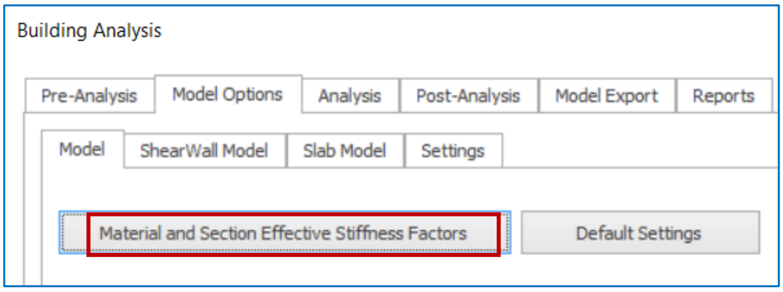
Wind and Storey Loads

Allow user to review & input lateral loads such as notional horizontal & wind loads. We will skip this for now.

Wind load cases and combinations must be generated first. For more guidance, kindly refer to Prota Help Centre : [Wind and Storey Loads]

Building Analysis Model Options

- *Go to the **Model Options** → **Model** → **Material & Section Effective Stiffness Factors** and review the stiffness factors as shown below.*



Effective Material and Section Stiffness Factors

	Elasticity Modulus	Axial Area		Bending Stiffness	Shear Area	Torsional Constant
Shearwalls (Shell)	1.00	1.00	In Plane	1.00	1.00	1.00
	1.00	1.00	Out of Plane	1.00	1.00	1.00
Shearwalls (Frame)	1.00	1.00	Major	1.00	1.00	1.00
	1.00	1.00	Minor	1.00	1.00	1.00
Basement Walls	1.00	1.00	In Plane	1.00	1.00	1.00
	1.00	1.00	Out of Plane	1.00	1.00	1.00
Slabs	1.00	1.00	In Plane	1.00	1.00	1.00
	1.00	1.00	Out of Plane	1.00	1.00	1.00
Columns	1.00	1.00		1.00	1.00	1.00
Beams	1.00	1.00		1.00	1.00	0.01
Coupling Beams	1.00	1.00		1.00	1.00	1.00

You can modify the elasticity modulus, section areas, moment of inertias and torsional constants of the member groups to be used in the analysis model. For example, you can enter 0.05 to reduce the moment of inertia values by 95% to reduce the lateral stiffnesses of the columns.

Note: In order to apply these factors, building analysis must be repeated. These factors will be applicable only for load cases for which cracked section properties are used.

The stiffnesses table will only be applied for load cases with **Used Cracked Sections** checked in the respective **Load Case Editor**, except for seismic load cases where cracked sections are always assumed.

All factors can be modified, except those that are greyed out, which must be auto-calculated.

Building analysis must be repeated each time the effective stiffness factors are changed.

To prevent large torsional forces in primary beams due to secondary beams, the global **Torsional Stiffness Factor** is by default set to **0.01** (1%) in the Stiffnesses settings.

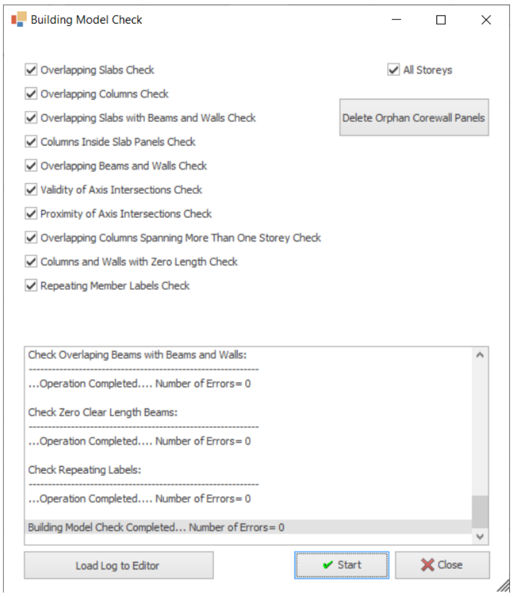
Stiffnesses of individual members can be changed by selecting a member → **Right-click** → **Edit Section/Material** → **Properties** tab.

Running Analysis

➢ *Go to the **Analysis tab.***

Before running the analysis, it's always recommended to check the model's validity.

➢ *Click **Building Model Check** & pick **All Storeys** & click **Start.***



The building model check will pick up the most obvious modelling errors as indicated.

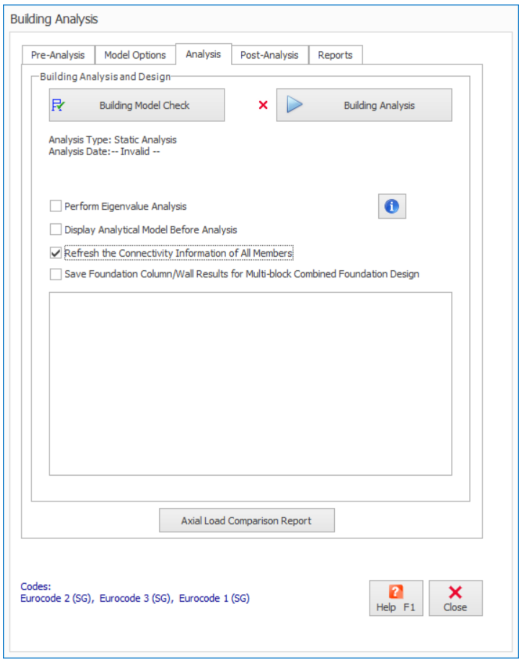
The warnings / errors are mostly self-explanatory. The exact member(s) affected will be stated.

You should review any warnings / errors and correct the model before proceeding to perform analysis.

Click "**Load Log to Editor**" to open the list of warnings/errors in a separate screen in Notepad for ease of reference.

➢ *Click **Close** to exit the check.*

The building analysis can now be performed.



➢ *Click **Building Analysis***

Eigenvalues analysis can be done to obtain the natural frequencies and mode shapes of the structure.

The **Batch Design Options** dialog will appear.

- It allows you to perform the design of column/wall & beams automatically after the analysis.
- We suggest these to be unchecked for 1st few analyses, as result should be verified before proceeding to design.
- These options can be useful for subsequent design, example minor amendment in model.

➢ *Pick **Building Analysis** to analyse & design the model*

The analysis will also check for instability and large deformations and there will warning messages if any are found.

The **Analysis Summary Report** will appear at the end of the analysis, summarizing the essential results.

Axial Load Comparison Report

An important check on the validity of the analysis is the **Axial Load Comparison Report**. This report sums all the dead and live load applied at each storey and displays the axial forces in the columns and shear walls. These values need to agree within a tolerance limit of 5%. If they do not, the reason for the discrepancy should be investigated.

➢ *Select Axial **Load Comparison Report** (in the Analysis tab)*

TOTAL LOADS (Based On Slabs Loads):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	93.80	295.31	297.39	1193.77	0.00	1880.28
3 (+9.00m)	93.80	295.31	759.36	1035.96	0.00	2184.43
2 (+6.00m)	93.80	295.31	759.36	1035.96	0.00	2184.43
1 (+3.00m)	93.80	295.31	759.36	1035.96	0.00	2184.43
Total						8433.56

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	0.00	0.00	0.00	155.47	0.00	155.47
3 (+9.00m)	0.00	0.00	0.00	546.86	0.00	546.86
2 (+6.00m)	0.00	0.00	0.00	546.86	0.00	546.86
1 (+3.00m)	0.00	0.00	0.00	546.86	0.00	546.86
Total						1796.05

TOTAL LOADS (Decomposed to Beams):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	93.80	509.53	1277.83	0.00	0.00	1881.16
3 (+9.00m)	93.80	402.42	1689.09	0.00	0.00	2185.31
2 (+6.00m)	93.80	402.42	1689.09	0.00	0.00	2185.31
1 (+3.00m)	93.80	402.42	1689.09	0.00	0.00	2185.31
Total						8437.10

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	0.00	28.13	127.34	0.00	0.00	155.47
3 (+9.00m)	0.00	56.25	490.61	0.00	0.00	546.86
2 (+6.00m)	0.00	56.25	490.61	0.00	0.00	546.86
1 (+3.00m)	0.00	56.25	490.61	0.00	0.00	546.86
Total						1796.05

BUILDING ANALYSIS COLUMN AND WALL AXIAL LOADS:

Storey	G	Delta G	Q	Delta Q
4 (+12.00m)	1881.16	1881.16	155.47	155.47
3 (+9.00m)	4066.48	2185.31	702.33	546.86
2 (+6.00m)	6251.79	2185.31	1249.19	546.86
1 (+3.00m)	8437.10	2185.31	1796.05	546.86
Total		8437.10		1796.05

Total Base Reactions: G = 8437.1 kN Q = 1796.05 kN

Table 1: **TOTAL LOADS (Based on Slab)** is the sum of dead and live loads of all members with the slab load not yet decomposed or calculated on the beam. You can take this as the input weight of the structure.

Table 2: TOTAL LOADS (Decomposition to Beams) consider the decomposition of the slab load onto the beams based on either yield-line or FE Decomposition. The beam load now includes the slab loads (and hence zero values are shown under the slab column).

Table 3: BUILDING ANALYSIS COLUMNS AND WALL AXIAL LOADS sums up the actual column and wall axial loads after building analysis.

Firstly, check **Table 1**. Total values are similar to **Table 2**. This ensures that beams accurately capture all slab loads, i.e., no lost slab loads.

Then verify **Table 2** total values are similar to **Table 3**. This ensures that all the super-structure weight is entirely captured by the columns and walls down to the foundation.

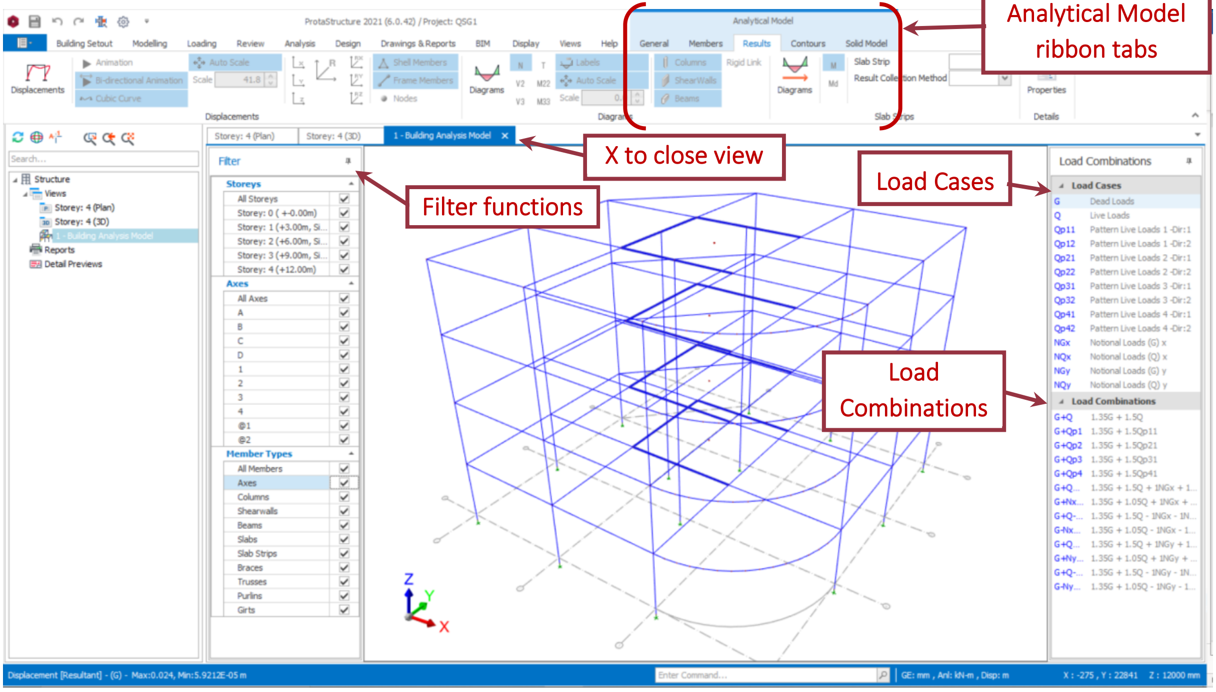
Analytical Model

The **Analytical Model** enables you to review the results of the analysis graphically. It is essential to check & verify the analytical model to ensure the model the correctly set up & results are reasonable.

- *Go to the **Post-Analysis** tab of the Building Analysis dialog.*
- *Click **Display Analytical Model***

Alternatively, go to **Analysis** tab > click **Analytical Model** icon.

A new **Analytical Model tab** set will appear with the **Analytical Model** view.

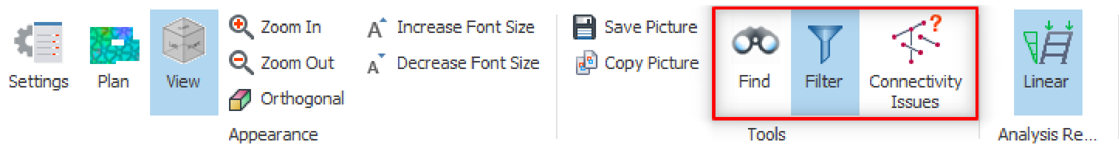


The analytical wireframe is shown in **blue**. Load cases and combinations can be selected on the right-side pane.

If too much information is displayed, the screen can appear cluttered. You can create a more meaningful display view by using the various filter buttons and the view settings.

- *In the **General** tab, click on the **Filters** command (if it's not activated)*

  The filter options will appear on the left-hand side of the view. There are options to filter the display by Storeys, Axes, and Member Type.

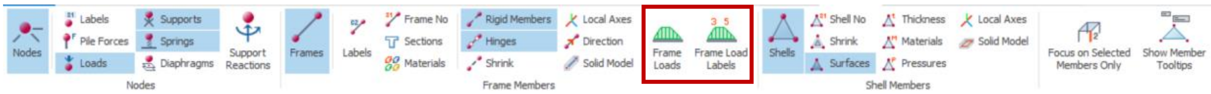


The **Find** command allows you to find a particular Node, Frame, or Shell number.

Connectivity Issues lists all the frames with unsupported nodes and highlights them when selected.

➢ *Click the **Members** tab.*

This is where you can show node labels, rigid diaphragms, element labels, and beam loads.



➢ *Click **Frame Loads** & **Frame Load Labels** icons to show loads calculated or decomposed on the beams (& then turn it off).*

This may include all the slab loads, brick wall loads, beam self-weight & other manual input loads.

➢ *Click **Diaphragms** → **Select All** will show the rigid diaphragm formed by the slabs (& then turn it off).*

The **grey** lines join nodes constrained by the diaphragm to the centre of the diaphragm mass.

➢ *Click on the **Results** tab.*

➢ *Click **Displacement** → The **red** lines show the structure's displacement (deflection).*

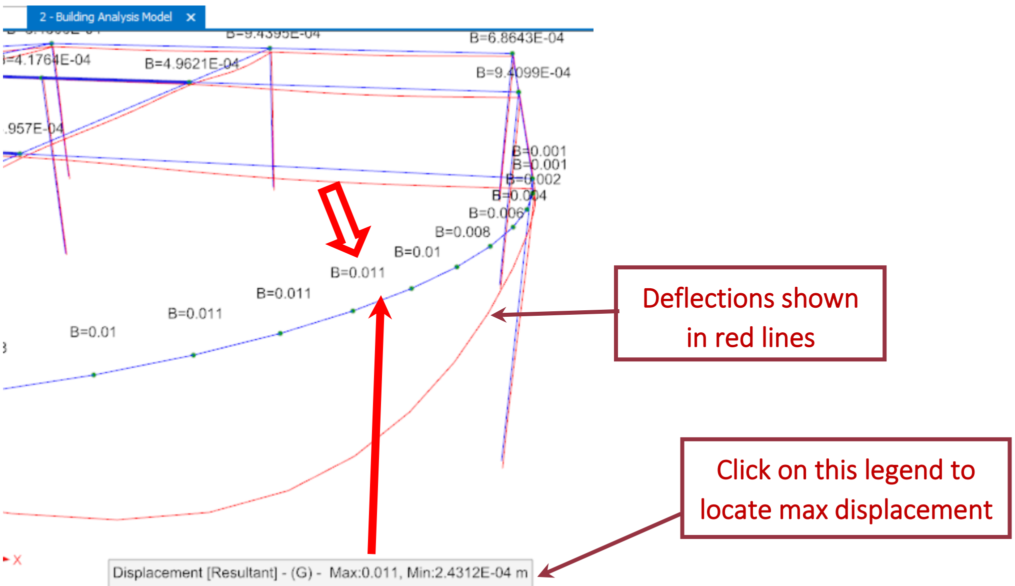
By default, **Auto Scale** is activated. You can deactivate this and then type in your **Scale**.

> Displacement Unit can be changed in the **Settings Centre** > [Units and Format]()

➢ *Click on the various **directional displacement** values by selecting **x, y, z** & **R** (Resultant). **Rx, Ry, Rz** is the rotation of joints in radians.*

The **Maximum** and **Minimum** displacement is shown at the bottom of the screen.

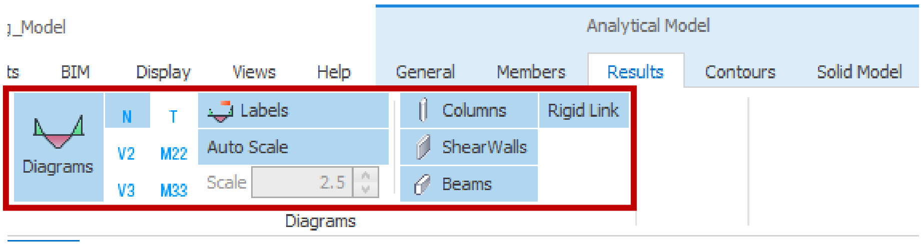
➢ *Click on this **legend** → an arrow will appear to pinpoint the node of maximum displacement.*

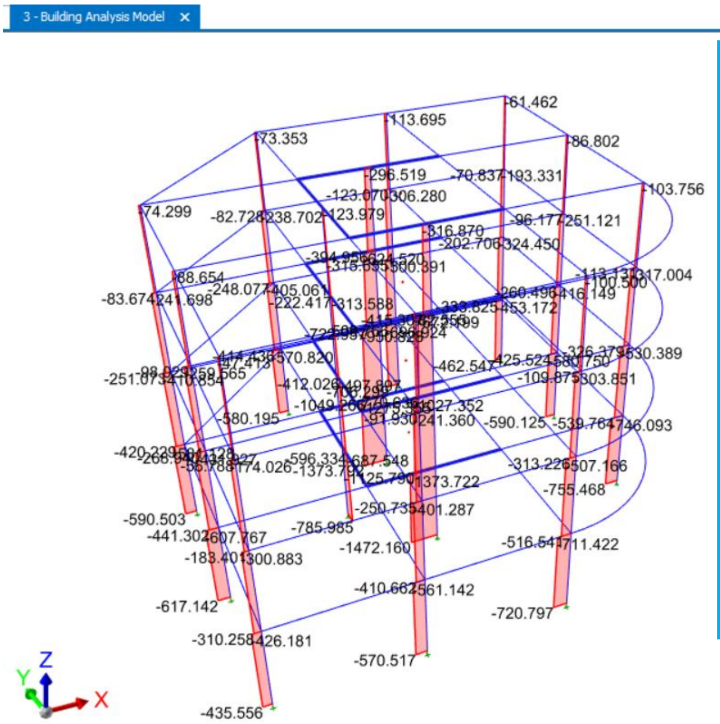


➢ *Click **Animation** to visualize the deformation of the structure.*

➢ *Switch off the **Displacements** display and click on the **Diagrams** button.*

➢ *Click on the Axial force **N** icon to display the **Axial Force diagram** (for G Load Case).*





Diagrams Legend

N - Axial forces

V2 – Major shear force of beams and walls. Shear of column along dir 1

V3 – Minor shear force of beams and walls. Shear of column along dir 2

M2 – Minor moment of beams and walls. Moment of column along dir 1

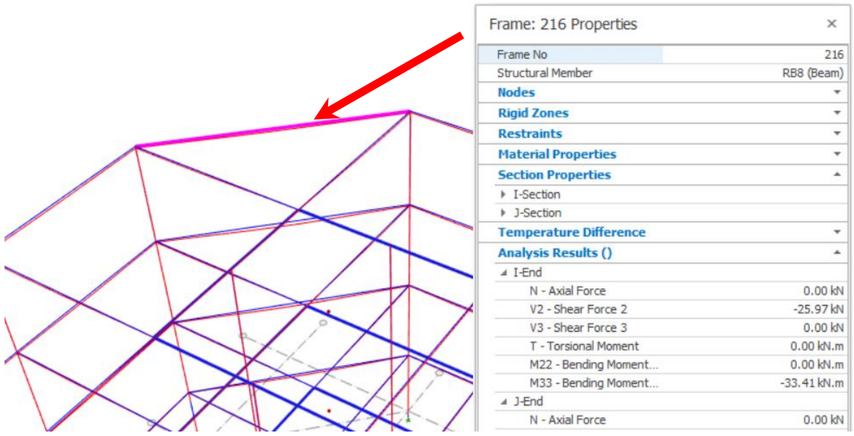
M3 – Major moment of beams and walls. Moment of column along dir 2

T - Torsion

- *Experiment with the various effects of the diagrams.*

You can change the diagram scale by de-activating **Auto Scale** & typing the **desired Scale**.
You on/off the diagrams on Column, Shear walls, Beams & Rigid link by clicking on the respective icons.

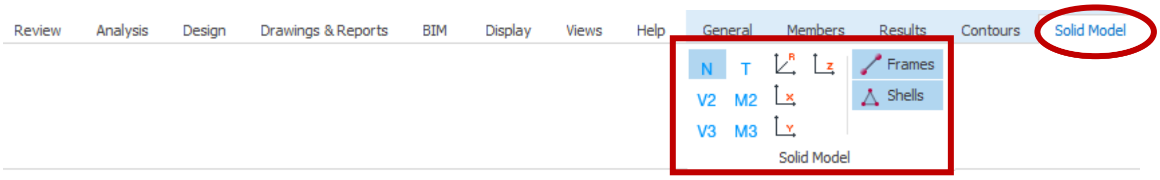
- *Click* ***Properties*** *icon* Properties *&* ***select*** *a frame member will show the detailed member properties, including material, section properties & summary of analysis forces at both members ends.*

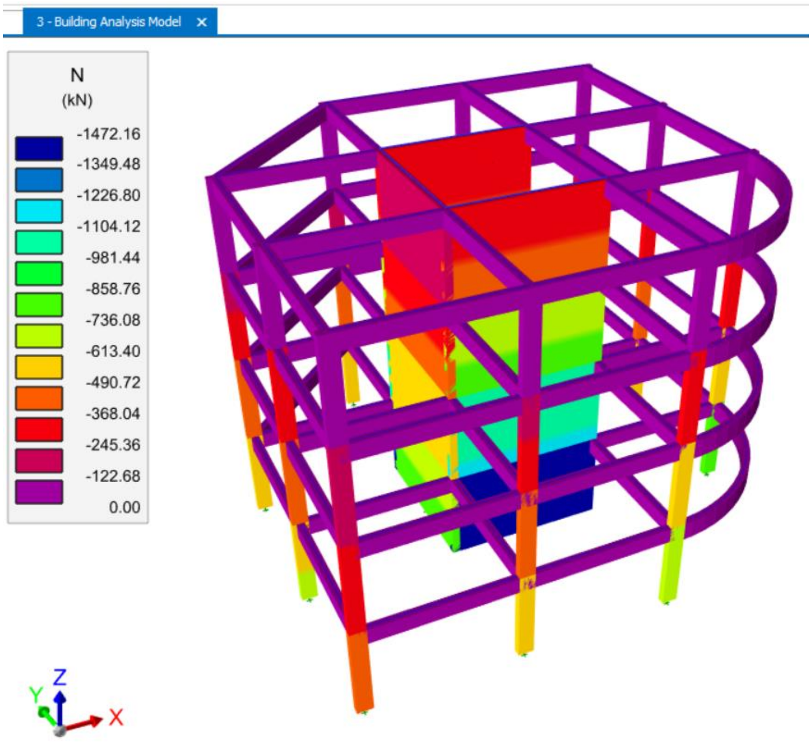


Contours tab will show contour results for shell elements which will only be visible if finite element shell walls is assumed, or floor slabs are selected to be meshed in Building Analysis.

