PROTA

ProtaStructure® 2021

Quick Start Guide

For support & training please contact

Support : asiasupport@protasoftware.com
Training : asiasales@protasoftware.com

www.protasoftware.com

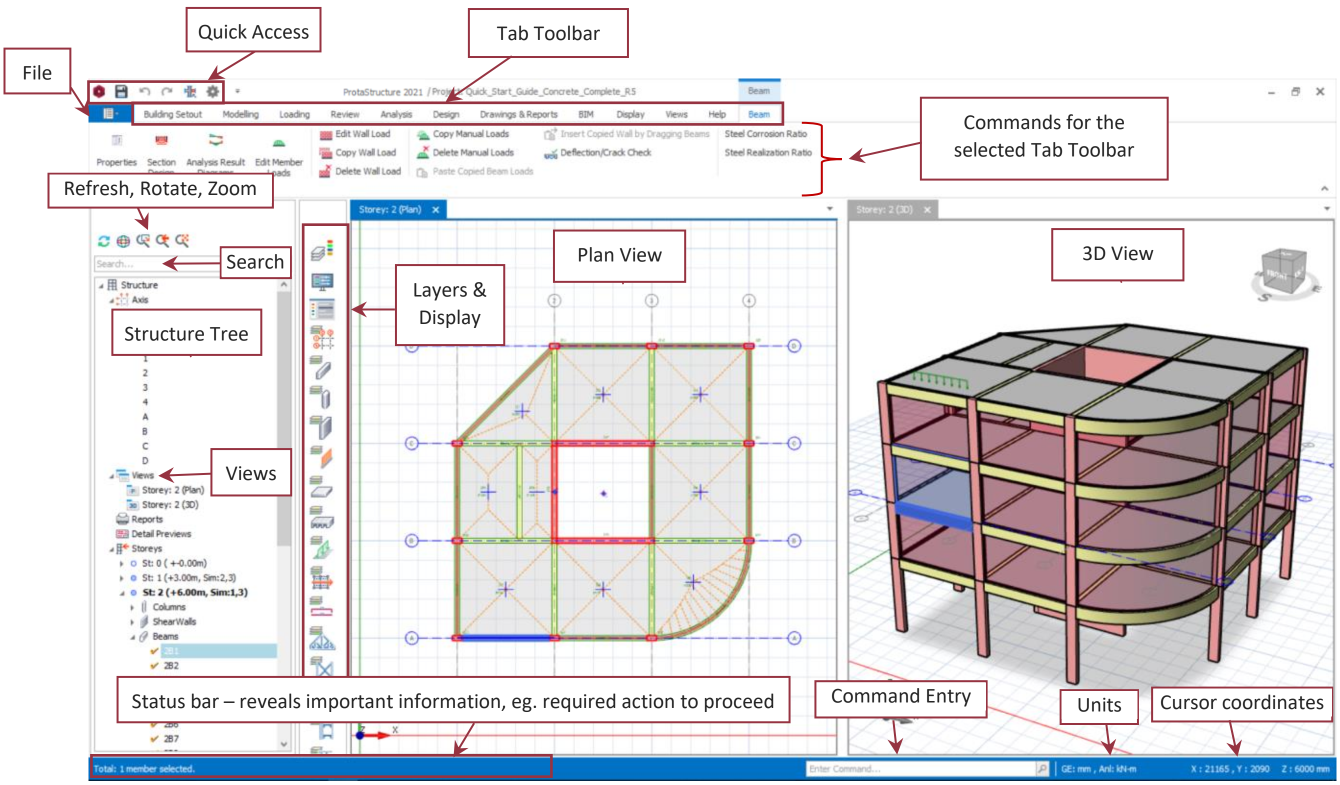
Table of Contents

1. Introduction

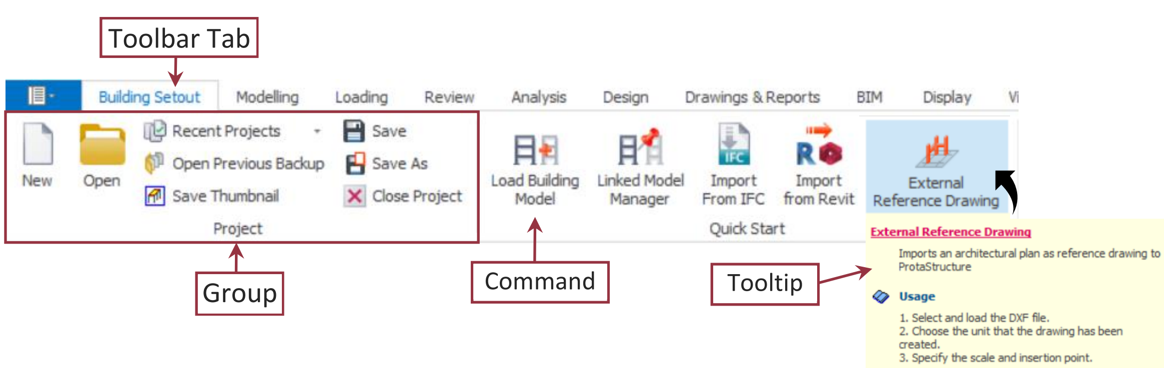
Thank you for choosing **Prota**Structure. This Quick Start Guide aims to get you up and running quickly. You should be able to model, analyze and review the results for this simple model in around 1 hour.

2. User Interface

ProtaStructure 2021 welcomes you with a modern & efficient user interface designed from scratch for ease of use. The various components of **Prota**Structure user interface are as shown below:



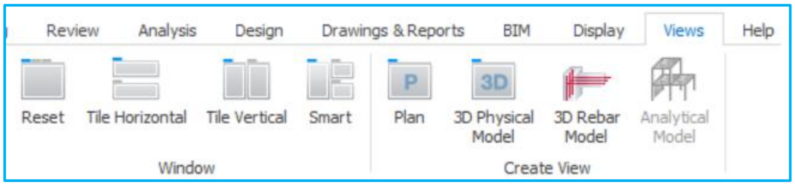
Ribbon User Interface



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 a command will reveal the Tooltip explaining how to use the function.

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

Views

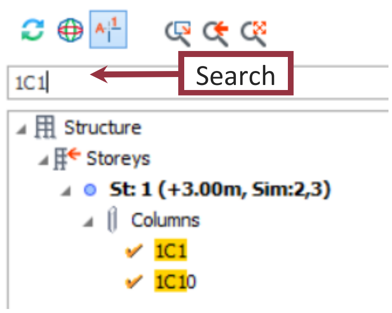


Create as many views as you need, such as plan, 3D physical Model, Analytical Model & 3D rebars. Views can be organized using smart window layout options.

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 screenscape, just move one of the views to another screen.

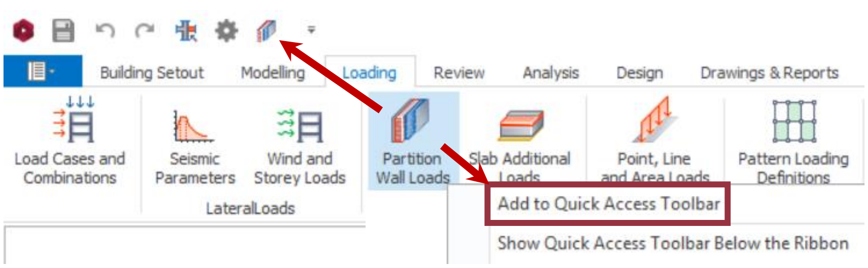
Structure Tree Search Option

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.

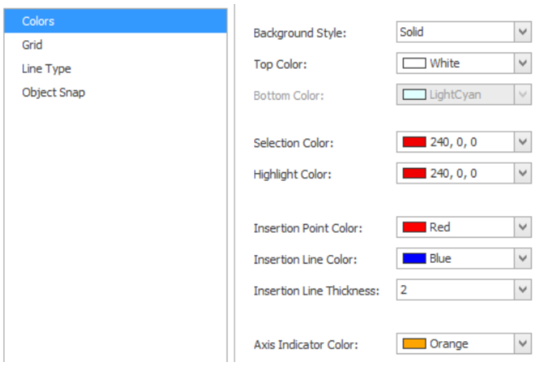


Quick Access Toolbar

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***.

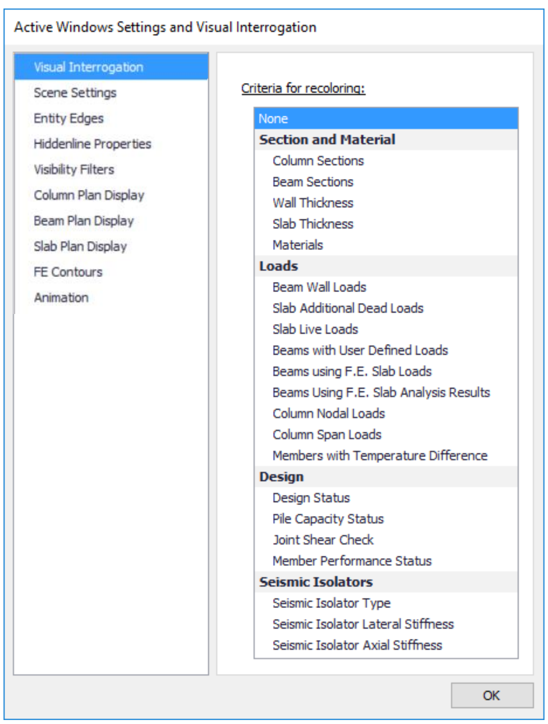


Display Settings



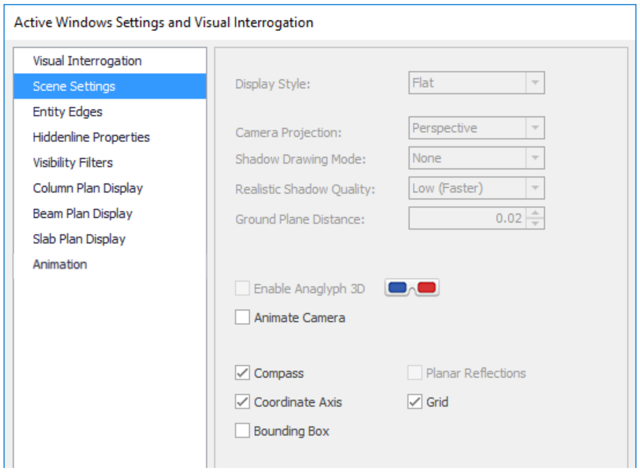
Colors	Choose the color of background and various active modelling objects
Grid	Sets the spacing of the guiding grid system to allow ease of modelling as objects can snap to intersection of the grids
Line Type	Sets the Line Type Scale
Object Snap	Choose the various snap options such as Start/End/Corner, Perpendicular or Orthogonal Grid, etc.

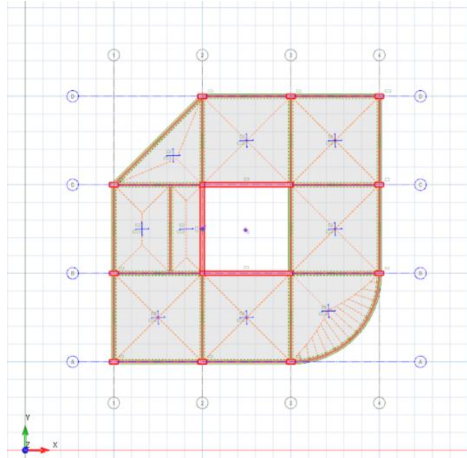
Active Window Settings & Visual Interrogation



Visual Interrogation	Color-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 result of the model.
Scene Settings	Control the graphics of the views and allows you to switch on/off the guiding Grids & Coordinate Axis, etc.
Visibility Filters	Filter to the specific storey, axis or member type.
Column Plan Display	Allows 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	allows the dead and live load values to be shown on the plan view.
FE Contours	Shows the FE contours exported from the FE Analysis Post-Processing
Animation	Animate by spinning the model in 3D view

TIP : Each modelling window can have separate view settings. For example, you might want to color the slab live loads on the plan view & at the same time color design status in the 3D view.





TIP: In the **Scene Settings** tab, **Grid** allows you to switch on/off the grey rectangular grids in the background. **Coordinate Axis** allows you to switch on/off the coordinate symbol.

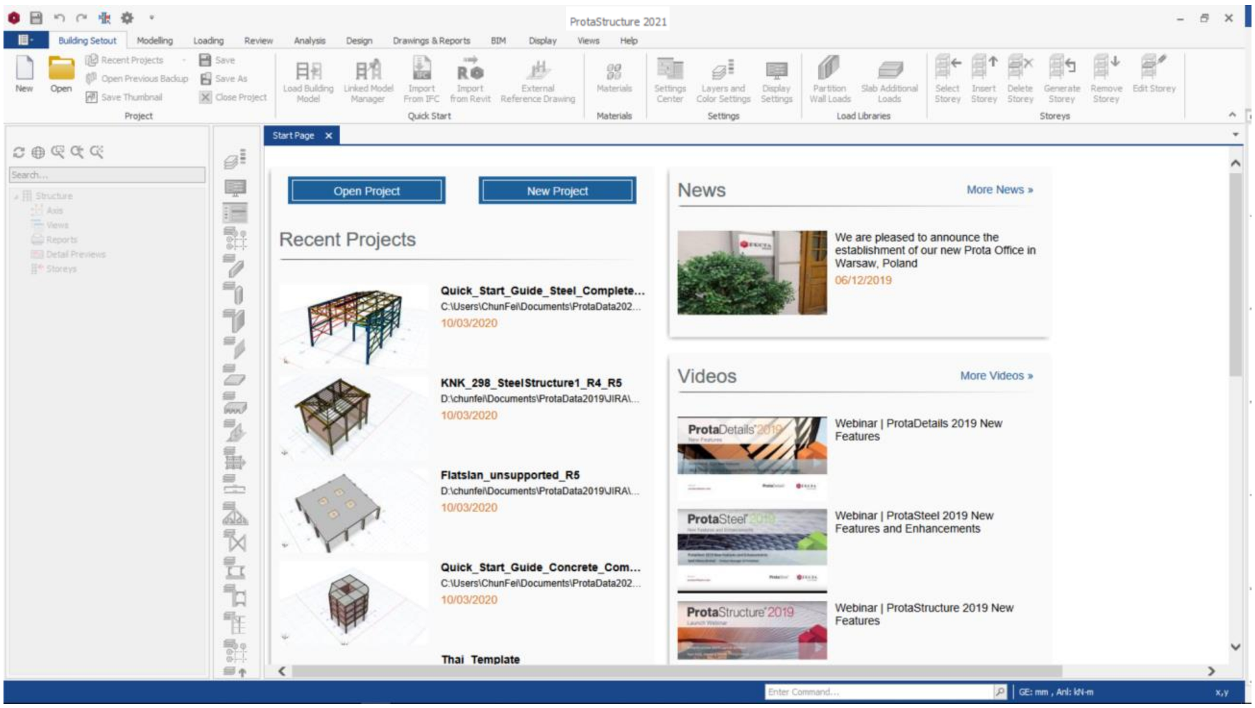
Layer Tool bars (under Display Tab)

Layer and color settings		switch on/off layers and modify the name, color, 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
Wall Layer Group		switch on/off the wall layer
Partition Wall Layer		switch on/off the partition / brick wall layer
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
Reinforcement Layer		switch on/off the slab reinforcement layer
Steel Member Layer		Switch on/off steel members such as truss, brace, etc
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 the all the texts
Footing Layer Group		switch on/off the footing layer

3. Start Page

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



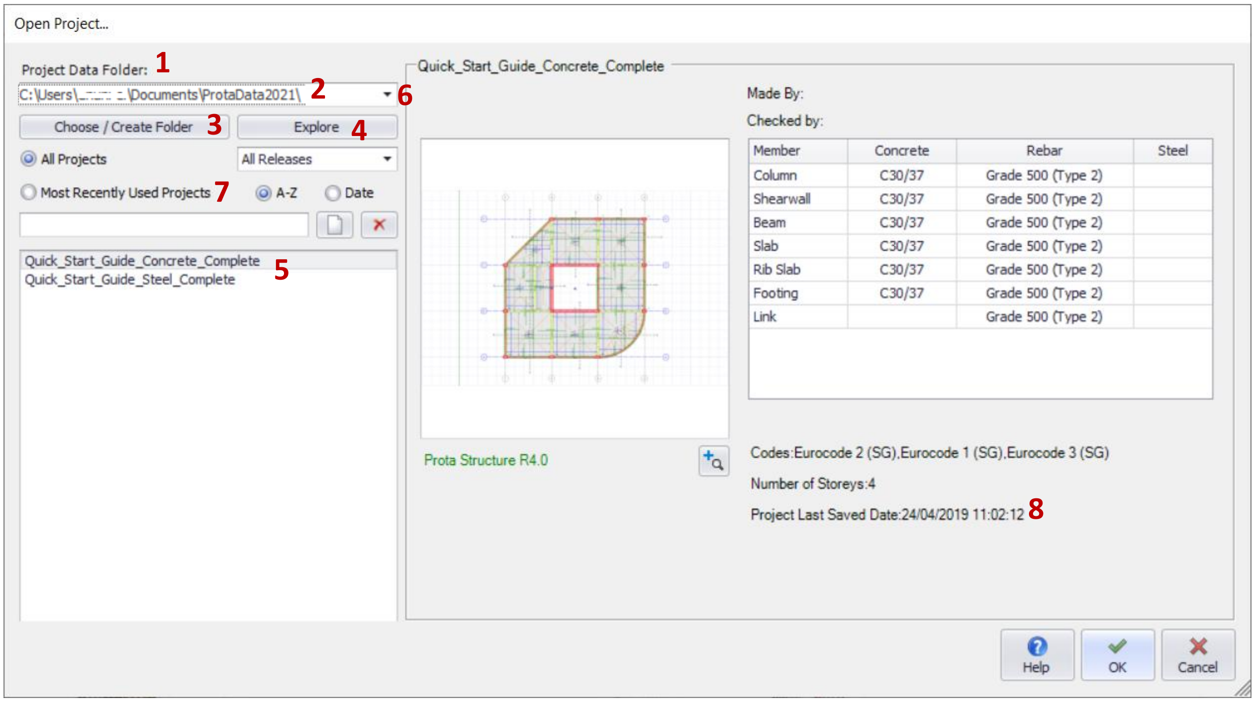
The Start Page contains the following functions:

- Open existing project
- Start a new project
- Read Prota News and watch Prota Videos
- Access Prota resources such as Help Center, What' New document & Quick Start Guide
- Read important Notification and download new Software Update
- View subscription status

4. 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 :

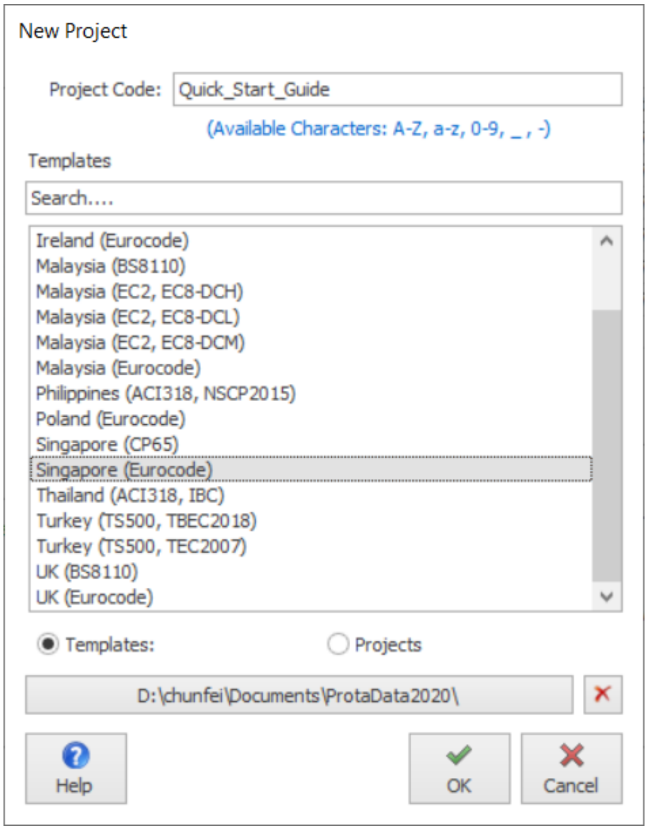


1. The **Project Data Folder** is the main folder where the project models are stored.
   The project model consists of several files stored in a single folder called **Project folder**. This **Project folder** is created as a subfolder in the **Project Data Folder.** The project folder name will always be created exactly the same as **project name**.
2. By default, a **Data Folder** called **ProtaData2021** will be installed under **My Documents**
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 it the project list
   After selecting the project, a preview of the project will be shown in the right-hand side
   Double-clicking on the project name or click **OK** will open the project.
6. Click on the **dropdown** list just under the above to expose most recent accessed Data Folders.
7. Tick ***Most Recently Used Projects*** to quickly show the list of recent project opened.
8. You can also read ***Project Last Saved Date*** to quickly locate the saved project.

Tip : Previous versions of ProtaStructure model can be opened directly in PS 2021:

➢ 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.
➢ Please note that newer projects can't be opened in older versions of ProtaStructure (not backward compatible).

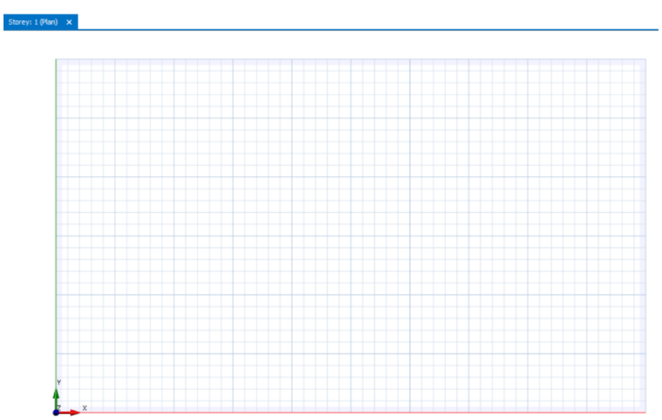
- *Click **New Project** and type the project name as shown using the '_' character for spaces.*
- *Select **Singapore (EuroCode)** template*



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**.*

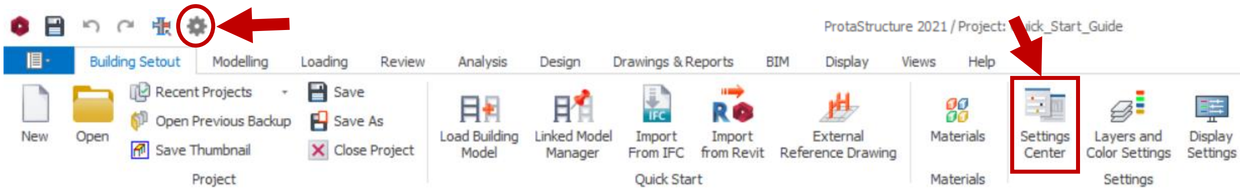
The main modelling area will now show a set of rectangular grids in the background.



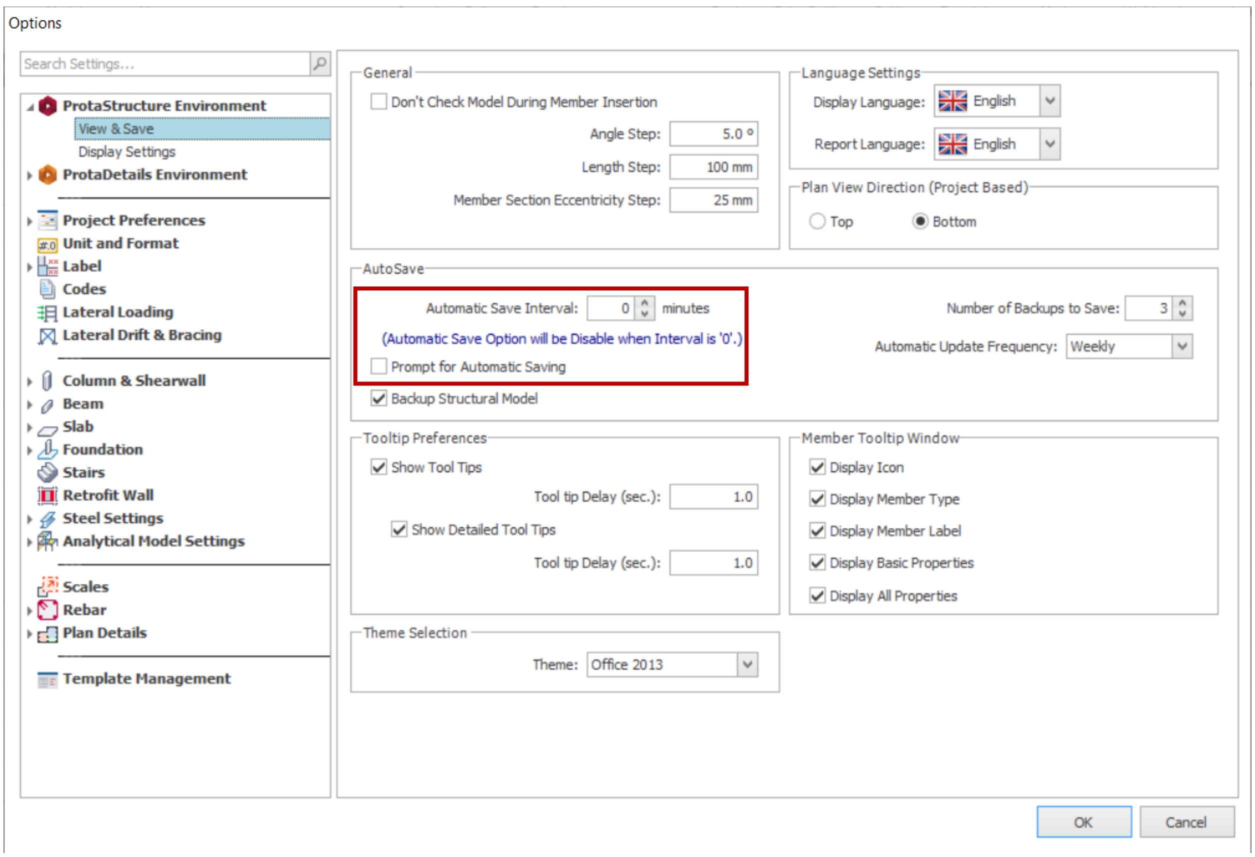
These girds spacing are by default 1m with major grid colored darker every 5m. The defaults can be changed via **Display Setting** button

5. 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 default settings of the program 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 **View & Save** dialog. By default, "0" means it's disabled.

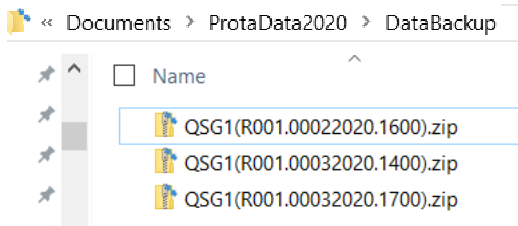
➢ *Change the **Automatic Save Interval** to **10 minutes** to ensure the model is auto-saved.*

➢ *Untick **Prompt for Automatic Saving** to set the auto-save to run silently in the background.*

Number of Backups to Save

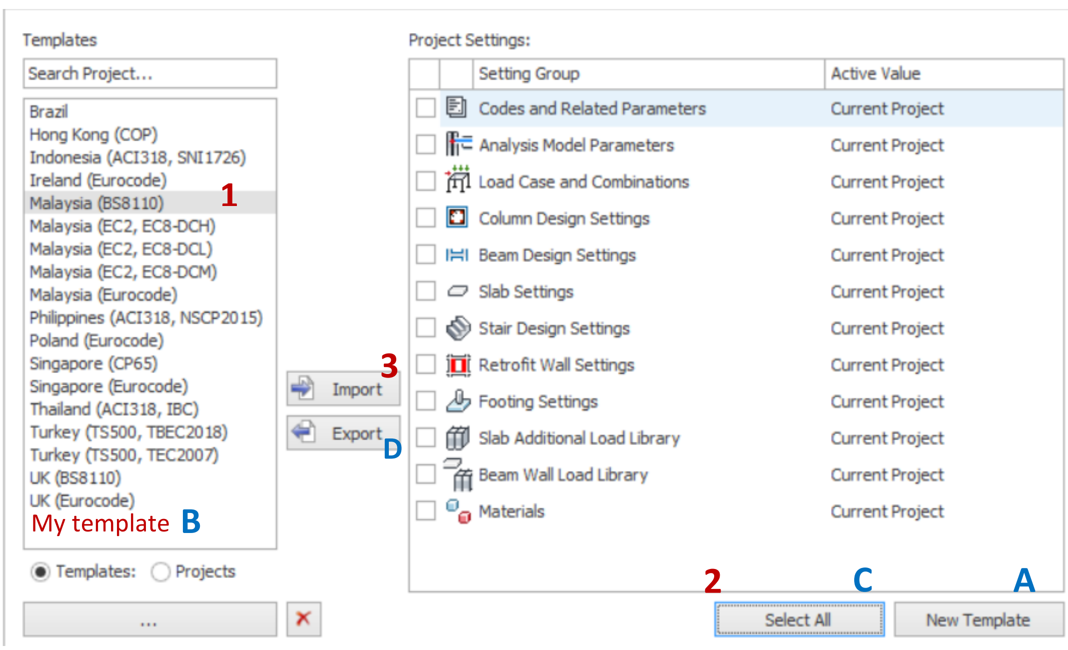
The **Number of Backups to Save** refers to the automatic & silent additional backup created every full hour. Example, the 1st backup file will be created at 9:00 am sharp, 2nd backup at 10:00 & 3rd backup at 11:00. At 12:00 pm, the backup will overwrite the 1st backup and the cycle continues.

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)



Project Template

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



Importing an existing Template (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 imported template will apply to the current project.

Exporting a new Template (refer to step **A,B,C,D** above)

You can save the settings of your current project as your own 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)**

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

6. Selection Methods

Selection is performed using the selection button in the Member toolbar. Although there is no entity to select now, the information is important 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 **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 **completely 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** which allows you to edit and perform other task related to that entity, example Properties, Delete etc.

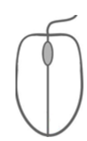
Pressing **ESC** will deselect all entities.