➢ *Go to the **Solid Model** tab.*

This allows the various effects such as **Axial Load** & effects to be color-coded on the physical model.





➢ ***Close** the Analytical Model view by clicking on the "**x**" sign next to the view name.*

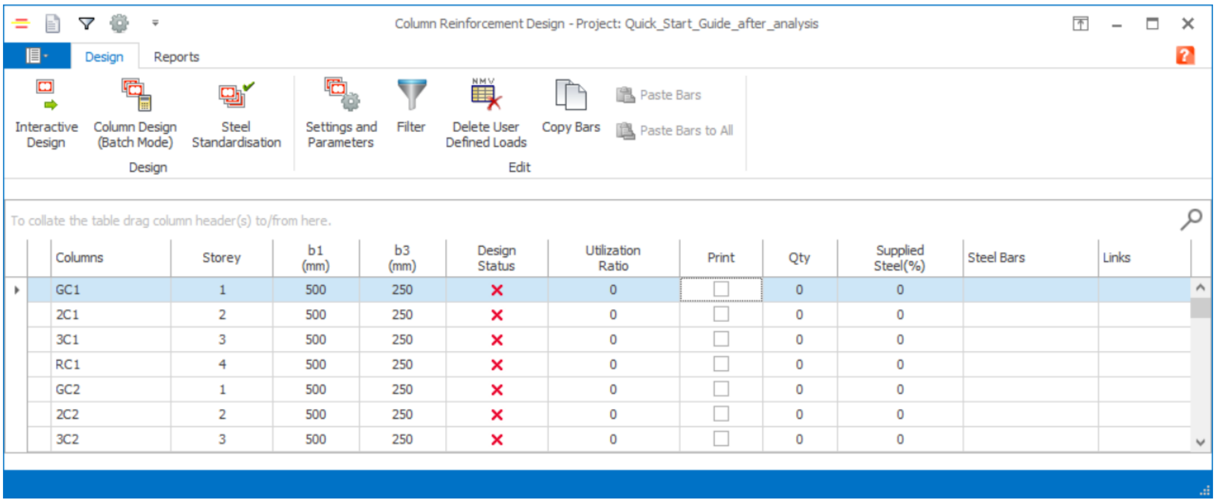
➢ *Alternatively, you can click to switch to **plan view*** Storey: 4 (Plan) *and exit the Analytical Model interface.*

You must interrogate the Analytical Model and check its validity as that is the actual analytical model from which the design forces will be based.

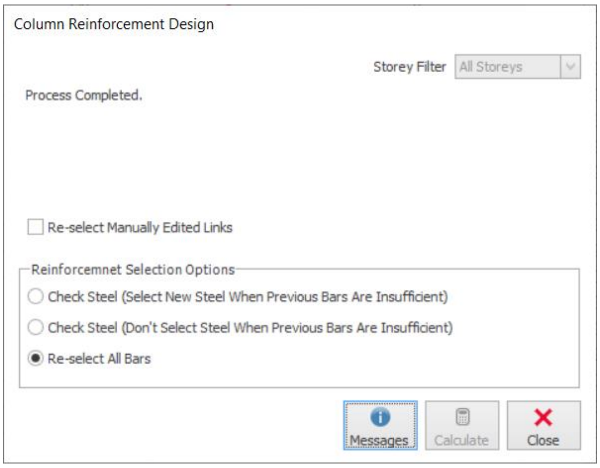
Column & Wall Design

➢ *Navigate to the **Column Section Design** under the **Design** tab.*

The columns and walls are not designed as indicated by a red cross under **Design Status** header.



➢ *Click **Column Design (Batch Mode)*** 



Three design options are :

Check Steel (Select New Steel When Previous Bars are Insufficient) : Assuming there is previous design, only change rebars that is insufficient. Bars that are sufficient will not be changed at all.

Check Steel (Don't Select Steel When Previous Bars Are Insufficient) : Freeze all existing rebars, insufficient column will be marked as failed.

Re-select All Bars : Discard all previous rebars and redesign all the beams with current beam design settings.

➢ *Pick **Re-select All Bars*** to design all columns.

Once batch design is completed, the "Message" icon will appear.

➢ *Click on **Message** icon to review the design summary.*

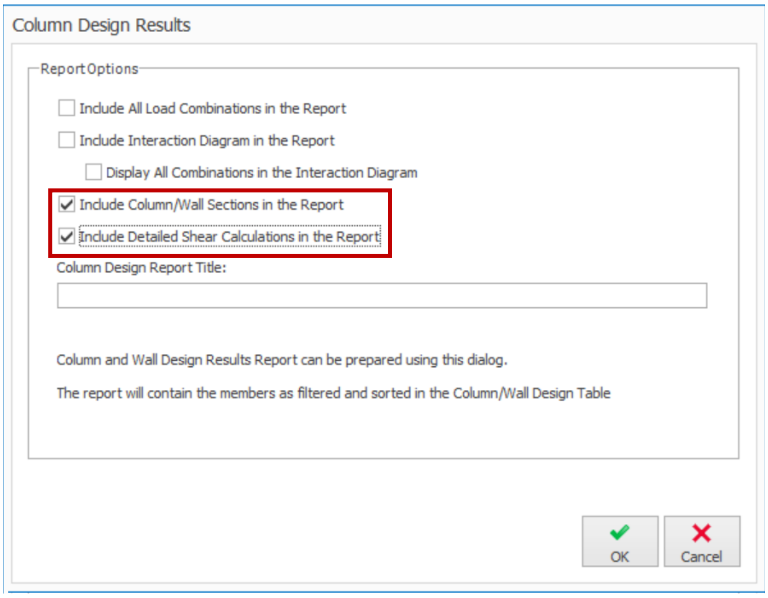
If there are any failures, the exact beam & reason will be stated.

➢ *Close the notepad & batch design dialog.*

The design menu will be updated with pass status, UR = Utilisation Ratio and other steel bar information.

➢ *Go to the **Reports** tab → Choose **Design Report*** 

A tick in the Print column marks the columns included in the report. Columns can be added or removed from the report by checking or unchecking the **Print** checkbox. Furthermore, icons **Mark All Columns** and **Remove Print Marks** can be applied to all columns.

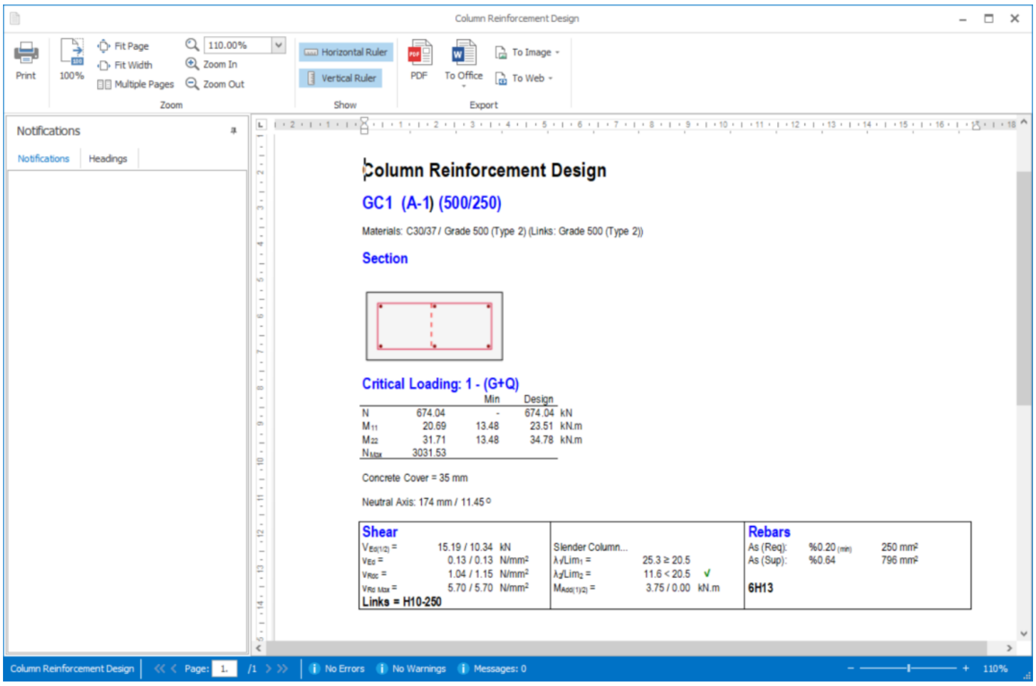


- *Check "Include Column/Wall Sections in the Report" to draw the column section detail.*
- *Check "Include Detailed Shear Calculations in the Report".*

You can also give the report a title.

- *Pick **OK** to generate the report.*

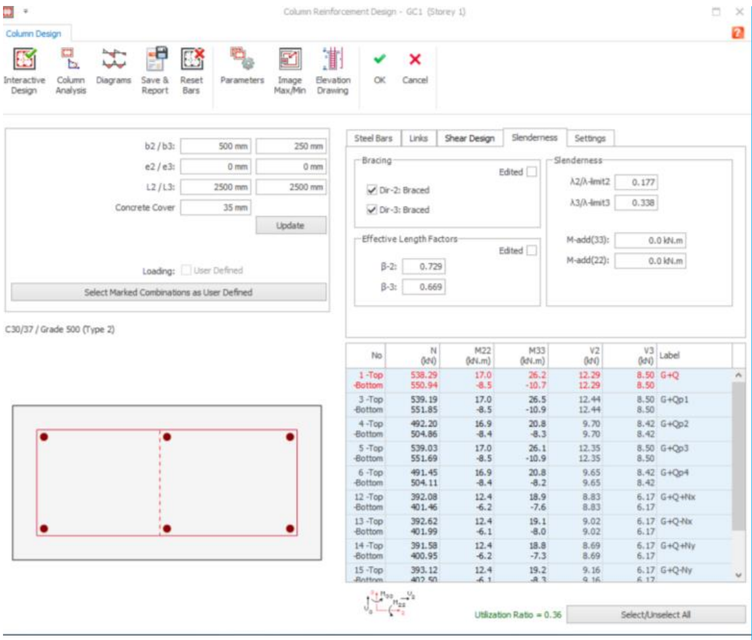
  All reports can be exported in PDF, MS Word, image, or webpage (HTML) format.



- ***Exit** & this report will automatically be saved & made available for compilation in **Report Manager***

The Interactive Column Design allows you to examine & alter individual column designs in more detail.

- ***Double click** on **GC1** in the list of columns in the Column Design screen.*



Section Dimension shows the parameters of the column such as size, clear height & concrete cover

Steel Bars shows the number and layout of the rebar. These can be changed.

Load Combination table list down all the load combination. The most critical (governing) is highlighted in red.

Required & *Supplied As* are calculated from the critical load combination.

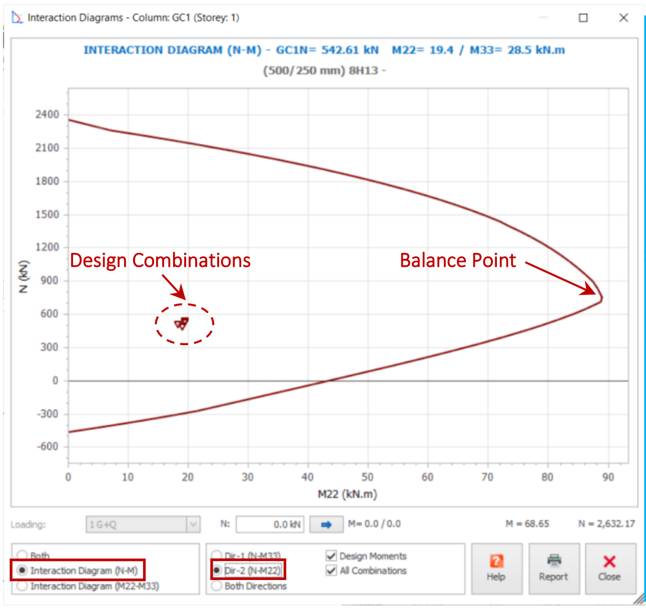
Slenderness tab shows the bracing assumption, calculated effective lengths & additional moments.

If there are changes in the assumption of this column, you can click the **Interactive Design** button to re-perform the reinforcement design.

You can change the dimensions (b1, b2) above & click **Update**. This is handy if a column fails due to insufficient size & you want to try out if a bigger size works. However, due to the change in column size, the weight & stiffness of the model will change. Hence, you should re-run the analysis and re-check the column design to ensure it is sufficient finally.

➢ *Click **Column Analysis** to inspect the Interaction Diagrams (capacity curves).*

The **Interaction Diagrams (N-M)** show the possible combination of axial force and moment that cause failure to a given column dimension and rebars. This "capacity curve" indicates the maximum axial capacity of a reinforced column is dependent on the co-incident or applied moment (vice versa). The axial capacity of the column decreases as the applied moment increases until the 'Balance point', i.e., the maximum theoretical moment capacity. After this point, moment capacity decreases with decreasing applied axial load.



➢ *Select **Interaction Diagram (N-M)***

For rectangular columns, there will be 2 capacity curves in direction 1 and direction 2 respectively.

The governing design will be the direction with load combinations closer to the curve; for this column in will be along dir-2, the shorter dimension & hence weaker capacity.

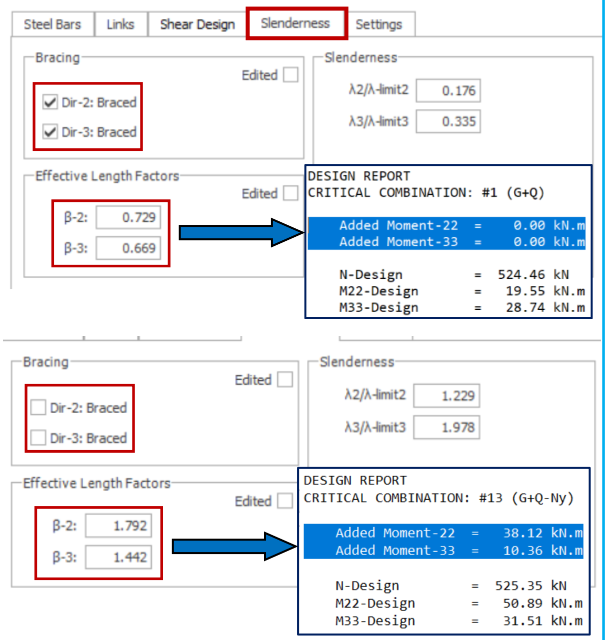
➢ *Select **Dir-2 (N-M22)***

To verify that this column passes with the rebar selected, all the design combination points must be within the curve, especially if you have manually changed the rebars in Rebar Table.

The **Interaction Diagrams (M22-33) plots** the component directional moment capacity of the column at a given applied axial load.

➢ *Close the Interaction Diagram and go to the **Slenderness** tab.*

The **Slenderness** tab shows the bracing conditions of the column/wall, effective length factors & additional moment (M-add) due to slenderness.



For this column, it is braced in both directions, the calculated effective lengths **β** are less than 1 & hence M-add is zero or minimal.

You can change the bracing condition manually.

➢ *Untick both **Dir-1-Braced** & **Dir-2 Braced***

➢ *Click **Interactive Design** to refresh the design*

➢ *Click **Design Report** for detail report*

➢ *Go back to **Slenderness** tab to review*

Notice the unbraced condition of the column now results in effective lengths > 1. As a result the additional slenderness moment, '**Added Moment**' is calculated and auto-added to the base design moment to become the final design moment.

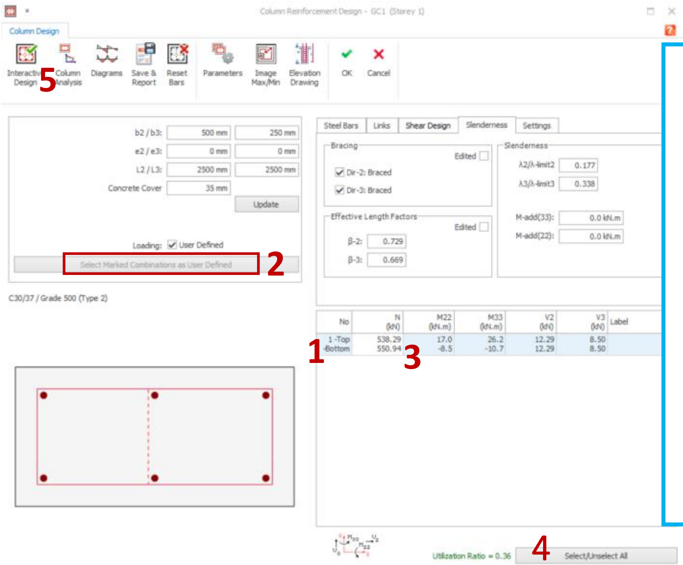
If you want to manually save either the Bracing or β factors, ensure to tick "**Edited**". This means the design module will not auto-calculate these values anymore.

➢ *Click **Diagrams** to view the shear force, bending moment, or deflection diagrams.*

➢ *Click **Elevation Drawings** to view the reinforcement of the column in elevation.*

Manually Specifying Column Design Forces (for info)

The column design forces are automatically linked and updated to the latest analysis run. You can over-write the column design forces manually.



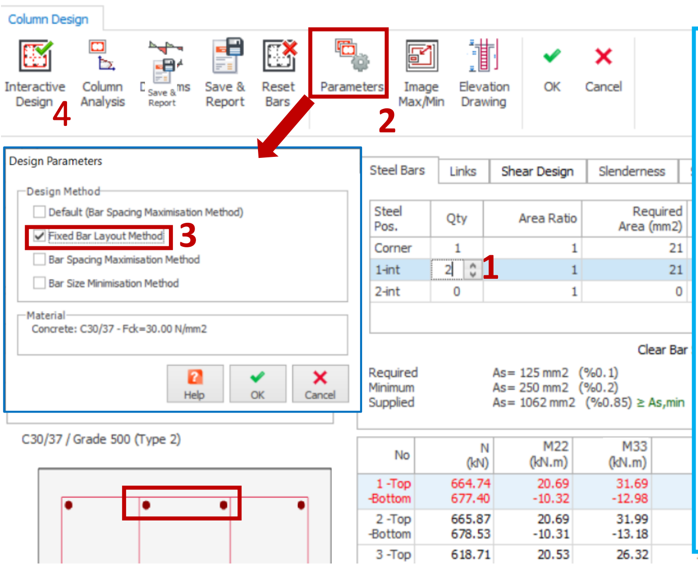
To manually enter column design forces :

1. *Select the first row of column design forces*
   To select more rows, hold down **CTRL** key
2. *Pick **Select Marked Combination as User Defined***
3. *Click on figures under "**N**" axial load & enter figures for top and bottom, e.g. 1000,1000*
4. *Pick **Select / Unselect All** to select all rows*
5. *Click **Interactive Design** to refresh the design*
6. *Untick **User Defined Loads** to auto-link back to analysis forces.*

Manually Change Column Reinforcement (for info)

The column design module will automatically design the column to pass status. However, there are situations where you might want to specify the reinforcement manually & check if the column passes.

To manually fix the layout & number of rebars & let column design auto-select rebar size:



1. *Input "**2**" for **1-int** under Qty column*
   Notice diagram now shows 2 nos. internal horizontal bars on both sides
2. *Pick **Parameters***
3. *Pick **Fixed Bar Layout Method** → OK*
   This option ensures that the number of bars will not change during re-design
4. *Click **Interactive Design** to re-design*
   The column will be re-designed to 8 number of bars as specified. If this reinforcement is insufficient, the program will auto-increase the bar size, say from H13 to H16.

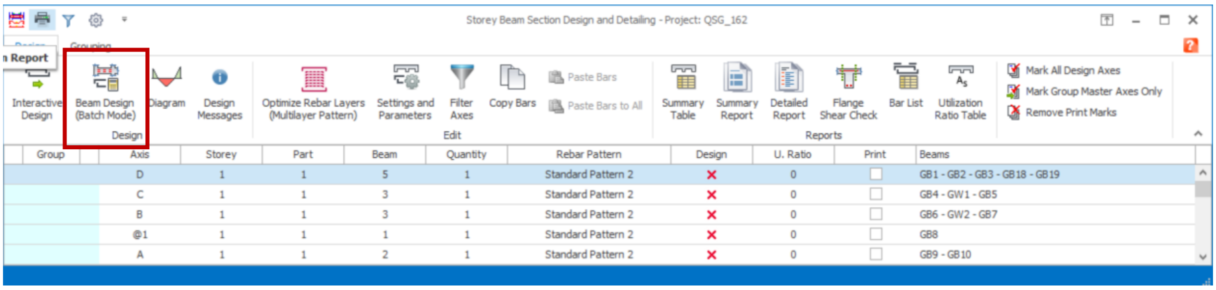
You can also manually change both the number of bars and rebar sizes (i.e., both Qty & Diameter). **Do not** click on **Interactive Design,** which will change the rebar sizes. You must then manually check the following to ensure the column passes:

- *Check **As Supplied** is more than **As required** and **Min Steel %***
- *Click **Column Analysis** → Check all design points are within the Interaction Diagrams N-M22 & N-M33*
- *Click **Cancel** to exit the Interactive Column Design dialog and **Close** the window.*

Beam Design

➢ *Go to **Design** tab → pick **Storey Beams***

Since we have selected beam reinforcement design as part of Building Analysis, all the beams are already designed as indicated by green ticks.



➢ Click ***Beam Design (Batch Mode)***

The same 3 options of design which were presented for columns are shown.

➢ Pick ***Re-select All Bars*** to auto design all beams at one go with the current design settings.

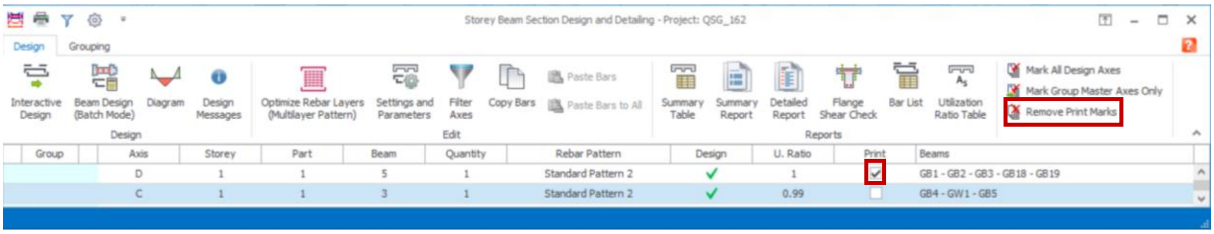
➢ Pick **OK** when *prompted if you want to use the current rebar pattern set in beam design settings.*

➢ After completion of batch beam design, close the batch beam design dialog.

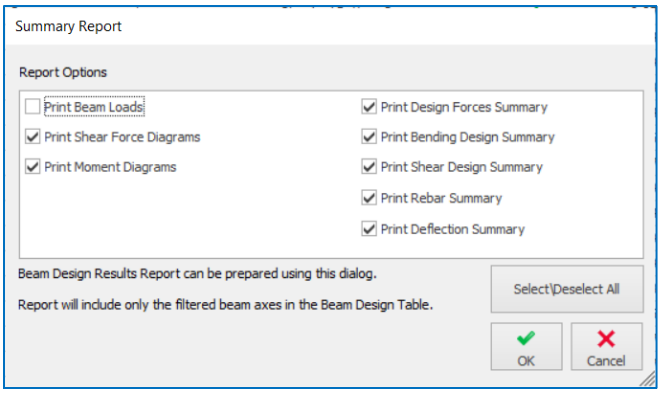
The beam design dialog will be updated showing pass status for all beams. A tick in the **Print** column marks the beams included in the report. Beams can be added or removed from the report by checking or unchecking the **Print** checkbox.

➢ *Choose **Remove Print Marks** to exclude all beams in the report.*

➢ ***Check/tick** the checkbox under **Print** for the first-row beam axis **D** only.*



➢ *Choose **Summary Report***



The Beam Design Report is in rich text format.

Print Beam Loads – List all the loads on the beams in table format. This will significantly increase the size of the report.

Print Shear / Moment Diagrams - Print the envelope shear / moment diagrams in pictures.

Print Rebar and Axis Image – Prints reinforcement design as shown in interactive Reinforcement Data interface. This will significantly increase the size of the report.

- *Uncheck **Print Beam Loads** (for faster generation of report)*
- *Pick **OK** to generate the report.*

Beam Reinforcement Design Rev: 1	Prota Software Asia (21838) Calc. By: Checked By:

Beam Reinforcement Design

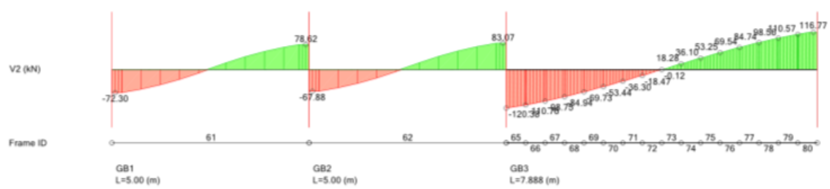
Eurocode 2 (SG) **(EN 1992) Design of Concrete Structures (Singapore Annex)**

Notation	Definitions
f_{ck}	Compressive (Cylinder) Strength of Concrete
f_{yk}	Yield Strength of Longitudinal Reinforcement
f_{ywk}	Yield Strength of Transverse Reinforcement
E_s	Modulus of Elasticity of Reinforcement
b_w	Web Width
h	Depth
d	Effective Depth
$c_{c\text{-}compression}$	Compression Reinforcement Clear Cover
$c_{c\text{-}tension}$	Tension Reinforcement Clear Cover
$c_{compression}$	Compression Reinforcement Cover
$c_{tension}$	Tension Reinforcement Cover
A_g	Gross Cross Section Area

Axis : A

Analysis Results

Shear Diagram (Envelope)



- ***Exit** & this report will automatically be saved & made available for compilation in **Report Manager***
- *Click **Detailed Report** → uncheck all options for faster generation → OK*

The report recreated has more detail containing clause by clause check, example below.

Rectangular Section Details (Top Section)

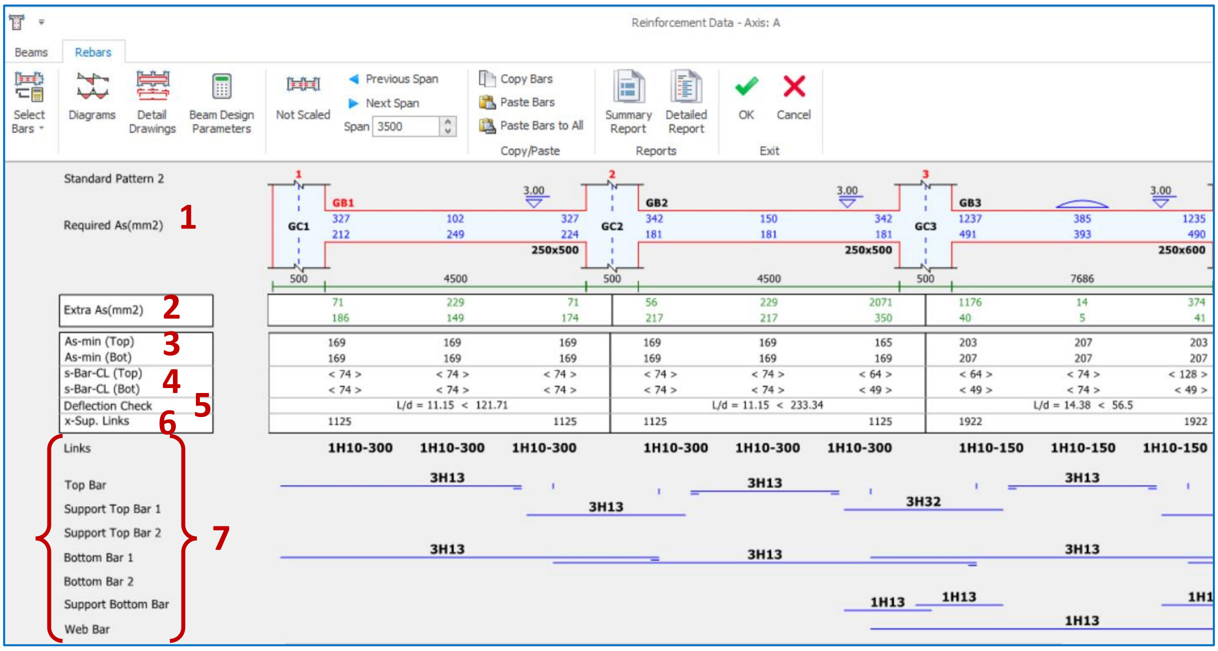
b_w (mm)	h (mm)	$A_g = b_w h$ (m^2)	d (mm)	$c_{c\text{-}compression}$ (mm)	$c_{c\text{-}tension}$ (mm)	$c_{compression}$ (mm)	$c_{tension}$ (mm)
250	500	0.1250	449	35	35	52	52

Bending Design (Top Section)

Design Bending Moment	M_{Ed} = 61.5 kN.m (Negative Moment) → ***Tension steel will be placed at the top of the section.***
Limiting Value of the Ratio of the Neutral Axis Depth to the Effective Depth	$(x/d)_{Limit} = (\delta - k_1)/k_2 = 0.5$ (**Eurocode 2 (SG) 5.5.(4)**) → $f_{ck} \le 50.00$ N/mm²
Maximum Distance from Extreme Compression Fiber to Neutral Axis	$x_{max} = (x/d)_{Limit}\, d$ = 202 mm
Maximum Allowable Depth to the Neutral Axis	$a_{max} = \lambda\, x_{max}$ = 161 mm
Equivalent Rectangular Compression Block Depth	$a = d - \sqrt{(d^2 - 2 M_{Ed} / (\eta f_{cd} b_e))}$ = 34 mm
	$a < a_{max}$ → ***Tension-controlled section. Compression reinforcement is not required.***
Required Area of Tension Reinforcement	$A_s = M_{Ed} / (f_{yd} (d - a/2))$ = 327 mm²
Minimum Area of Tension Reinforcement	$A_{s,min}$ = Max [$0.26 f_{ctm} / f_{yk}$, 0.0013] $b_w d$ = 169 mm² (**Eurocode 2 (SG) 9.1N**)
Maximum Area of Tension Reinforcement	$A_{s,max} = 0.04 A_g$ = 5000 mm² (**Eurocode 2 (SG) 9.2.1.1(3)**)

➢ ***Double click** on any of the beam axis in the list of beams in the Beam Design dialog.*

The Interactive Beam Design allows you to examine individual beam design in more detail.



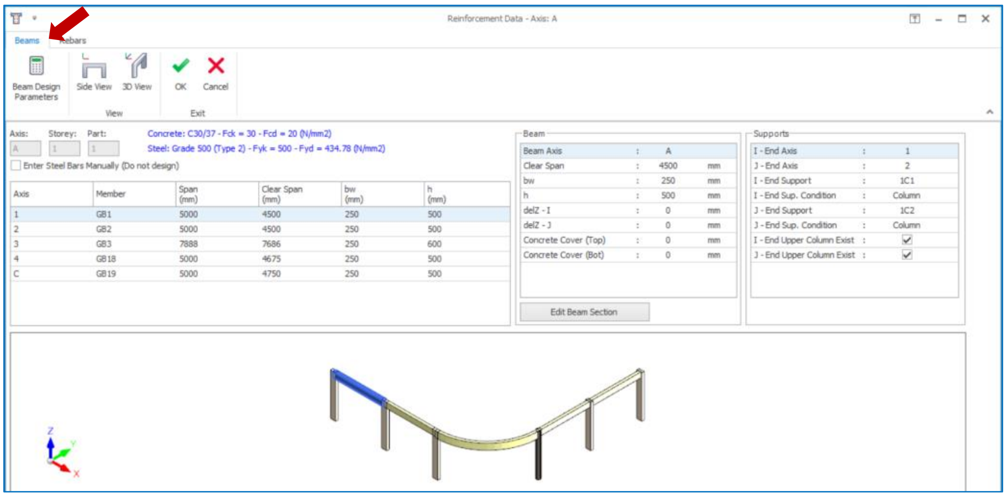
Reinforcement Data window shows the details of the beam design :

1. Required As is shown in blue as is based on design envelope of all load combination.
2. Extra As shows the additional or surplus area of steel based on selected steel.
3. As-min (Top & Bottom) is calculated minimum area of steel.
4. s-Bar-CL (Top)/(Bot) shows the spacing of the rebars at the outmost layer.
5. Deflection Check shows the actual vs allowable span/effective depth. The modification factor is automatically applied.
6. x-Sup. Links shows the distance of the support region for links
7. *Rebars* designed are shown in the various rows including shear links, main bars & web/side bars. These can be manually changed if desired.

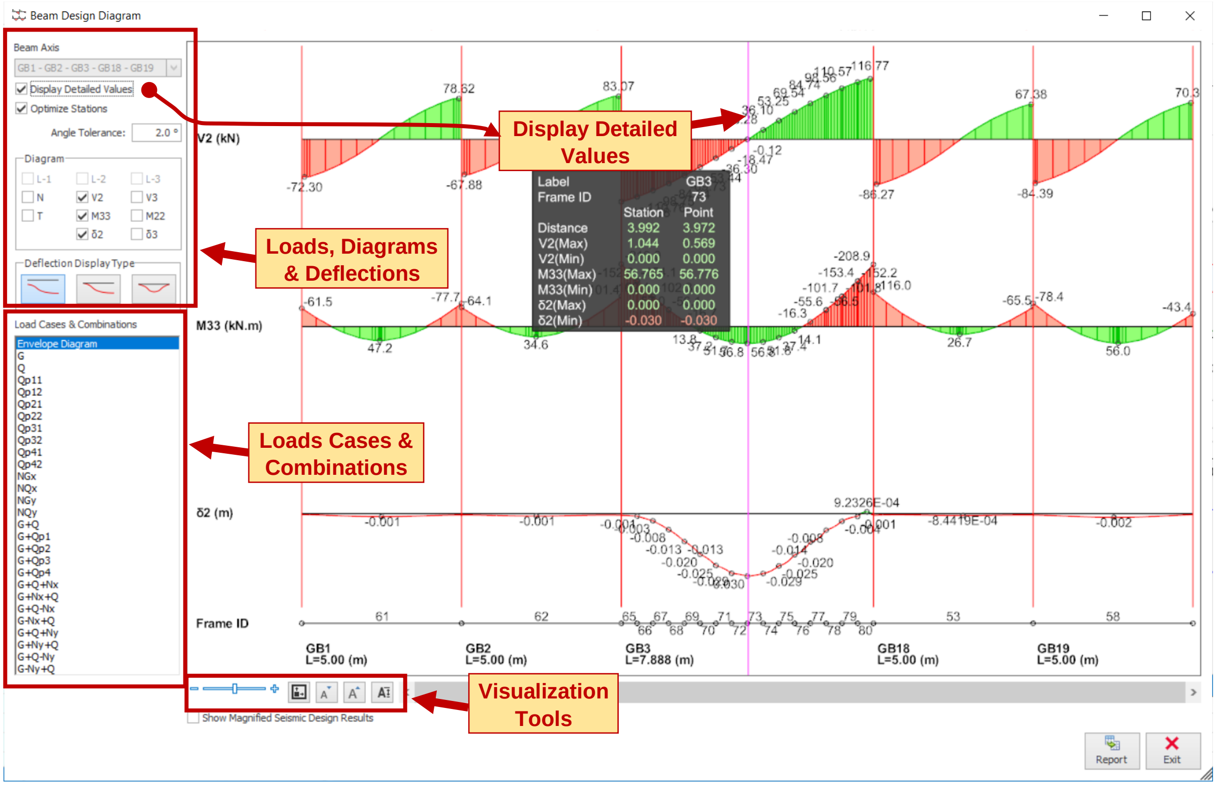
Any figures in red indicate a failure to meet the design criteria and should be investigated.

➢ *Click on the **Beams** tab at the top right corner.*

This shows the summary of the beams parameters along this design axis including a 3D view.



➢ *Go back to Rebars tab → Click on the **Diagrams** button to see the loading and design forces.*



Loads, Diagrams & Deflections

❖ Display Detailed Values:

- Checked: A tracing window will appear showing the exact values of the diagrams, e.g., shear & moment, when the mouse cursor is placed at a particular location on the member.

❖ Optimize Stations:

- Unchecked: The diagrams are displayed using the default maximum number of stations.
- Checked: The number of stations will be reduced & optimized to maintain similar accuracy.

❖ Diagrams:

- L1/ L2 / L3: Check to show slab loads decomposed & user-defined loads on beams.
- N = Axial force ; T = Torsion
- V2 = major shear ; M33 = major moment ; δ2 = major deflection
- V3 = minor shear; M22 = minor moment ; δ3 = minor deflection

❖ Deflection Display Types :

Deflections are calculated along the member span for each load case and combination. Deflections can be displayed in three different ways:

- **Absolute:** The absolute rotation and deflection values display the deflected shape.
- **Normalized:** The absolute shape is normalized to the value calculated at the first point.
- **Relative:** The deflected shape is normalized to both start and endpoints. This is particularly useful in determining the deflection relative to both ends in serviceability checks.

Visualization Tools & Report

❖ Horizontal Scale  → Increase or decrease horizontal Scale of diagrams.

❖ Default Display Scale → click to reset to the default scale.

❖ Increase / decrease font size

❖ Default Font Size → Click to rest to default font size.

❖ Report → Generate a report in tabular format with/without diagrams.

➢ ***Exit** the diagrams & **Close** the beam design dialog*

The same beam diagrams can be assessed by :

➢ *Select a beam on plan view → Right-click → **Analysis Results Diagram***

The same diagrams are used for columns & walls :

➢ *Select a column or wall on plan view → Right-click → **Analysis Results Diagram***

➢ *Alternatively, access the **Interactive Column Design** → pick '**Diagrams**'*

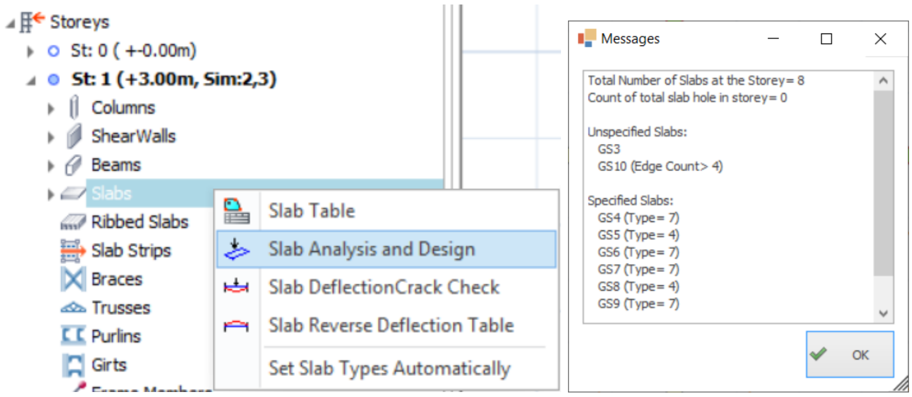
Slab Analysis & Design

For beam/slab models, slab reinforcement can be obtained by inserting slab strips in the X and Y directions. This process uses the moment coefficient method of selected code. This is independent of the general building analysis and can be carried out before or after the building analysis.

To use the moment coefficient method, it is essential to set all the **Slab Types** correctly per the provisions of selected code. This can be done automatically in batch mode by right-clicking on the slab icon in the structure tree and choosing **Set Slab Types Automatically**.

➢ *From the **Structure Tree,** double click on storey **St: 1** to return to the 1st Storey plan view.*

➢ ***Right-click** on Slab → **Set Slab Types Automatically** → Choose defaults options → **OK***



Slabs that do not have 4 edges is beyond the scope of the moment coefficient method. It will be entirely up the user to set the appropriate slab type or use alternative finite element method to analyse & design these slabs.

➢ *Click **OK** to close the Messages dialog.*

➢ *Go to the **Modelling** tab & click on the **Slab Strip***

The Slab Strip Properties will be displayed:

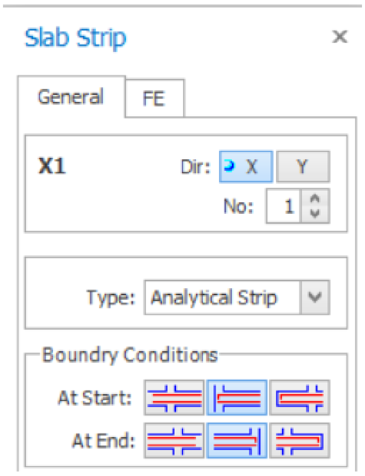
- Slab Strip **Type**: **Analytical Strip** is used for design based on the Code Method. **FE Strip** is used for innovation based on Finite Element Analysis results. **Manual strip** is used to manually input rebars without any analysis & design.

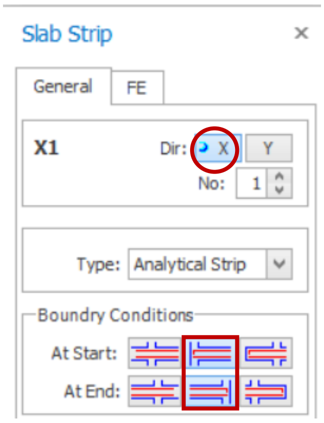
- The correct "At Start" and "At End" boundary conditions must be specified when drawing the strips. The three options are:

  Slab - The Strip starts or ends inside a slab. The bottom steel for the slab in question is not designed, but the span of the slab can be defined, and this value is used in determining the support steel extension.

  Bob - The Strip starts or ends beyond an edge beam or wall. The support steel at the edge is bent down into the beam/wall.

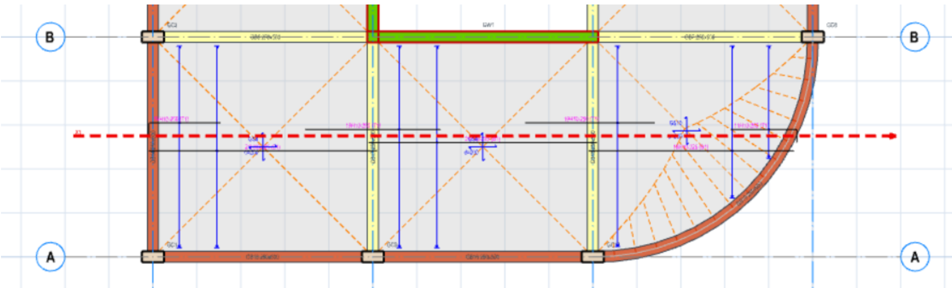
  Cantilever - The Strip starts or ends beyond a cantilever slab.



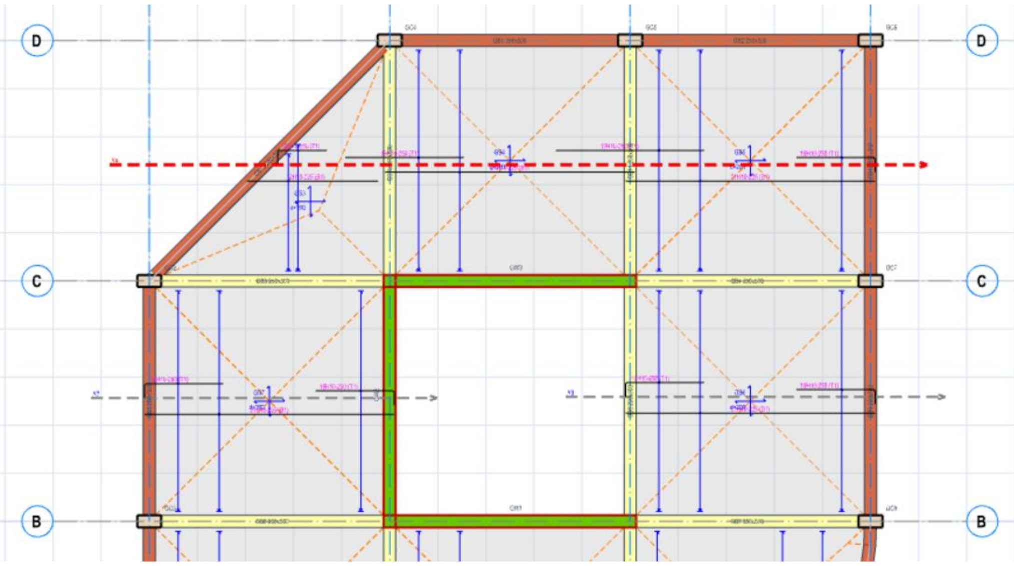


- *Set the **Dir:** to **X***
- *Set the **No:** to '**1**' so that the strip label is X1*
- *Set **Type** to **Analytical Strip***
- *Set the **At Start** condition to **Bob.***
- *Set also the **At End** condition to **Bob.***

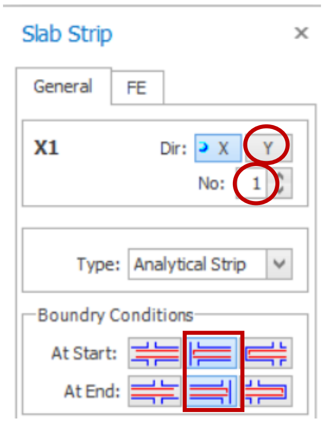
  Pick the slab strip's start point outside the floor plan, as shown below.
- ***Hold down on the CTRL key** and then click the end of the strip to the right of **Axis 4***



- *Draw another similar slab strip **X2** to obtain the design for the slabs between **Axis B-C/1-2***
- *Draw similar slab strip **X3** to obtain the design for the slabs between **Axis B-C/3-4***
- *Draw slab strip **X4** to obtain the design for the slabs between **Axis C-D/1-4***

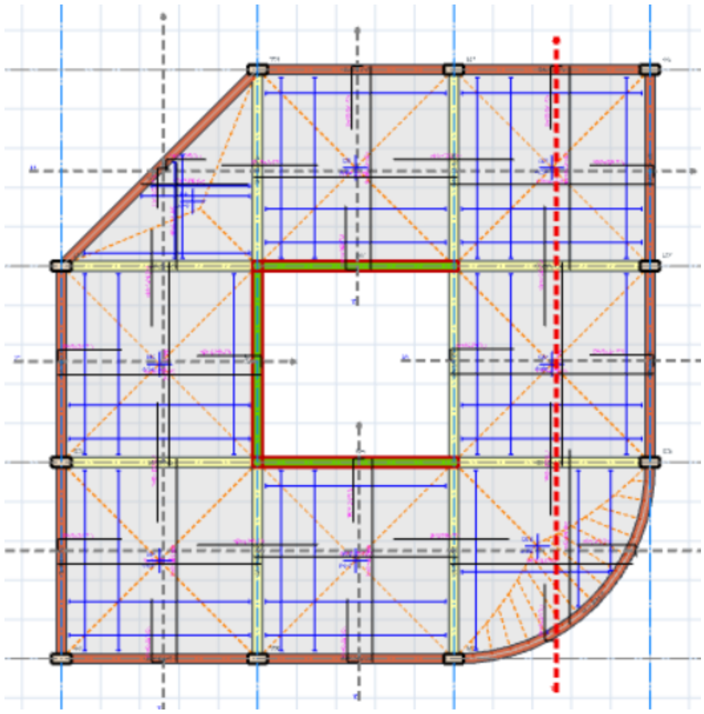


Now draw the **vertical strips** to design Y-direction rebar for the slabs.