7. Zoom & Pan Methods

The useful functions are:

Zoom Window **CTRL+W** → Zoom into the area defined by dragging a rectangle.
Zoom Previous **CTRL+O** → Zoom to the previous view.
Zoom Extents **CTRL+E** → zooms to the selected entities. If no entities are selected, then it will zoom to show all entities.
Zoom Limits **CTRL+L** → zooms to show the limits of the grids.

You will find it easiest to use the mouse wheel to:
Zoom in → scroll your mouse wheel **up**
Zoom out → scroll mouse wheel **down**
Pan (move) → **hold down** the middle mouse wheel and **drag**



8. Modelling Axes

The very first step to build a model is to define axes. Axes intersections then becomes the nodes at which members are inserted. Hence, it's critical that axes are created correctly. There are 3 ways to model axes:

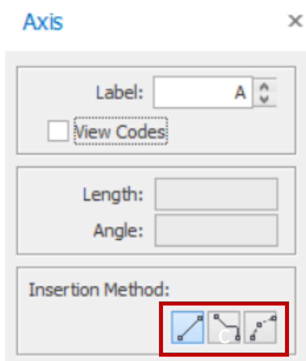
1. **Axis toolbar** to build axes individually
2. **External Reference Drawing** to import all axes from dxf drawing file.
3. **Orthogonal Axis Generator** to build a system of axes quickly

9. Axis / Grid Tool

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

- *Click on **Grid** in the **Modelling** tab.*

There are several insertion methods for axis:

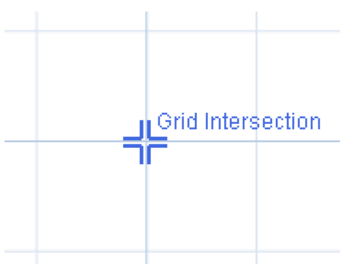


Single Segment allows you to create a straight-line axis by clicking on the start and end point of the axis.

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

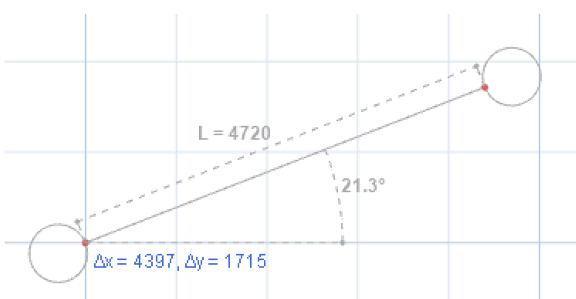
Curve Axis can be created by specifying the radius

- *Ensure **Single Segment** insertion is selected*
- *Put the mouse cursor on any of the grey grid intersection & the "Grid Intersection" snap is shown*



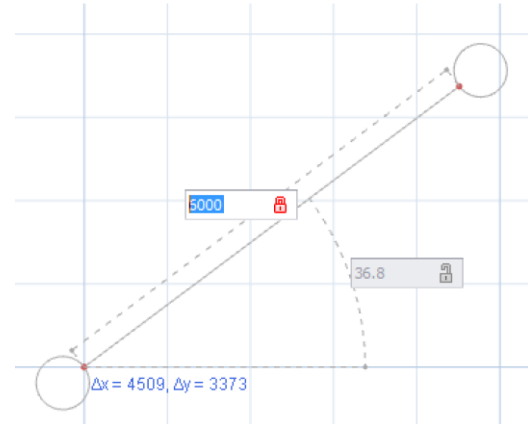
- ***Left click** (& release) to confirm the start point*

A "rubberband" will appear as you move your mouse cursor to specify the end point

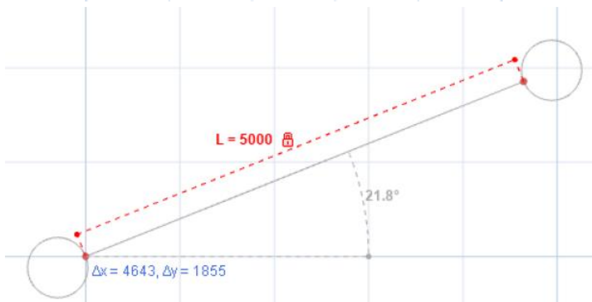


During the rubberband operation, the length (L) and local angle will be displayed. In addition, the relative distance **Δx** & **Δy** with respect to the local UCS will also be shown.

- *Press **F2** to enable input of the length (L) of axis via Dynamic Input System (DIS)*
- *Press **TAB** to cycle to the next input of angle.*

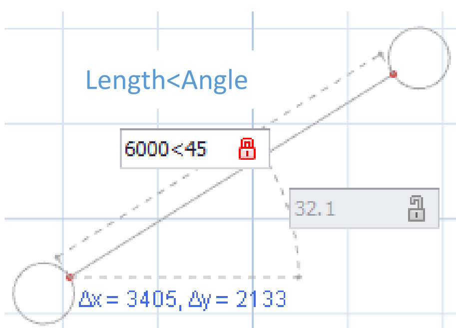


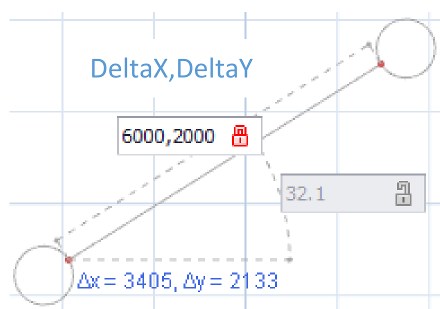
- If a value is entered in the textbox, the related parameter will be locked. You can unlock by pressing ***ESC*** or pressing the lock icon at the right of the text box.
- If a textbox is locked and ***F2*** or ***ENTER*** is pressed then DIS will be deactivated. However, rubberband operation now continues 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 of the text boxes are locked then pressing **ENTER** will accept the operation and candidate point will automatically be picked.
- In either of the text boxes, you can use shortcut notation **Length<Angle** or **DeltaX**, **DeltaY** (without the need to switch by TAB.)





- ➢ *After specifying the length and/or angle, pressing* ***ENTER*** *or left-click will accept the end point*
- ➢ *Try inserting a multi-segment axis*  *by defining points continuously*

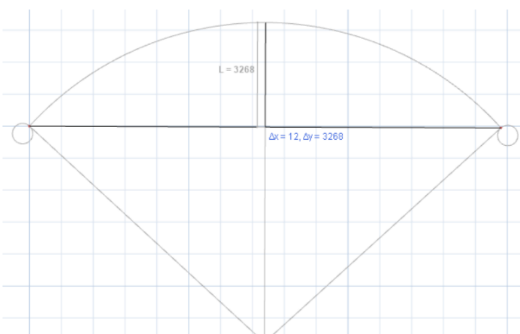
- Use **F2**, **TAB**, **ENTER** to define destination points as mentioned earlier
- Pressing **ESC** will cause a smart roll-back. If parameters are locked, then first the DIS is unlocked then operation is cancelled. If a pick-continuous command is active, **ESC** will unpick points.

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

- When the axis property dialog shown, you are in axes creation mode. **Close** it if you would like to end creation of axis. This applies to all member properties dialog.

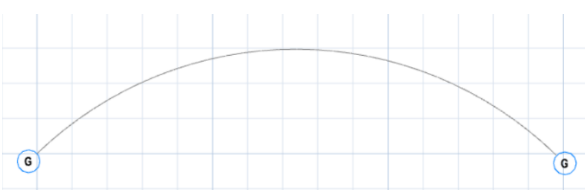
➢ *Insert a curve axis by clicking on the curve axis icon*

➢ *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 press **F2** to specify the offset length manually.

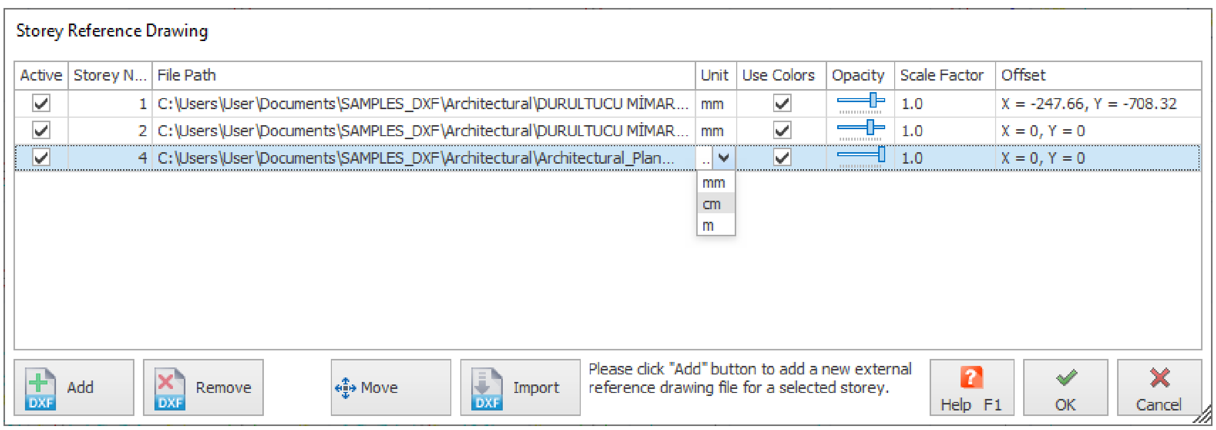
10. External Reference Drawing

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

ProtaStructure allows you to load external DXF drawings and show them as ghost reference layers under your model. *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 can be restored; the next time 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 unit column of the table right after the loading of the file. The drawing will be scaled immediately.

Storey No

The imported file will be assigned to the active story initially. You can use the '*Story No'* column to assign it to any other story. Only one drawing can be attached to a specific story.

Use Colors

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

Opacity

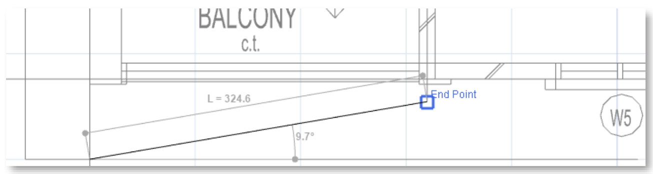
This parameter controls the opacity of the colors. This field applies only if drawing colors are selected to be used.

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 at the '*Offset'* column of the table.



Import

The selected reference drawing can be imported into the assigned story. '*Import DXF'* interface will be loaded with pre-defined Storey and file unit values in this case. The drawing can be imported on top of the existing model in this mode.

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

11. Import DXF

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

Three options are available for DXF import into **ProtaStructure** which can be accessed from the *QuickStart* Menu, *Import DXF* button :

1. *Floor Plan (a pre-defined structural floor-plan for example)*
2. *3D Physical Model*
3. *3D Analytical Model*

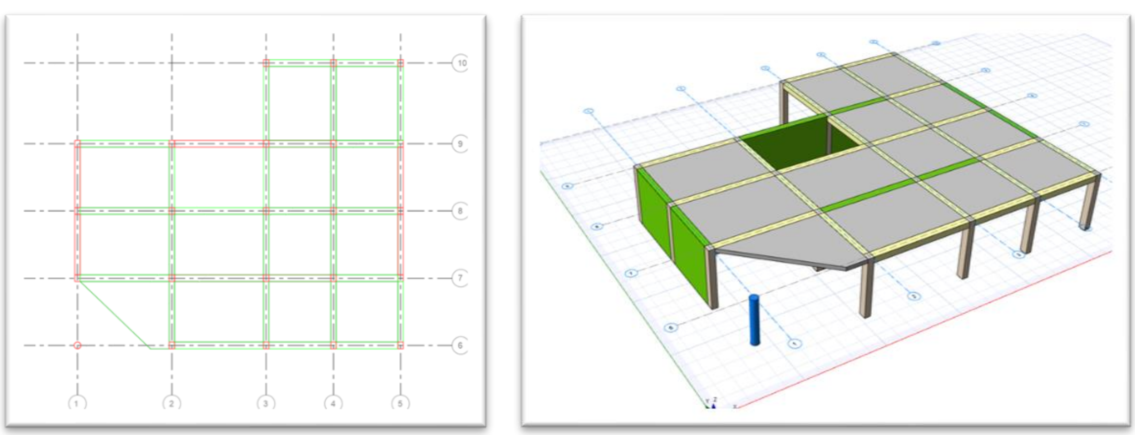
If you are importing a drawing which you also wish to use overlay against your model for coordination purpose, please use the *'External Reference Drawing'* option.

Please note that member definitions usually vary significantly from one software to another.

ProtaStructure uses grid systems to be able to create a high-precision structural model out of physical members for analysis, design, and detailing purposes. Since DXF is not a data-rich format, its not always possible to capture all the modeling information such as connectivity and grid relations, sections, and eccentricities. We generate these from the primitive data read from the file. However, you may have to edit and refine information further once you have it in **ProtaStructure**.

Importing Floor Plans

Floor plans can be imported and transformed into 3D modeling elements.



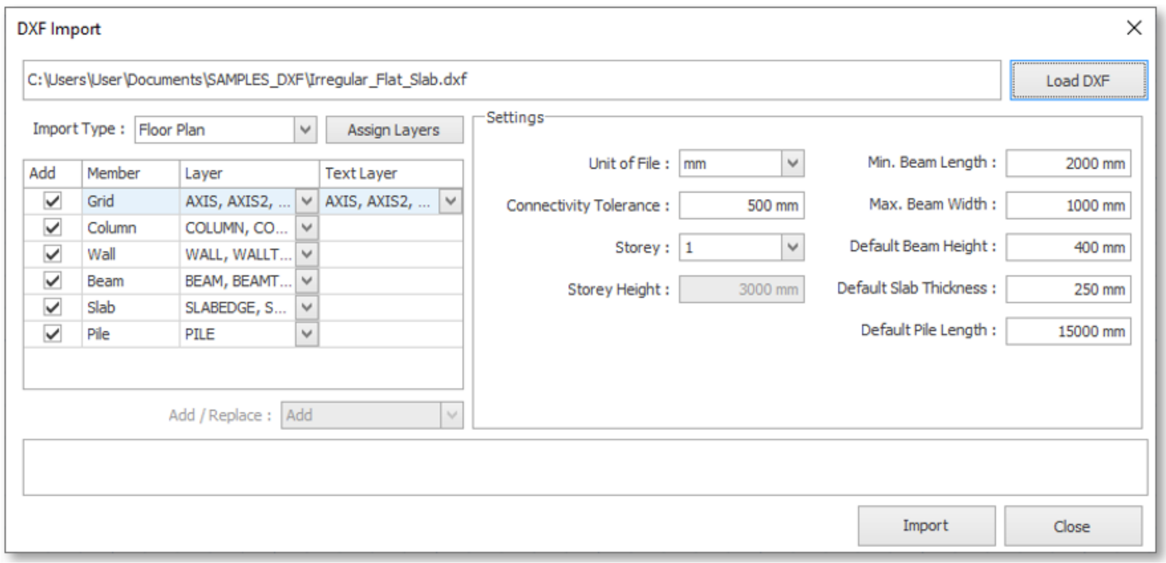
DXF > ProtaStructure

DXF import module reads primitive CAD objects and converts them to structural members as grids, columns, beam, shearwalls, slabs, and piles. The below table shows supported CAD entities for 2D drawings.

Structural Member	Supported CAD Entity Type
Grid	Polyline, line
Column	Polyline, line, circle, block
Shearwall	Polyline, line
Beam	Polyline, line
Slab	Polyline
Pile	Polyline, circle

Tip: Grid definitions are not a MUST. If they are not provided, they will be generated automatically, considering all the imported structural members.

Please select the DXF file using the "**Load DXF**" button. The file will be read immediately.



Layers

Each member type should be defined on different layers in the DXF file.

The layers in the file are scanned for keywords to spot possible layers that may be used for entities defining structural members in **ProtaStructure**.

The layer detection is done automatically upon the DXF file load.

If layers for a specific member type are not recognized, please use the dropdown list to assign the layers manually.

You can make multiple layer selection for each member type. At least one layer should be selected to enable that member type for import.

You can use the ***"Assign Layers"*** button anytime to reset all layers to the ones found by the program.

Text Layer is only necessary to relate grid labels with grids.

Add/Replace

To prevent discrepancies, Add/Replace functionality is disabled for *Floor Plan Import.* Members will be added to the existing model. Existing members in the model (if any) will not be removed.

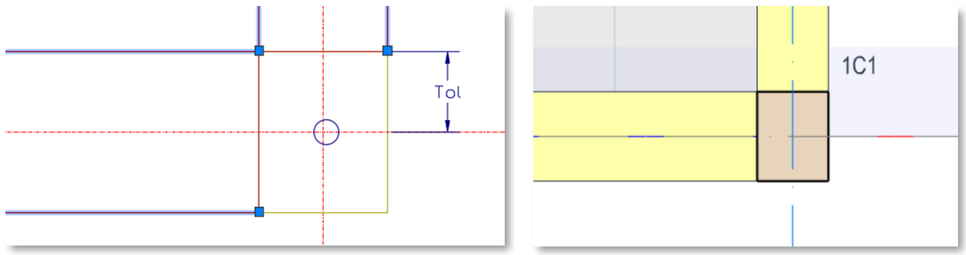
Unit of File

Please select the unit of the DXF file you load.

Connectivity Tolerance

This tolerance helps **ProtaStructure** to correctly assign grids to shearwalls, beams, and columns, even if nodes of the grids are outside the borders of those members' geometrical definition.

Connectivity tolerance should be larger than the distance measured from the beam or shearwall boundary to the grid intersection.



DXF > ProtaStructure

Storey Information

Select the Storey on which the model is to be generated. The current Storey will be selected upon the form opening.

If you are going to create a full model out of 2D floor plans, you'll need to create the stories first. You may also find it easier to use the External Reference Drawing import command for this. It helps to manage all drawings from a single interface and links to the DXF import interface.

Min Beam Length

The length of a DXF line entity should be higher than this value to be eligible to form a beam.

Max.Beam Width

The import module checks all parallel lines against possible beam formation. Distance between two parallel lines should be less than this value to form a beam.

Default Beam Height

Beam sections will be generated using the measured beam width and this value.

Default Slab Thickness

This parameter can be used to assign the default thickness for the imported slabs.

Default Pile Length

This parameter can be used to assign default length for the imported piles.

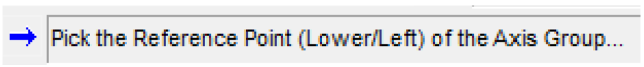
For *3D Physical Model & 3D Analytical Model* import, kindly refer to Prota Help Center.

12. Orthogonal Axis Generator

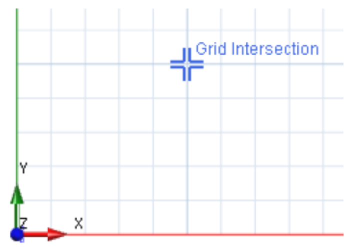
Let us now start the new model with creation of the required model axes. We will define multiple axes in one go using the **Orthogonal Axis Generator**.

- *__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.

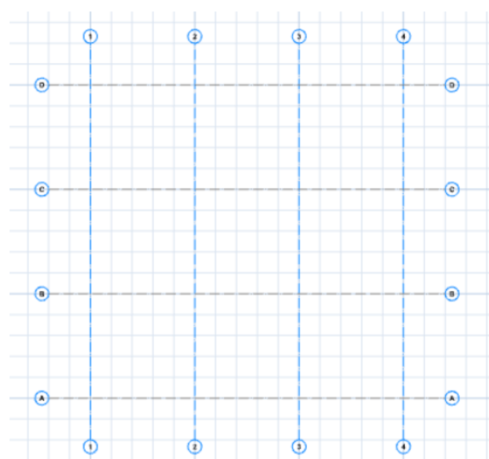


- *Pick the __intersection__ of the major grid near the origin (as shown below)*



The Orthogonal Axis Generator will appear. **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).

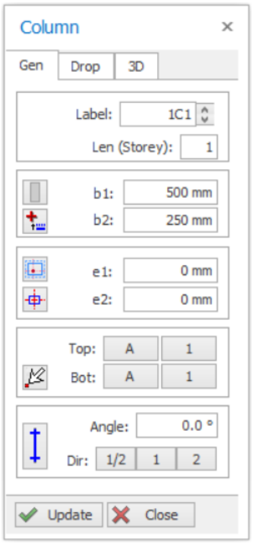
- *Accept all the __default__ values in the inputs and click __OK__*

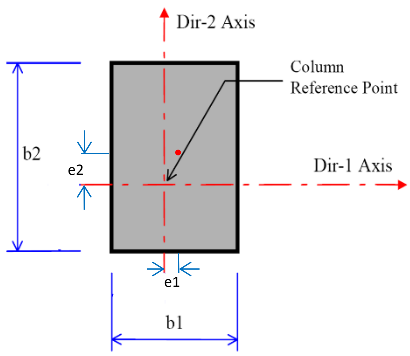


For horizontal and vertical axes with spacing of 5m will be created.

13. Columns Creation

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

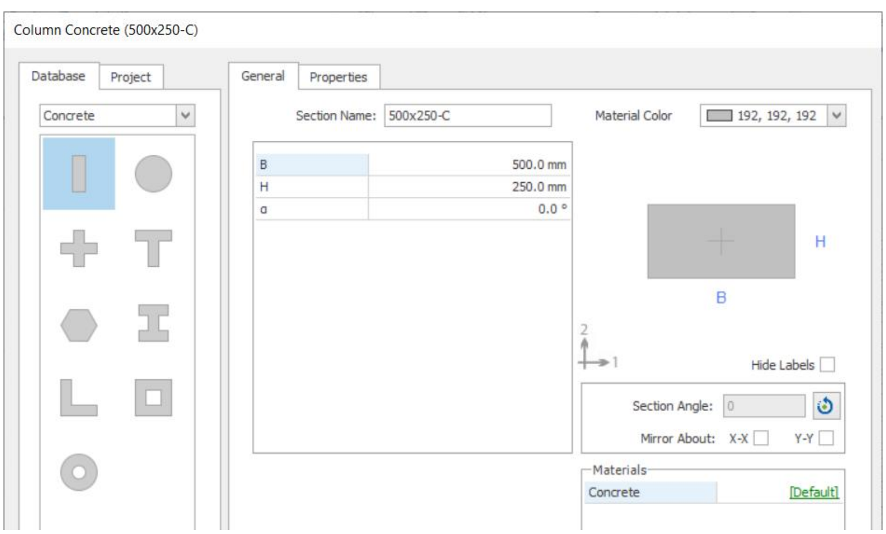




e1 & e2 is measure from the centroid of the column

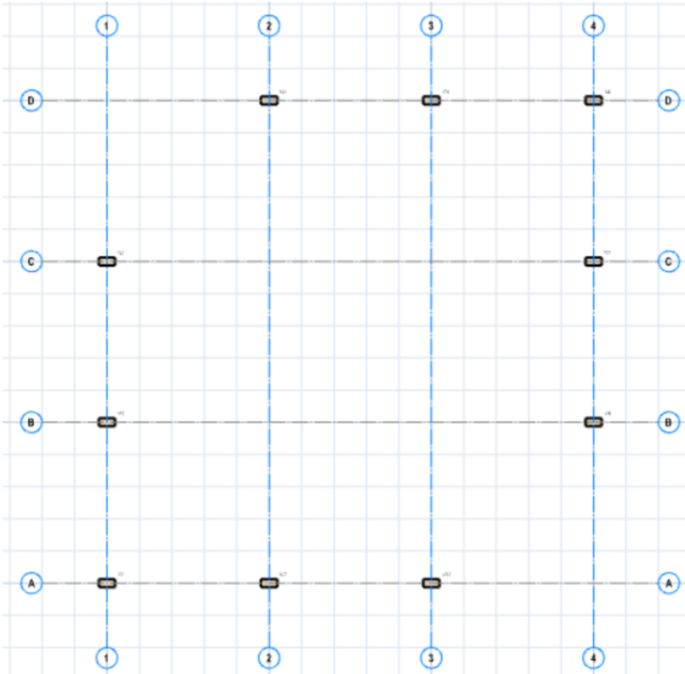
Section Manager icon allows you to access other sections types such as **circular**, "**L**", "**T**".

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 inserting by **drag a box** around the intersection of axes*

Using the 2 methods, create **10 nos.** of columns at position shown below.



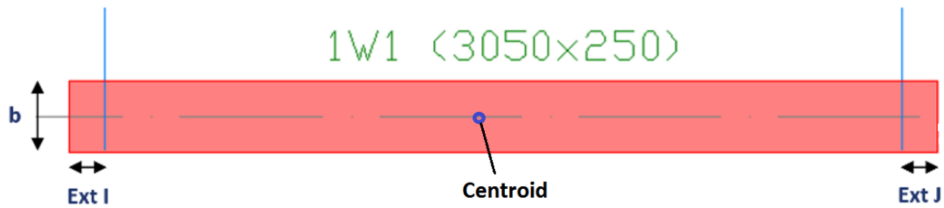
- ***Close** the column properties.*

TIP : Always close member properties when you finish member creation.

14. Walls Creation

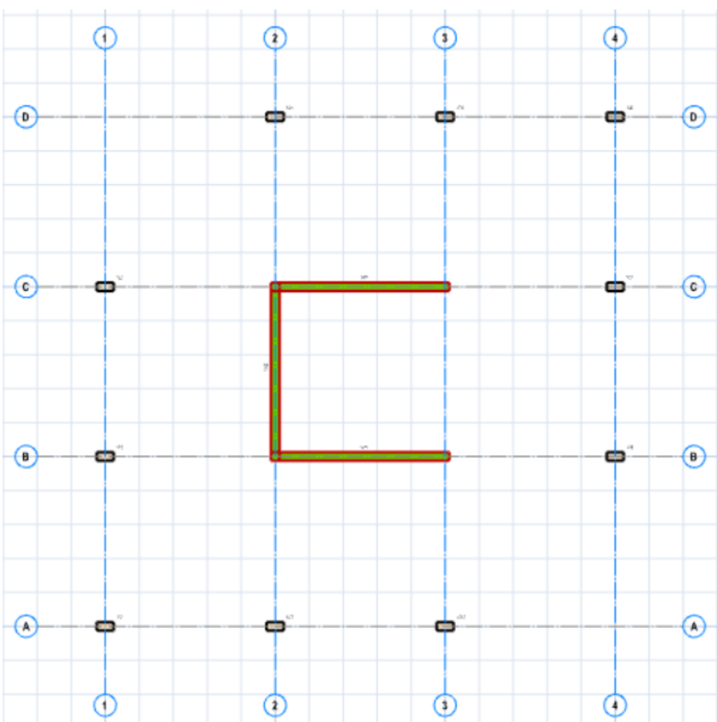
- *Click on the **Wall** icon*  *& the wall properties will appear*
- *Use the default wall thickness **b = 250 mm** & **e = 0 mm***

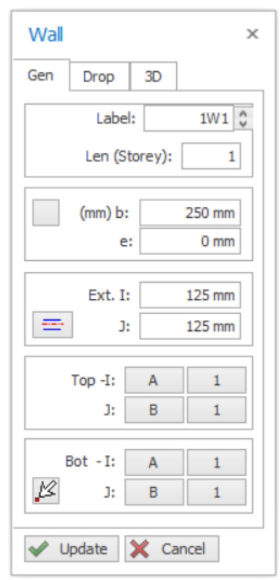
The parameters are explained in the diagram below



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

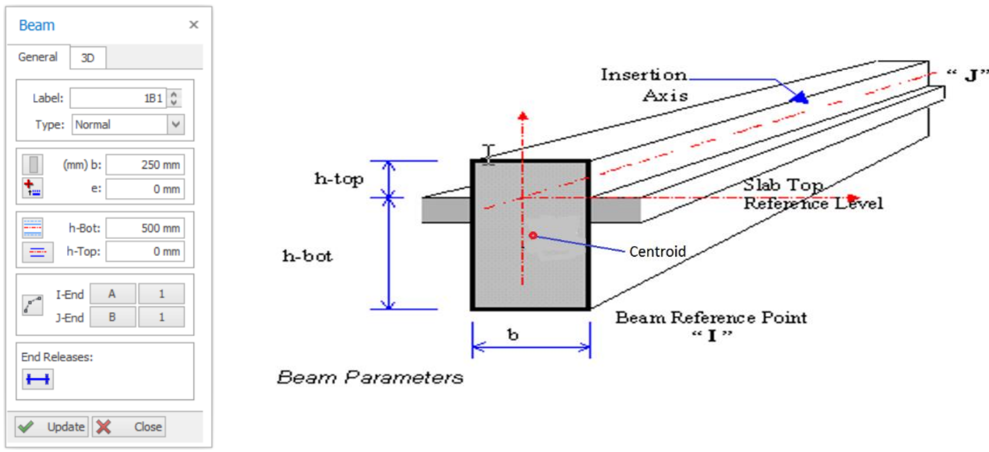
- *Insert the 3 nos. of wall by simply clicking on the start and then end of the wall.*



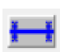


15. Beams Creation

➢ *Click on the **Beam** icon*  *& the beam properties will appear*



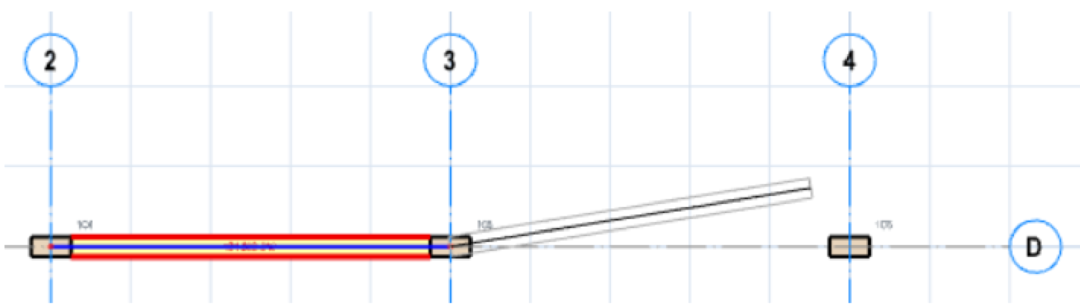
e is measured from the centerline of the beam to its sectional area centroid. e = 0 means that centerline of the beam coincides its area centroid.

 **Beam End Condition** : Beam ends are fixed by default. You can apply hinges to left and/or right by clicking successively on this icon.