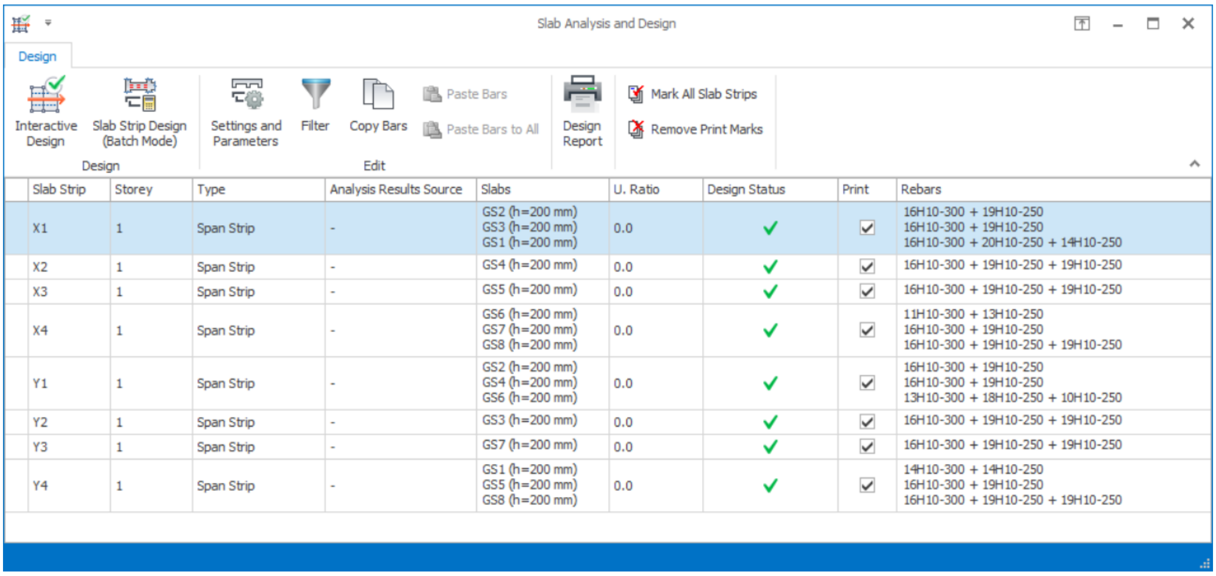
- *Change the* ***Dir*** *to* ***Y***
- *Reset the* ***No:*** *to* ***1*** *again so that the strip label is* ***Y1***
- *Keep* ***Type*** *as* ***Analytical Strip***
- *Keep the* ***At Start*** *condition as* ***Bob***
- *Also, keep the* ***At End*** *condition as* ***Bob***
- *Draw* ***4*** *nos. of the vertical strip through all of the slabs*

The final layout of strips & slab reinforcements should be as shown below.



We can now re-check the strips as a batch and create a slab analysis & design report.

- *Go to the* ***Design*** *tab → choose* ***Slab Analysis and Design***



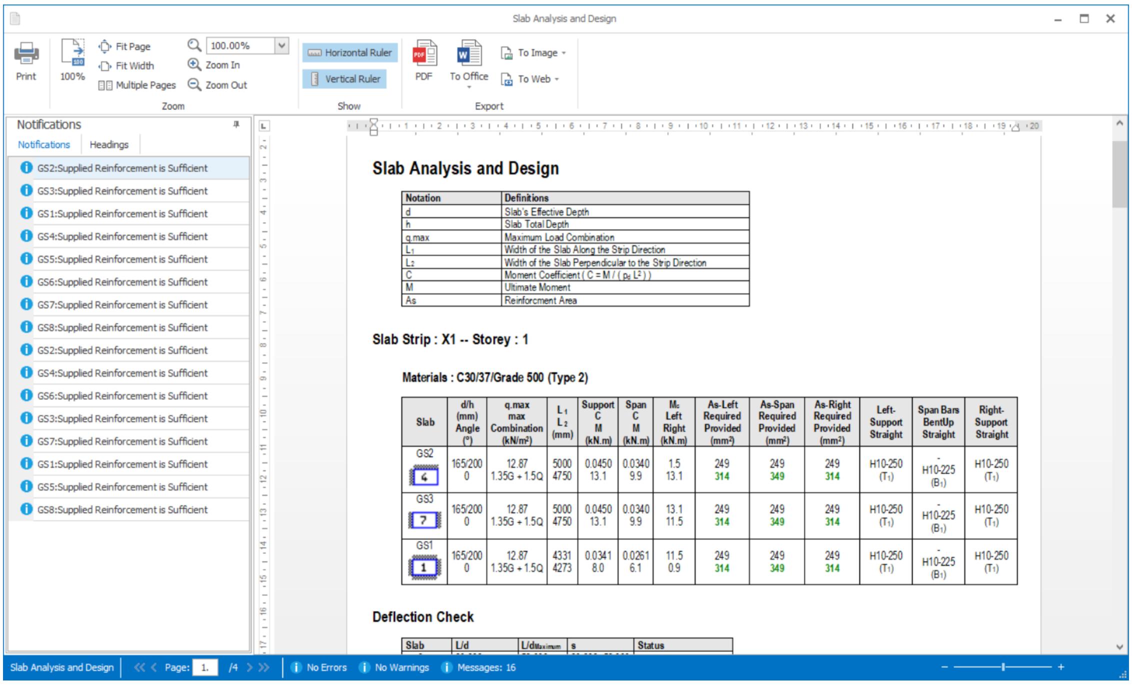
Slab Strip	Storey	Type	Analysis Results Source	Slabs	U. Ratio	Design Status	Print	Rebars
X1	1	Span Strip	-	GS2 (h=200 mm) GS3 (h=200 mm) GS1 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 16H10-300 + 19H10-250 16H10-300 + 20H10-250 + 14H10-250
X2	1	Span Strip	-	GS4 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 + 19H10-250
X3	1	Span Strip	-	GS5 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 + 19H10-250
X4	1	Span Strip	-	GS6 (h=200 mm) GS7 (h=200 mm) GS8 (h=200 mm)	0.0	✔	☑	11H10-300 + 13H10-250 16H10-300 + 19H10-250 16H10-300 + 19H10-250 + 19H10-250
Y1	1	Span Strip	-	GS2 (h=200 mm) GS4 (h=200 mm) GS6 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 16H10-300 + 19H10-250 13H10-300 + 18H10-250 + 10H10-250
Y2	1	Span Strip	-	GS3 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 + 19H10-250
Y3	1	Span Strip	-	GS7 (h=200 mm)	0.0	✔	☑	16H10-300 + 19H10-250 + 19H10-250
Y4	1	Span Strip	-	GS1 (h=200 mm) GS5 (h=200 mm) GS8 (h=200 mm)	0.0	✔	☑	14H10-300 + 14H10-250 16H10-300 + 19H10-250 16H10-300 + 19H10-250 + 19H10-250

The slab analysis & design dialog will appear with a summary of pass / fail status, utilization ratio & reinforcement. The functions at the top menu bar are :

- **Interactive Design**: Select a slab strip → Interactive Design will allow access to detail strip design
- **Slab Strip Design (Batch Mode)**: Redesign or re-check the existing design of all strips at once
- **Settings & Parameter**: Opens up the slab design settings in the Settings Center
- **Filter**: Allows filter by a storey, slab thickness, and design status
- **Copy Bars & Paste Bars**: Allow copy & paste of bars of similar slabs marked with **=** symbol
- **Design Report**: Prints out the design report

➢ *Click **Design Report***

The **Slab Reinforcement Design** report will be displayed. Options are available to configure and then print it. Any failure in the design will be highlighted in the **Notifications** pane at the left.



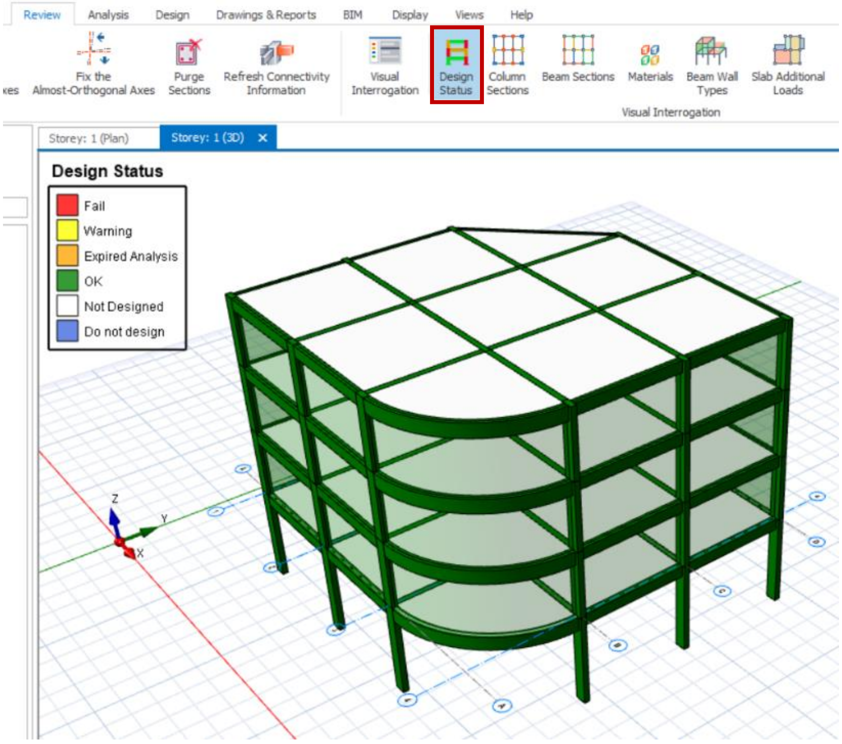
➢ ***Review** the report and then **Exit***

Design Status

The design status can be displayed graphically in the plan or 3D window.

➢ *Click on the **3D view** to make it active*

➢ *Go to **Review** Tab → pick **Design Status** → **OK***



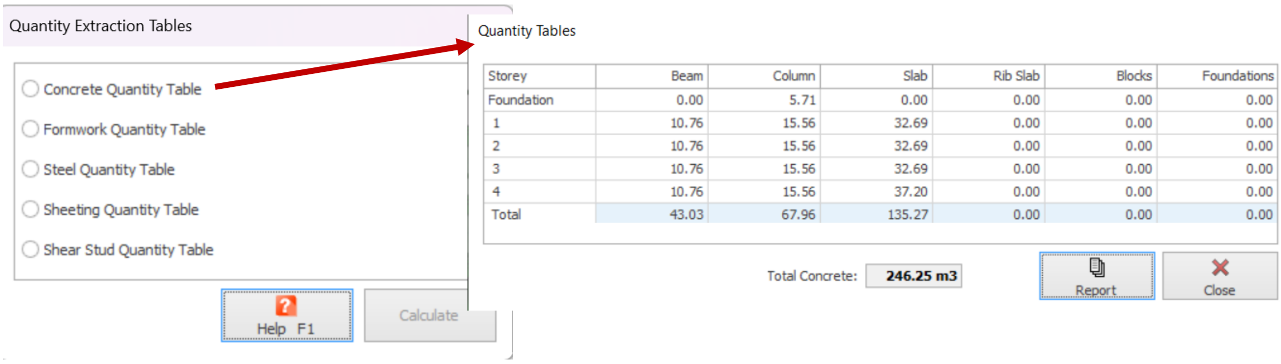
Quantity Extraction Tables

The concrete and Formwork quantity reports can be produced.

➢ *Go to the **Drawings & Reports** tab → Pick **Quantity Extraction Tables***

➢ *Choose **Concrete Quantity Extractions Table** → click **Calculate***

As shown below, this produces a concrete quantity report with member type and storey breakdown.

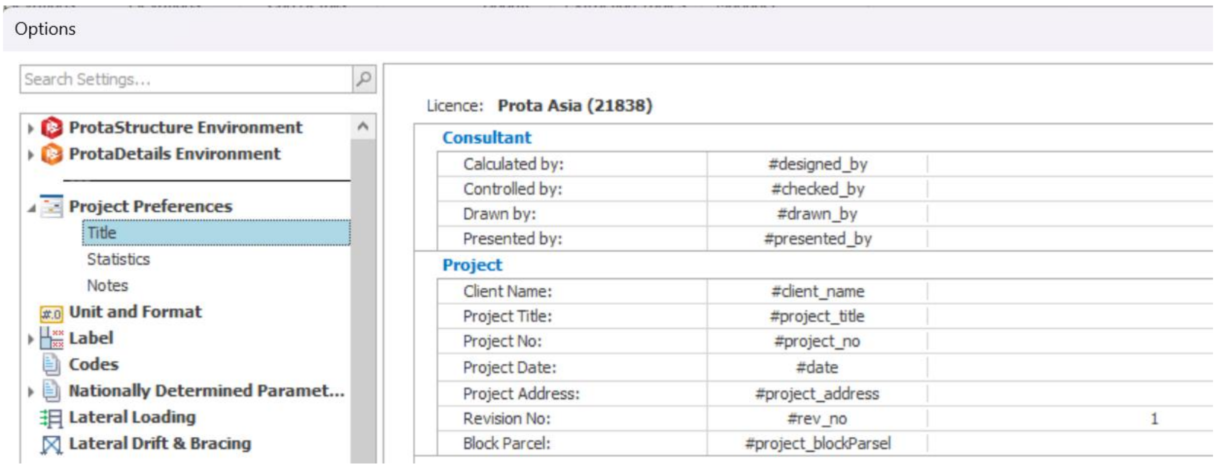


➢ *Click **Report** to produce a detailed report*

Project Preferences

The Project Preferences dialog allows you to enter the details of the project model.

➢ *Go to **Building Setout** tab → Click on **Settings Center** → **Project Preferences** → **Header***



Title - Enter the Consultant and Project details to be displayed in the header of all the reports.

Statistics - Show graphical chart of essential model information such as the total number of members, member types, grids & storeys. In addition, it shows the number and types of foundation, beam & column end releases. A text file can be created.

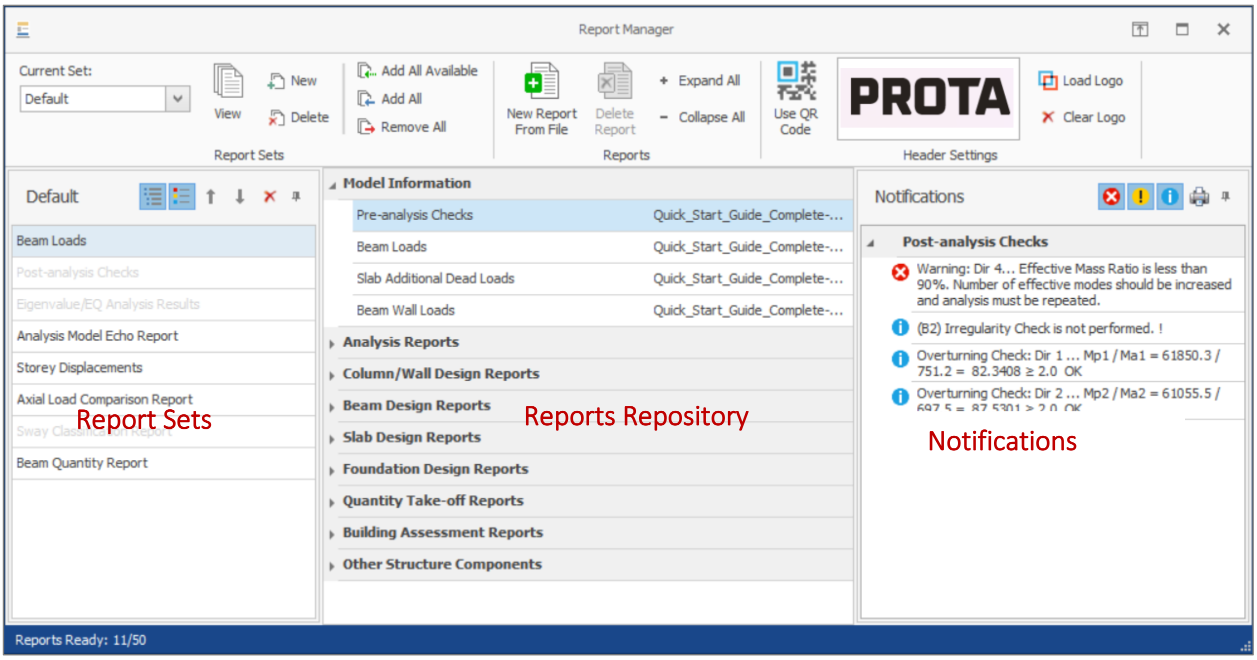
Notes - Enter additional information on this model, such as revision changes, etc.

➢ *Key in the various information in the **Title** tab as desired*

Report Manager

The **Report Manager** is the central report manager where all analysis and design reports can be accessed and managed.

➢ *Go to the **Drawings & Reports** tab → choose **Report Manager***



- **Report Sets** (left): This shows the constituent reports, which will be complied & combined into a single final report.
- **Reports Repository** (middle): All the individual reports generated and available to be included as a Report Set.
- **Notifications** (right): Important notes and warnings from the model's analysis are shown for easy reference.

You can expand or collapse the main folder by clicking on topic icons. To create a combined report:

➢ *Select **New*** New *in the Report Sets tab and give the new report set a name.*

➢ *Select **Add All Available** reports in the Reports window to include only available and generated reports in the report repository.*

Alternatively, click and drag a report from the Reports Repository into the report Sets pane.

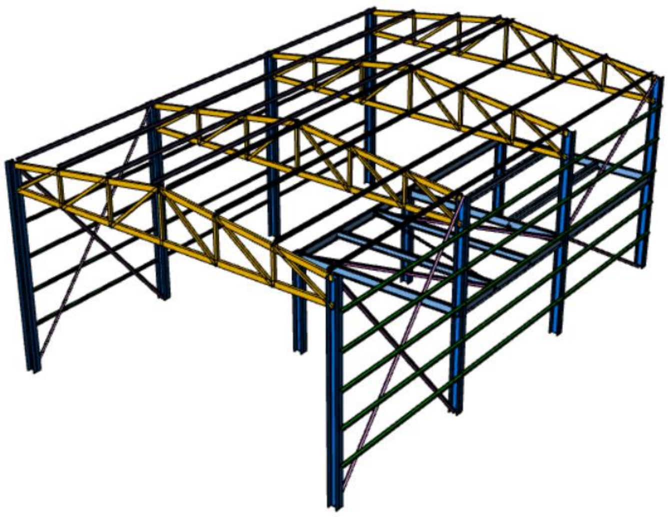
➢ *Choose to insert **Table of Contents** and **Summary Report** using icons below the report Sets tab.*

➢ *Select **View** to generate and view the report set.*

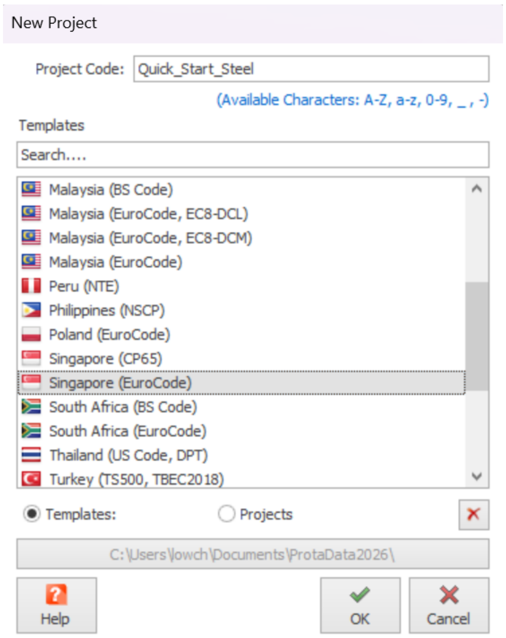
Other useful functions include loading of company logo : **Load Logo**, inserting **QR Code** & inserting external .doc or .rtf files : **New Report from File**.

Steel Model

We will now start a new steel model. This section will cover steel members, trusses modelling, analysis & design. The below is the screenshot of the completed model, which can also be found in the default Project Data folder.