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

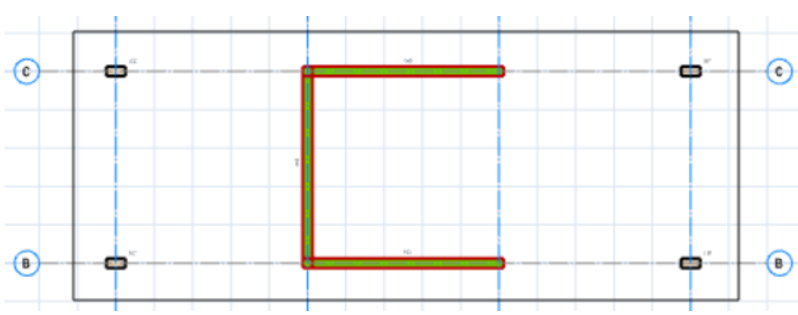
➢ *To create a beam, click on the **intersection of axes** for the start & then end of the beam*

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

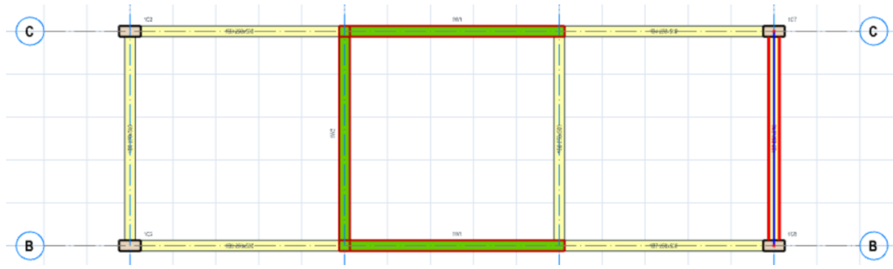


➢ ***Right-click** to end the beam insertion after inserting the 2 beams as shown above*

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.

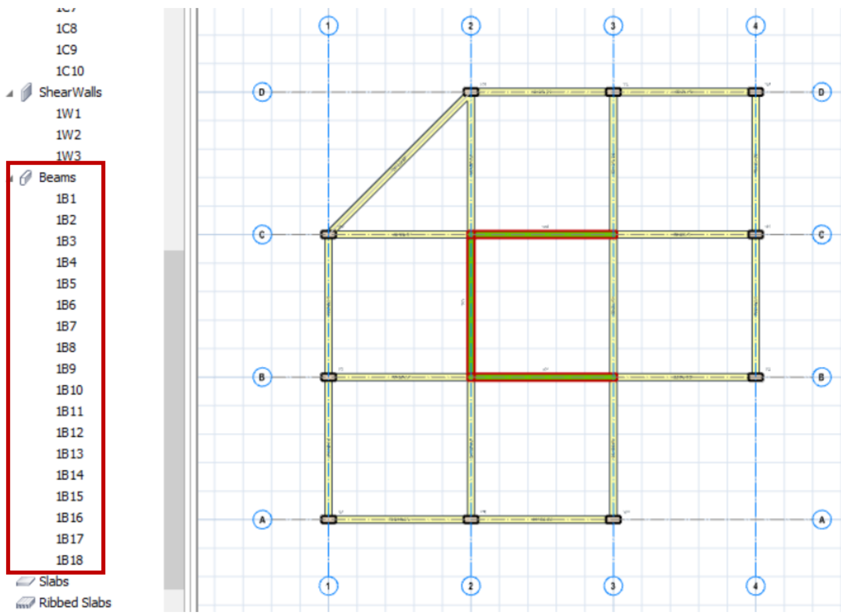


- *Click and drag box enclosing the columns & walls shown above.*



Seven (7) new beams will be created.

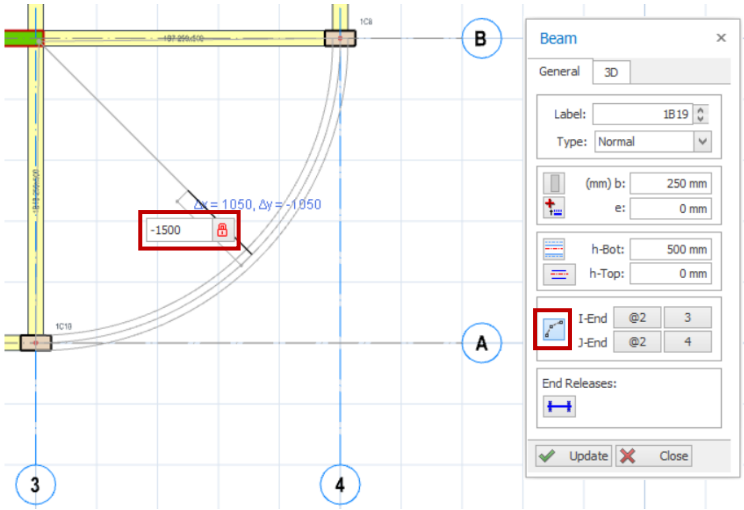
- *Insert the rest of the beams by any method as shown below*



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.

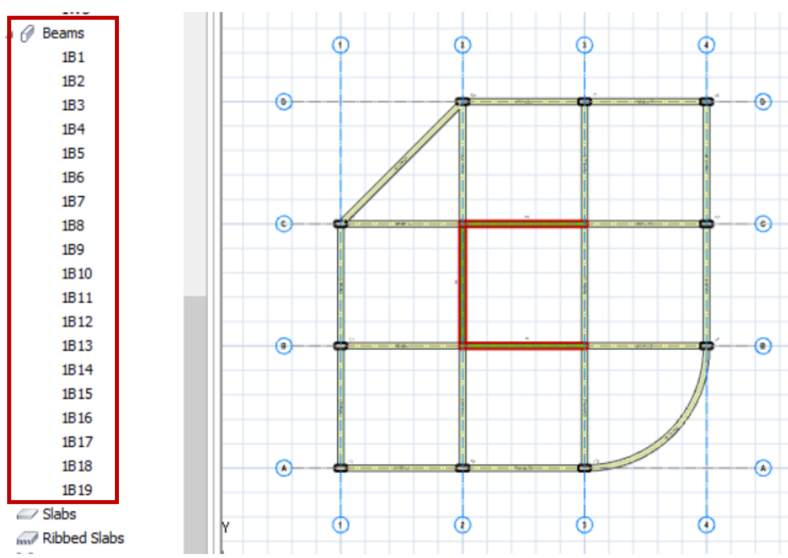
- *Click on the **curve beam** insertion icon*  *in the Beam Properties dialog*
- *Click on the intersection of **axes A/3** and then intersection of **axes B/4** (ie. start & end of the beam)*



You define the radius of the curve by simply moving the mouse cursor and the preview of curve beam in grey will show automatically.

➢ *Press **F2** to define the radius of the exactly to **-1500** mm & press **ENTER***

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

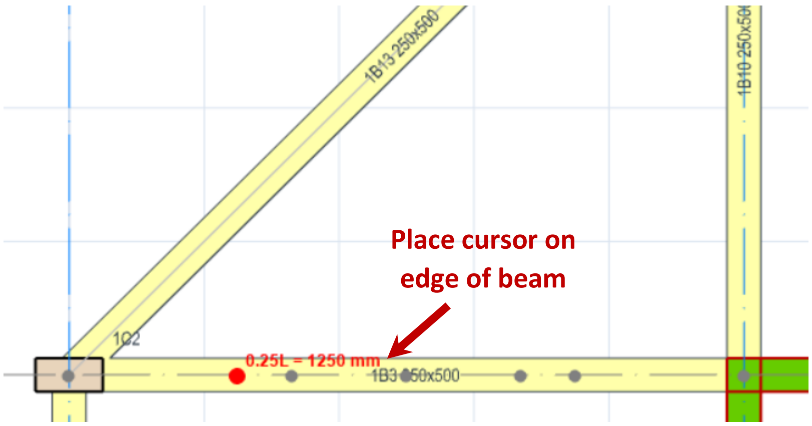


16. Beams Creation using dynamic snap points

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

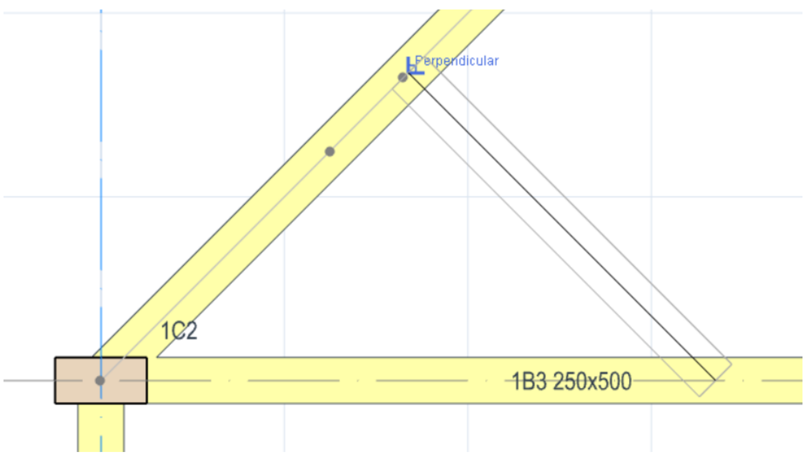
Secondary beams can easily be inserted using the dynamic snap points (without having to create axes).

- *Click **Beam** icon & place the cursor on the **edge** of 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.

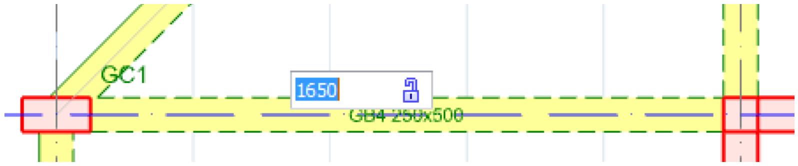
- *Click on **0.67L** as the start point of the secondary beam.*
- *Place the cursor on the destination beam*



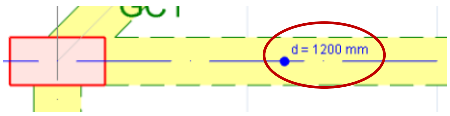
Notice that **0.25L**, **0.33L**, **0.5L**, **0.67L**, **0.75L** & **perpendicular** point will appear

- *Select the desired point as end point 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 the exact distance from the start of the beam.

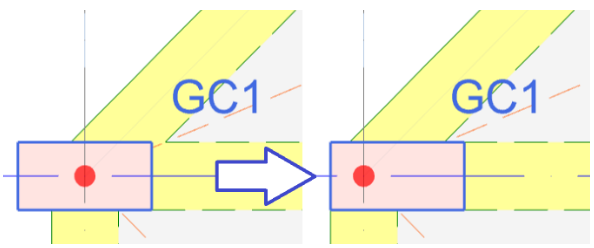


Pressing ***CTRL*** key while the cursor is placed on the primary beam will expose distance of increment 100mm. Left click to select the desired distance (while still holding down the CTRL key).



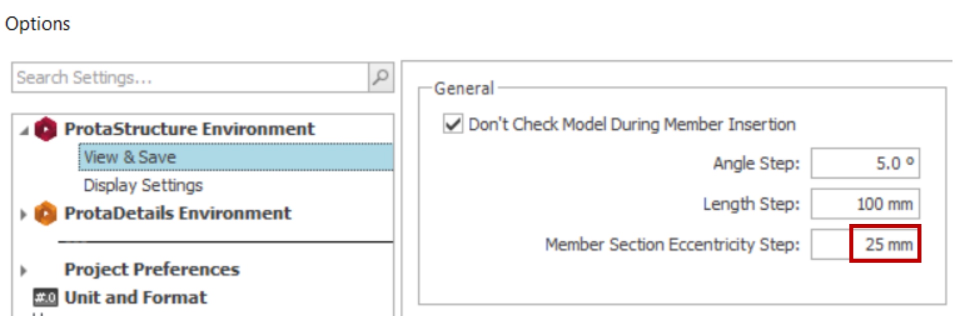
Handy Tip to adjust the position of columns and beams

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



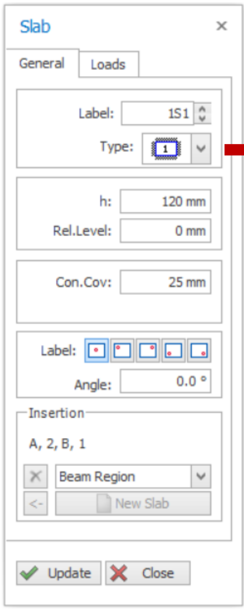
- ***Select*** *column GC1*
- ***Press*** *right arrow key → to move the column to the right*

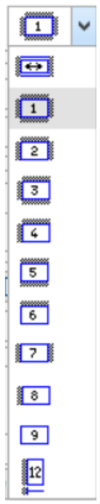
The step by which the member is moved can be set in ***Building Setout*** → ***Settings Center*** → ***View & Save*** → ***Member Section Eccentricity Step*** (by default 25mm).



17. Slab Creation

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

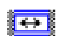




➢ *Click on the* ***Type*** *box*

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

The slab type 1 to 12 relates to table 3.14 in the BS 8110 and is used in the design of the slab reinforced based on the coefficient method.
Type 1 to 12 does not affect the slab load calculation on the supporting beams.

 One-way slab. The slab load will only to be transferred to the 2 supporting beams only in the direction of the span.
The direction of one-way span must be specified in the **Angle** input box : 

Label icons activates the slab label plus control the position of the slab label.

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***

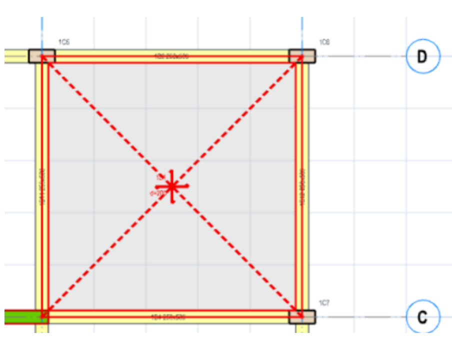
For ease in creating this model we will initially leave Slab Type as 1 for all slabs. Once they have been created & when we are ready to design the slab, there is a function to automatically set the correct type for each slab (covered later).

➢ *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 → Slab Additional Loads library**.

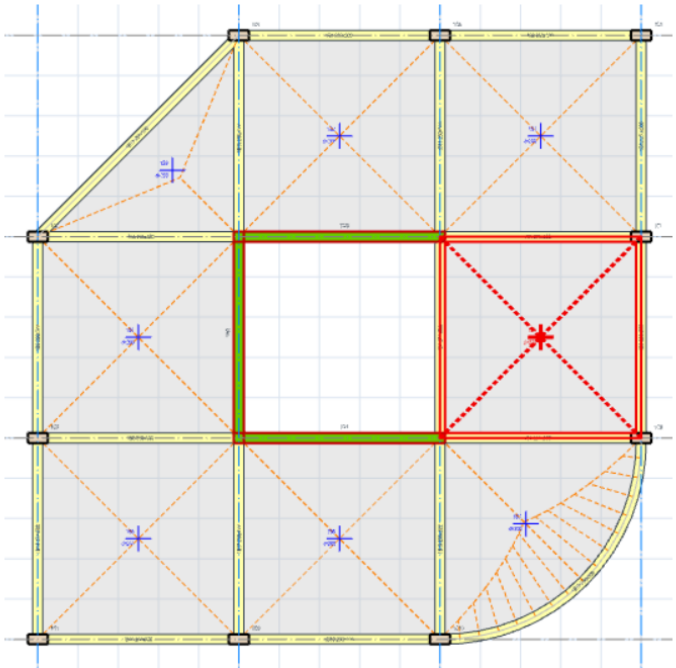
➢ *To create a slab, position the cursor in the region bounded by beams @GL 3,4,C,D and left-click.*



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 **7 more slabs** to give the layout shown below*

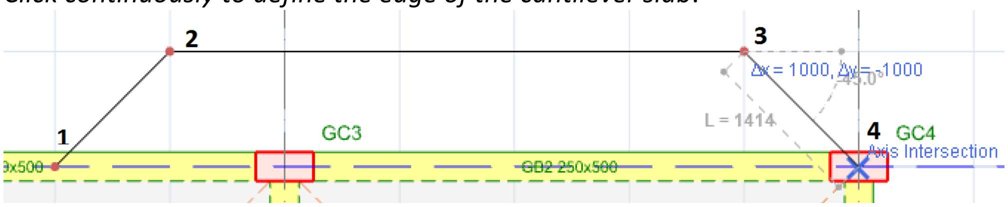


Polyline Slab/Column Edge

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

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

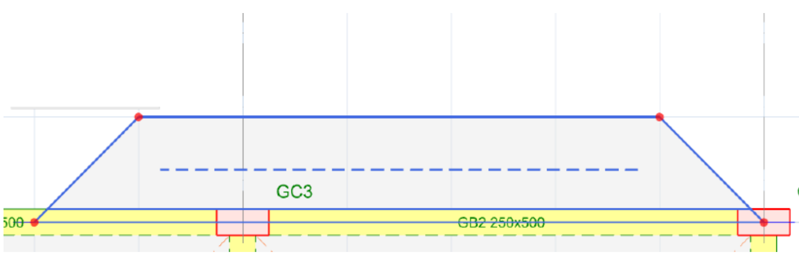
- *Click **Polyline Slab/Column Edge** under **Modelling***
- *Click continuously to define the edge of the cantilever slab.*



The method to draw the edge lines are similar to 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.*

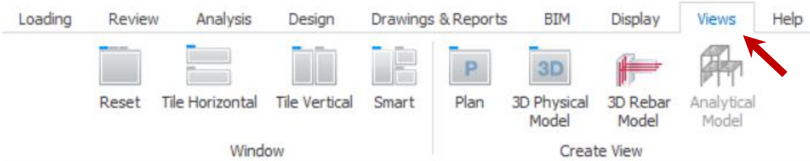


Note : For regular cantilever slab, it is recommended to use **Type 12** in slab properties (instead of slab edge lines)

TIP : 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-clck → **Convert to Column** or **Slab Hole/drop**

18. Views Creation

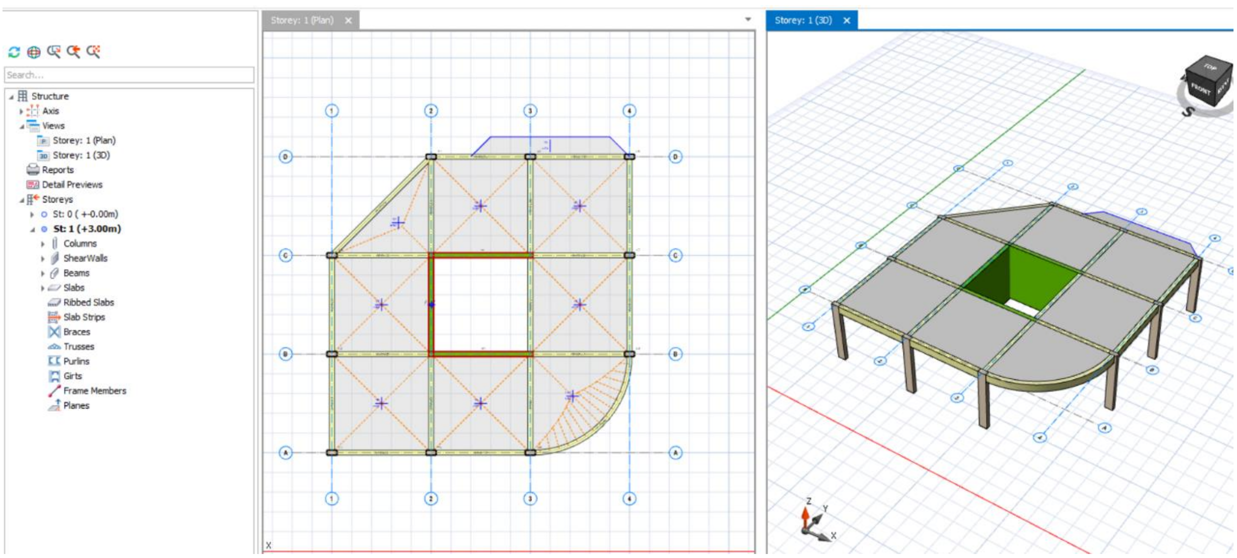
The graphical editor supports multiple windows; this allows you to create different views in separate windows. The Views tab commands can be used to create new views and to arrange the views.



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

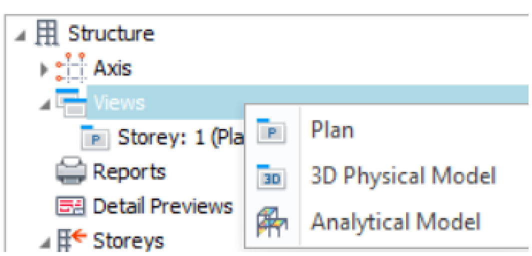
  This will create a 3D physical model view.

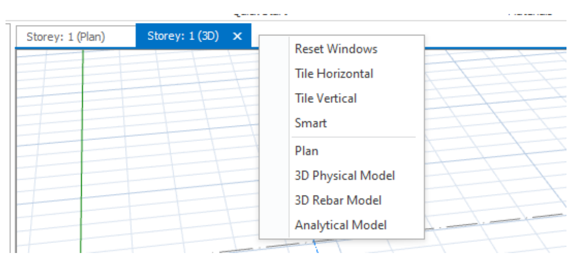
- *Click **Tile Vertically** → This will tile the plan and 3D view side by side*



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

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



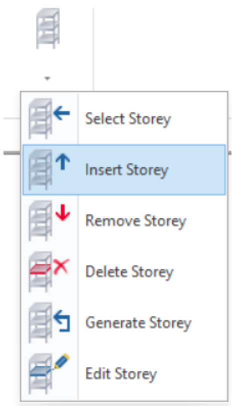


- ***Select** the curve beam in the 3D view. Notice the same member will be selected in the plan view.*
- ***Right click** (to access the context menu) → **Properties***
- *Change the depth of the beam **h-bot** to **600 mm***
- *Click **Update** and **Close** the beam properties*

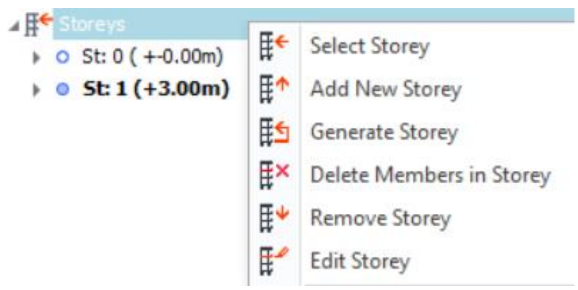
19. Inserting Storeys & Defining Building Parameters

The final model will be a 4 storey building. We will now insert the other floors.

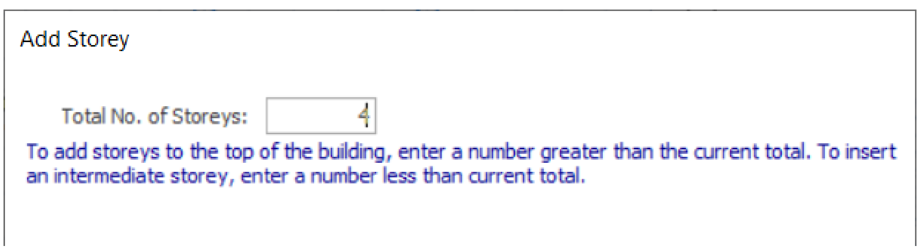
- *Go to **Building Setout** → **Storeys** drop down → **Insert Storey**.*



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



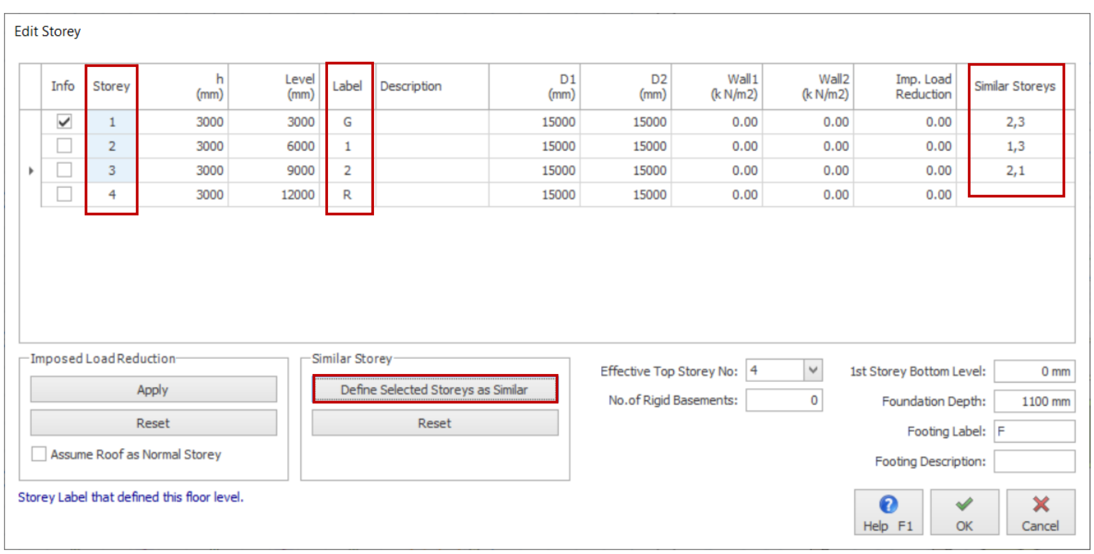
- *Input **Total No. of Storeys = 4** → **OK***



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

This automatically inserts storey 2, 3 & 4. We will now edit the information of the storeys.

- *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 storeys as similar**"*



This means that storey 1, 2 & 3 will now be identical. Since we have already inserted members in storey 1, these members will be automatically copied to storey 2 and 3. In addition, changes to a particular similar storey will be applied automatically to all similar storeys.

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

- *Under **Labels** column, change member label to as shown above, eg. "**G**" for ST01, "**R**" for ST04.*

This means that ST01 members will be labelled GB1, GC1, etc.

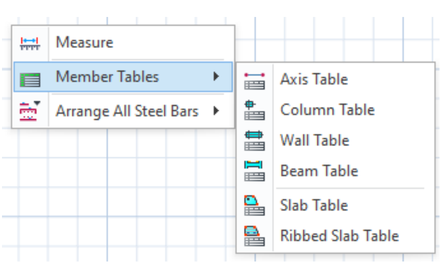
- *Click **OK** to exit and notice that the 3D view now shows storey 1, 2 & 3 with identical members.*

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

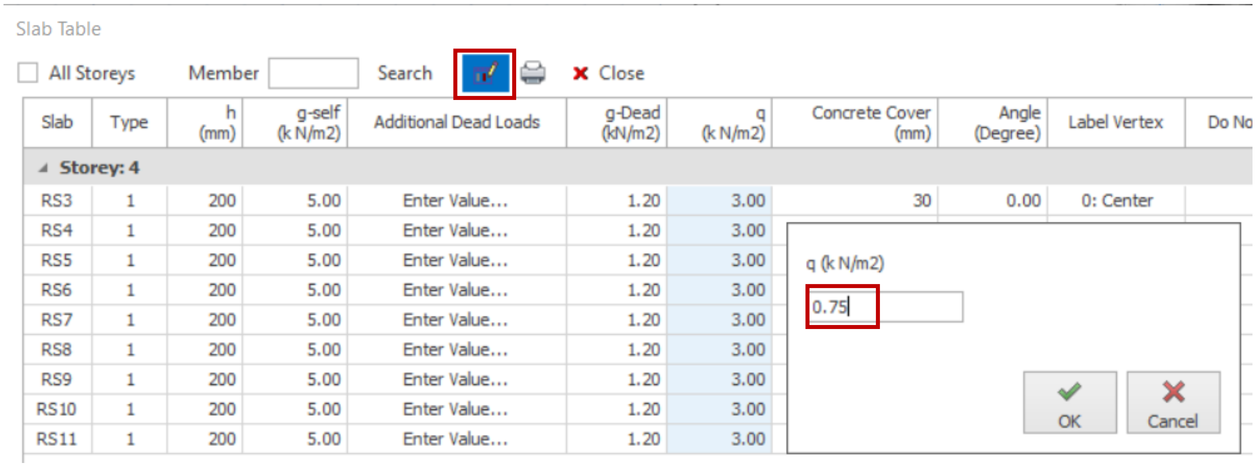
- *Pick **Generate Storey** from the same **Storey** dropdown*

- *Under **Source Storey**, pick **Storey 1** & then pick **Storey 4** as **Target Storey***
- *Click **OK** and members will be copied from ground floor to 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** 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 **Columnwise Edit** icon at the top*
- *Enter **0.75** kN/m^2 → press **ENTER** (all the slabs live load values will be changed)*
- ***Close** the Slab Table*

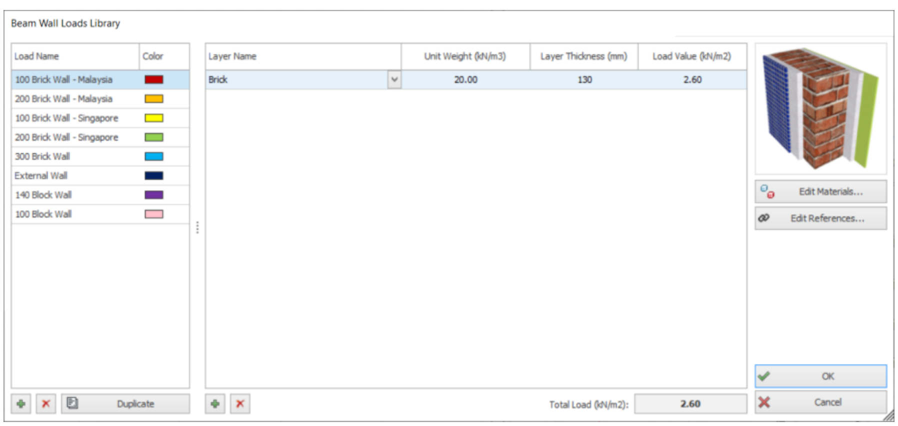


Slab	Type	h (mm)	g-self (k N/m2)	Additional Dead Loads	g-Dead (kN/m2)	q (k N/m2)	Concrete Cover (mm)	Angle (Degree)	Label Vertex	Do No
Storey: 4										
RS3	1	200	5.00	Enter Value...	1.20	3.00	30	0.00	0: Center	
RS4	1	200	5.00	Enter Value...	1.20	3.00				
RS5	1	200	5.00	Enter Value...	1.20	3.00				
RS6	1	200	5.00	Enter Value...	1.20	3.00				
RS7	1	200	5.00	Enter Value...	1.20	3.00				
RS8	1	200	5.00	Enter Value...	1.20	3.00				
RS9	1	200	5.00	Enter Value...	1.20	3.00				
RS10	1	200	5.00	Enter Value...	1.20	3.00				
RS11	1	200	5.00	Enter Value...	1.20	3.00				

20. Wall Loads Library & Inserting Brickwall Loads

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

- *Go **Building Setout** → **Partition Wall Loads***



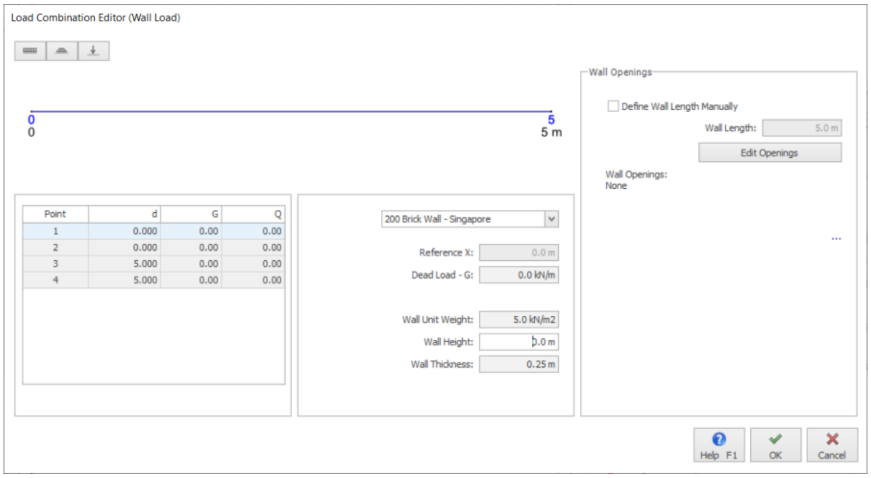
Defaults wall types are automatically setup but you can add any new user-defined wall type.