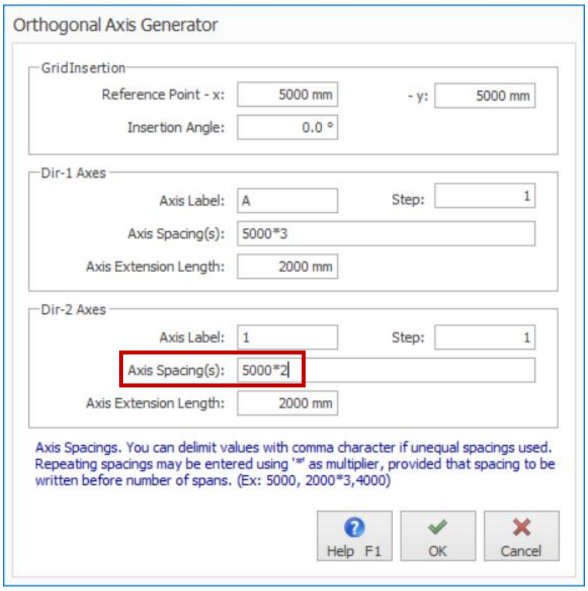
➢ *Click New Project*  *and type the project name as shown using the '_' character for spaces.*

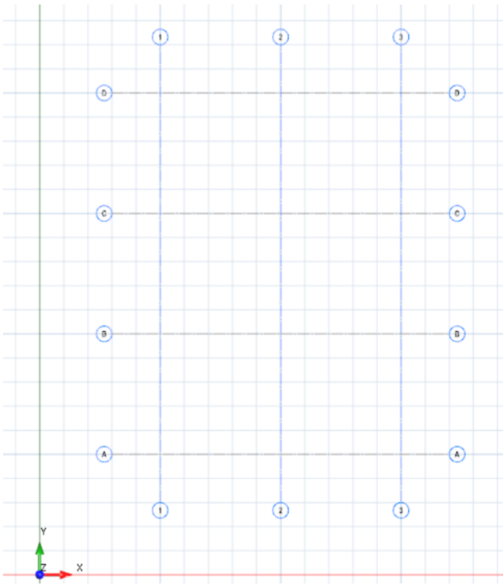


➢ *Select **Singapore (Eurocode)** template and then **OK***

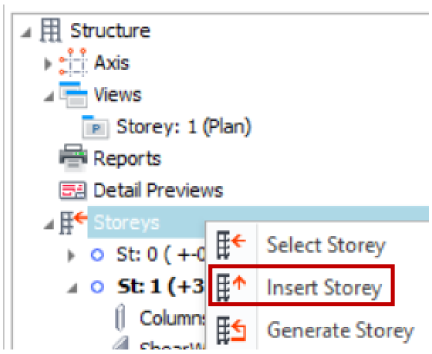
Axis Creating & Storey Insertion

- *Go to Modelling tab → Pick* ***Orthogonal Axis Generator*** *in the* ***Modelling*** *tab.*
- *Pick the* ***intersection*** *of the primary grid near the origin (5000,5000 coordinate).*
- *In Orthogonal Axis Generator, change* ***Dir-2 Axis Spacing*** *to* ***5000*2*** *& click* ***OK****.*

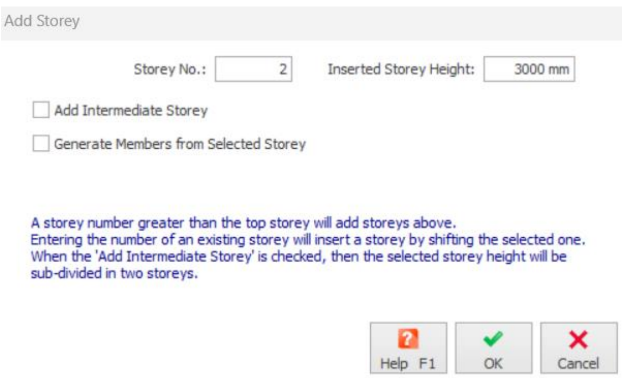




- *Right-click on* ***Storeys*** *in the Structure Tree and pick* ***Add new Storey****.*



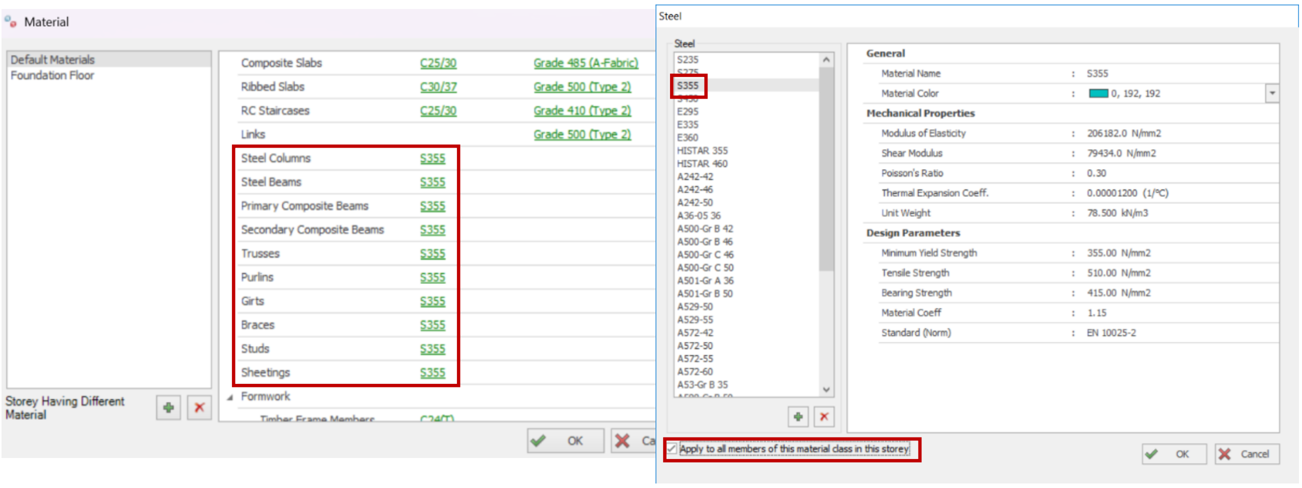
- *Input* ***Storey No.: = 2*** *→* ***OK.*** *Leave* ***Add Intermediate Story*** *and* ***Generate Members from Selected Storey*** *unchecked.*



- *When prompted to confirm → Pick* ***Yes*** *→ The plan view will now change focus to Storey 2*

Materials & Load Case / Combination Generator

- *Go to **Analysis** tab → **Building Analysis** → **Pre-Analysis** tab.*
- *Pick **Edit Materials** → Change the steel grade of all members to **S355** (as shown below)*

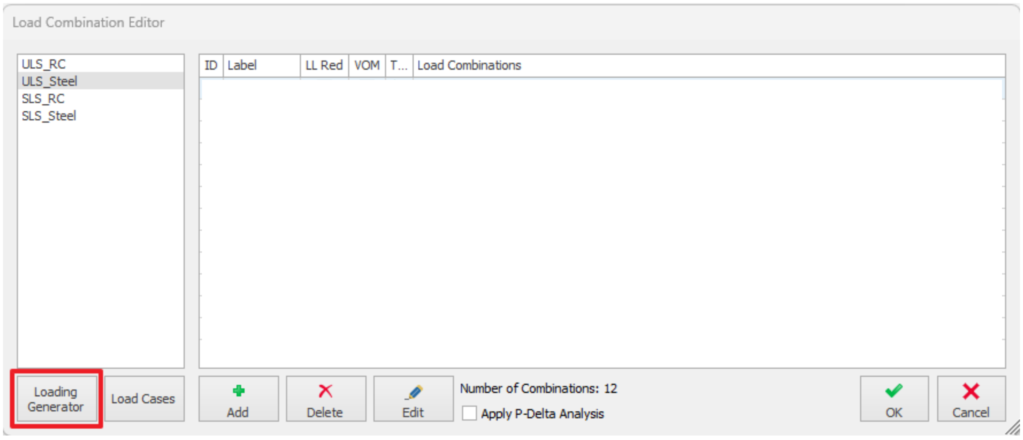


The global materials grade of the truss, purlins, girts & brace members are shown & can be changed.

Individual member material can also be changed in their respective **Section Manager** dialog by selecting the member → Right-click → **Edit Section / Material**.

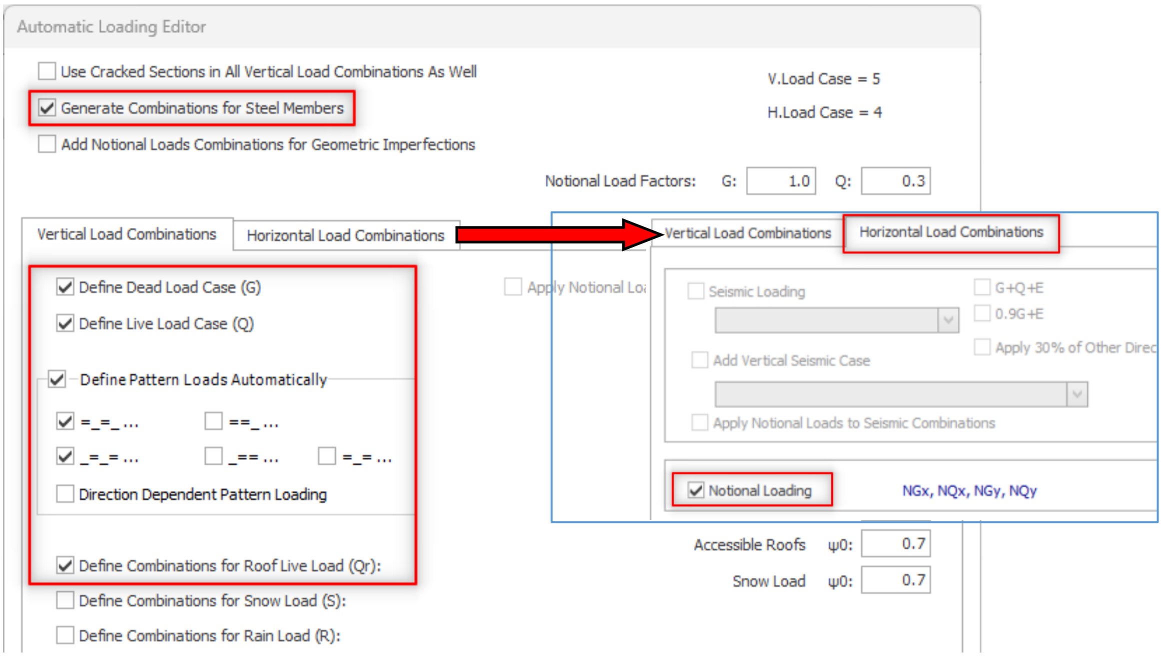
We will now auto-generate the load cases and combination (required to add roof purlins loads).

- *Pick **Loading Cases and Combinations** to access the Load Combination Editor.*



- *Pick **Loading Generator** to access the **Automatic Loading Editor**.*

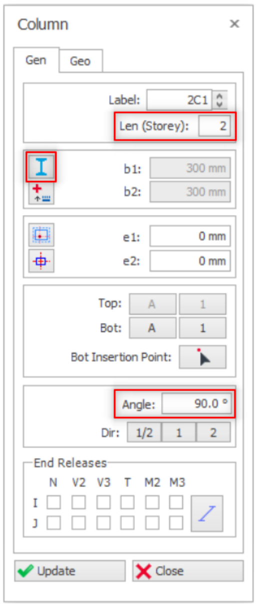
➢ *In the Automatic Loading Editor, pick the options as shown below → click **OK**.*



➢ *Load cases & combinations will be generated. Pick **OK** & **Close** the building analysis menu.*

Steel Columns Creation

➢ *Click on **Steel Column** in the **Modelling** tab.*



➢ *In **Column Properties**, change **Len (Storey)** to **2***

This means the column will span 2 storeys with effective length of 2 storey height. Thus, we do not need to model any column in ST01.

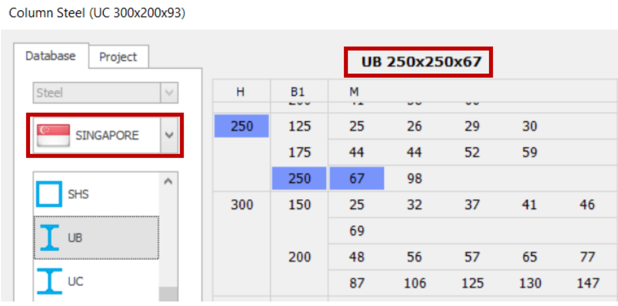
➢ *Change the **Angle** of rotation to **90 degrees**.*

This will rotate the steel section 90 degrees on plan view.

➢ *Pick **Section Manager** icon*

This will launch the **Section Manager** dialog which allows us to pick a section profile.

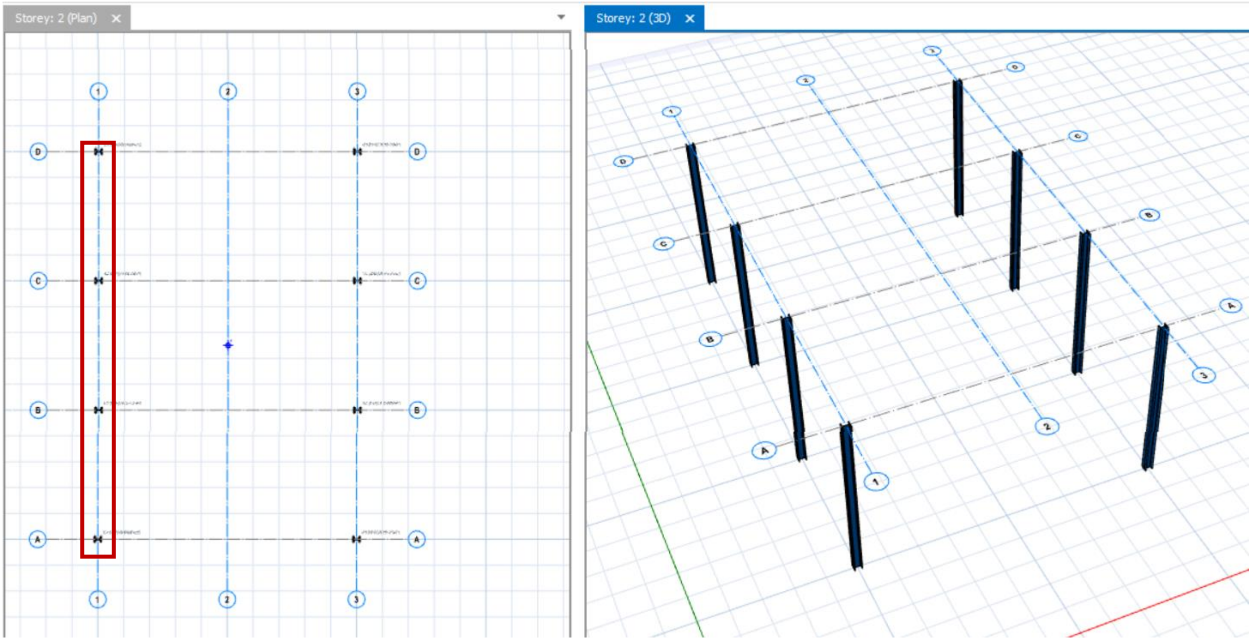
➢ **Column End Condition**: Columns ends are fixed by default. You can apply hinges to top and/or bottom by clicking successively on this icon.



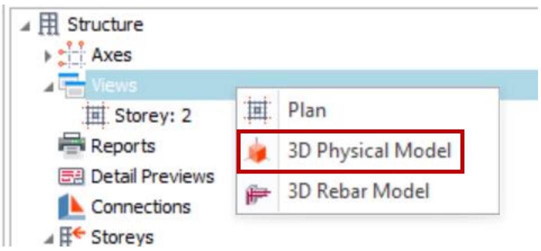
- *In* ***Section Manager*** *dialog, pick* ***Singapore.***

  This will access to Singapore Steel Profiles (Continental)
- *Pick* ***UB 250x250x67*** *→* ***Select.***
- *Pick* ***OK*** *to close the dialog.*

- *Enter* ***eight nos*** *of columns along Axis* ***1/A-D*** *&* ***3/A-D***



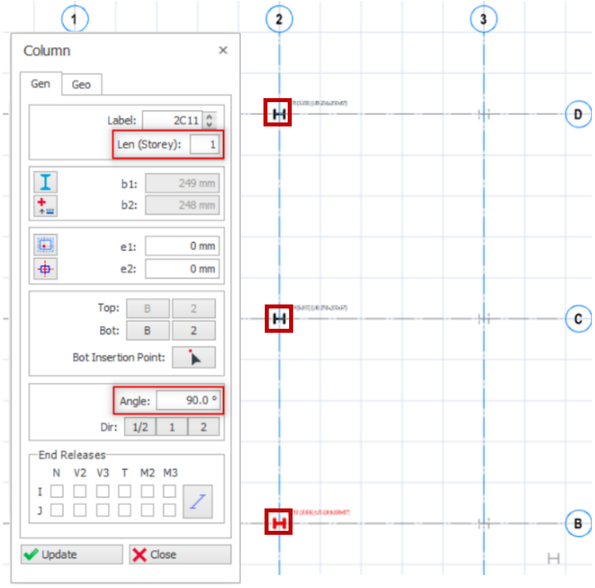
- *Right-click on* ***Views*** *in the Structure Tree and pick* ***3D Physical Model****.*

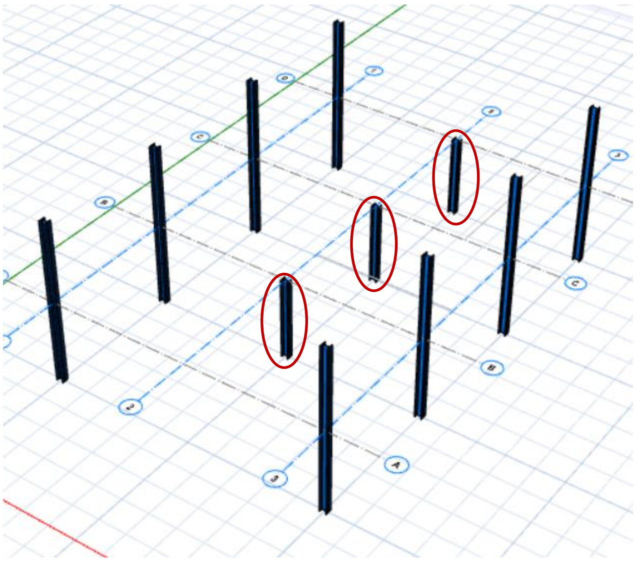


This command will create a separate window showing the 3D view.

- *Go to* ***Views*** *tab →* ***Tile Vertical***

  This command will arrange the 3D view and plan view in 2 windows.
- *Click on the* ***Plan*** *view to make it active (the active view title & border will be darker)*
- *Double-click on* ***Storey 1*** *in the* ***Structure Tree*** *to switch focus to Storey 1*





- *Click on the **Steel Column** icon*
- *Change **Len (Storey)** to **1** in **Column Properties***

  This property means that the column will only span a single storey.
  The columns with “Len = 2” are grey in ST01 because they were inserted in ST02.

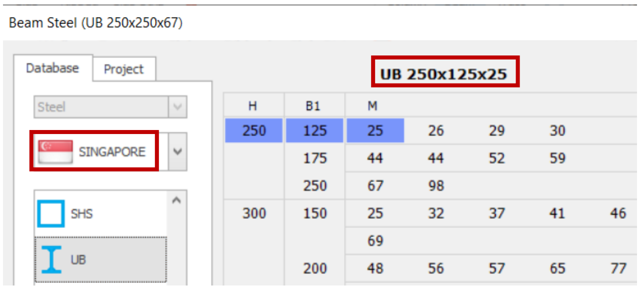
- *Ensure the **angle** of rotation is **90 degrees**.*
- *Pick **Section Manager** icon*
- *Ensure that the same **UB 250x250x67** is selected.*
- *Insert **three** nos. of the column along GL2/B to D.*

*Check the **3D view** to ensure the columns were properly.*

Steel Beams Creation

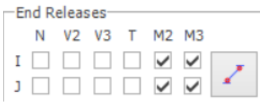
- *Click on the **3D view** to make it active (the active view title & border will be darker)*
- *Double-click on **Storey 1** in the **Structure Tree** to switch focus to Storey 1.*
  Notice the grids will move to storey 1. This operation ensures beams to be created in St: 1 in the 3D view.
- *Click on **Frame** icon*
- *Then select **Steel Beam***

  Steel Beam
- *Click on **Section Manager** in Beam Properties*

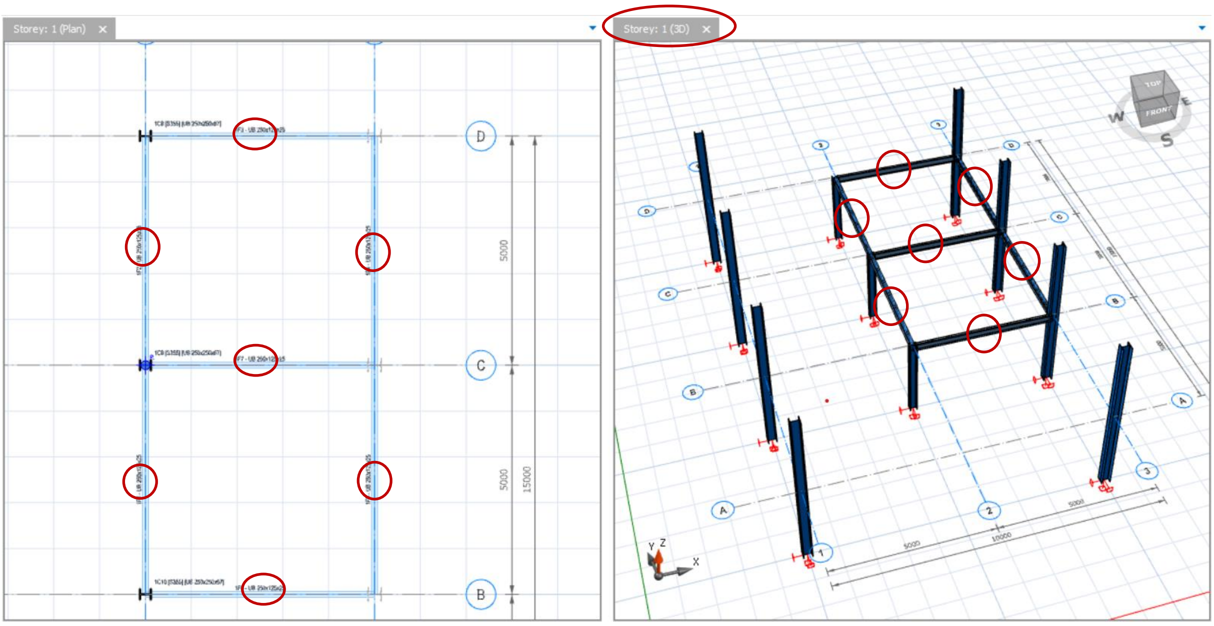


- *In **Section Manager** dialog, pick **Singapore flag.***
  This will access to Singapore Steel Profiles (Continental)
- *Pick **UB 250x125x25 → Select.***
- *Pick **OK** to close the dialog.*

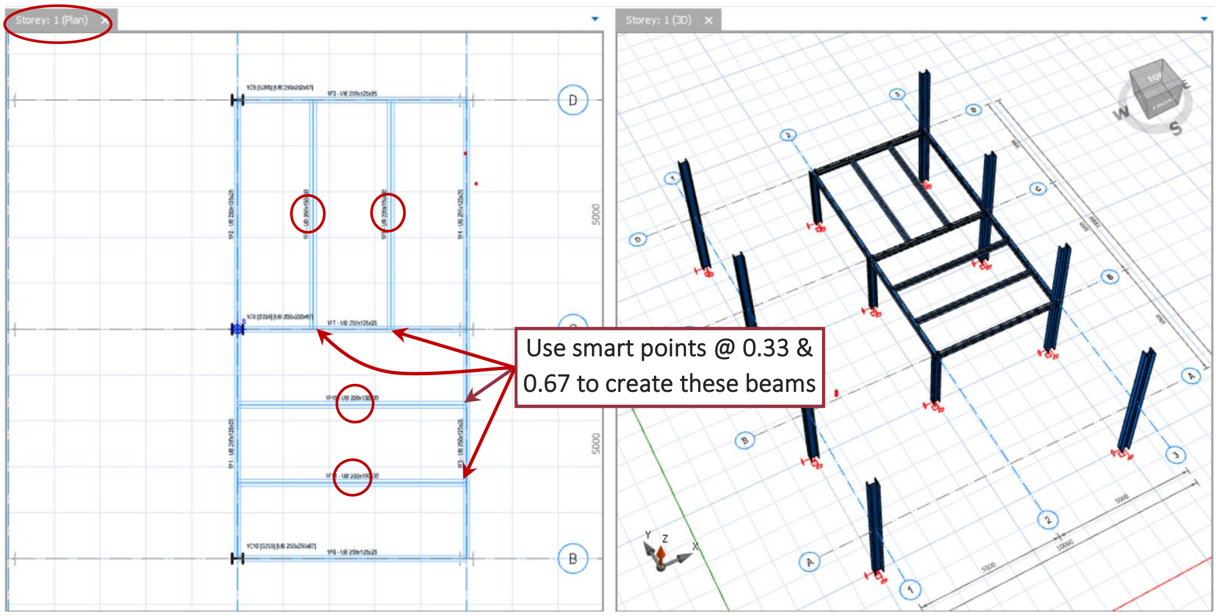
- ***End Releases**: Click successively to **hinge** both ends of the beam*



- *Click on the 3D view to make it active > **Double-click** on St: 1 to make it the active storey.*
- *In the 3D view, create **seven nos.** of beams in the region bounded by **GL B, D,2** & **3** (as shown below)*



- *Go to the **plan view** of **ST01** → Create **four nos.** of **UB 200x150x30** secondary beams in the region bounded by GL **B, D,2** & **3** (as shown below)*
- *Ensure both ends of the beams are hinged*



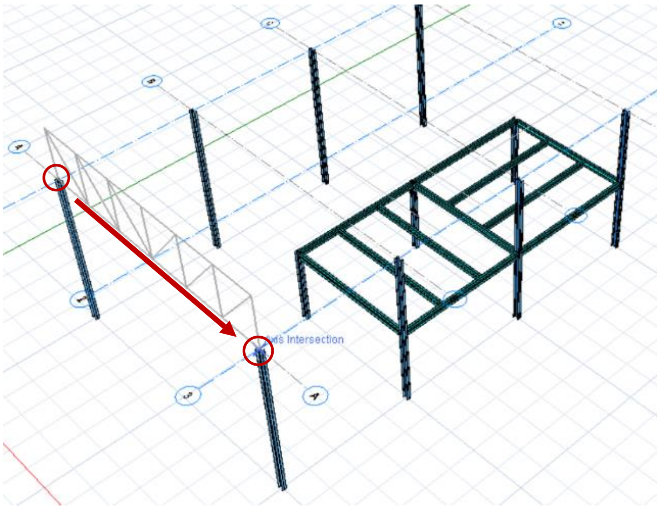
Use smart points to create secondary beam. With beam properties out, place the mouse cursor at the edge of the primary beam (not center) & the smart points will appear at **0.25L**, **0.33L**, **0.5L** & **0.75L**.

Use **Beams Sections**  in the **Review** tab to show beams of different sizes by colour. Assign colours to members in **Section Manager** dialog → **Material Colour**

Steel Truss Creation

We will now insert roof trusses. Trusses are best inserted using the 3D view.

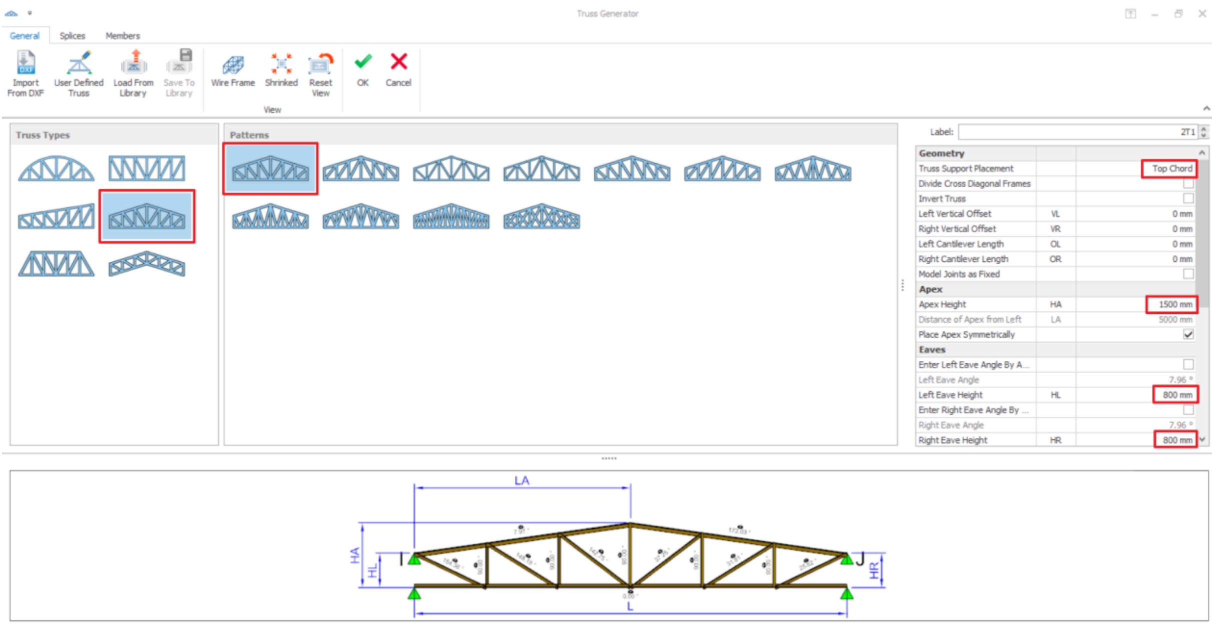
- *Go to the **3D view** → **Double-click** on **St: 2** in Structure Tree to make it **active** & grids are at **St: 2***
- *Click on the **Truss** icon* 



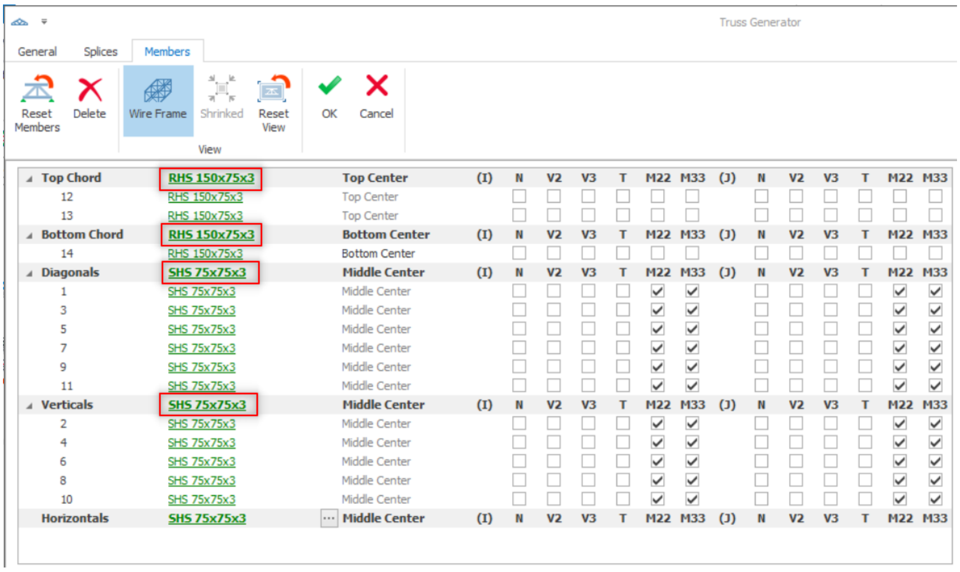
- *Pick **grid intersections A/1** to define the **start** of the truss.*

  A dynamic rubber band will appear with the default truss type if no truss is defined before.
- *Pick **grid intersections A/3** to define the **end** of the truss.*

  After you pick the second point, the truss generator dialog will appear.

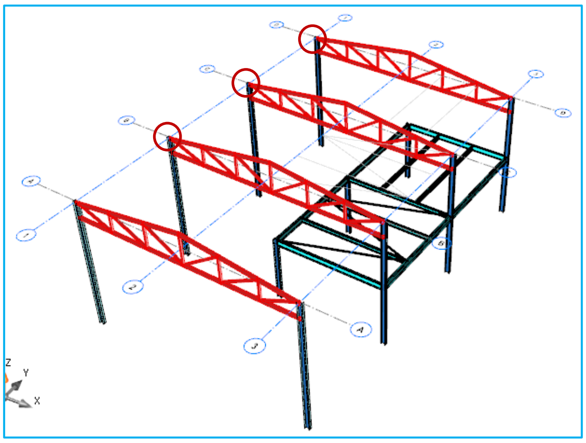


- *In **Geometry**, pick **Duo-Pitched** truss → Change **Truss Support Placement** to **Top Chord.***
- *Change **Apex Height** to **1500 mm**, **Left & Right Eaves Height** to **800 mm.***
- *Click on the **Members** tab*



- *Change the truss member sections to the following by click on the section name:*
  - *Top & bottom chord = **RHS 150x75x3** (Singapore)*
  - *Diagonals & Verticals = **SHS 75x75x3** (Singapore)*
- *Click **OK** to exit the Truss Generator dialog.*

The truss will be inserted. The truss insertion will automatically switch into copy mode & you can pick the next point of insertion.



- *Insert **3 nos**. of similar trusses by clicking grid intersection **B/1**, **C/1** & **D/1***
- *Press **ESC** or **Right-Click** to end the copy operation.*

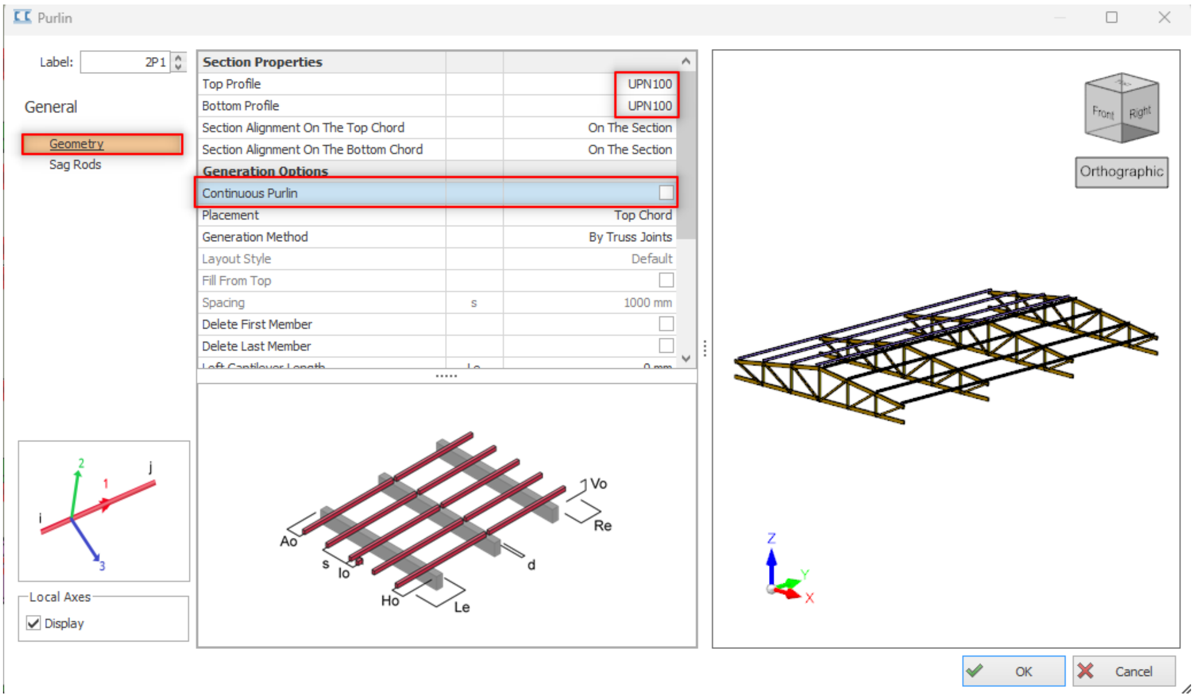
To enter copy mode manually, select the truss → Right-click → Pick **Copy**

- *Select the basepoint @ intersection of **A/1***
- *Insert **3 nos**. of similar trusses by clicking grid intersection **B/1**, **C/1** & **D/1***

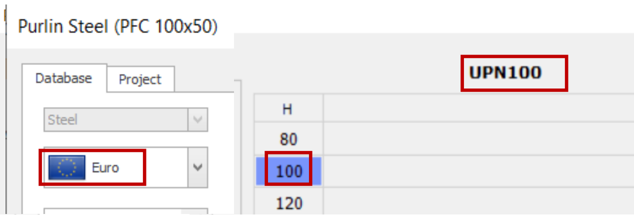
Purlins Creation

We will now insert purlins on the roof trusses. Purlins are best inserted using the 3D view.

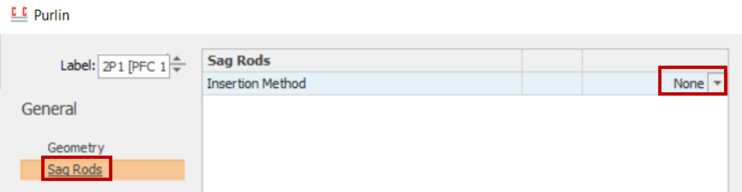
- *Click on **Purlin** toolbar button → **Insert Using Two Members***
- *Select the **first** truss @ **GL A** → Select the **last** truss @ **GL D** (Intermediate trusses will be automatically found)*
- *Another insertion method is by Insert Using Multiple Members. Click Enter after all the members needed are selected.*
- *On **Purlin Dialog**, you can specify the following:*
  - Profile / Section of the for the Purlin.
  - Section Alignment: On the Section / Under the Section / Centered / Align To Top / Bottom.
  - Generation Method: By Truss Joints / By Spacing.
  - To design the purlin as continuous, tick Continuous Purlin. Edit Continuous Purlin max length in Settings Center > Steel Settings > Member Design Tab > Max. Member Length.
  - There are many other options. Refer to the diagrams as a guide.
  - The 3D diagram on the right can be rotated (right-click & drag) & zoom in/out (mouse wheel)



- *Choose section **UPN100** (under **Steel** → **European sections** → **UPN**)*



- *In the **Sag Rods** dialog → **Insertion Method** → Choose **None** to remove all sag rods (for simplicity)*



- *Pick **OK** to exit the purlins dialog.*

We will insert the roof loading using the more specific Cladding function.

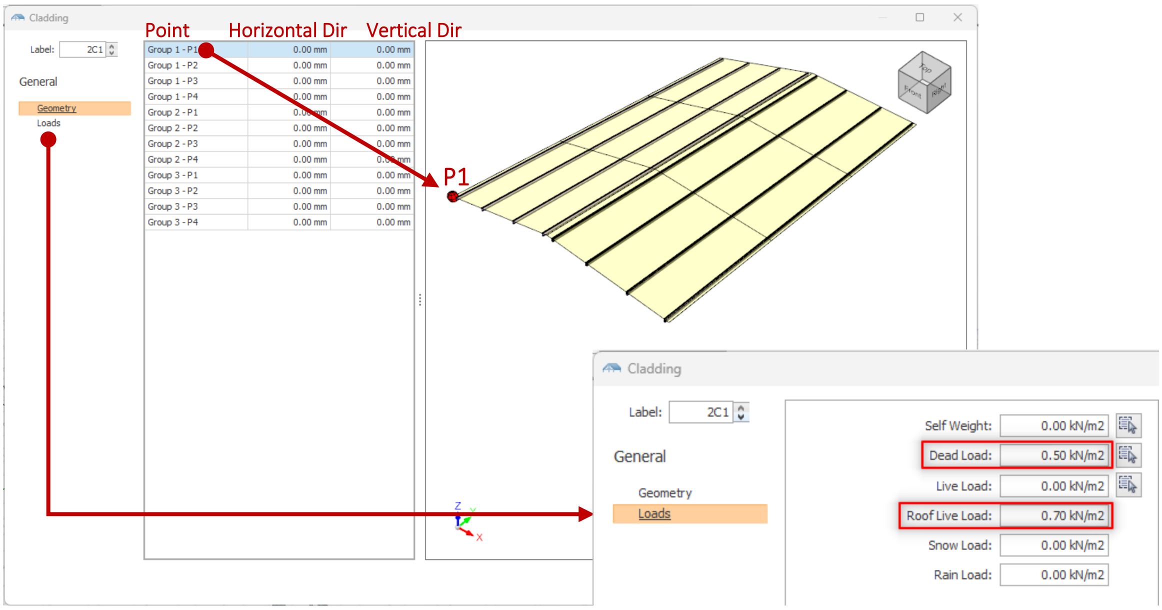
Creating Cladding & Loads

- *Go to **Modelling** tab → Pick **Cladding** → Select any purlin.*

The Cladding dialog will appear to insert roof loads on the purlins.

In the **Geometry** dialog, the cladding geometry can be adjusted by selecting the point and changing the values in the vertical and horizontal direction.

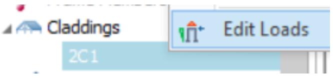
In the **Loads** dialog, loading of the cladding can be input here.



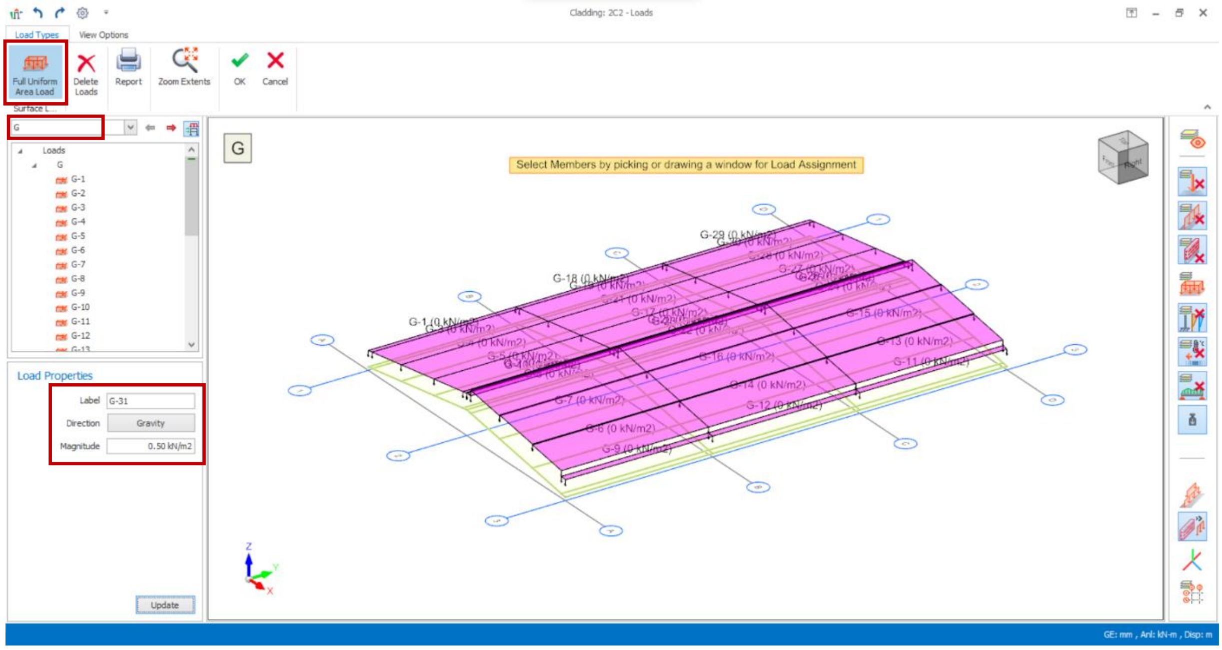
- Input the loads value in the **Loads** Dialog; Dead Load (G) = **0.50** kN/m2 and Roof Live Load (Qr) = **0.70** kN/m2. The loads input will be assigned onto the whole cladding area.
- *Click **OK** to exit the cladding dialog > The cladding will be rendered on top of the purlins.*

➢ *Alternatively, to input cladding load, go to the **Structure Tree** > **Cladding** > **Select** the cladding label*

➢ ***Right-click** > Pick **Edit Loads***



The load editor dialog will appear, which allows you to insert area load on the cladding.



➢ *At the top, pick **Full Uniform Area Load***

➢ *Pick **G Load Case** in the left pane.*

➢ *In **Load Properties**, ensure **Load Direction = Gravity** & **Magnitude = 0.5 kN/m**^{2}*

➢ *Click & drag window over the entire roof cladding to apply all panels.*

➢ *Pick **Qr Load Case (Roof Live)** in the left pane & **Full Uniform Area Load***

➢ *In **Load Properties**, ensure **Load Direction = Gravity** & **Magnitude = 0.7 kN/m**^{2}*

➢ *Click & drag window over the entire roof cladding to apply all panels.*

➢ *Click **OK** to save & exit the load dialog.*

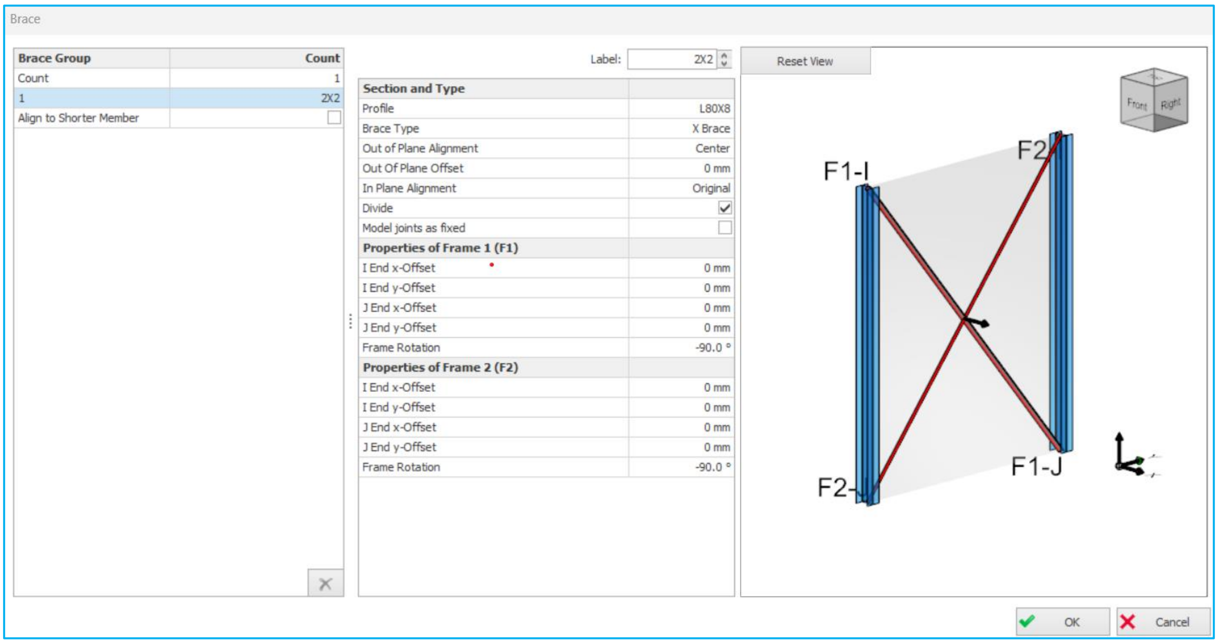
➢ *You can switch off the **cladding Layer** by going to the **Display** tab > **Layers**.*
This operation will make the view less cluttered & make the selection of other members easier.