- *Choose **Cancel** to exit*

We will now insert brickwall 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 a perimeter **Beam** → Click **Edit Wall Load*** Edit Wall Load *in the **Beam** tab that appears.*

Alternatively, you can select the beam → Right-click → Edit Wall Load



Tips

Partial wall can be entered by checking "***Define Wall Length Manually***" :

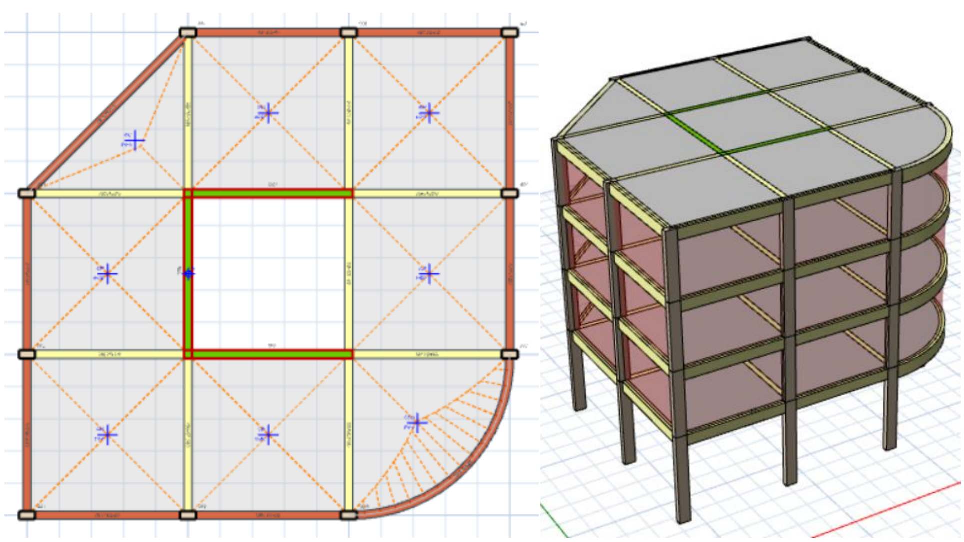
- *Enter **Wall Length***
- *Enter "**Reference X**" for position of start of the wall load*

Wall Opening, eg. of windows & doors can be inserted by click "***Edit Openings***".

- *Choose **200 Brick Wall – Singapore***
- *Enter **Wall Height** = **3.00 m** → Click on any other box for the diagram to refresh → **OK***

Examine the **3D view** and note that the wall load is also copied to ST02 & ST03 as they are similar. We will now copy the brick wall load to all the perimeter beams.

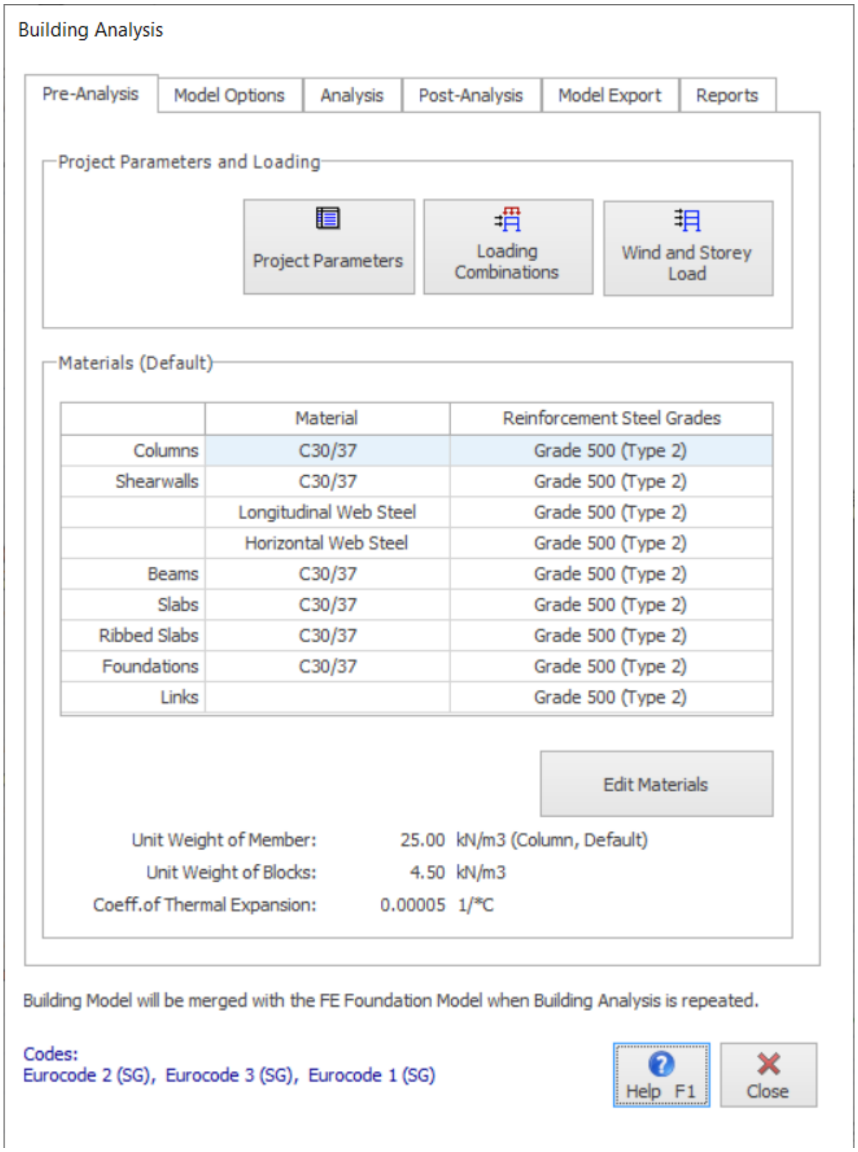
- *Ensure the beam with wall load is selected → right-click → **Copy Wall Load***
- ***Multiple select** all the perimeter beams by holding down the **CTRL** while selecting the beams. You can also drag a box to select the perimeter beams (it does not matter if columns are selected).*
- ***Right-click** → **Paste Copied Beam Loads***
- *Choose **Yes** to copy the loads*



- *Examine the 3D view to ensure all wall loads are inserted correctly. The model is now complete and we are ready to run the analysis.*

21. Building Analysis

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



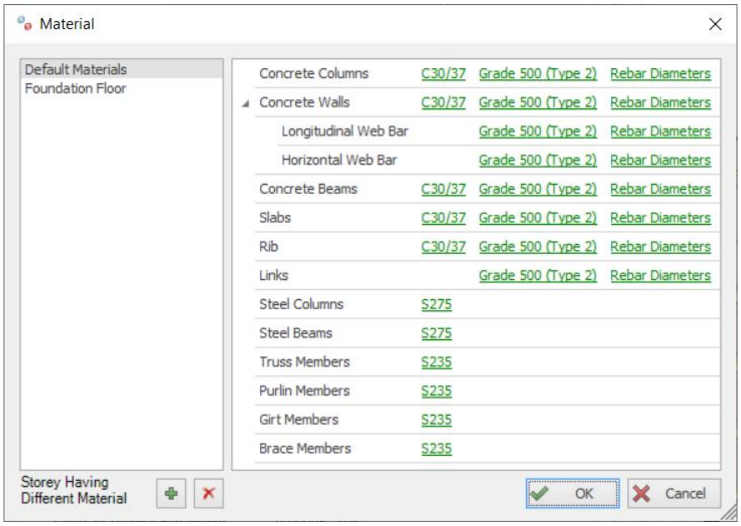
The Pre-Analysis settings is where main assumptions of the analysis can be defined :

- **Project Parameters** : review or modify the analysis parameters such as the design code and notional horizontal load
- **Load Combination** : 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.

22. Materials

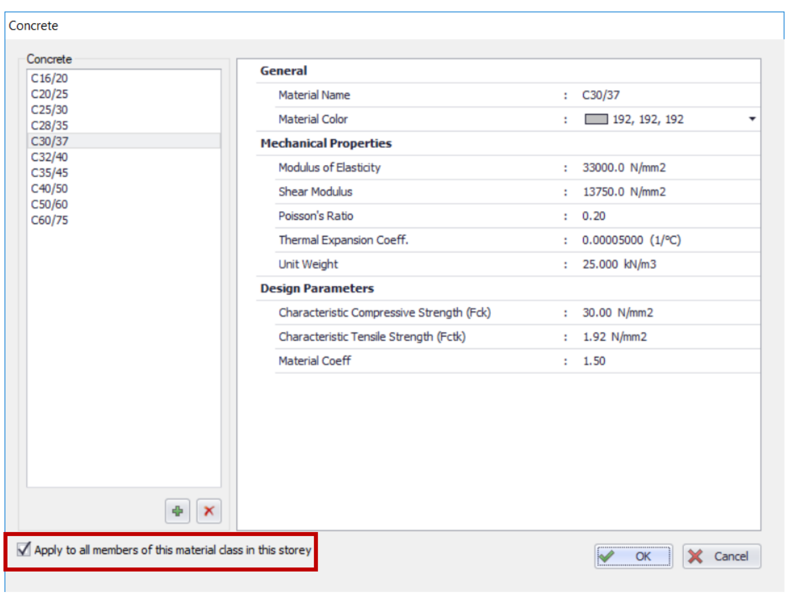
We will now define the materials for this project via Building Analysis > **Pre-Analysis** dialog.

- *Pick **Edit Materials***



The materials and reinforcement steel grades are shown for all member type. ***Default Material*** is used for super-structure floors. A separate ***Foundation Floor*** category is automatically created for foundation members only.
If there is different material for a particular storey, a separate material settings 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 member in this Storey** → **OK***

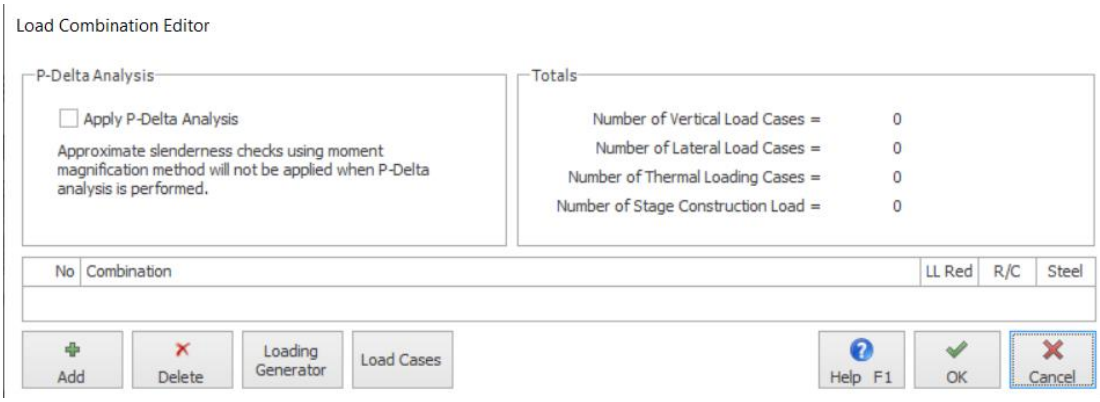


- *Pick **Steel Grade** and ensure that **Grade 500 (Type 2)** is selected and applied to all member types*
- *Pick **Diameter** and select the desired **rebar diameter** to be used in the member design*
- *Pick **OK** to return to the Building Analysis dialog*

23. Load Combinations

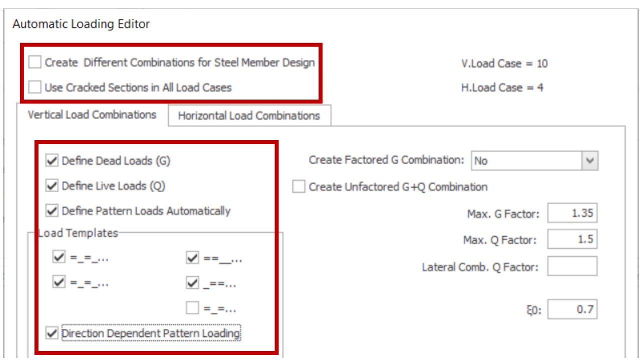
We will now auto-generate the load cases and load combination.

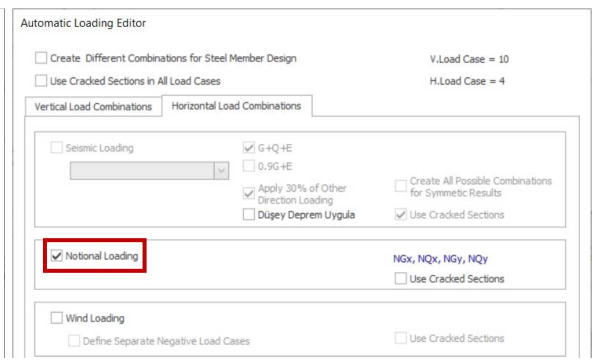
➢ *Pick* ***Loading Combination*** *to launch the Load Combination Editor*



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

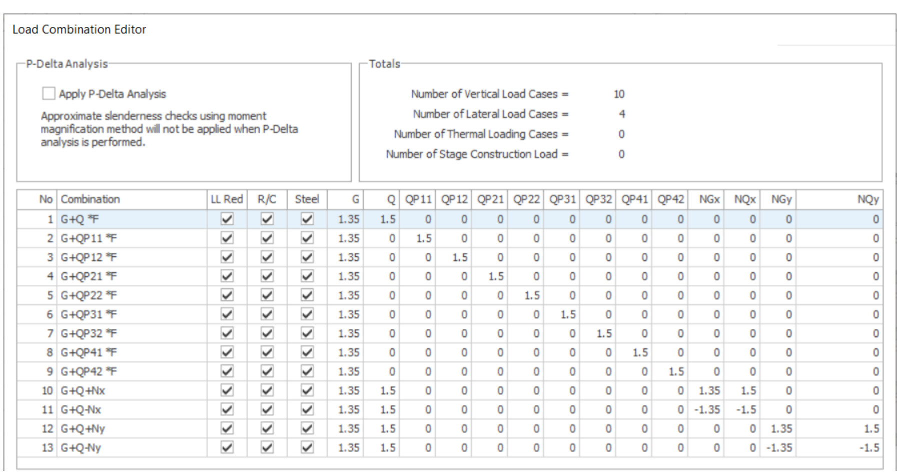
➢ *Pick* ***Loading Generator*** *& use the options as shown below and click* ***OK***





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

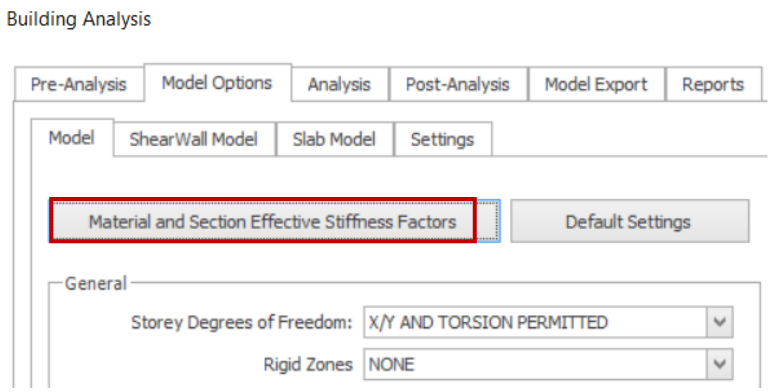
If "**Use Cracked Sections**" is checked, **Section Stiffness Factors** will be applied for that load case. All the load cases and combinations will automatically be generated as shown below.

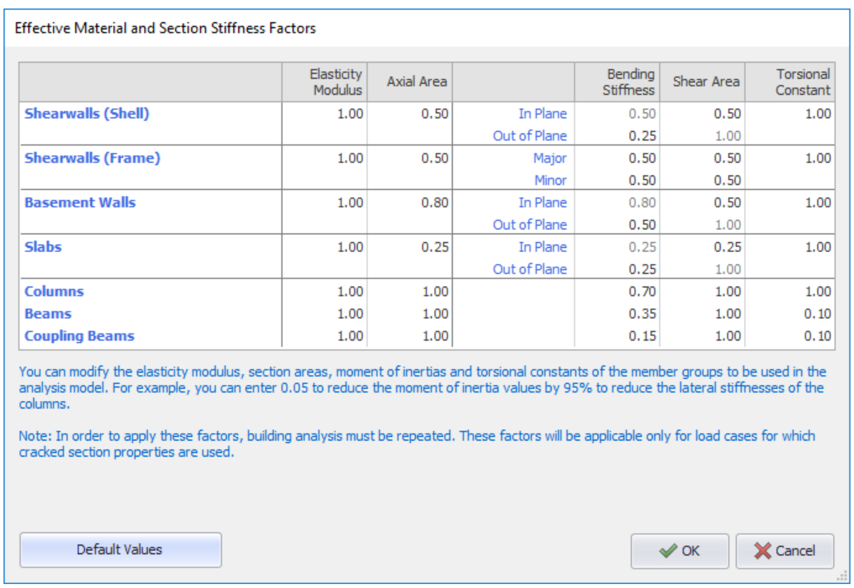


No	Combination	LL Red	R/C	Steel	G	Q	QP11	QP12	QP21	QP22	QP31	QP32	QP41	QP42	NGx	NQx	NGy	NQy
1	G+Q *F	✓	✓	✓	1.35	1.5	0	0	0	0	0	0	0	0	0	0	0	0
2	G+QP11 *F	✓	✓	✓	1.35	0	1.5	0	0	0	0	0	0	0	0	0	0	0
3	G+QP12 *F	✓	✓	✓	1.35	0	0	1.5	0	0	0	0	0	0	0	0	0	0
4	G+QP21 *F	✓	✓	✓	1.35	0	0	0	1.5	0	0	0	0	0	0	0	0	0
5	G+QP22 *F	✓	✓	✓	1.35	0	0	0	0	1.5	0	0	0	0	0	0	0	0
6	G+QP31 *F	✓	✓	✓	1.35	0	0	0	0	0	1.5	0	0	0	0	0	0	0
7	G+QP32 *F	✓	✓	✓	1.35	0	0	0	0	0	0	1.5	0	0	0	0	0	0
8	G+QP41 *F	✓	✓	✓	1.35	0	0	0	0	0	0	0	1.5	0	0	0	0	0
9	G+QP42 *F	✓	✓	✓	1.35	0	0	0	0	0	0	0	0	1.5	0	0	0	0
10	G+Q+Nx	✓	✓	✓	1.35	1.5	0	0	0	0	0	0	0	0	1.35	1.5	0	0
11	G+Q-Nx	✓	✓	✓	1.35	1.5	0	0	0	0	0	0	0	0	-1.35	-1.5	0	0
12	G+Q+Ny	✓	✓	✓	1.35	1.5	0	0	0	0	0	0	0	0	0	0	1.35	1.5
13	G+Q-Ny	✓	✓	✓	1.35	1.5	0	0	0	0	0	0	0	0	0	0	-1.35	-1.5

24. Building Analysis Model Options

➢ *Go to the **Model Options** → **Model** → **Material & Section Effective Stiffness Factors** and review the assumption as shown below*





Effective Material and Section Stiffness Factors

	Elasticity Modulus	Axial Area		Bending Stiffness	Shear Area	Torsional Constant
Shearwalls (Shell)	1.00	0.50	In Plane	0.50	0.50	1.00
			Out of Plane	0.25	1.00	
Shearwalls (Frame)	1.00	0.50	Major	0.50	0.50	1.00
			Minor	0.50	0.50	
Basement Walls	1.00	0.80	In Plane	0.80	0.50	1.00
			Out of Plane	0.50	1.00	
Slabs	1.00	0.25	In Plane	0.25	0.25	1.00
			Out of Plane	0.25	1.00	
Columns	1.00	1.00		0.70	1.00	1.00
Beams	1.00	1.00		0.35	1.00	0.10
Coupling Beams	1.00	1.00		0.15	1.00	0.10

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.

Default Values

OK

Cancel

Notes:

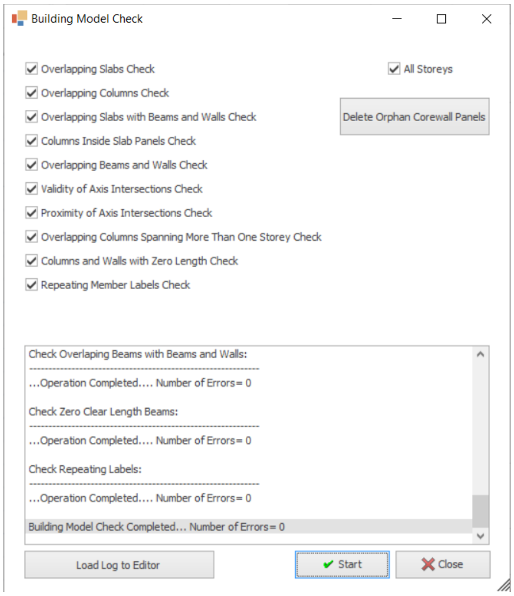
- The stiffnesses table will only be applied for load cases with '**Used Cracked Sections'** checked in the respective **Load Case Editor**, except for seismic load cases where cracked sections are always assumed.
- Each seismic code has its own set of default values. If an entered value is different than the default, then editor color will turn to **orange** to inform the user.
- 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 0.10 (10%) in the Stiffnesses settings.
- Stiffnesses of individual members can be changed by selecting a member → **Right-click** → **Edit Section/Material** → **Properties** tab.

25. Running Analysis

- *Go to the* ***Analysis tab***

Before running the analysis, it's always recommended that we check the validity of the model.

- *Click* ***Building Model Check*** *& pick* ***All Storeys*** *& click* ***Start***



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

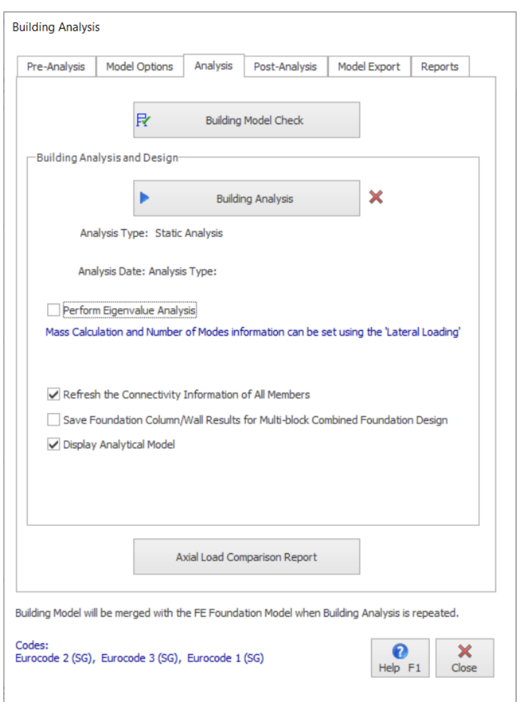
The warnings / errors are mostly self-explanatory. The exact member(s) affected will be stated.

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 and beams automatically after the analysis. We suggest that this should <u>only be done</u> if you are confident that the result is correct.

- *Check* ***Column/Wall Reinforcement Design***
- *Check* ***Beam Reinforcement Design***
- *Pick* ***Building Analysis*** *to analyze 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 end of the analysis stating summarizing the important results.

26. Axial Load Comparison Report

An important check on the validity of the analysis is the **Axial Load Comparison Report**. This report sums all of 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 with each other 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)*

Axial Load Comparison Report

TOTAL LOADS (Based On Slabs Loads):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	93.75	284.27	221.99	1130.15	0.00	1730.16
3 (+9.00m)	93.75	284.27	1091.41	1130.15	0.00	2599.58
2 (+6.00m)	93.75	284.27	1091.41	1130.15	0.00	2599.58
1 (+3.00m)	93.75	284.27	1091.41	1130.15	0.00	2599.58
Total						9528.91

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	0.00	-1.34	0.00	136.71	0.00	135.38
3 (+9.00m)	0.00	-5.34	-32.05	546.85	0.00	509.45
2 (+6.00m)	0.00	-5.34	-32.05	546.85	0.00	509.45
1 (+3.00m)	0.00	-5.34	-32.05	546.85	0.00	509.45
Total						1663.73

TOTAL LOADS (Decomposed to Beams):

G - Dead Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	93.75	384.65	1251.76	0.00	0.00	1730.16
3 (+9.00m)	93.75	384.65	2121.19	0.00	0.00	2599.58
2 (+6.00m)	93.75	384.65	2121.19	0.00	0.00	2599.58
1 (+3.00m)	93.75	384.65	2121.19	0.00	0.00	2599.58
Total						9528.91

Q - Live Loads:

Storey	Column	Wall	Beam	Slab	Ribbed Slab	Total
4 (+12.00m)	0.00	10.81	124.57	0.00	0.00	135.38
3 (+9.00m)	0.00	43.23	466.22	0.00	0.00	509.45
2 (+6.00m)	0.00	43.23	466.22	0.00	0.00	509.45
1 (+3.00m)	0.00	43.23	466.22	0.00	0.00	509.45
Total						1663.73

BUILDING ANALYSIS COLUMN AND WALL AXIAL LOADS:

Storey	G	Delta G	Q	Delta Q
4 (+12.00m)	1730.16	1730.16	135.38	135.38
3 (+9.00m)	4329.75	2599.58	644.83	509.45
2 (+6.00m)	6929.33	2599.58	1154.28	509.45
1 (+3.00m)	9528.91	2599.58	1663.73	509.45
Total		9528.91		1663.73

Table 1 : TOTAL LOADS (Based on Slab) is 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) takes into account 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 all slab loads are accurately captured by beams, i.e. no slab loads are lost.

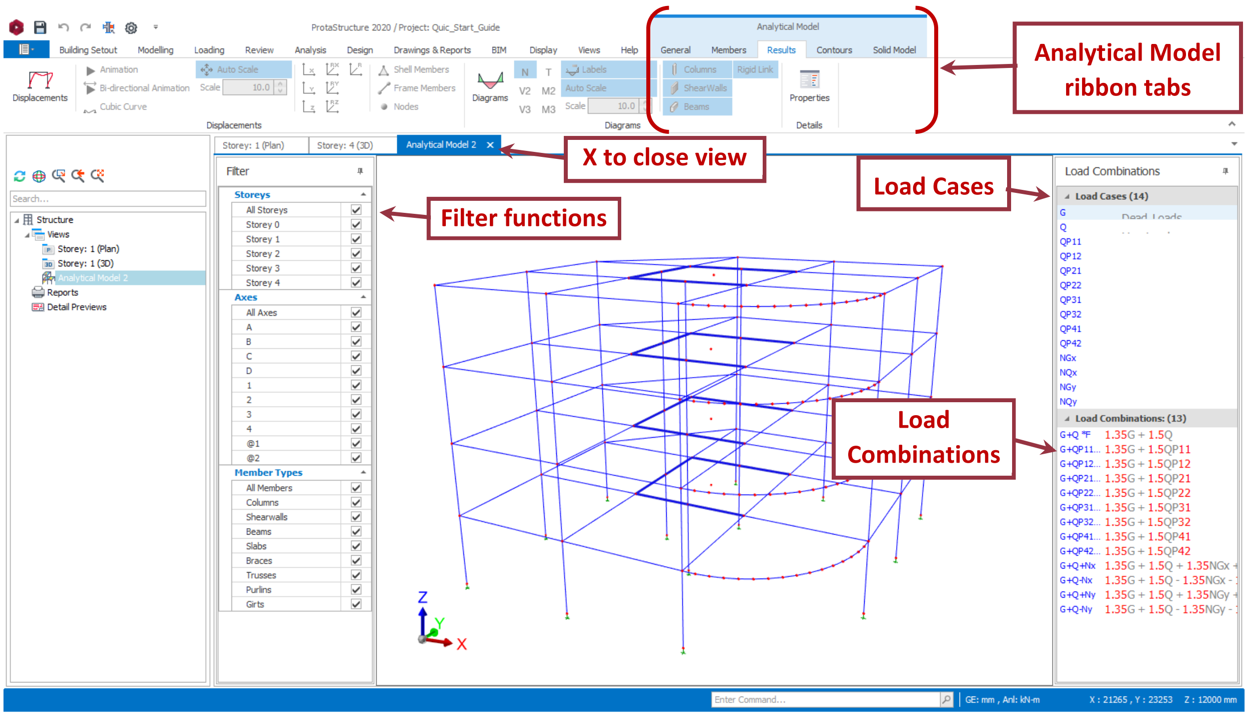
Then verify **Table 2** total values are similar to **Table 3**. This ensures that all the superstructure weight is completely captured by the columns and walls down to the foundation.