Braces Creation

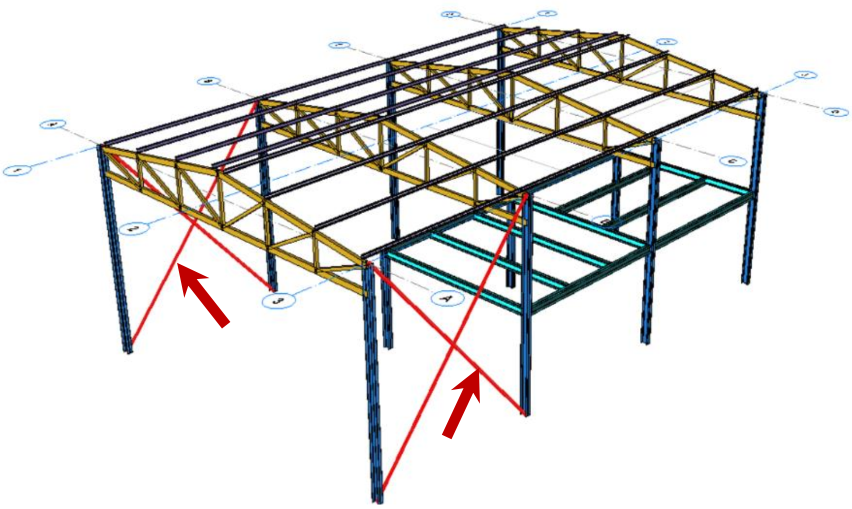
We will now insert some bracings between steel columns.

➢ *Click on the **Brace** toolbar icon*

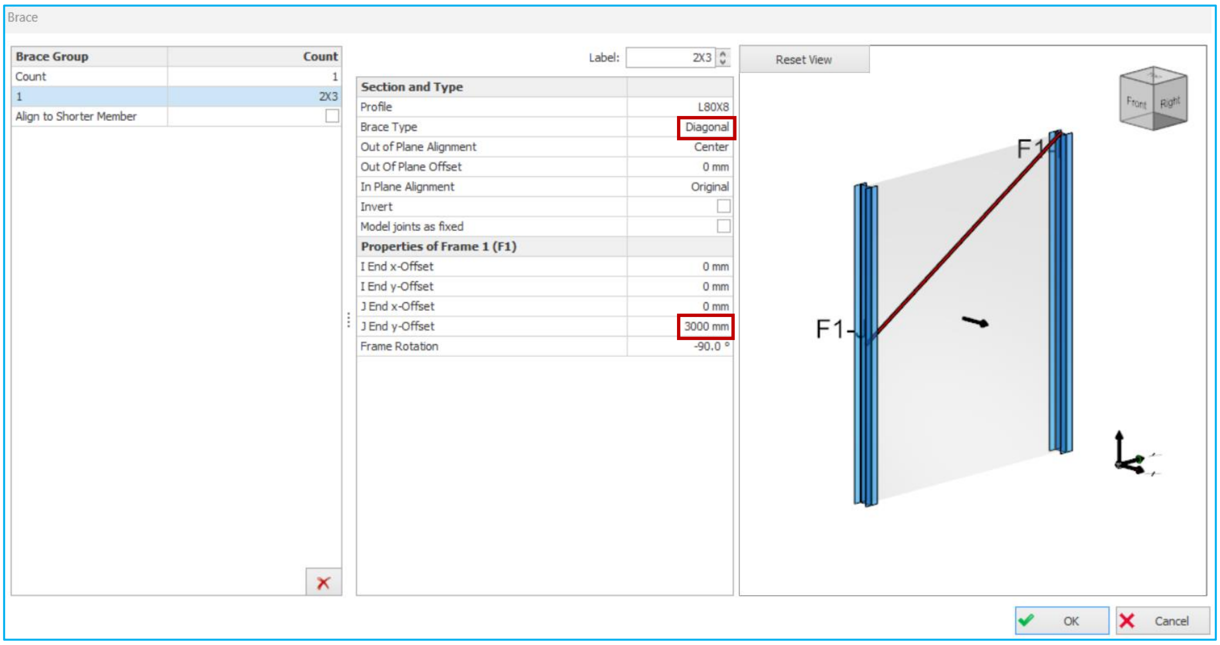
➢ *Pick two adjacent columns at **A/1** & **B/1** → Brace Dialog will appear.*



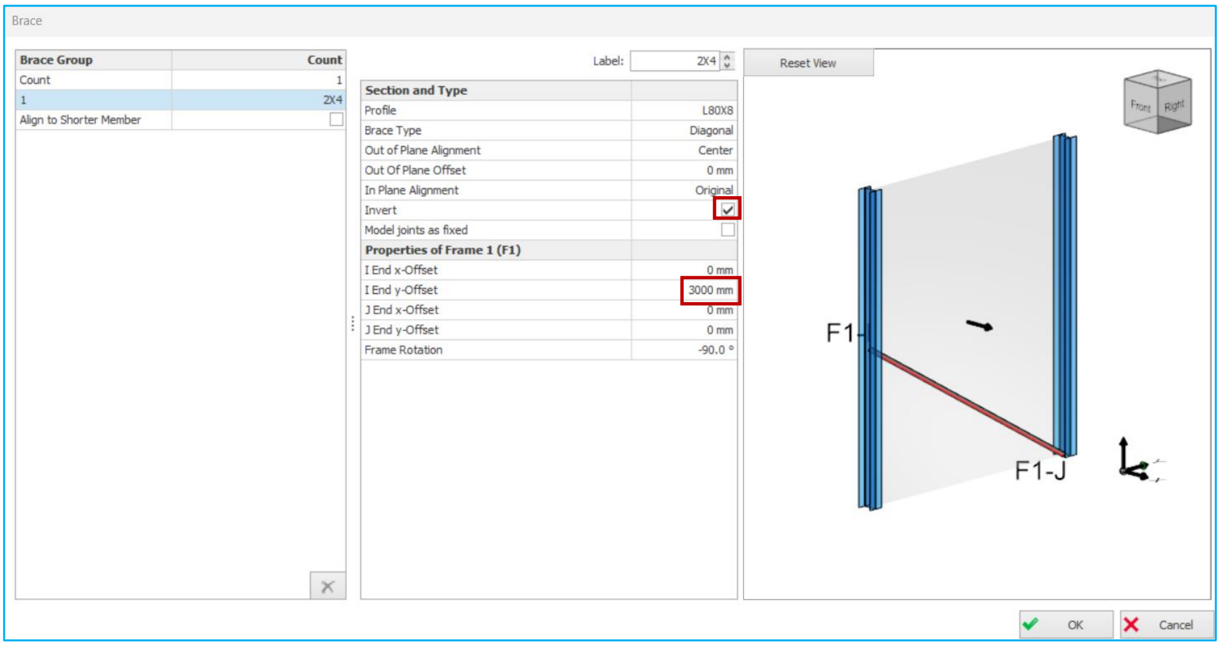
- *In the Brace dialog, you can specify the following:*
  - Profile / Section of the for the braces
  - Brace Type (Diagonal, X Brace, Y Brace or K Brace)
  - Alignment & Top/Bot Offsets
  - Apply to analysis: offsets will affect the analytical model.
- *Accept all defaults & click **OK.***
- *Insert the same bracing between columns **A/3** & **B/3***
- *Check the braces are correctly inserted, as shown below*



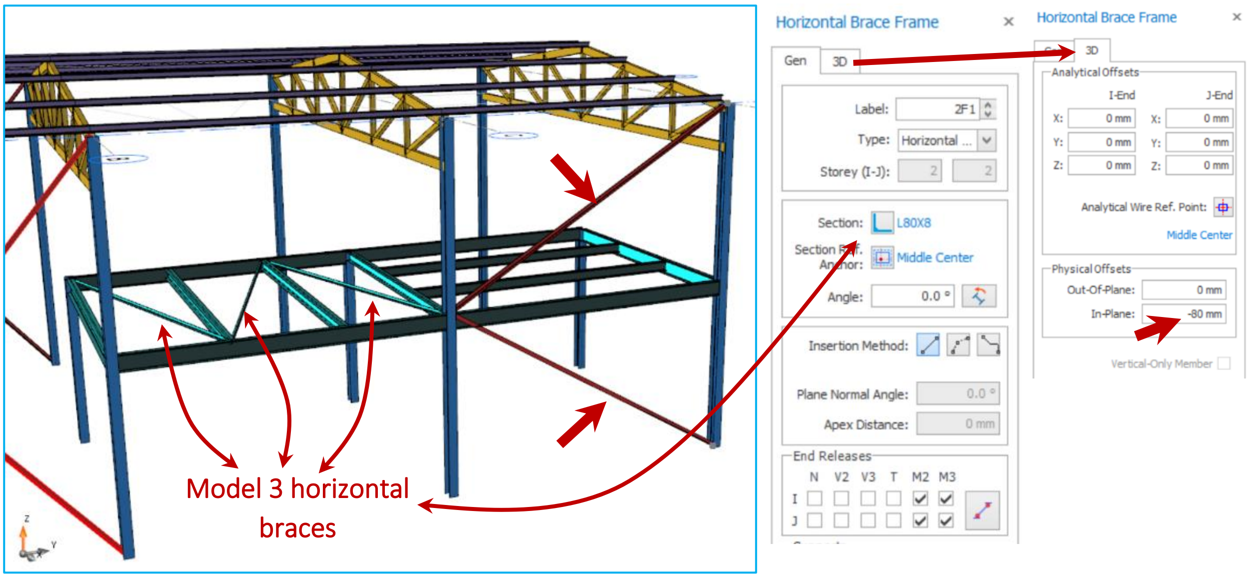
- *Insert new brace between columns **C/3** & **D/3***



- *Change **Brace Type** to **Diagonal***
- *Input **J End y-Offset** = **3000 mm** (rest remains 0 mm)*
- *Click **OK** → Diagonal Brace will be inserted.*
- *Insert another new diagonal brace between the same columns **C/3** & **D/3.***



- *Ensure **Brace Type** = **Diagonal***
- *Tick **Invert** → this will invert the diagonal.*
- *The input **I End y-Offset** = **3000 mm** (rest remains 0 mm).*
- *Click **OK** → A new diagonal brace will be inserted.*
- *Check the braces are correctly created in the 3D view as below.*



Horizontal braces can also be inserted between beams. Create 3 nos. horizontal braces connecting the beams in ST01, as shown above.

- *Go to Frame dropdown, pick **Horizontal Brace Frame*** Horizontal Brace Frame

Horizontal Brace Frame dialog will appear. We will just use the default "L" section.

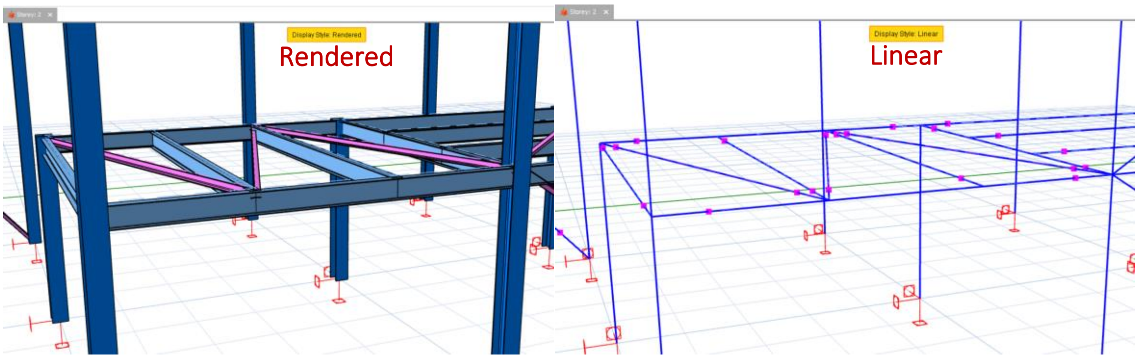
- *Go to **3D** tab > **Physical Offsets** > Change **In-Plane** = **-80 mm** (top of section aligns with beam top)*

Physical Offsets changes the position of a member in the 3D view but does not affect the analytical position in the analysis (changed by **Analytical Offsets**).

- ***Pick** the 2 insertion points for the brace ends to create **3 nos**. horizontal braces (as shown above).*
- *Click **OK** to close the dialog.*

When modelling steel members, it may be easier to switch **to analytical Linear** view :

- Go to **3D** view > By default a "Solid" or "Rendered" model is shown.



- Press "**CTRL + D**" consecutively to change to "**Linear**" to Display Style
- Ensure "**Linear**" is active > This is the 3D analytical wireframe model which will be analysed.
- Verify the members are connected properly by zooming in common joints of interest.

It is important to check member connectivity before the analysis, otherwise analysis may fail or yield many warnings of instability.

Girts Creation

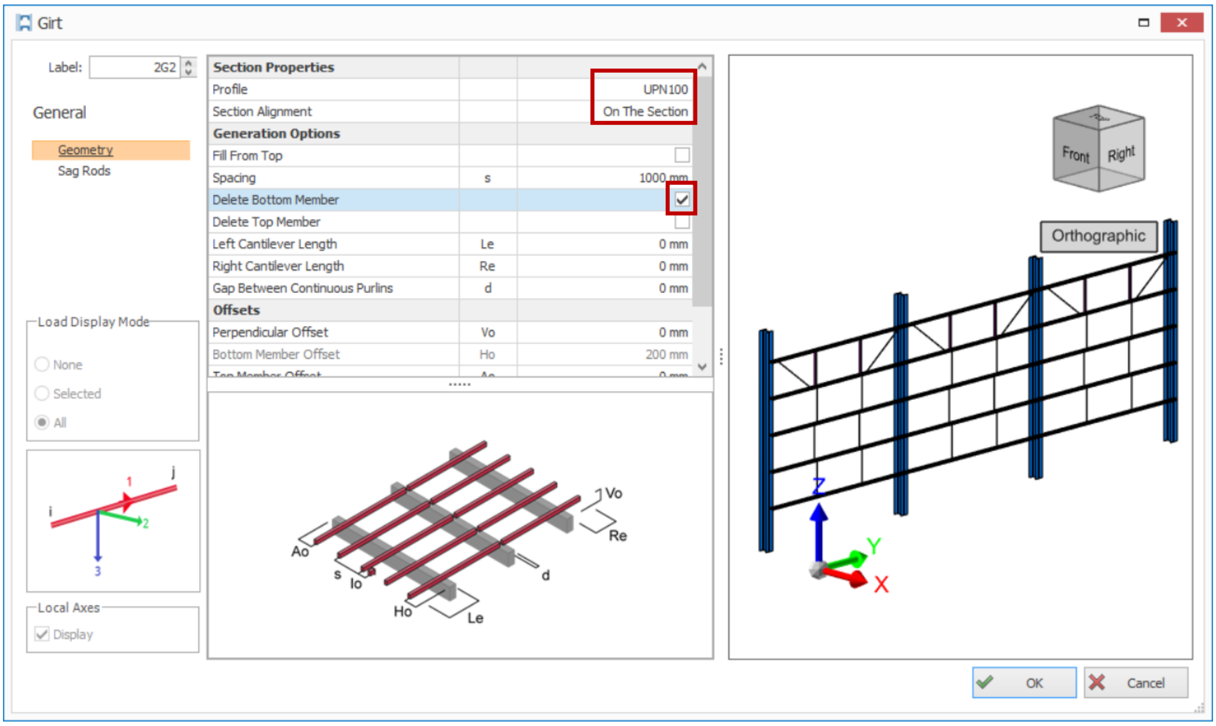
We will now insert some girts between steel columns.

- ➢ *Click on **Girt** toolbar button*
- ➢ *Select the **first column** at grid **A/3** → Select the **last column** at grid **D/3**.*

Intermediate columns will be automatically found.

- ➢ *In the **Geometry** dialog, you can specify the following:*

- Profile / Section of the Girt
- Section Alignment: On The Section / Under The Section / Centered / Align To Top / Bottom
- Spacing, Delete Bottom / Top Member, Left / Right Cantilever, Offsets



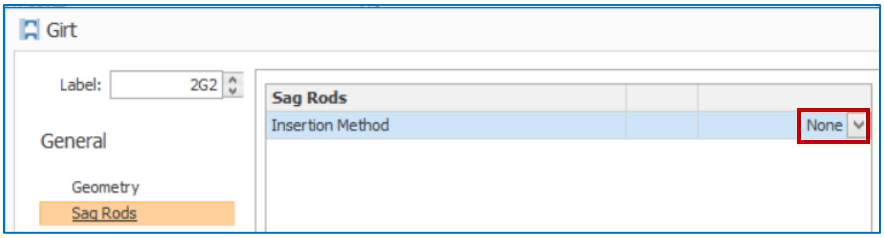
The insertion of the girts depends on the sequence of clicking the members as this will determine the correct *Section Alignment*. For example, if you click the column at A/1 followed by the column at D/1, the girts will be placed at the right-hand side of the column i.e., at the internal part of you structure when ***Section Alignment** = **On the Section.*** Therefore, you will have to change ***Section Alignment** = **Under the Section*** if you would like to have the girts modelled at the external part of the structure. Likewise, if you click the column at D/1 followed by the column at A/1, the girts will be placed at the left-hand side of the column i.e., at the external part of you structure when ***Section Alignment** = **On the Section.***

Inserting the girts b clicking on the members at any sequence are both correct so long as you define the section alignment correctly to ensure that the girts are modelled correctly.

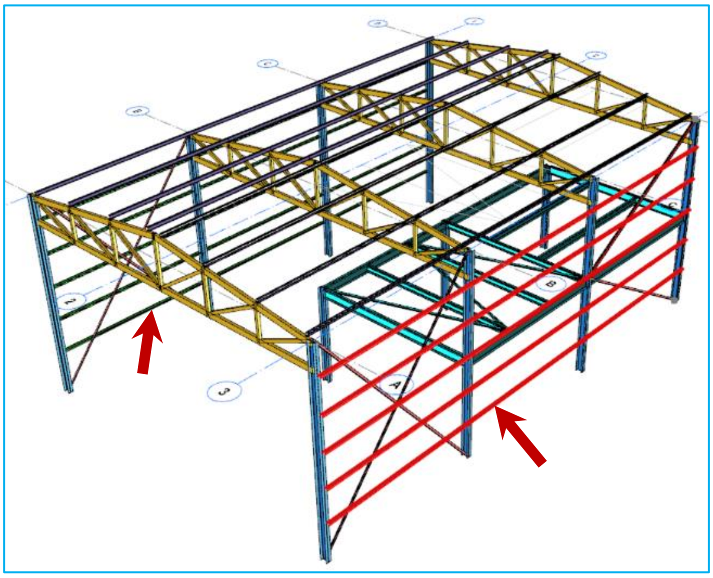
- ➢ *Ensure Profile is **UPN100** (under **Steel** → **European sections** → **UPN**)*
- ➢ *Ensure **Section Alignment** is **On The Section**.*
- ➢ *Tick "**Delete Bottom Member**" so that bottommost Girt is removed.*

In the **Sag Rods** dialog, you can specify the configuration of the sag rods. For simplicity, we will not insert the sag rods for this model, as they are minor elements.

- *In the **Sag Rods** dialog → **Insertion Method** → Choose **None** to remove all sag rods*

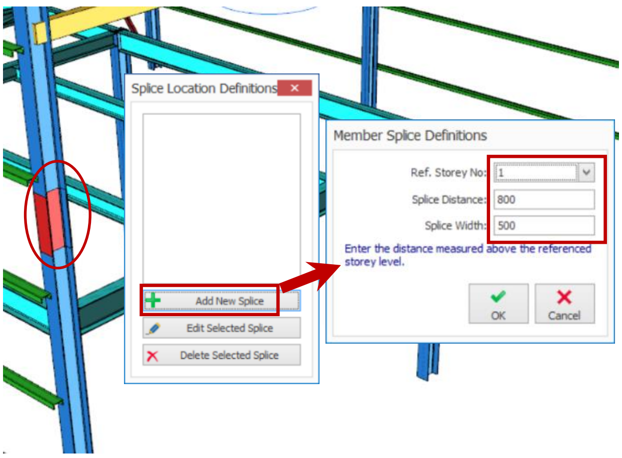


- *Click **OK,** and the girts will be inserted.*
- *Insert similar girts between columns **A/1** & **D/1** but **Section Alignment** = **Under the Section.***
- *Check the girts are inserted correctly at both sides of the building, as shown below.*



Loads can be applied to girds similar to Purlin by creating a cladding first. For simplicity, we will skip this.

Column Splice Creation



- *Press **ESC** to deselect all members.*
- *Select the corner **column** at **GL D/3***
- ***Right-click** → **Define Splice***
- *Pick **Add New Splice** in Splice Location Definitions*
- *In **Member Splice Definition** choose :*
  - Reference Strorey = 1
  - Splice Distance (from floor level) = **800** mm
  - Splice width = **500** mm
- *Click **OK***

  The splice will be created & listed in Splice Location Definitions.
- ***Close** Splice dialog*

Building Analysis

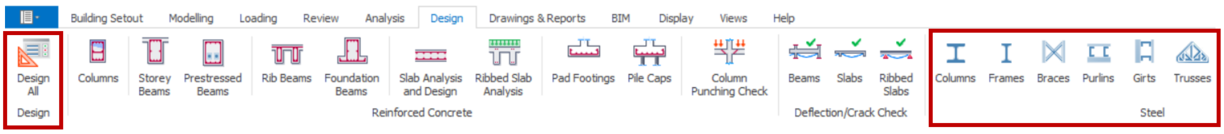
- *Go to the **Analysis tab** → Pick **Building Analysis***
- *In **Batch Design Options**, choose not the design any members → click **Building Analysis***

  Analysis should complete if there is no serious modelling mistakes.
- ***Review the results** in the **Analytical Model** view to your satisfaction as outlined in previous section.*

For example, check if the roof cladding loads have been correctly considered by switching on the **Frame Loads** & **Frame Load Labels**.

Steel Design

Steel member design commands can be accessed via the **Design** tab.



- *Go to **Design** tab → **Design All** → Pick **Steel Member Design Check** → **OK.***

❖ This will check all the adequacy of assigned all steel members in one go.

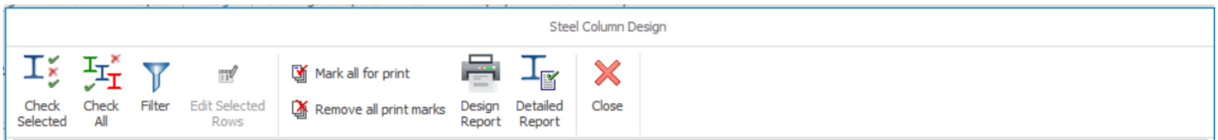
❖ The program does not automatically select the smallest section size.

❖ If the model is large, it is recommended to perform design check by member types separately.

❖ To access a summary of member design check, **select** the steel member icons in the steel group.