27. Analysis Model and Results Display

The **Model and Analysis Results Display** enable you to review the results of the analysis graphically. By default, a view will automatically open once Building Analysis is successfully com

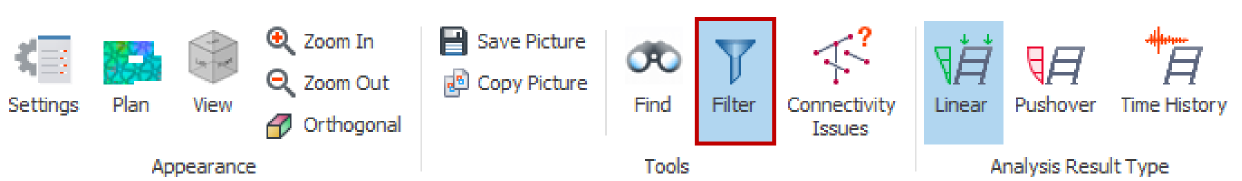
- *If not, go to the* ***Post-Analysis*** *tab*
- *Click* ***Display Analytical Model*** *→ Close the building analysis dialog*

A new **Analytical Model tab** set will appear together with **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. However, by making use of the various filter buttons and the view settings, you can create a more meaningful display view.

- *In the* ***General*** *tab, click on the* ***Filters*** *command (if it's not activated)*
  The filter options will appear in the left-hand side of the view. There are options to filter the display by Storeys, Axes and Member Type.

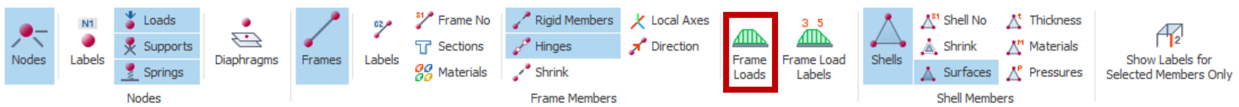


The **Find** command allows you to find a particular Node, Frame or Shell number.

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

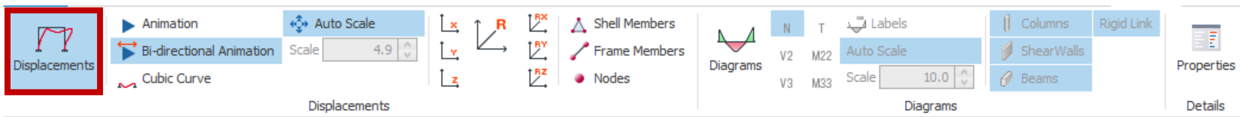
- *Click the* ***Members*** *tab*

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



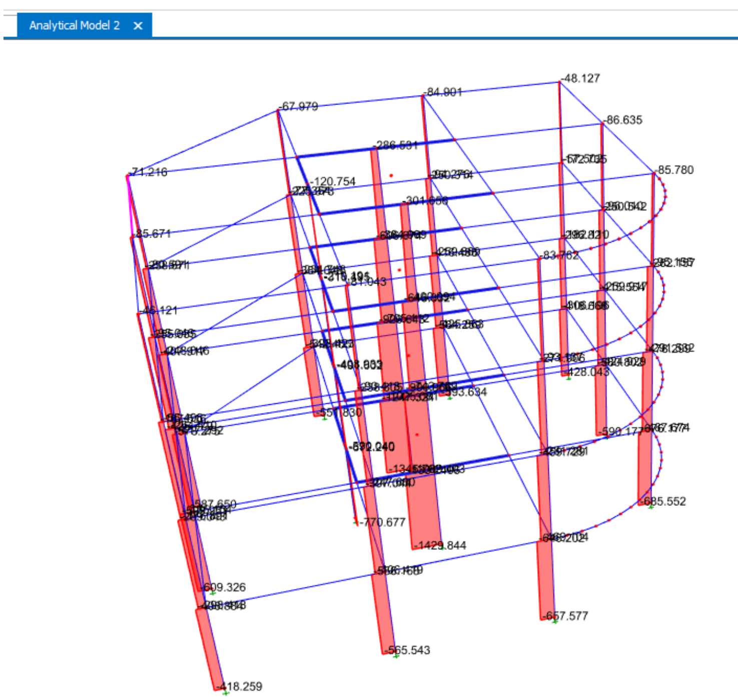
- *Click **Frame Loads** icons to show loads calculated or decomposed on the beams (& then turn it off).*

  This may include all the slab loads, brick wall loads, beam self-weight & other manual input loads.
- *Click on the **Results** tab*



- *Click **Displacement** → The **red** lines shows the displacement (deflection) of the structure.*

  By default, **Auto Scale** is activated. You can deactivate this and then type in your own Scale
- *Click on the various directional displacement values by selecting **x** (horizontal plan), **y*** (vertical plan), ***z*** *(along the storey) & **R** (Resultant). Rx,Ry, Rz is the rotation of joints in radians.*
- *Click **Animation** to visualise the deformation of the structure*
- *Switch off **Displacements** display and click on **Diagrams** button.*
- *Click on 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

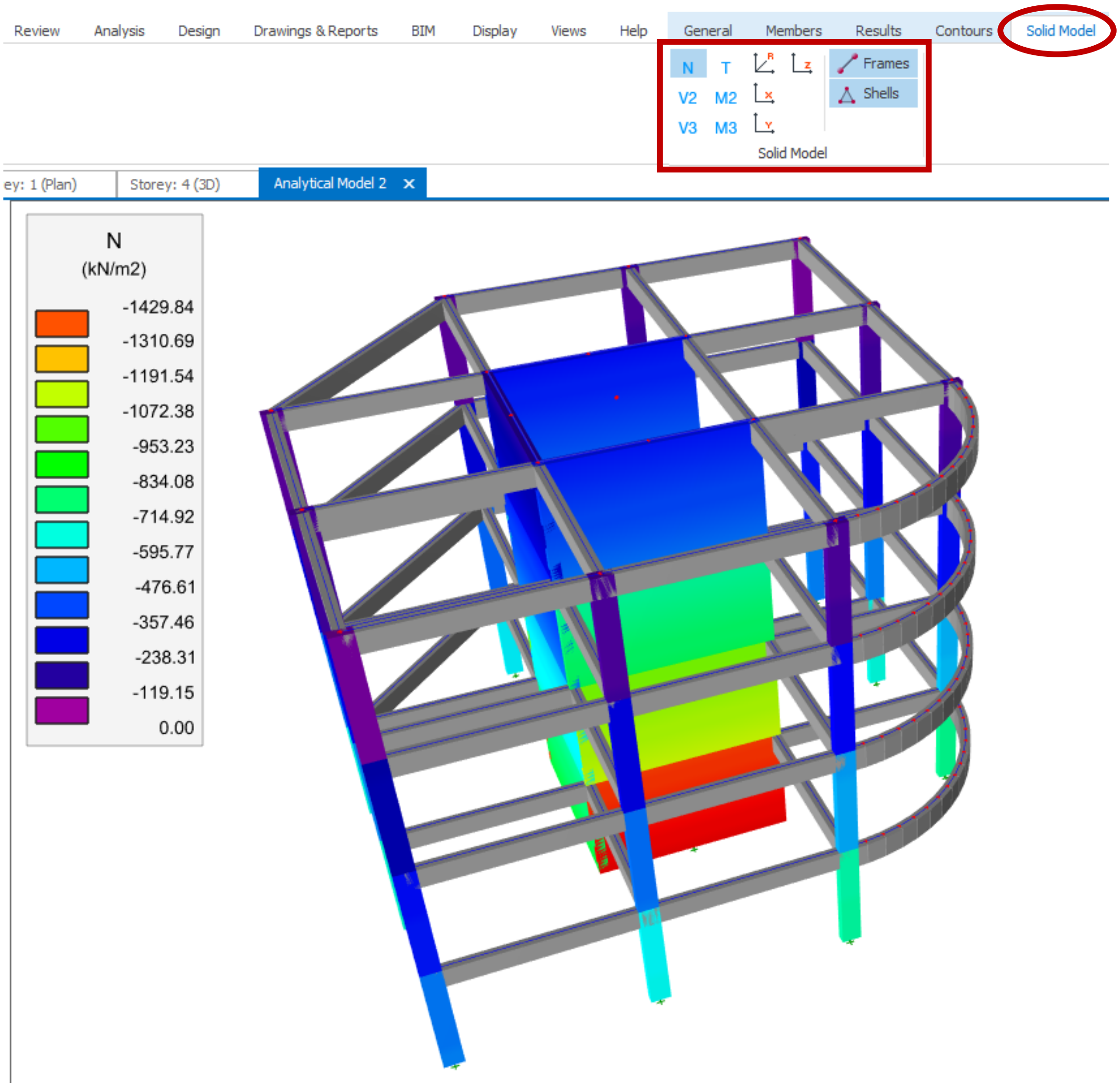
M3 – Major moment of beams and walls. Moment of column along dir 2

T - Torsion

- *Experiment with the various effects of the diagrams*

➢ *Go to the **Solid Model** tab*

This allows the various effects such **Axial Stress** (kN/m^2) to be color coded on the physical model



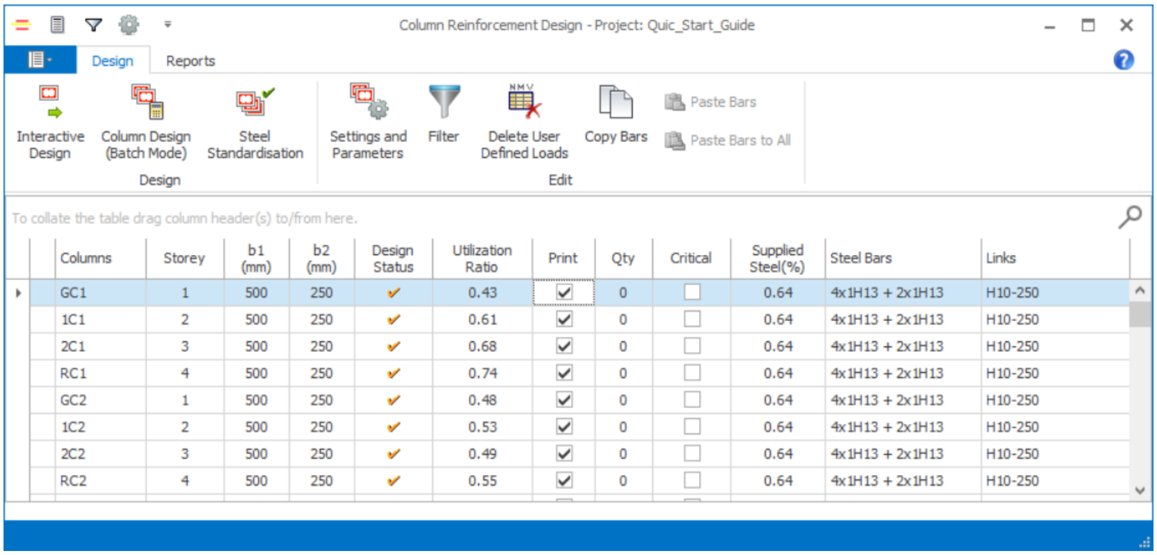
➢ ***Close** the Analytical Model view by clicking on the "**x**" sign next to the view name.*

It is essential that you interrogate the Analytical Model and check the validity of the model as that is the true analytical model from which the design forces will be based on.

28. Column & Wall Design

➢ *Go to **Design** tab → **Column Section Design***

Since we have selected to run column reinforcement design as part of Building Analysis, all the columns and walls are already designed as indicated by green tick.

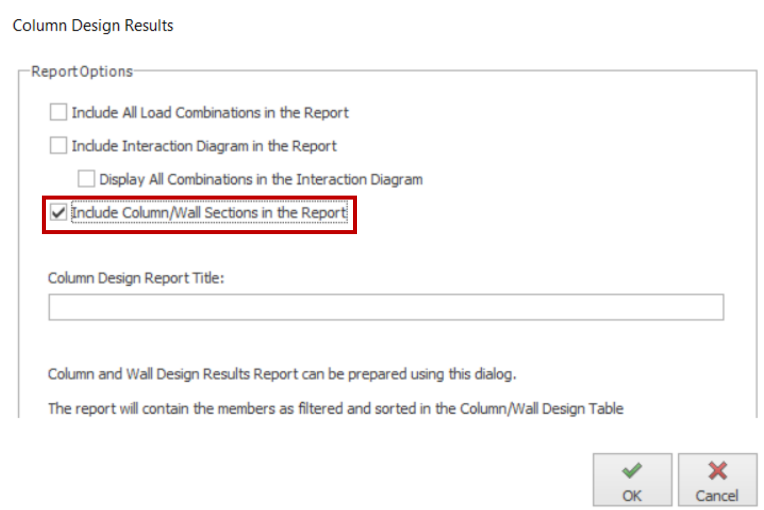


Columns	Storey	b1 (mm)	b2 (mm)	Design Status	Utilization Ratio	Print	Qty	Critical	Supplied Steel(%)	Steel Bars	Links
GC1	1	500	250	✔	0.43	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
1C1	2	500	250	✔	0.61	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
2C1	3	500	250	✔	0.68	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
RC1	4	500	250	✔	0.74	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
GC2	1	500	250	✔	0.48	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
1C2	2	500	250	✔	0.53	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
2C2	3	500	250	✔	0.49	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250
RC2	4	500	250	✔	0.55	☑	0	☐	0.64	4x1H13 + 2x1H13	H10-250

If they are not designed, you can choose ***Column Design (Batch Mode)*** to design all columns at one go.

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

The columns to be included in the report are marked by a tick in the **Print** column. Columns can be added or removed from the report by checking or unchecking the **Print** checkbox. Further, there are icons **Mark All Columns** and **Remove Print Marks** that can be applied to all columns.

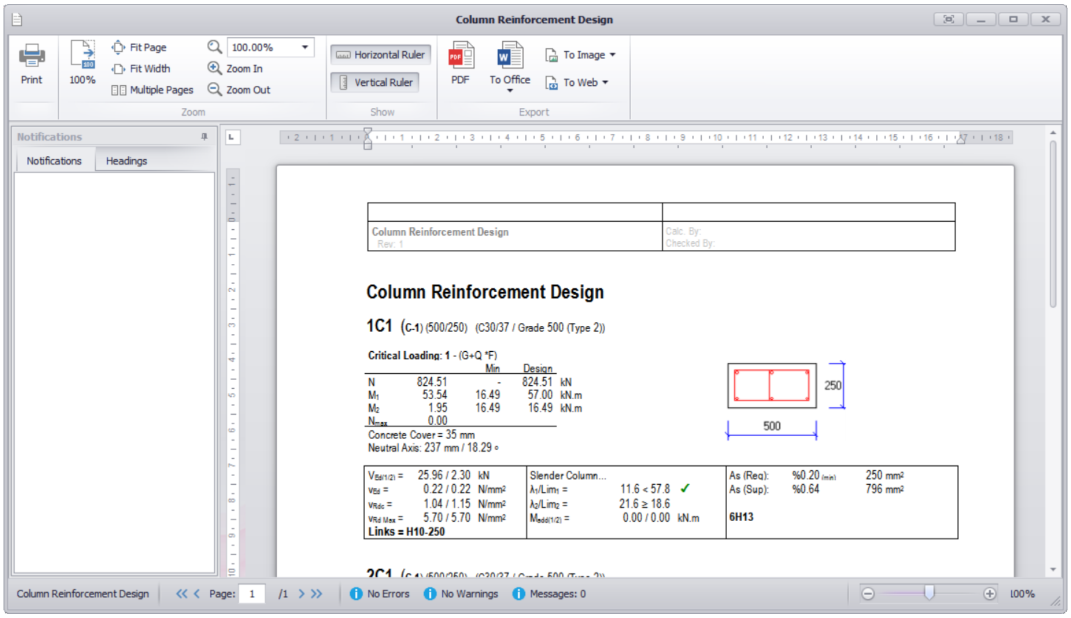


➢ *To draw the column section detail, check "**Include Column/Wall Sections in the Report**".*

You can also give the report a title.

➢ *Pick **OK** to generate the report*

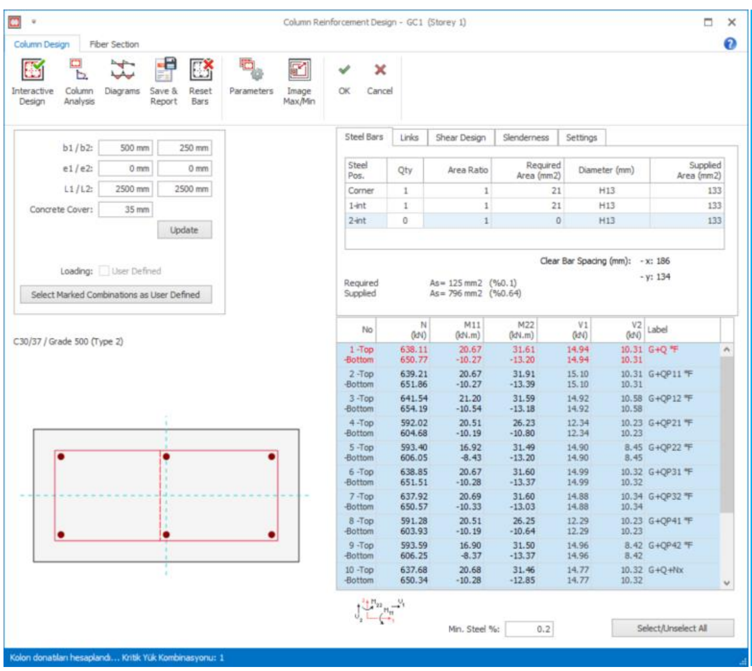
All reports can be exported as PDF, MS Word, image or webpage (html) format.



➢ ***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 design in more detail.

➢ ***Double click** on **1C1** 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 assumption of this column, you can click **Interactive Design** button to re-perform the reinforcement design.*

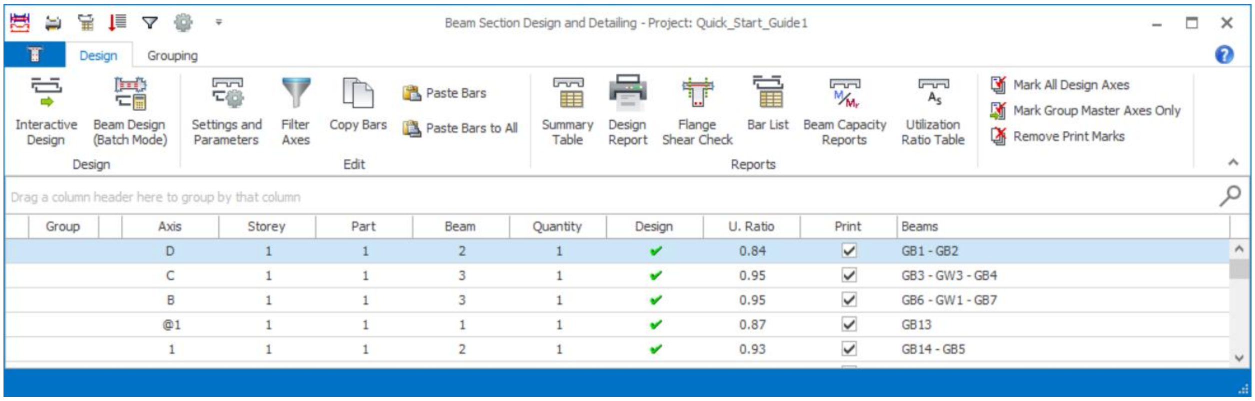
➢ *Click **Cancel** to exit the Interactive Column Design dialog and **Close** the Column Design screen*

![PROTA logo]

29. Beam Design

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

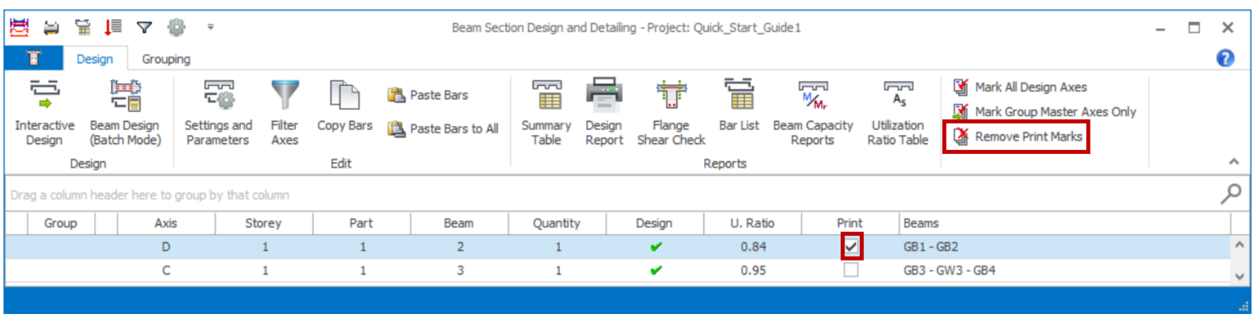
Since we have selected to run beam reinforcement design as part of Building Analysis, all the beams are already designed as indicated by green ticks.



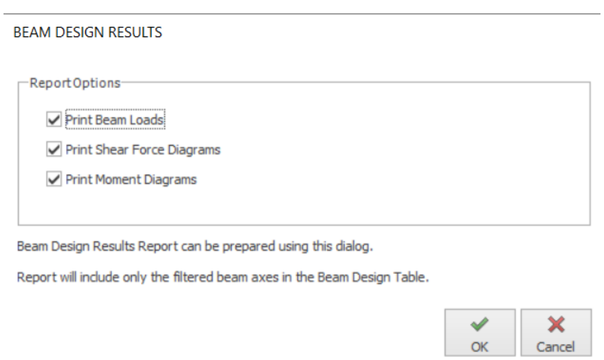
If they are not designed, you can choose ***Beam Design (Batch Mode)*** to design all beams at one go.

The beams to be included in the report are marked by a tick in the ***Print*** column. 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 A only*



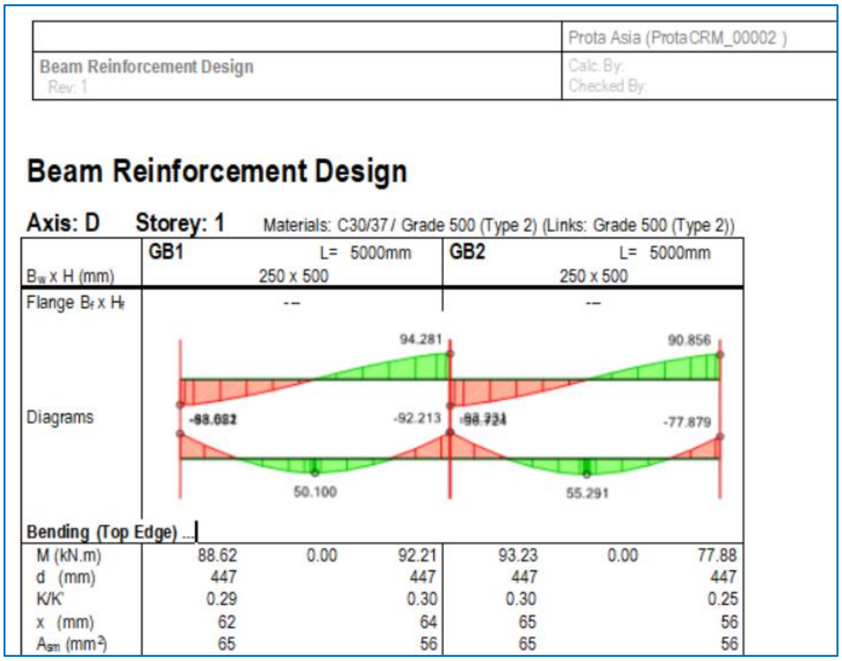
- ➢ *Choose **Design Report***



The Beam Design Report is in rich text format.

Choose the preferred Report Options to include the various force diagrams.

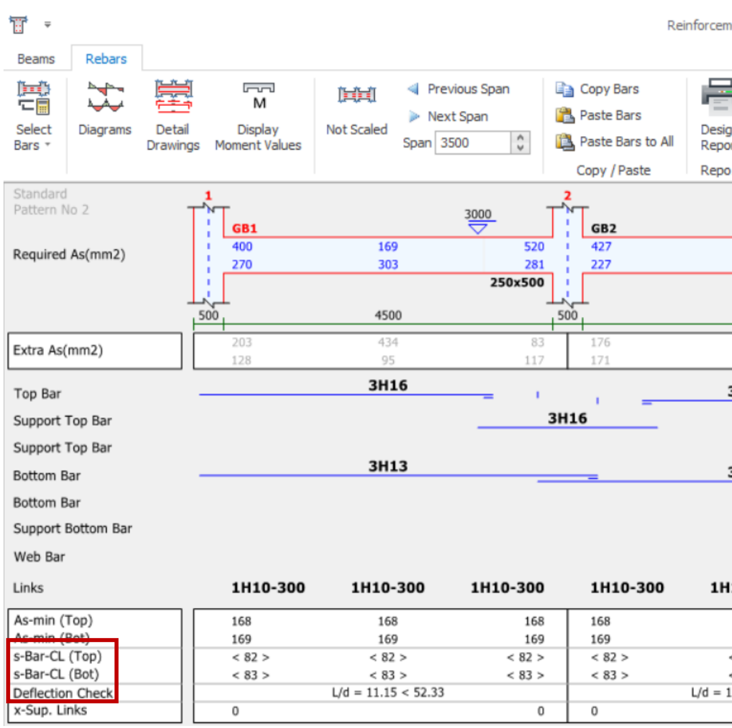
➢ *Pick **OK** to generate the report*



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

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

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



Reinforcement Data window shows the details of the beam design.

Required As is shown in blue as is based on design envelope of all load combination.

Extra As shows the additional or surplus area of steel based on selected steel.

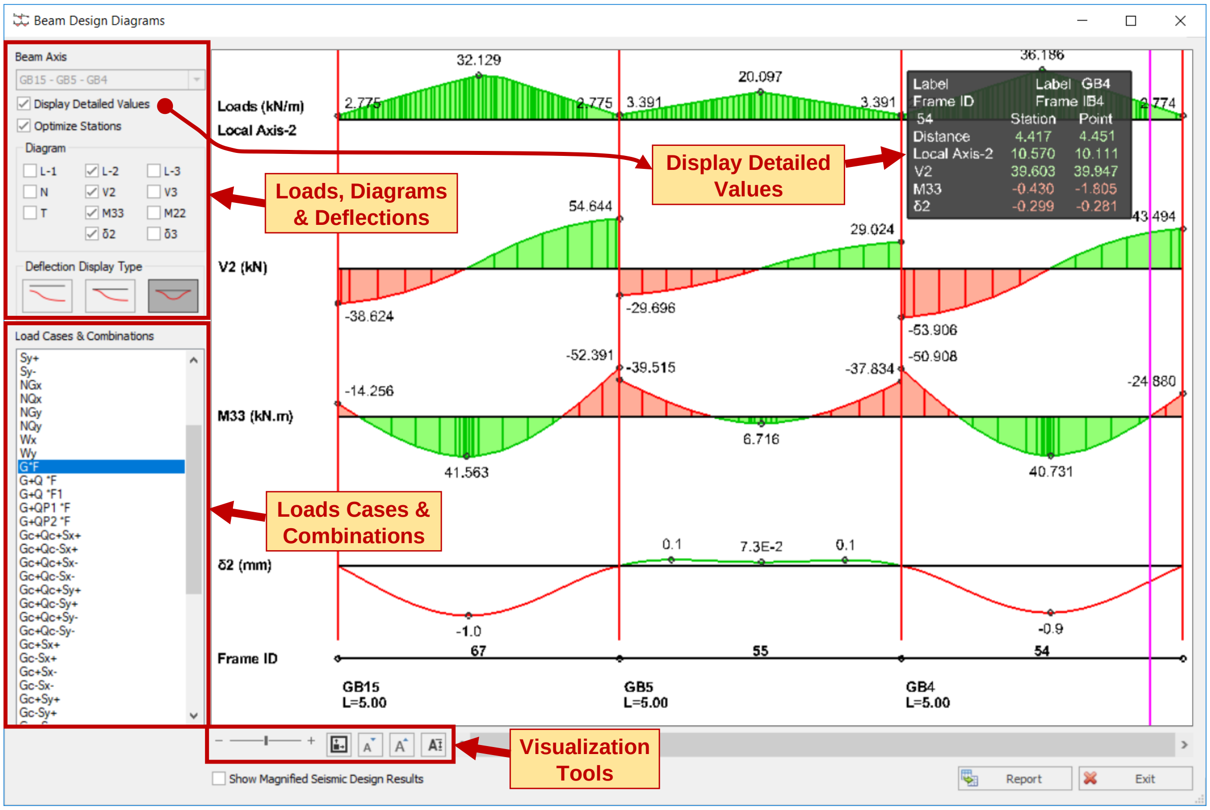
Rebars designed are shown in the various rows including shear links. These can be manually changed if desired.

s-Bar-CL (Top)/(Bot) shows the clear spacing of the rebars at the outmost layer.

Deflection Check shows the actual vs allowable span/effective depth. The modification factor is automatically applied.

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

➢ *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, eg. shear & moment, when the mouse cursor is placed at a particular location along the member.
- **Optimize Stations** :
  - Unchecked : The diagrams are displayed using 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 external 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 are used to display the deflected shape.
- **Normalized:** The absolute shape is normalized with respect to the value calculated at the first point.
- **Relative:** The deflected shape is normalized with respect to both start and end points. 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 default scale
- **Increase / decrease font size**
- **Default Font Size** → Click to rest to default font size
- **Report** Report → Generate a report in tabular format with / without diagrams.

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

Notes :

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 :

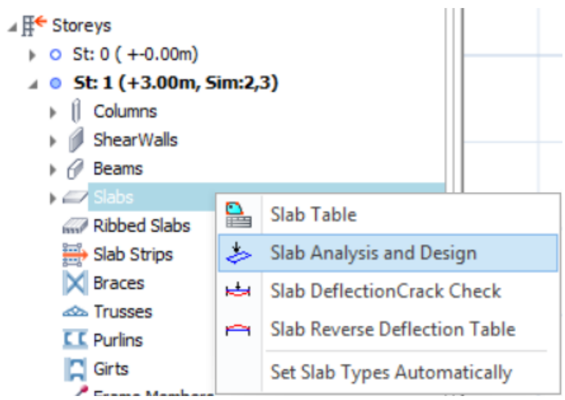
- *Select a column or wall on plan view → Right-click →* ***Analysis Results Diagram***
- *Alternatively, access the* ***Interactive Column Design*** *→ pick '****Diagrams****'*

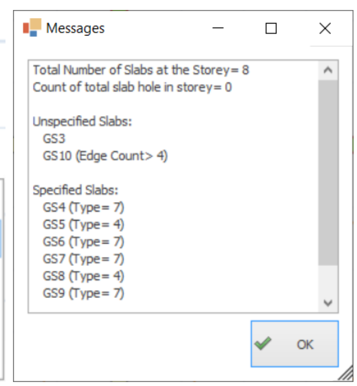
30. Slab Analysis & Design

For beam/slab models, slab reinforcement can be obtained by inserting slab strips in the X direction (horizontal plan) & Y direction (vertical plan). This process uses moment coefficient method from the tables in BS8110. This is independent of the general building analysis and can therefore be carried out before or after the general building analysis.

To use the moment coefficient method, it is important to set all of the **Slab Types** correctly in accordance with the tables in BS8110. This can be done automatically in a 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: St01** to return to the 1st Storey plan view*
- ***Right click** on Slab → **Set Slab Types Automatically** → Choose defaults options → **OK***

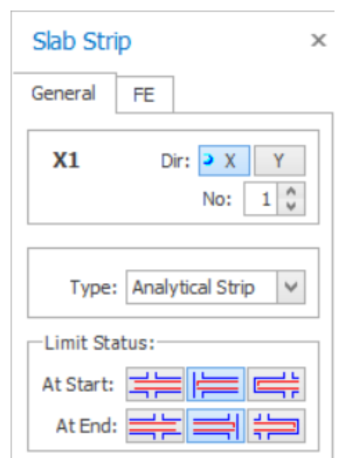




- *Click **OK** to close the Messages dialog.*
- *Go to **Modelling** tab & click on the **Slab Strip***

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 design based on FE Floor Analysis results.
- When drawing the strips, it is essential that the correct **At Start** and **At End** conditions are specified. The three options being:



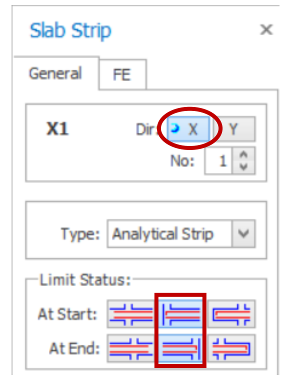
Slab - The strip starts or ends inside a slab. The bottom steel for the slab in question is not designed, but the span of the slab can be defined, and this value is used in determining the support steel.

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.

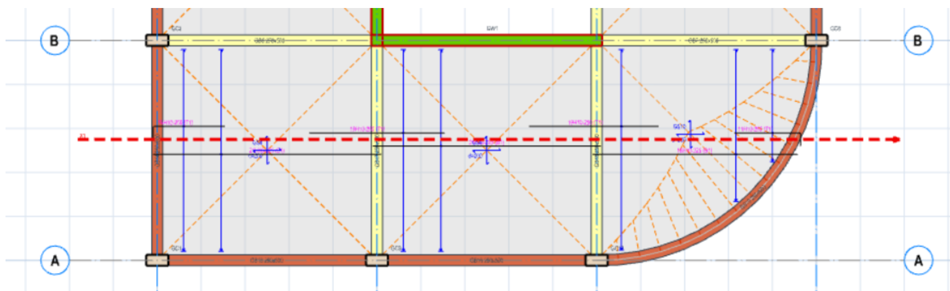
Cantilever - The strip starts or ends beyond a cantilever slab.

- *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***

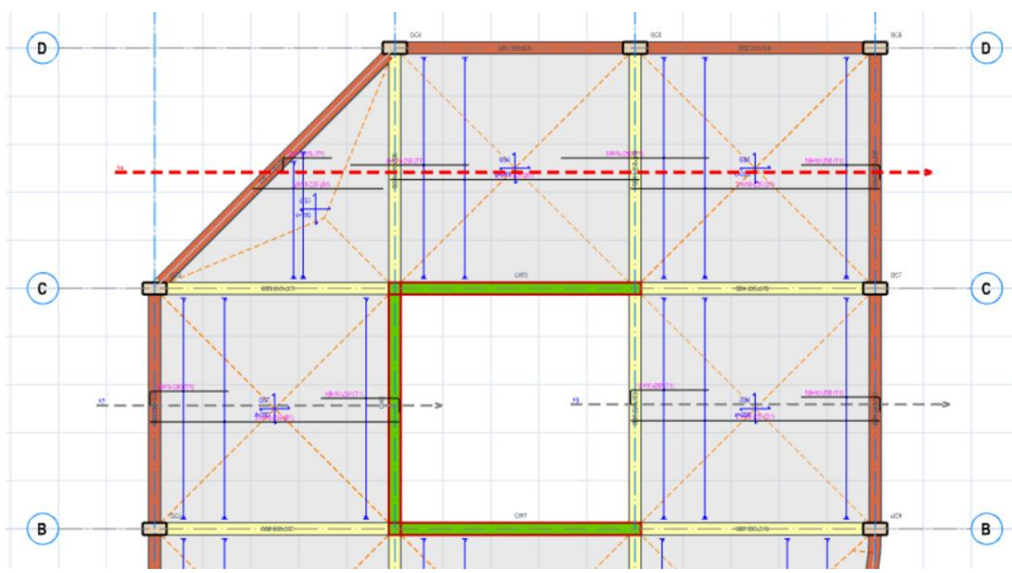
*Position your cursor between **Axis A and B**, but to the **left of Axis 1** (so that it is outside of the plan), then **click** to confirm the start of the strip*



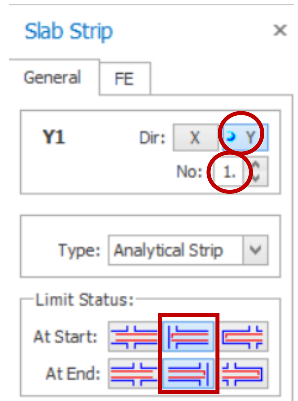
- ***Hold down on CTRL key** and then click the end of strip to the right of **Axis 4***



- *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***

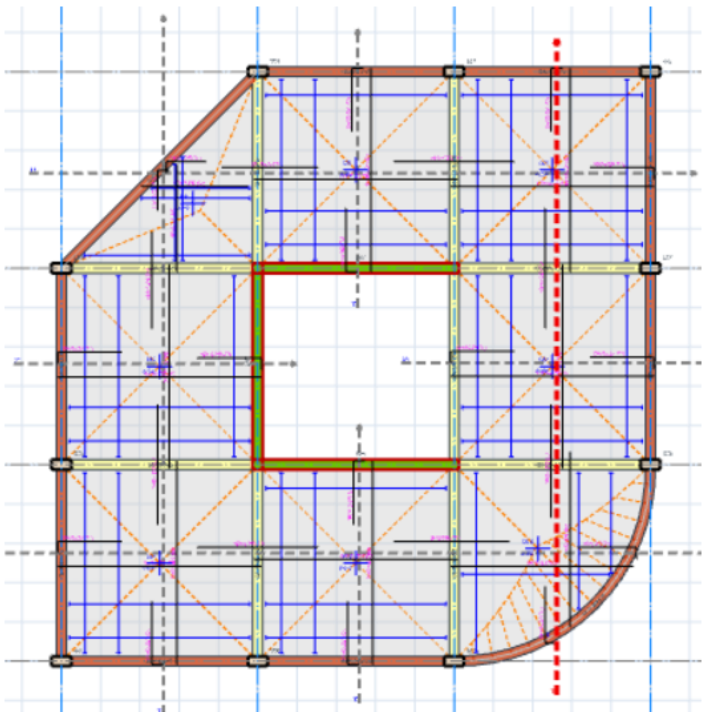


Now draw the **vertical strips** to design Y-direction rebar for the slabs.



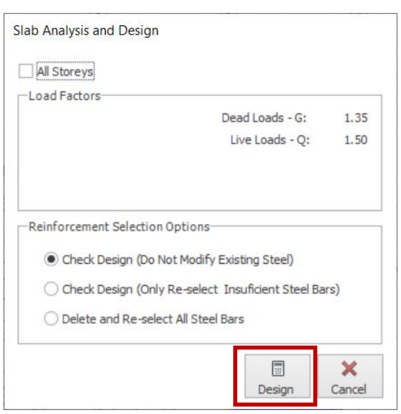
- *Change the **Dir** to **Y***
- *Reset the **No:** to **1** again so that the strip label is **Y1***
- *Keep **Type** as **Analytical Strip***
- *Keep the **At Start** condition as **Bob***
- *Also keep the **At End** condition as **Bob***
- *Draw **4** nos. of vertical strip through all of the slabs*

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



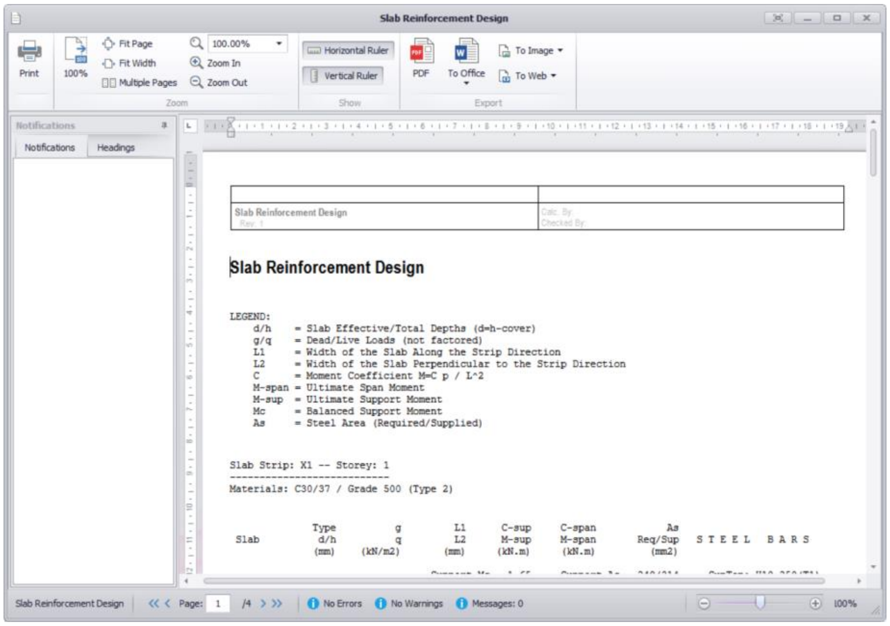
We can now re-check the strips as a batch and create a slab analysis & design report

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



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***

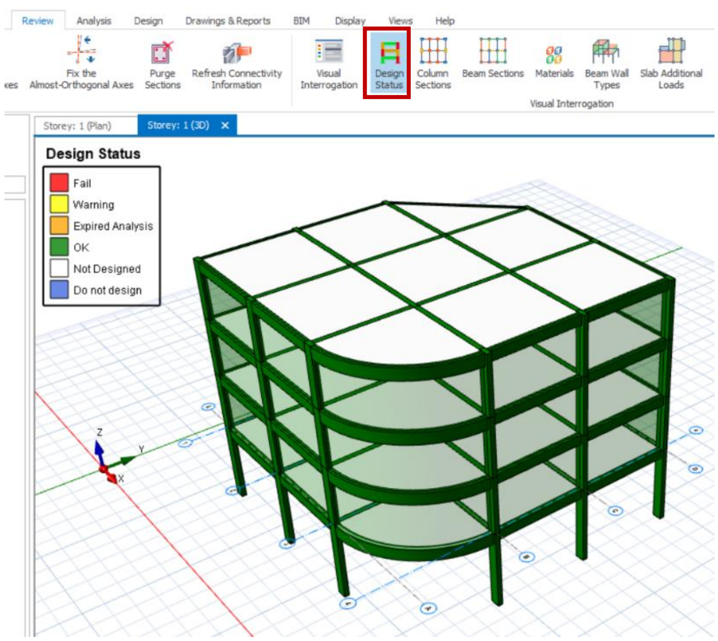


31. Design Status

The design status can be displayed graphically for in plan and/or 3D window

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

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

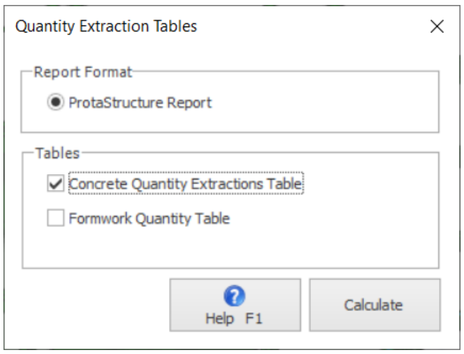


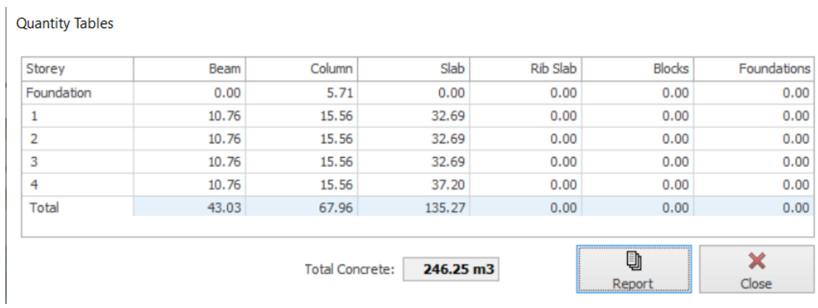
32. Quantity Extraction Tables

The concrete and Formwork quantity reports can be produced.

- *Go to **Drawings & Reports** tab → Pick **Quantity Extraction Tables***
- *Choose **Concrete Quantity Extractions Table** → click **Calculate***

This produces a concrete quantity report with member type and storey breakdown as shown below.





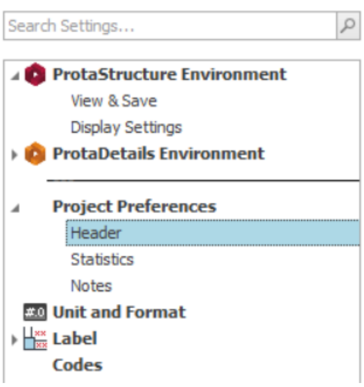
Storey	Beam	Column	Slab	Rib Slab	Blocks	Foundations
Foundation	0.00	5.71	0.00	0.00	0.00	0.00
1	10.76	15.56	32.69	0.00	0.00	0.00
2	10.76	15.56	32.69	0.00	0.00	0.00
3	10.76	15.56	32.69	0.00	0.00	0.00
4	10.76	15.56	37.20	0.00	0.00	0.00
Total	43.03	67.96	135.27	0.00	0.00	0.00

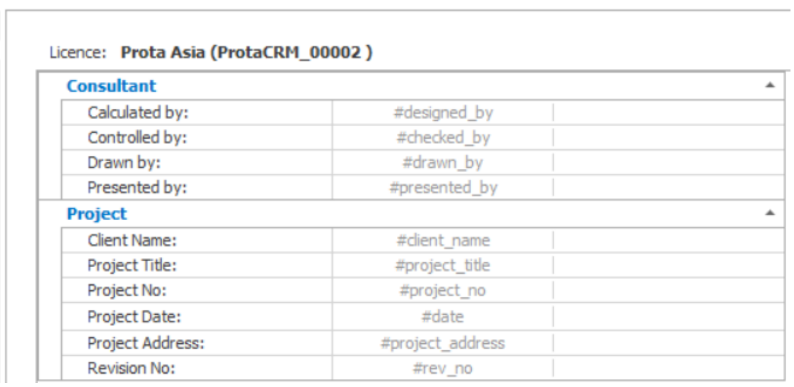
- *Click **Report** to produce a detail report*

33. Project Preferences

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

- *Click on **Settings Center** → expand **Project Preferences***





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

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

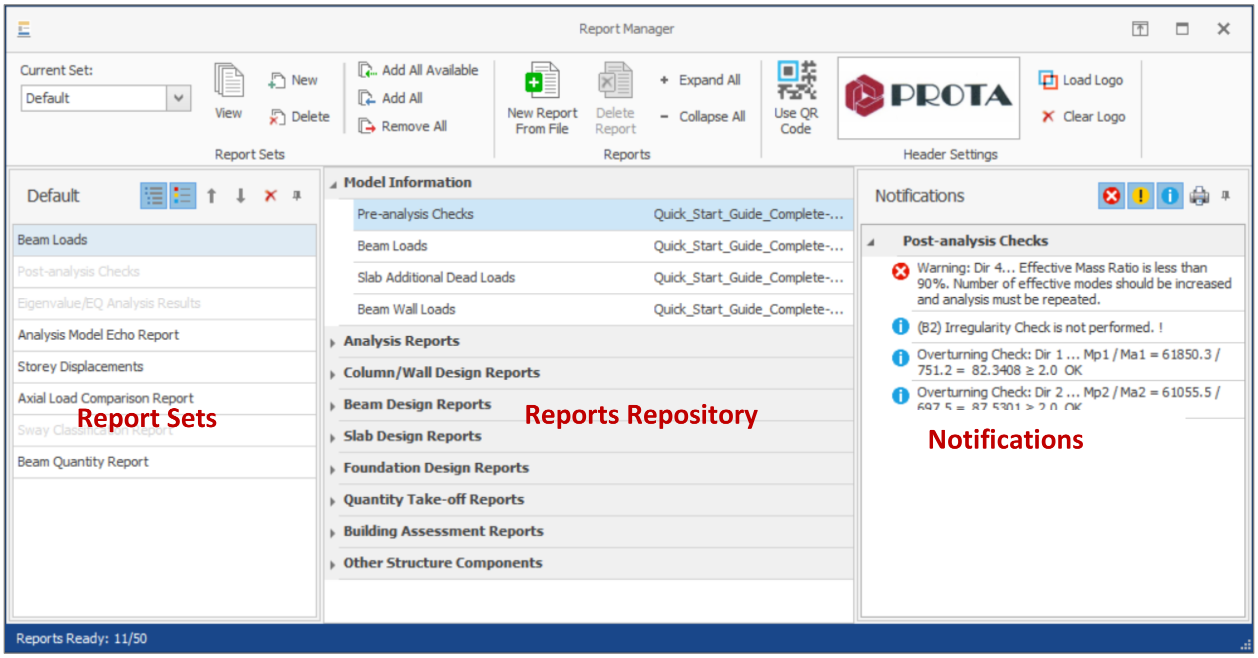
Notes - Enter additional information on this model, such as revision changes etc.

- *Key in the various information in the **Header** tab as desired*

34. Report Manager

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

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



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

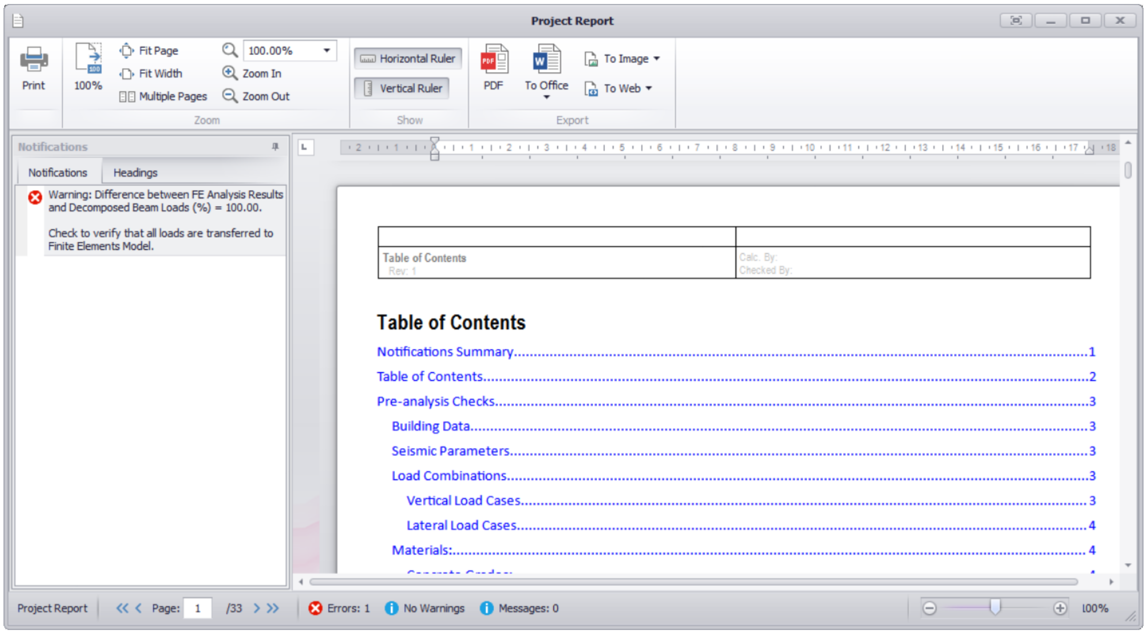
You can expand or collapse the main folder by clicking on topic icons. To create a combine report :

➢ *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 in include only reports that are available and generated 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 just below the Report Sets tab.*

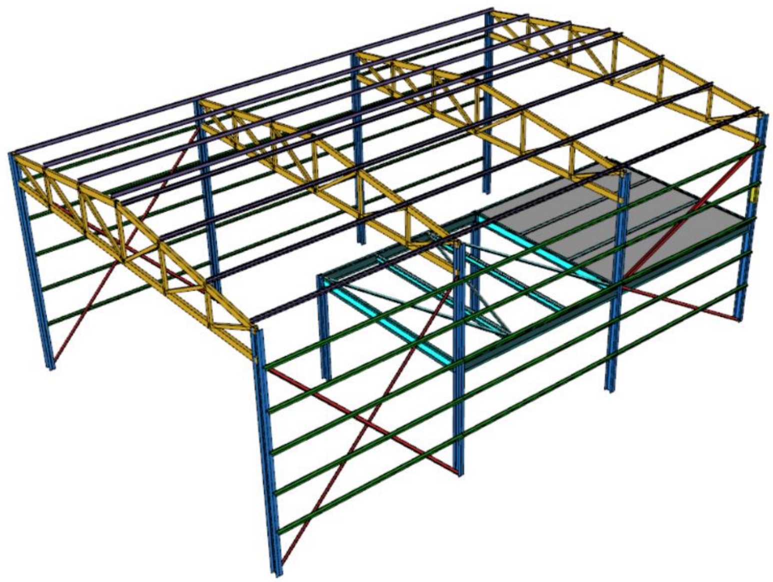
➢ *Select **View** to generate and view the report set*



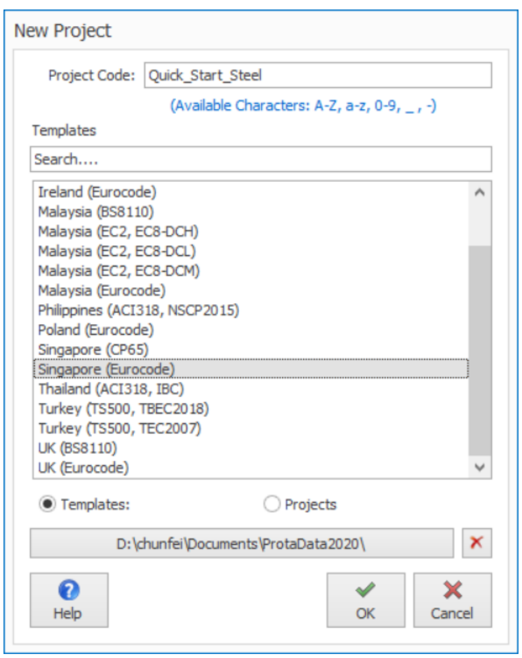
Other available functions include loading of company logo (**Load Logo**), inserting **QR Code** & inserting external reports (**New Report from File**).

35. 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.



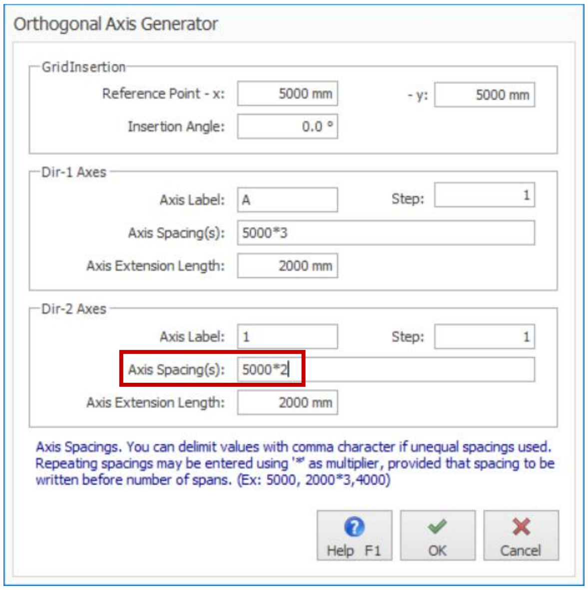
- *Click **New Project** and type the project name as shown using the '_' character for spaces.*

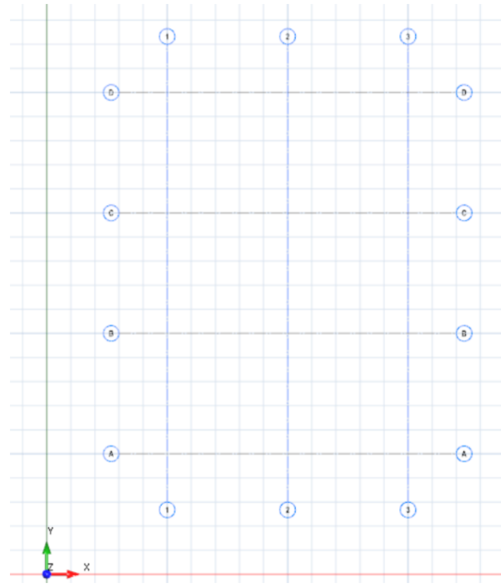


- *Select **Singapore (EuroCode)** template and then **OK***
- *Right click on **Axes** in the Structure Tree to expose the context menu*
- *Select **Orthogonal Axis Generator** in the **Modelling** tab*

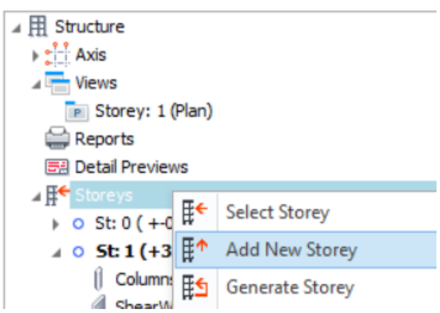
➢ *Pick the* ***intersection*** *of the major grid near the origin (5000,5000 coordinates)*

➢ *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***

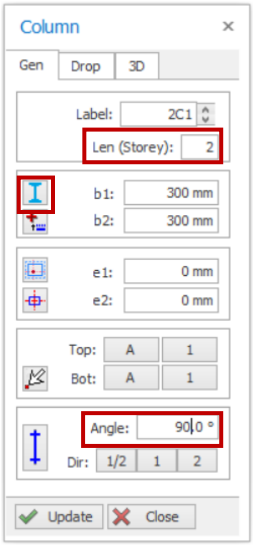


➢ *Input* ***Total No. of Storeys = 2*** *→* ***OK***

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

36. 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.

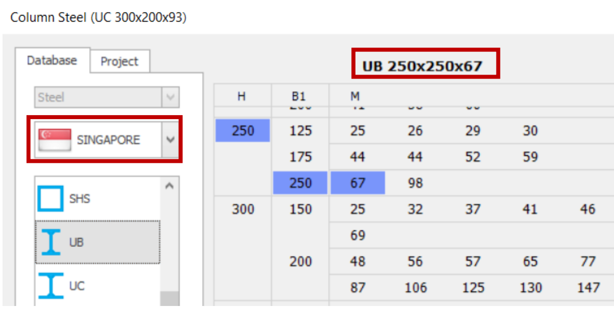
➢ *Change the **Angle** of rotation to **90 degrees***

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.