❖ To design only a single member, select it → Right-click → **Steel Member Design**

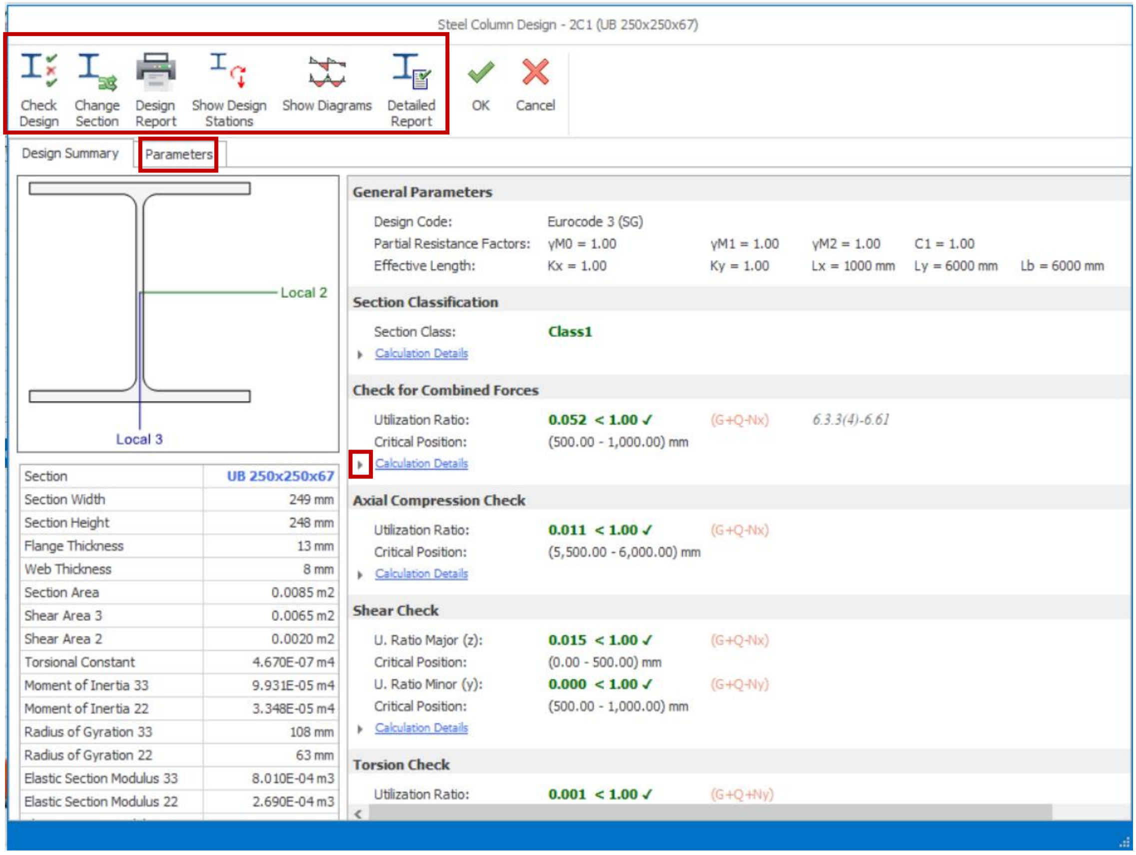
- *Pick steel **Column Design***  to access the design summary of all steel columns.



Member Label	Storey	Print	Section	Material	Section Class	Slenderness Ratio (kL/r)	Utilization Ratio	Design Status	Governing Check
2C1	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.05 < 1.00	Pass ✓	(Combined)
2C2	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.15 < 1.00	Pass ✓	(Combined)
2C3	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.15 < 1.00	Pass ✓	(Combined)
2C4	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.05 < 1.00	Pass ✓	(Combined)
2C5	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.05 < 1.00	Pass ✓	(Combined)
2C6	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.14 < 1.00	Pass ✓	(Combined)
2C7	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.13 < 1.00	Pass ✓	(Combined)
2C8	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.05 < 1.00	Pass ✓	(Combined)
1C8	1	☑	UB 250x250x67	S355	Class1	48 < 200	0.01 < 1.00	Pass ✓	(Combined)
1C9	1	☑	UB 250x250x67	S355	Class1	48 < 200	0.03 < 1.00	Pass ✓	(Combined)
1C10	1	☑	UB 250x250x67	S355	Class1	48 < 200	0.03 < 1.00	Pass ✓	(Combined)

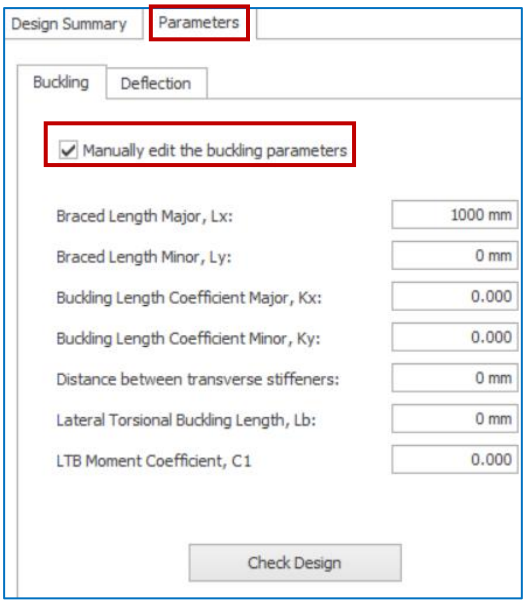
Total number of members: 11

- ***Double-click** on any column to review the detailed design checks.*



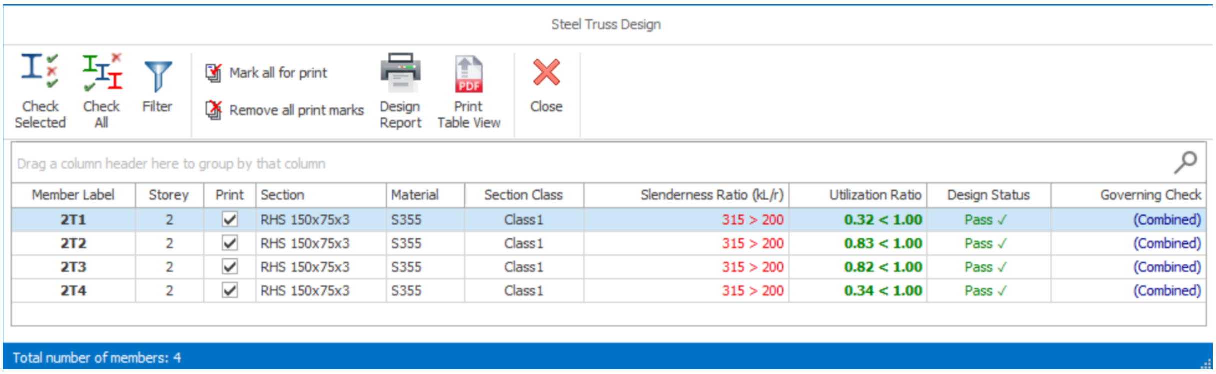
- ➢ *Click on the **triangle*** ▶ *next to the title check to drill down to more details.*
- ➢ ***Check Design** → Re-check the design of this column.*
- ➢ ***Change Section** → Allow you to pick another section → New section will be checked automatically for pass/fail.*
- ➢ ***Design Report** → Prepare the design report.*
- ➢ ***Show Design Stations** → List out all the design forces of all load combinations.*
- ➢ ***Show Diagrams** → Show design force diagrams for load cases, load combinations & envelope.*
- ➢ ***Parameters Tab > Buckling** → Input Braced length, Buckling Length Coefficient, Lateral Torsional Buckling length.*
- ➢ ***Parameters Tab > Deflection** → Manually edit deflection limits.*

Effective lengths are automatically detected & calculated by the program based on restraints provided by connected members. You can change them if you wish.



- ➢ *Go to the **Parameter** tab.*
- ➢ *Tick **Manually edit buckling parameters.***
- ➢ *Input various **Braced Length***
  - ❖ '0' (Zero) means auto-calculate values. Hence, only input values to over-write them.
  - ❖ It is highly recommended you review the auto-calculated values and amend to suit your design assumptions.
- ➢ *Go back to **Design Summary** to check the design.*
  - ❖ Deflection limits can be modified using similar steps.

- ➢ *Pick **OK** to close the column design dialog.*
- ➢ *Pick **Trusses** to design the trusses.*

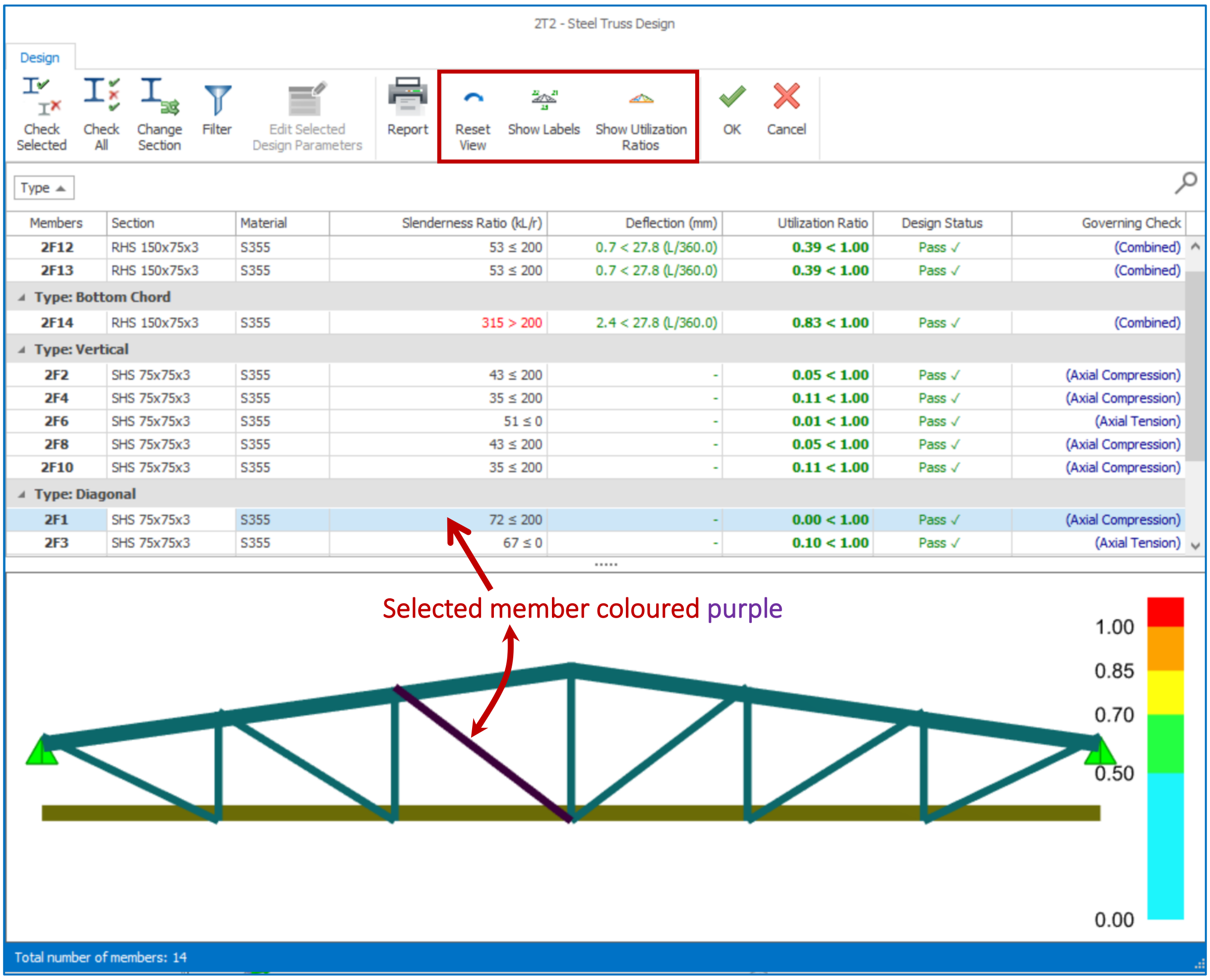


Member Label	Storey	Print	Section	Material	Section Class	Slenderness Ratio (kL/r)	Utilization Ratio	Design Status	Governing Check
2T1	2	✓	RHS 150x75x3	S355	Class1	315 > 200	0.32 < 1.00	Pass ✓	(Combined)
2T2	2	✓	RHS 150x75x3	S355	Class1	315 > 200	0.83 < 1.00	Pass ✓	(Combined)
2T3	2	✓	RHS 150x75x3	S355	Class1	315 > 200	0.82 < 1.00	Pass ✓	(Combined)
2T4	2	✓	RHS 150x75x3	S355	Class1	315 > 200	0.34 < 1.00	Pass ✓	(Combined)

The design status shows pass as Utilization Ratio is less than 1.

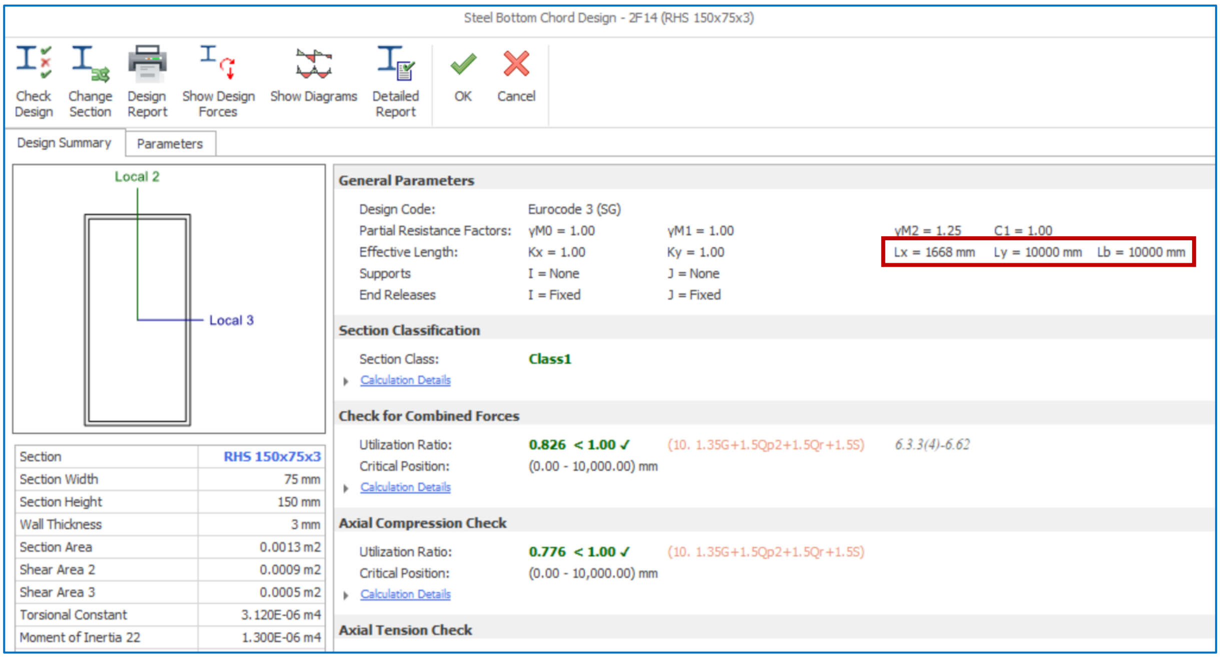
Slenderness Ratio check is a minor check to ensure sections are not too slender. It is up to the designer whether to increase the member size to pass this check.

- ➢ *Double-click on the truss **2T2** to access the detailed steel truss design.*



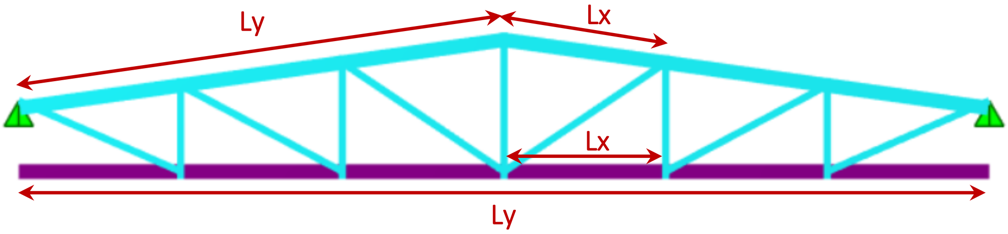
Members	Section	Material	Slenderness Ratio (kL/r)	Deflection (mm)	Utilization Ratio	Design Status	Governing Check
2F12	RHS 150x75x3	S355	53 ≤ 200	0.7 < 27.8 (L/360.0)	0.39 < 1.00	Pass ✓	(Combined)
2F13	RHS 150x75x3	S355	53 ≤ 200	0.7 < 27.8 (L/360.0)	0.39 < 1.00	Pass ✓	(Combined)
Type: Bottom Chord							
2F14	RHS 150x75x3	S355	315 > 200	2.4 < 27.8 (L/360.0)	0.83 < 1.00	Pass ✓	(Combined)
Type: Vertical							
2F2	SHS 75x75x3	S355	43 ≤ 200	-	0.05 < 1.00	Pass ✓	(Axial Compression)
2F4	SHS 75x75x3	S355	35 ≤ 200	-	0.11 < 1.00	Pass ✓	(Axial Compression)
2F6	SHS 75x75x3	S355	51 ≤ 0	-	0.01 < 1.00	Pass ✓	(Axial Tension)
2F8	SHS 75x75x3	S355	43 ≤ 200	-	0.05 < 1.00	Pass ✓	(Axial Compression)
2F10	SHS 75x75x3	S355	35 ≤ 200	-	0.11 < 1.00	Pass ✓	(Axial Compression)
Type: Diagonal							
2F1	SHS 75x75x3	S355	72 ≤ 200	-	0.00 < 1.00	Pass ✓	(Axial Compression)
2F3	SHS 75x75x3	S355	67 ≤ 0	-	0.10 < 1.00	Pass ✓	(Axial Tension)

- ➢ *Reset View → Reset the truss diagram to the original orientation.*
- ➢ *Show Label → Show the truss member label.*
- ➢ *Show Utilization Ratios → Show Utilization ratio for each member.*
- ❖ **Pass** / **Failed** members are shown clearly in Green / Red
- ❖ Slenderness Ratio failure will not result in FAIL status as it is not a critical criterion (more a warning)
- ❖ Fail members (UR > 1) will be coloured red in the truss diagram.
- ❖ The selected member in the table will be coloured purple in the truss diagram.
- ❖ The truss diagram at the bottom can be zoomed (mouse wheel) & rotated (right-click & drag).
- ➢ ***Double-click** on the bottom chord member → This will bring up detailed member design.*



In steel design, it is important to check the effective lengths detected as it will directly impact the Axial Compression & Combined Forces Check. In the results, **General Parameter**, notice the bottom chord, Lx = 1668 mm & Ly = 10,000mm.

- **Lx** is the **Major Braced Length** = truss panel length as there are vertical & diagonal members.
- **Ly & Lb** is the **Minor Braced Length** is the entire length of the bottom chord between column supports.



If there is failure axial compression check, you may increase the section size or thickness to solve the failure.

You may also reduce the minor bracing lengths bottom chord, by adding bracing members at intermediate locations joining bottom chords of the adjacent truss.

Alternatively, go to the **Parameter** > **Buckling** tab to manually edit the bracing lengths.

- ➢ If there is a bracing member at the mid-span of the bottom chord, you can change the braced length Minor, **Ly** & Lateral Torsional Buckling Length, **Lb** to 5000 mm.
- ➢ Click on **Design Summary,** and the member will automatically be re-checked.

The assumption of **top chord** is:

- **Lx** is the **Major Braced Length** is auto determined = truss panel length
- **Ly** is the **Minor Braced Length** is auto determined = purlin spacing

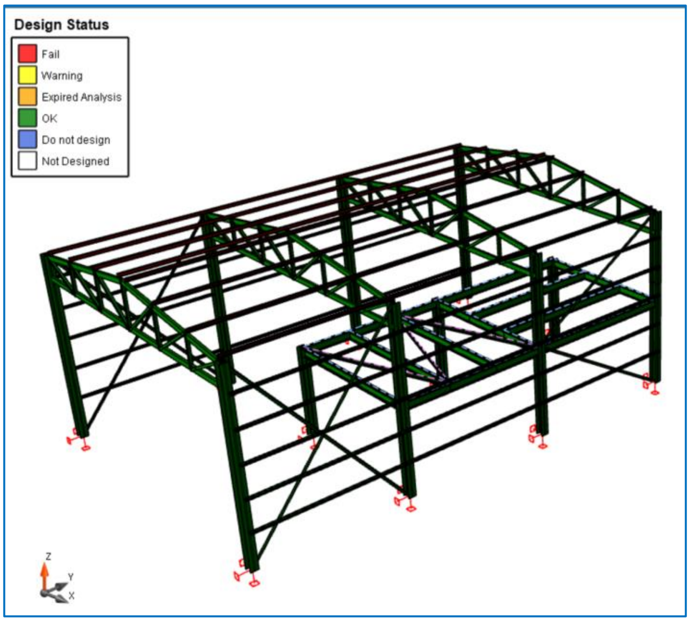
You may want to check the other frame member design. The design interface is similar to the column design.

Design Status & Design Utilization

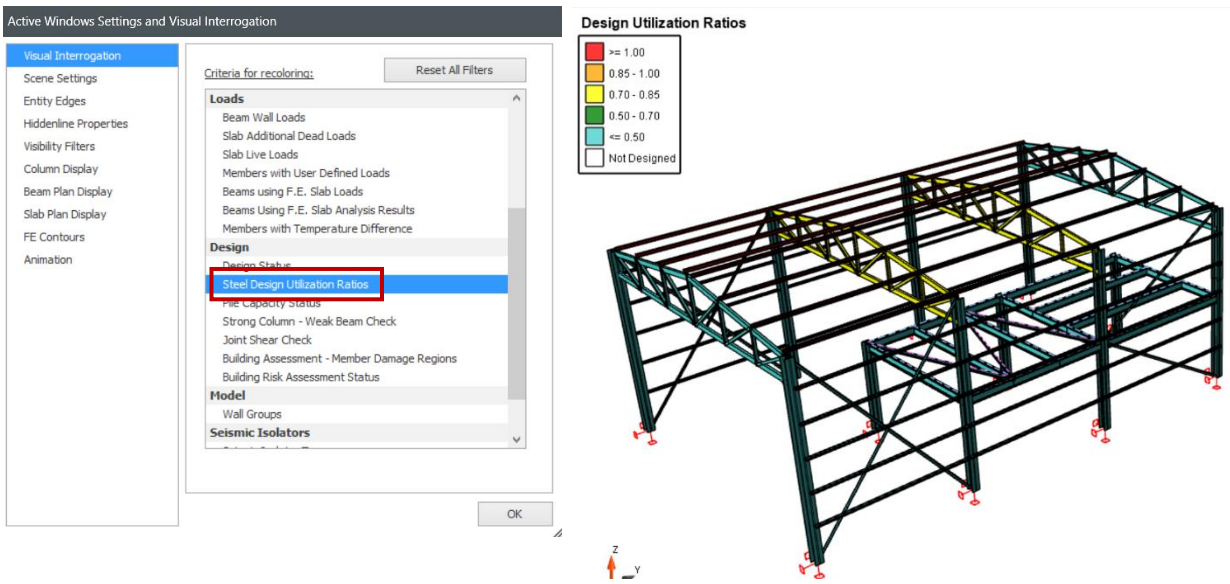
We can view **Steel Design Status** and **Steel Design Utilization** to review pass/failure status & design efficiency quickly.

The design status can be displayed graphically in the plan or 3D window.

- *Click on the **3D view** to make it active.*
- *Go to the **Review** Tab and pick **Design Status** → **OK.***



- *Click **Visual Interrogation** & choose **Steel Design Utilization Ratios** → **OK***



Thank You

Thank you for choosing the ProtaStructure Suite product family.

Our top priority is to make your experience excellent with our software technology solutions.

Should you have any technical support requests or questions, please do not hesitate to contact us at all times through globalsupport@protasoftware.com and asiasupport@protasoftware.com

Together with our responsive technical support team, our dedicated online support center is available to help you get the most out of Prota's technology solutions.

The Prota Team