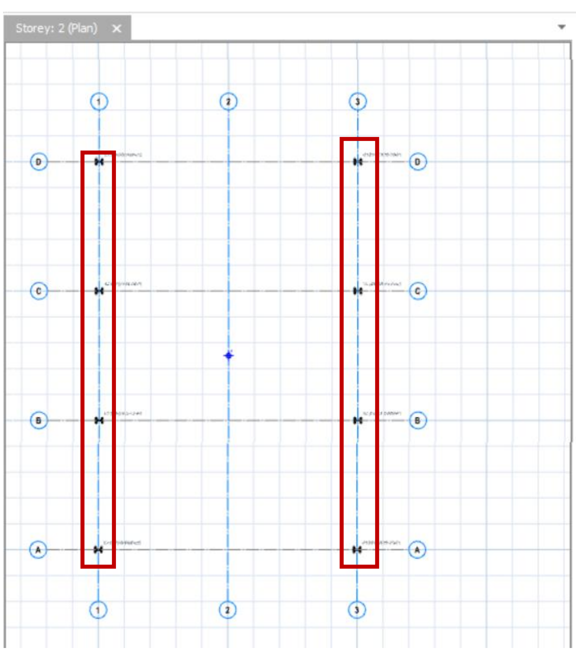
➢ *In **Section Manager** dialog, pick **Singapore***

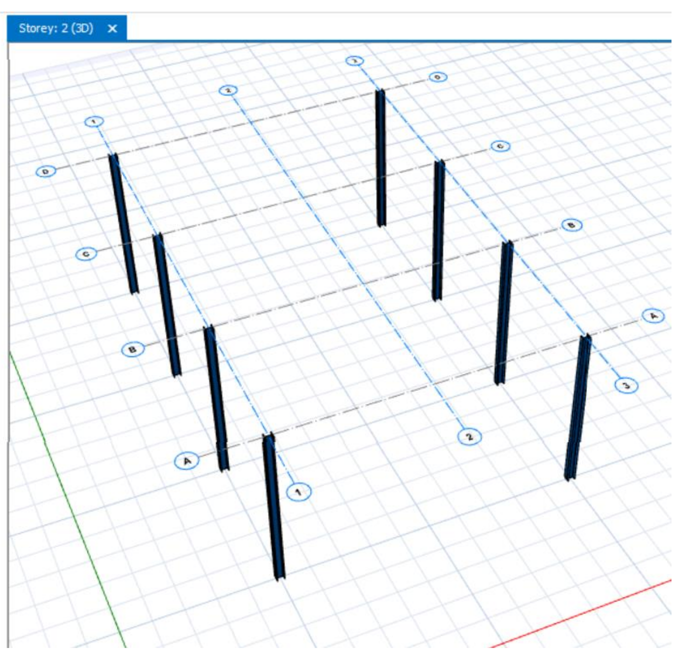
This will access to Singapore Steel Profiles (Continental)

➢ *Pick **UB 250x250x67** → **Select***

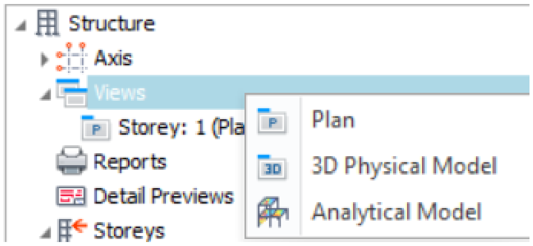
➢ *Pick **OK** to close the dialog*

➢ *Enter **8 nos** of columns along Axis **1/A-D** & **3/A-D***





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

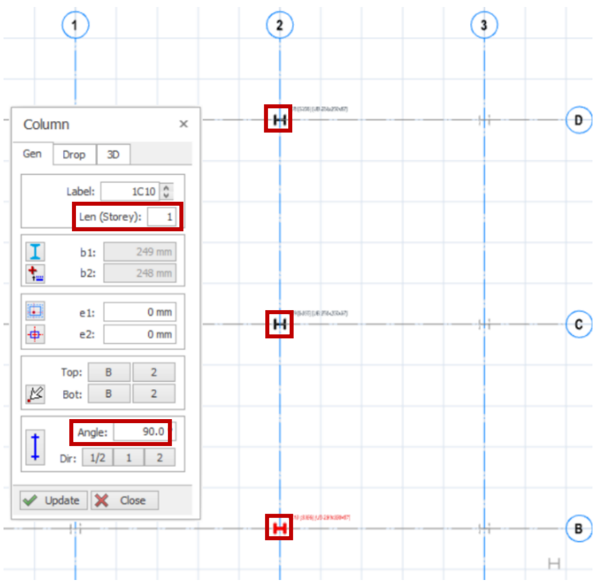


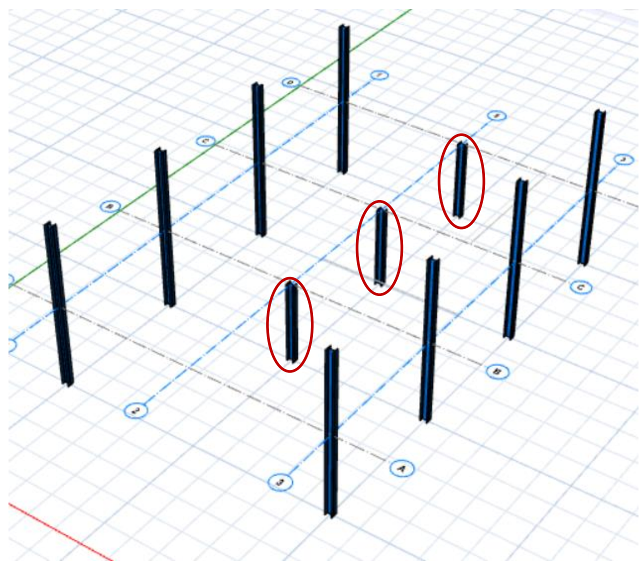
This will create a separate window showing the 3D view.

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

  This will arrange the 3D view and plan view in 2 windows.

- *Click on the* ***Plan*** *view to make it active (the active view border will be darker)*
- *Double-click on* ***Storey 1*** *in the* ***Structure Tree*** *to switch focus to Storey 1*





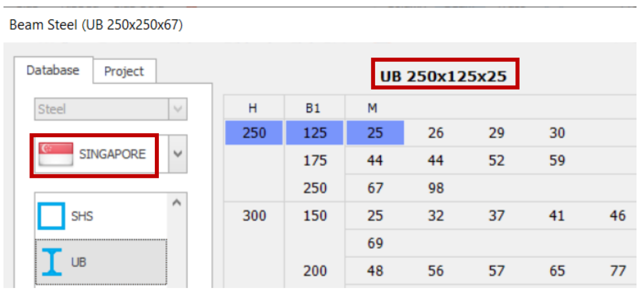
- *Click on* ***Steel Column*** *icon*
- *Change* ***Len (Storey)*** *to* ***1 in Column Properties***

  This means the column will only span one storey.
  The columns with len = 2 are shown in grey because they were inserted in ST02.

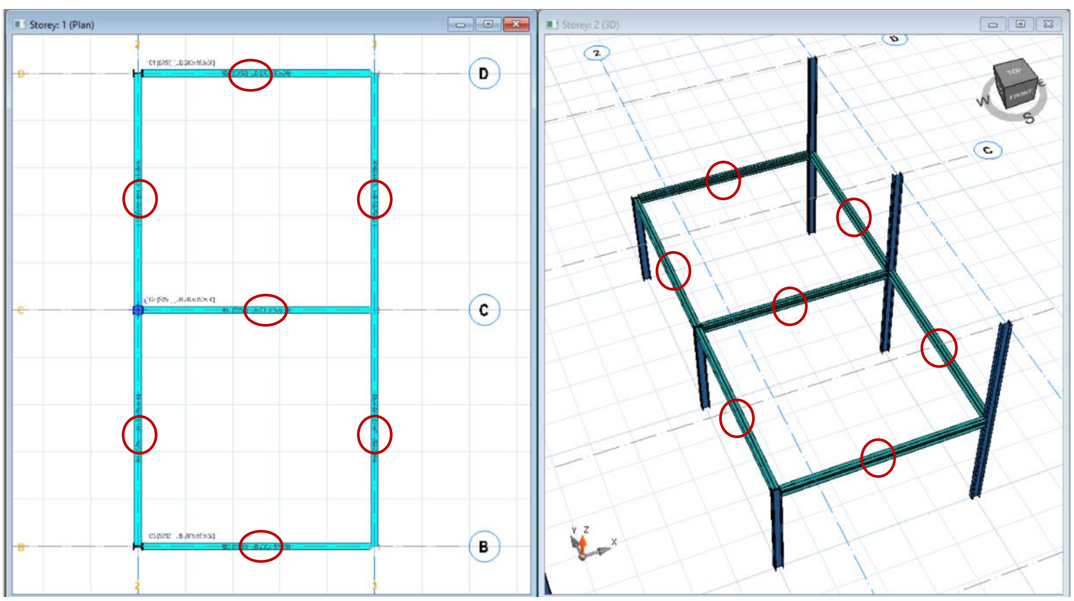
- *Ensure the* ***Angle*** *of rotation is* ***90 degrees***
- *Pick* ***Section Manager*** *icon*
- *Ensure that* ***UB 250x250x67*** *is selected*
- *Insert* ***3*** *nos. of column along GL2/B to D*
- *Check the* ***3D view*** *to ensure the columns were properly*

37. Steel Columns Creation

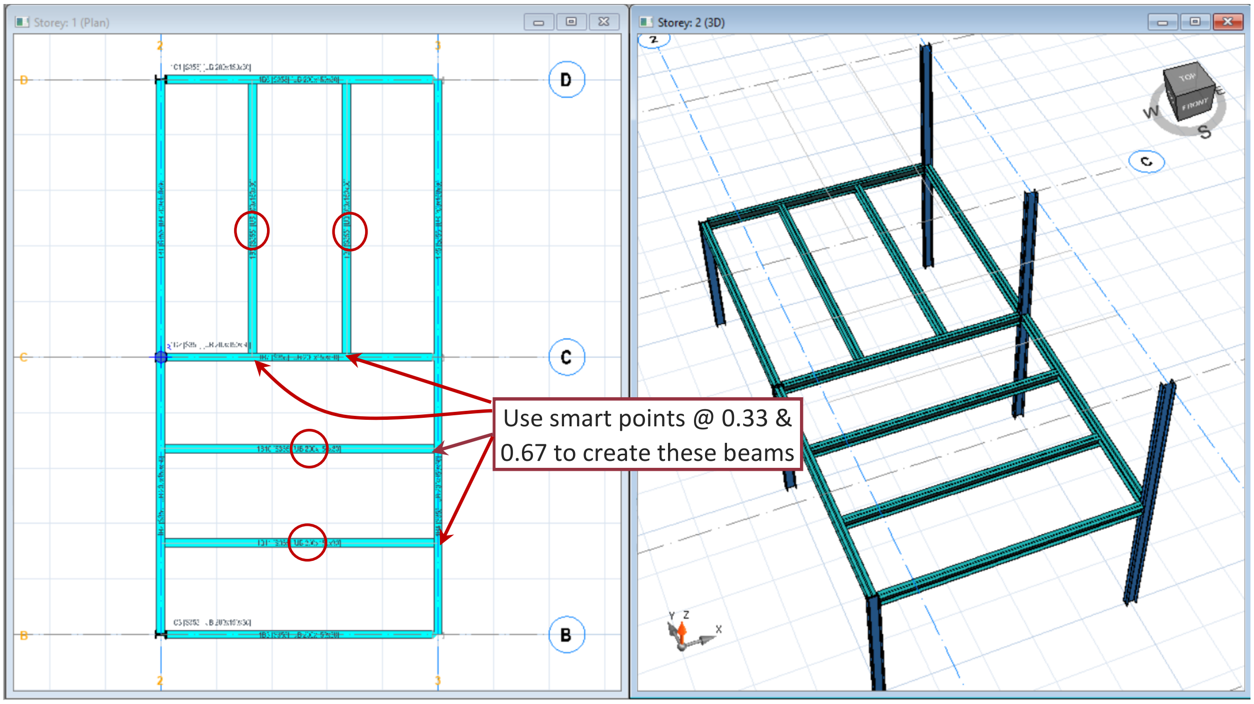
- *Click on **Steel Beam** icon*
- *Click on **Section Manager** in Beam Properties*



- *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*
- *Click on the 3D view to make it active (the active view border will be darker)*
- *Double-click on **Storey 1** in the **Structure Tree** to switch focus to Storey 1.*
  This is because we will now create beams in the 3D view.
- *In the 3D view, create **7 nos.** of beams in the region bounded by GL B,D,2 & 3 (as shown below)*



- *Go to the **plan view** of **ST01** → Create **4 nos.** of **UB 200x150x30** secondary beams in the region bounded by GL **B,D,2** & **3** (as shown below)*



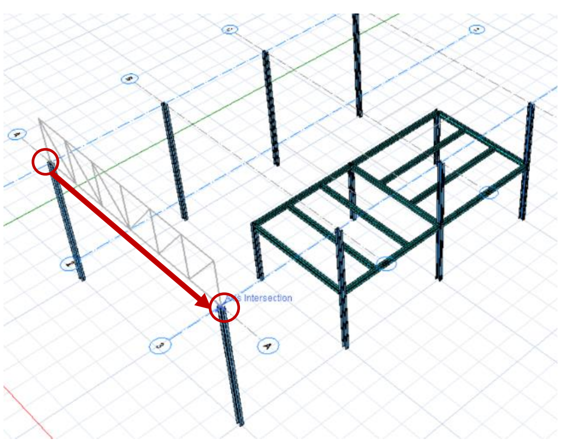
Tip : Use smart points to create secondary beam. With beam properties out, place the mouse cursor at the edge of the primary beam (not center) & the smart points will appear at **0.25L**, **0.33L**, **0.5L** & **0.75L**.

Use **Beams Sections** in the **Review** tab to show beams of different sizes by color. Assign colors to members in **Section Manager** dialog → **Material Color**

38. Steel Truss Creation

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

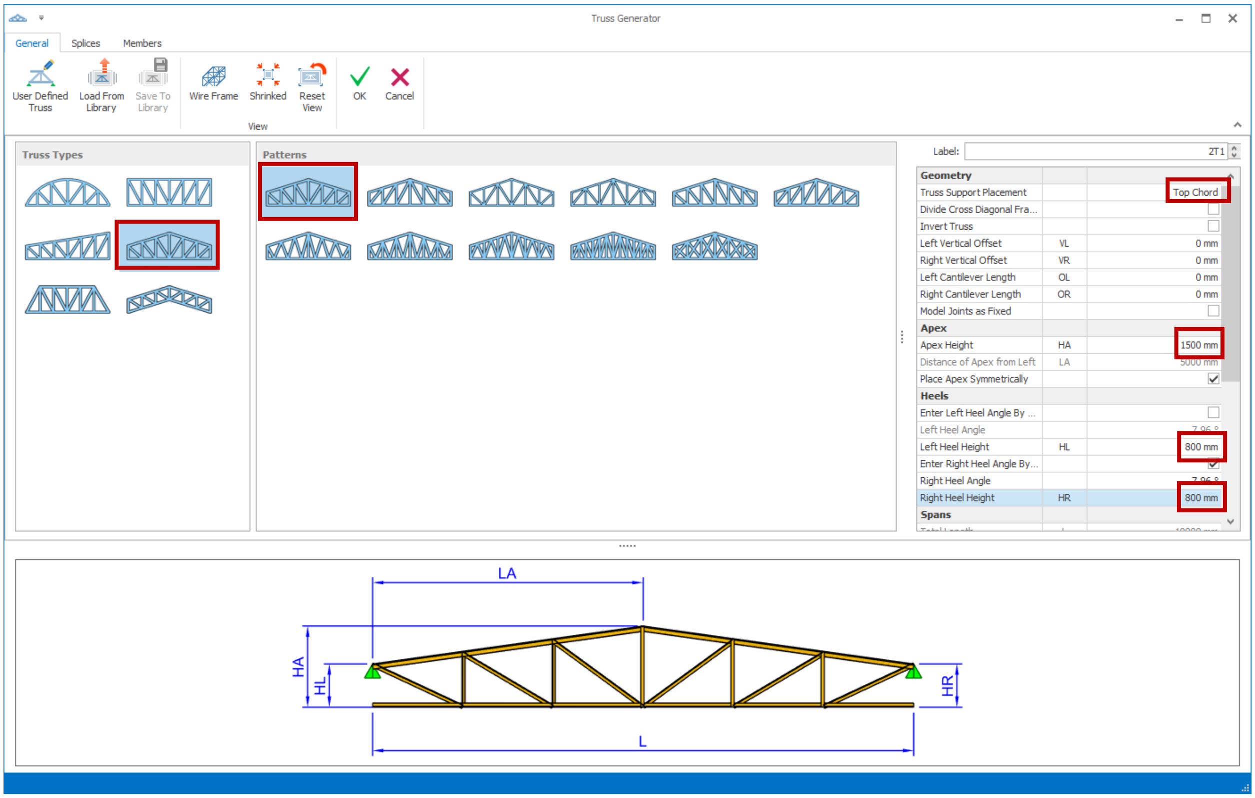
- *Go to the* ***3D view*** *→ Double-click on* ***ST02*** *in Structure Tree to make it active*
- *Click on* ***Truss*** *button*



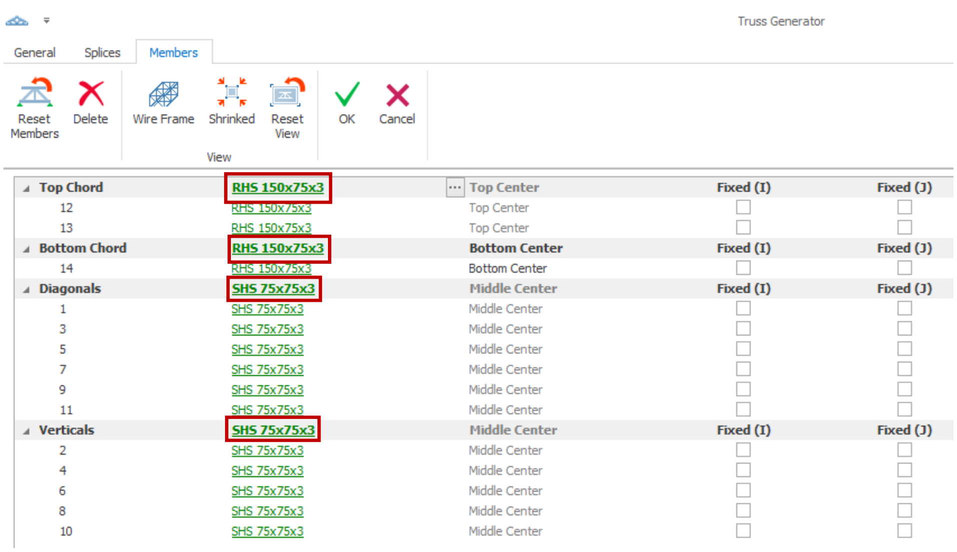
- *Pick* ***grid intersections A/1*** *to define the* ***start*** *of the truss*

  A dynamic rubberband 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.



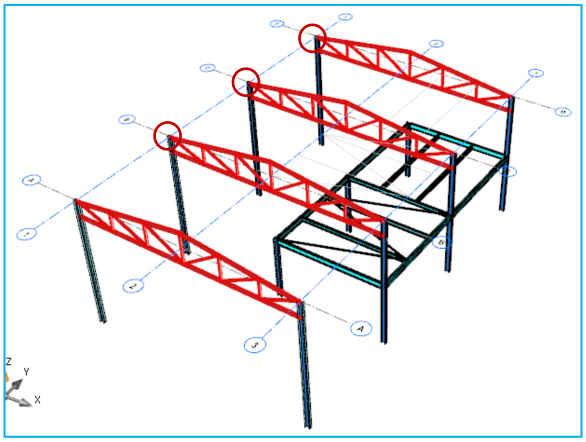
- In **Geometry**, pick **Duo-Pitched** truss → Change **Truss Support Placement** to **Top Chord**
- Change **Apex height** to **1500mm**, **Left & Right Heel height** to **800mm**
- Click on the **Members** tab



- Change the truss member sections to the following by click on the section name:
  - Top & bottom chord = **RHS 150x75x3** (Singapore)
  - Diagonal & Vertical = **SHS 75x75x3** (Singapore)

- *Click **OK** to exit the Truss Generator dialog.*

The truss will be inserted. We now copy the truss to the rest of the columns.

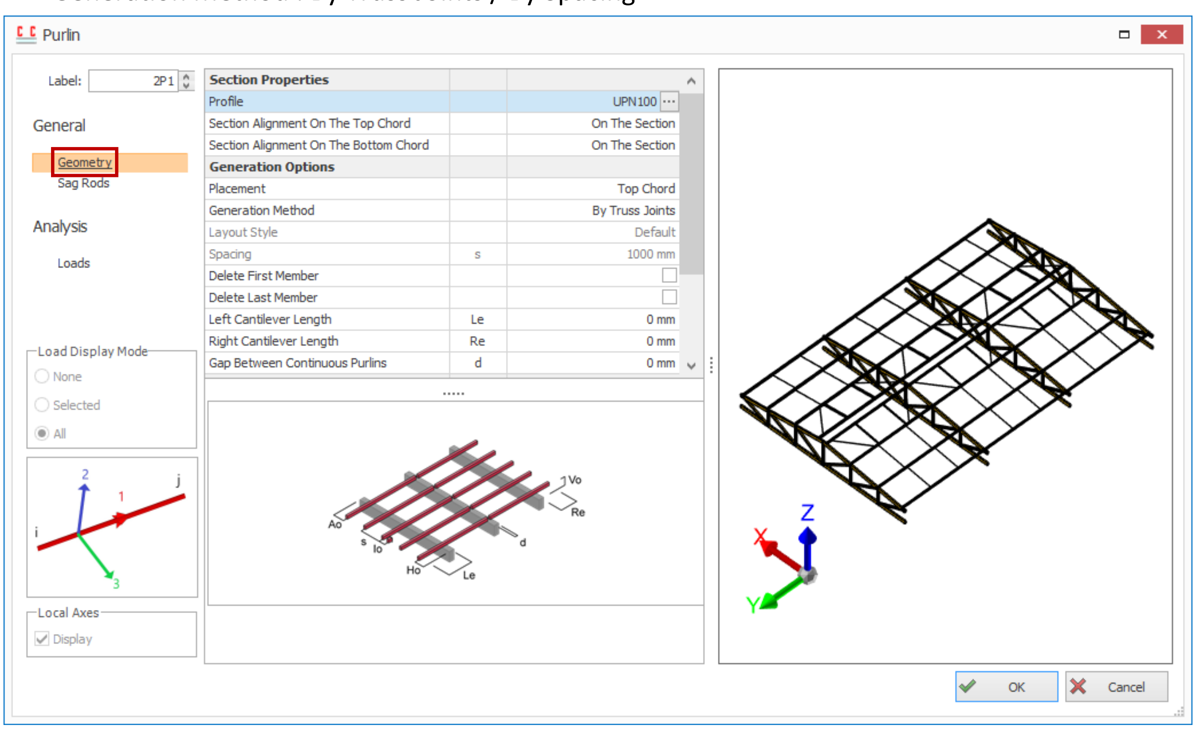


- *Select the truss → Right-click → Pick **Copy***
- *Select the basepoint @ intersection of **A/1***
- *Insert **3 nos**. of similar trusses by clicking grid intersection **B/1**, **C/1** & **D/1***
- *Press **ESC** or **Right-Click** to end the copy operation*

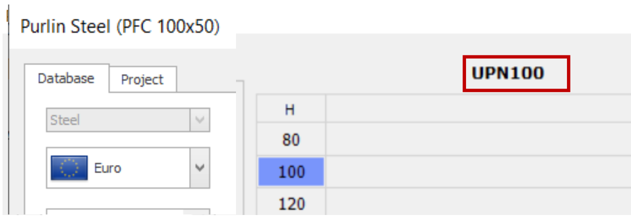
39. Purlins Creation

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

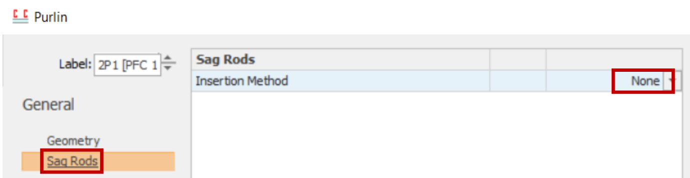
- *Click on **Purlin** toolbar button*
- *Select the **first** truss @ **GL A** →Select the **last** truss @ **GL D** (Intermediate trusses will be automatically found)*
- *On **Purlin Dialog**, you can specify the following:*
    - Profile / Section of the for the purlin
    - Section Alignment: On the Section / Under the Section / Center
    - Generation Method : By Truss Joints / By Spacing



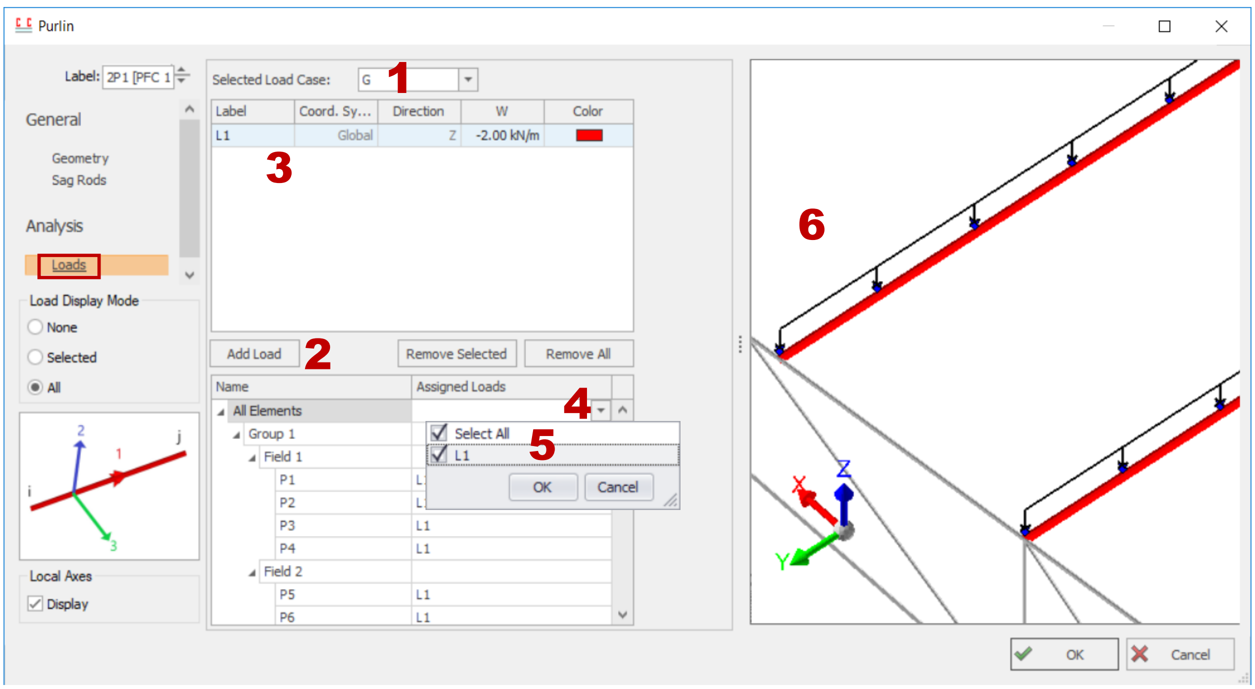
➢ *Choose section **UPN100** (under **Steel** → **European sections** → **UPN**)*



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



➢ *In the **Loads** dialog, you can apply loads to purlins :*



1 *Select the **Load Case** to insert the loading*

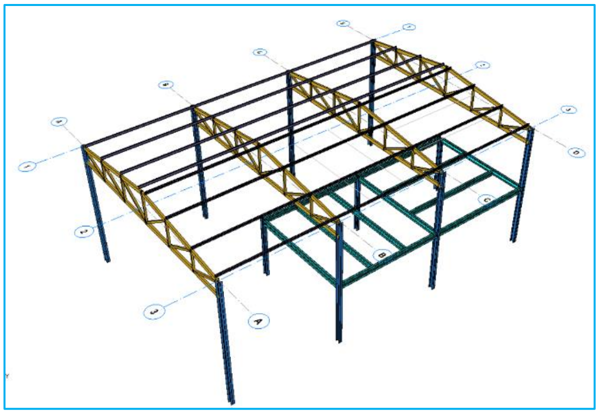
2 *Click **Add Load** → Load Label **L1** will be created*
You can continue adding load as required, eg. the last end-span purlin will have lesser load

3 Input the loads **Coordinate System**, **Direction** & **Value**

4 Assign the loads by picking on **Assigned Loads** for **All Purlins** or invididual **Purlin** (P1, P2, etc)

5 Pick the **Load Label** to assign → **OK**
You may find it easier to first assign the most common load to all the purlins & then change specific purlins later.

6 **Check** the diagram to ensure the loads are applied correctly (zoom in using mouse wheel)



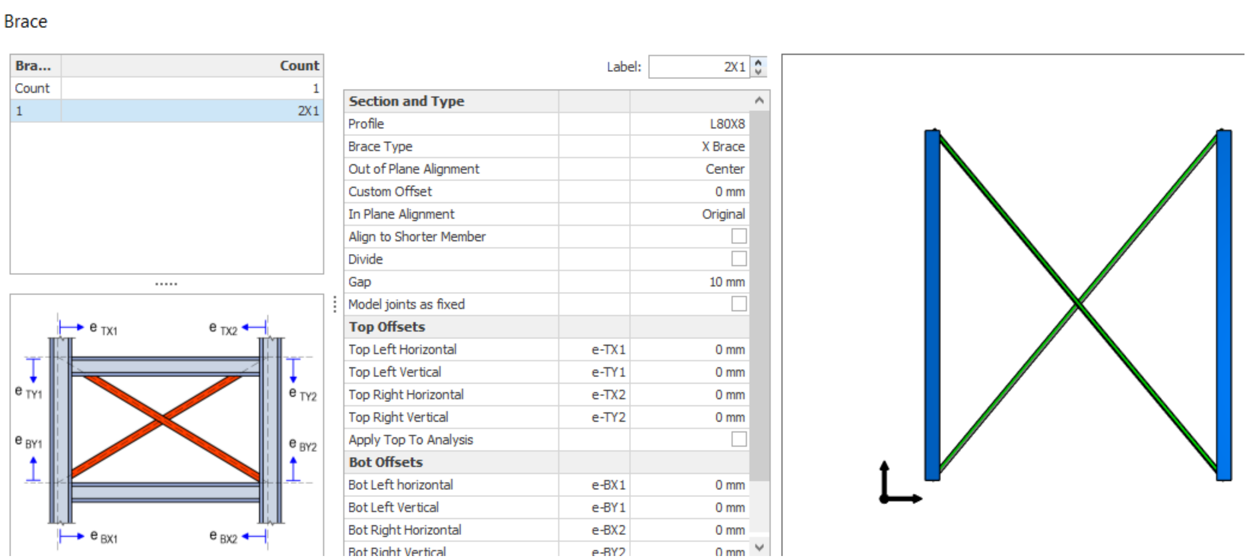
- ➢ *Following the above steps, add a **direction Z** load of **-2.00 kN/m** for **G Load Case***
- ➢ ***Assign it** too all purlins*
- ➢ *Check the right diagram to ensure the loads are applied correctly (item 6 above)*
- ➢ ***OK** to close the purlin dialog*

Purlins will be inserted on top of all the trusses.

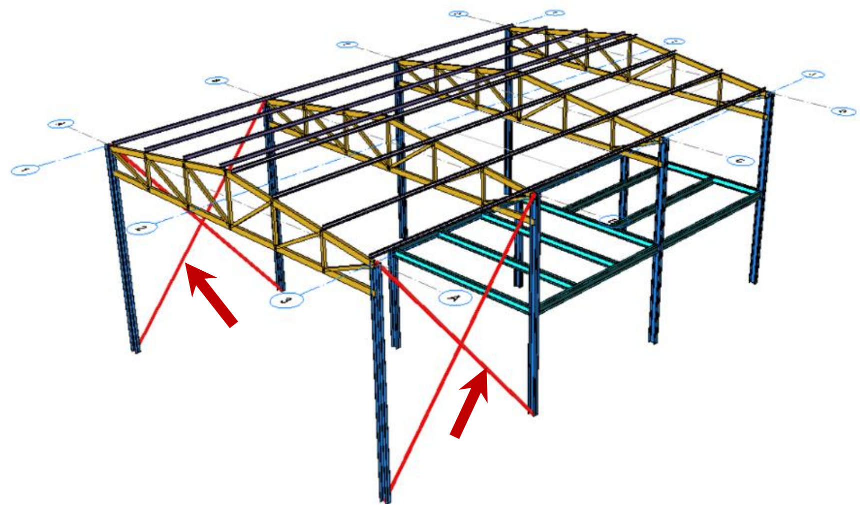
40. Braces Creation

We will now insert some bracings between steel columns.

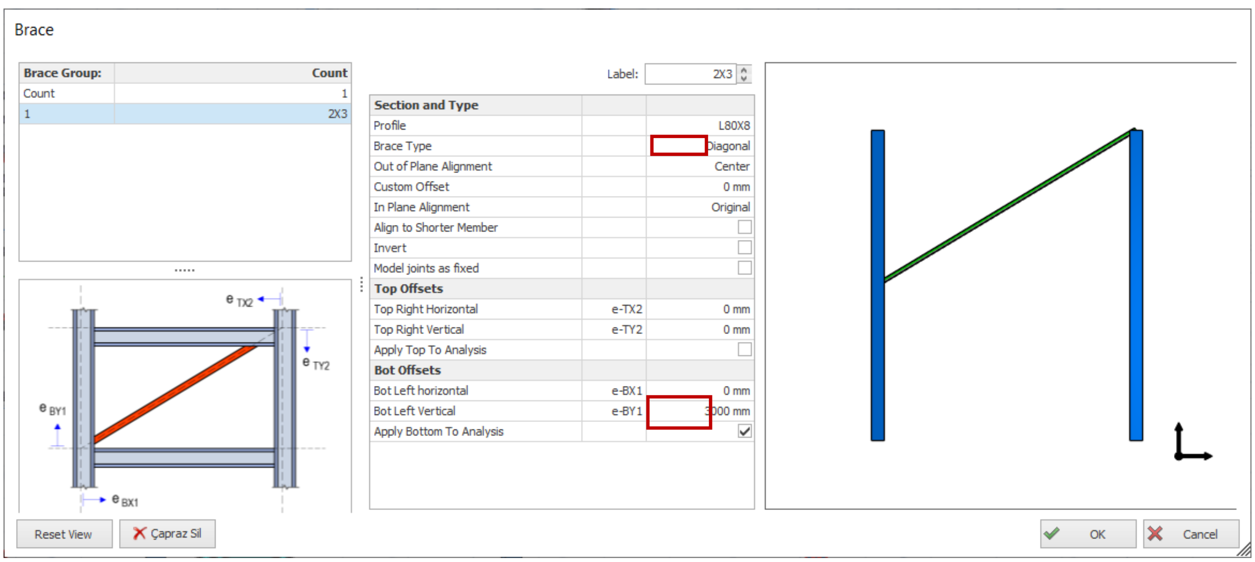
- ➢ *Click on the **Brace** toolbar icon*
- ➢ *Pick two adjacent columns at **A/1** & **B/1** → Brace Dialog will appear*



- ➢ *In Brace dialog, you can specify the following:*
  - Profile / Section of the for the purlin
  - Brace Type (Diagonal, X Brace, Y Brace or K Brace)
  - Alignment & Top/Bot Offsets
- ➢ *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***

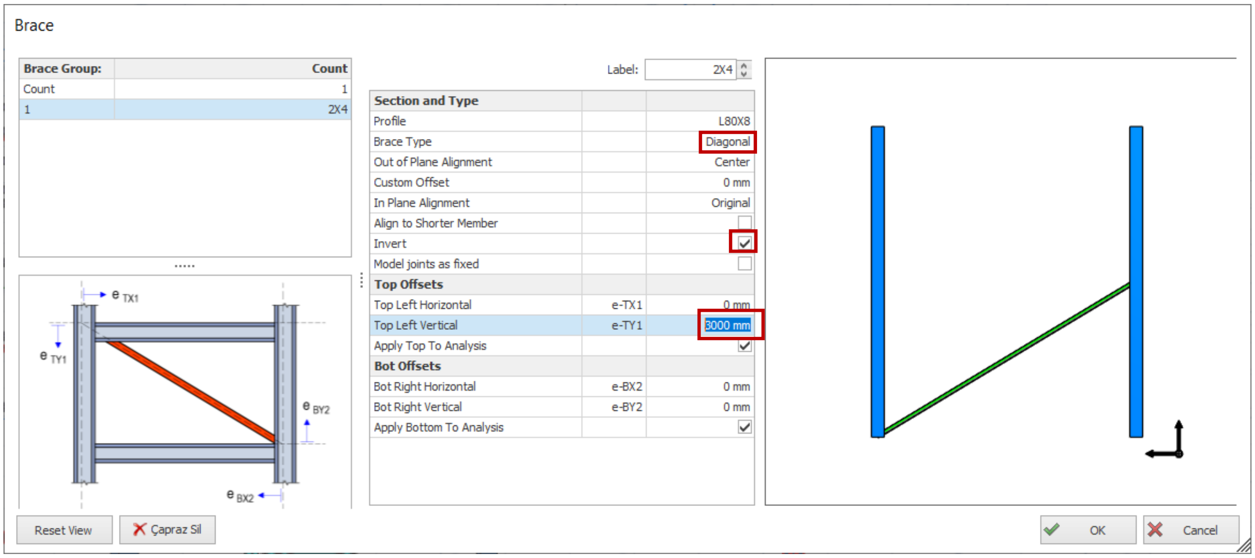
➢ *Input **Bot Let Vertical Offset** = **3000 mm***

➢ *Tick **Apply Bottom To Analysis***

This will ensure the analysis frame will accurately consider this offset.

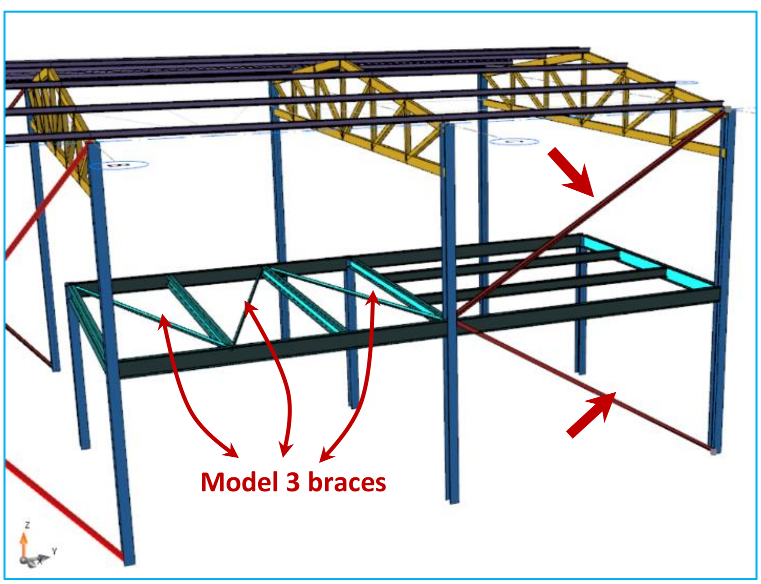
➢ *Click **OK** → Diagonal Brace will be inserted*

➢ *Insert another new brace between same columns **C/3** & **D/3***



- *Ensure **Brace Type** = **Diagonal***
- *Tick **Invert** → this will invert the diagonal*
- *Input **Top Let Vertical Offset** = **3000 mm***
- *Tick **Apply Top To Analysis***

  This will ensure the analysis frame will accurately consider this offset.
- *Set all **Bot Offsets** to **0 mm***
- *Click **OK** → A new diagonal brace will be inserted*
- *Check the braces are correctly created in the 3D view as below*



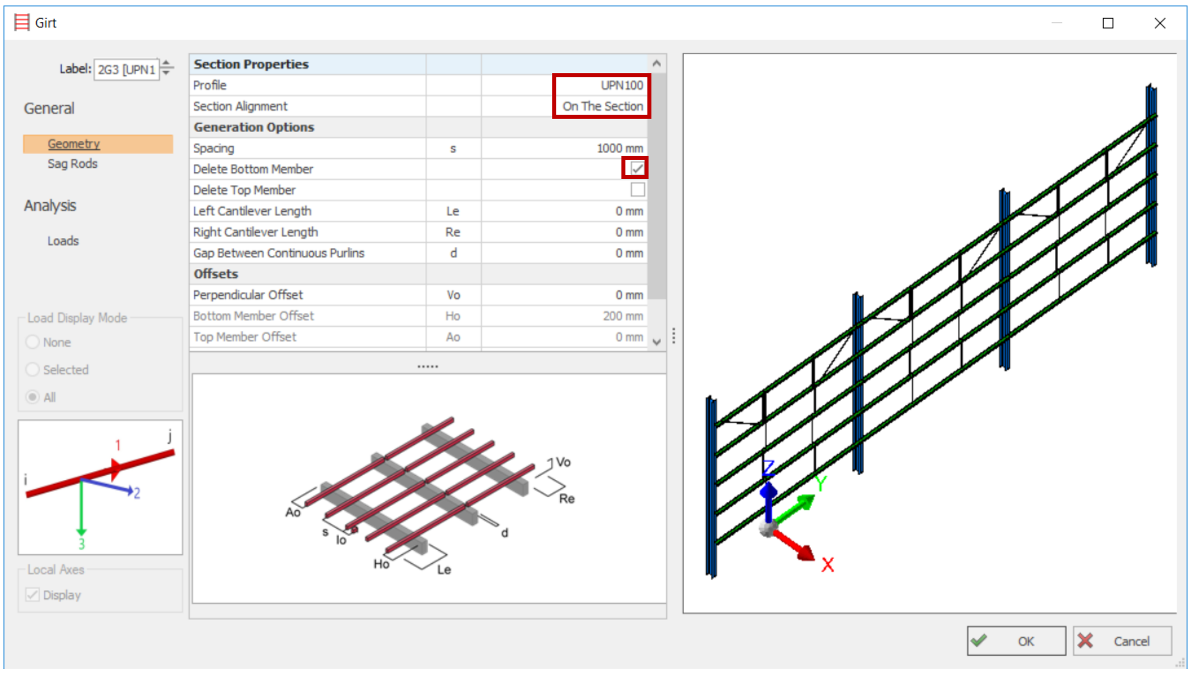
Braces can also be inserted between beams. Create 3 nos of horizontal brace connecting the beams in ST01 as shown in the left figure:

- *Pick 2 adjacent beams*
- *In the Brace dialog, pick **Brace Type** = **Diagonal***
- *Tick / Untick **Invert** as required*
- *Ensure all **Top** & **Bottom Offsets** = 0*
- *Click **OK***
- *Check the brace is correctly created in the 3D view.*

41. Girts Creation

We will now insert some girts between steel columns.

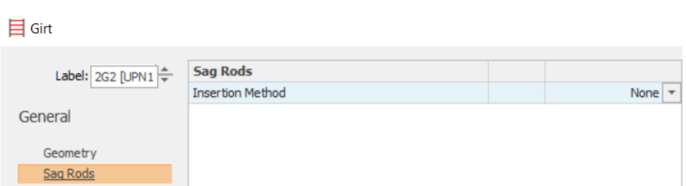
- ➢ *Click on* ***Girt*** *toolbar button*
- ➢ *Select the* ***first column*** *at grid* ***A/1*** *→ Select the* ***last column*** *at grid* ***C/1****.*
  Intermediate columns will be automatically found.
- ➢ *In the* ***Geometry*** *dialog, you can specify the following:*
  - Profile / Section of the girt
  - Section Alignment : Under The Section / On The Section
  - Spacing, Delete Bottom / Top Member, Left / Right Cantilever, Offsets



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

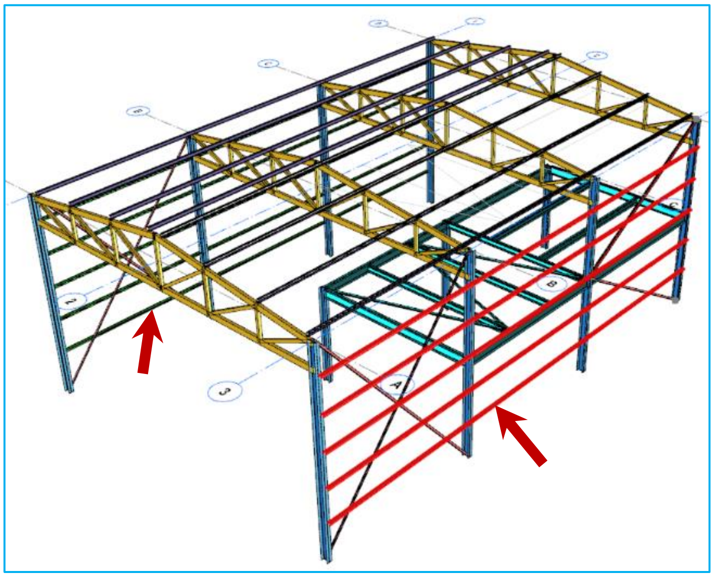
In the Sag Rods dialog, you can specify the configuration of the sag rods. For simplicity, for this model we will not insert the sag rods.

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

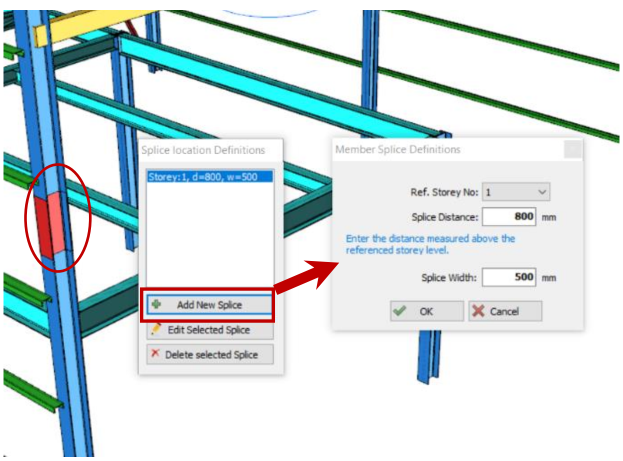


- ➢ *Click* ***OK*** *and the girts will be inserted.*
- ➢ *Insert similar girts between columns* ***A/3*** *&* ***C/3*** *but* ***Section Alignment*** *=* ***On the Section***

- *Check the girts are inserted correctly at both sides of the building as shown below*



42. Column Splice Creation

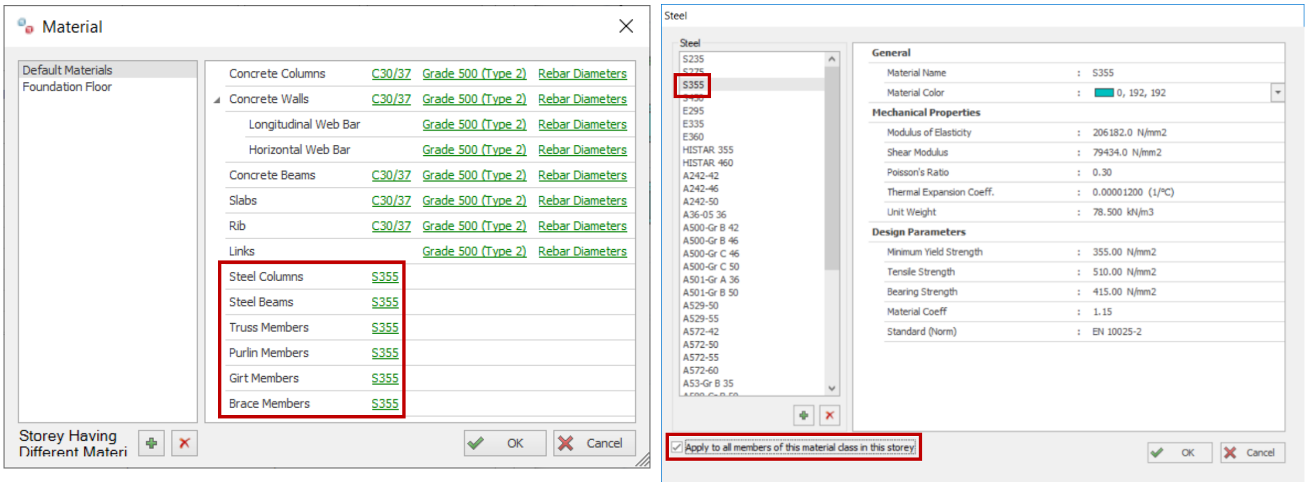


- *Select the corner* ***column*** *at* ***GL D/3***
- ***Right-click → Define Splice***
- *Pick* ***Add New Splice*** in *Splice Location Definitions*
- *In* ***Member Splice Definition*** *choose* ***:***
  - Reference Strorey = 1
  - Splice Distance (from floor level) = **800** mm
  - Splice width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

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

43. Building Analysis

- *Go to **Analysis** tab → **Building Analysis** → **Pre-Analysis** tab*
- *Pick **Edit Materials** → Change the steel grade of all member to **S355***

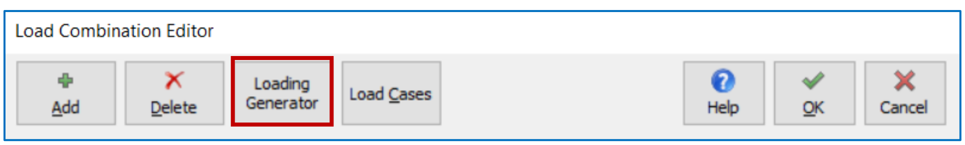


❖ The global materials grade of the truss, purlins, girts & brace members are shown & can be changed.

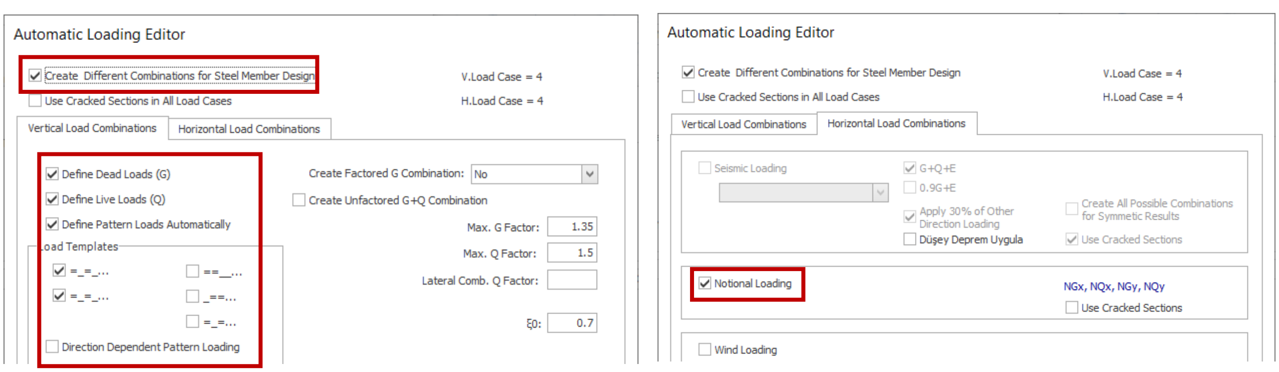
❖ Individual member material can also be changed in their respective **Section Manager** dialog by selecting the member → Right-click → **Edit Section / Material**.

We will now auto-generate the load cases and load combination.

- *Pick **Loading Combination** to access the Load Combination Editor*



- *Pick **Loading Generator** → Pick the options as shown below → click **OK***



All the load cases and combination will be generated automatically.

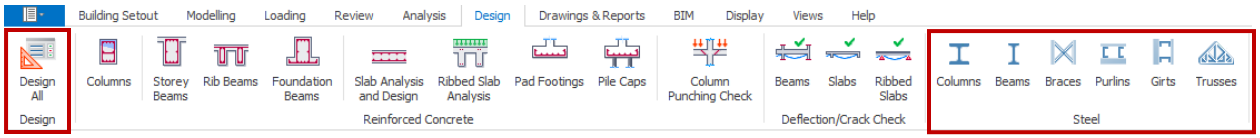
- *Go to **Analysis tab** → Tick **Building Analysis** → **Start***
- *In Batch Design option, choose not the design any members → click **Building Analysis***

  Once analysis is completed, the **Analytical Model** view will open automatically.

- ***Review the results** to your satisfaction as outlined in the previous section*

44. Steel Design

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

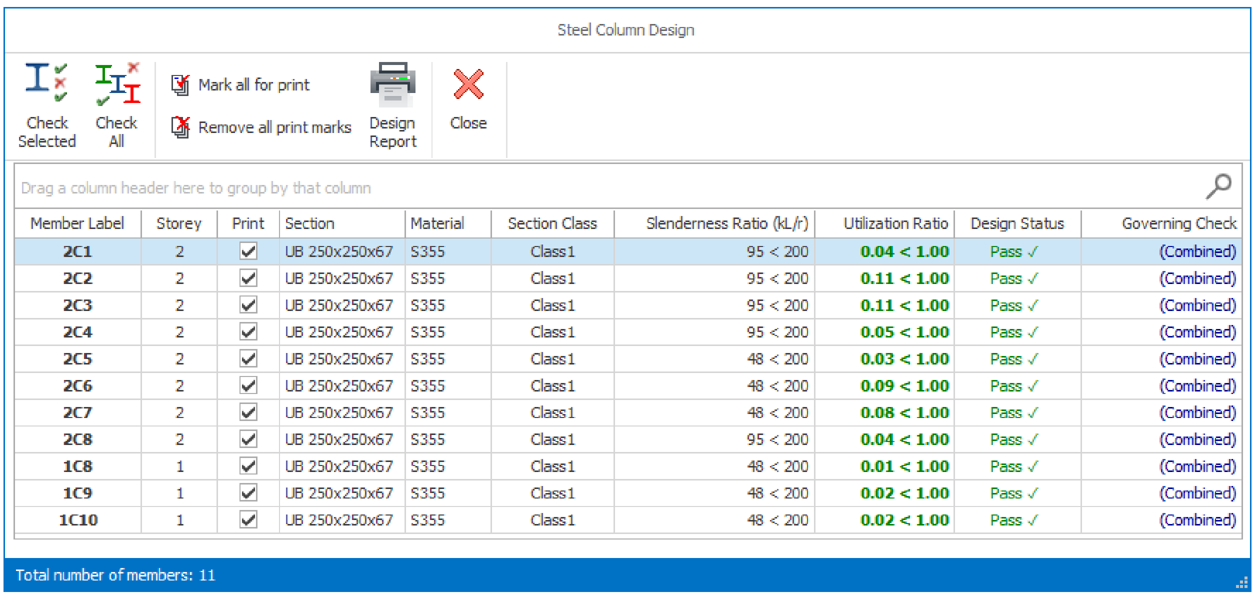


- *Go to **Design** tab → **Design All** → Choose **Steel Member Design Check** → **OK***

Notes:

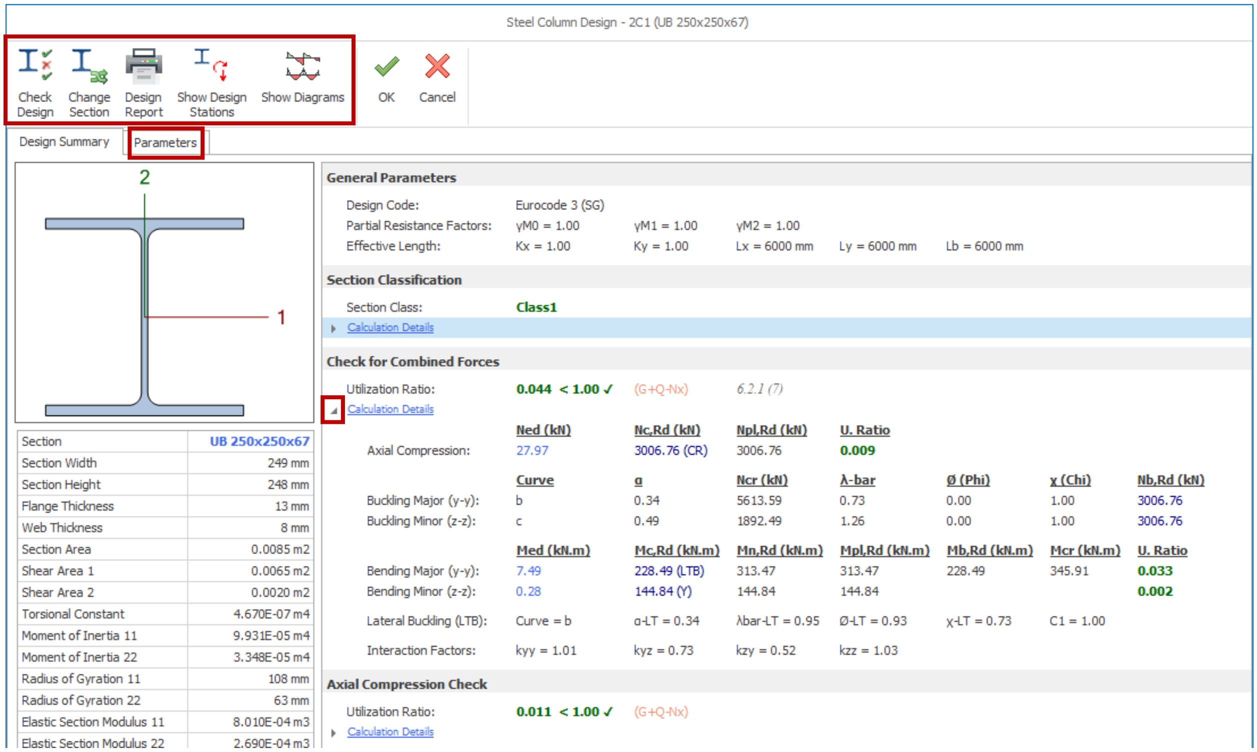
- If the model is large, it is recommended to perform design check by member types separately.
- To check the design of a particular member, **select** the steel member icons in the steel group.

- *Pick steel **Column Design***



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	0.04 < 1.00	Pass ✓	(Combined)
2C2	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.11 < 1.00	Pass ✓	(Combined)
2C3	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.11 < 1.00	Pass ✓	(Combined)
2C4	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.05 < 1.00	Pass ✓	(Combined)
2C5	2	☑	UB 250x250x67	S355	Class1	48 < 200	0.03 < 1.00	Pass ✓	(Combined)
2C6	2	☑	UB 250x250x67	S355	Class1	48 < 200	0.09 < 1.00	Pass ✓	(Combined)
2C7	2	☑	UB 250x250x67	S355	Class1	48 < 200	0.08 < 1.00	Pass ✓	(Combined)
2C8	2	☑	UB 250x250x67	S355	Class1	95 < 200	0.04 < 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	0.02 < 1.00	Pass ✓	(Combined)
1C10	1	☑	UB 250x250x67	S355	Class1	48 < 200	0.02 < 1.00	Pass ✓	(Combined)

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



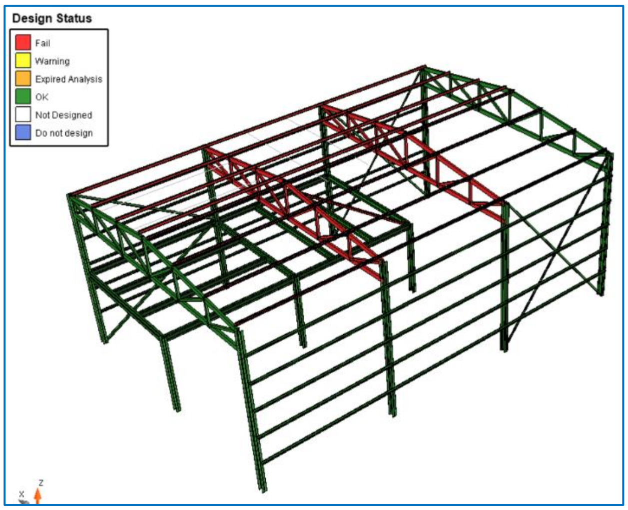
- *Click on the triangle ▶ next to the title check to drill down to more details*
- ***Check Design*** *→ Re-check the design of this column*
- ***Change Section*** *→ Allow you to pick another section → New section will be checked automatically for pass / fail.*
- ***Design Report*** *→ Prepare the 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*** *→ Input Braced length, Buckling Length Coefficient, Lateral Torsional Buckling length*

45. Design Status & Design

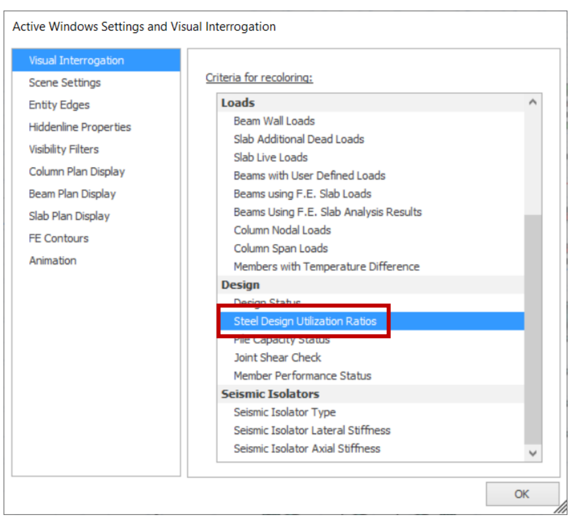
We can view **Steel Design Status** and **Steel Design Utilization** to quickly review pass / failure status & design efficiency.

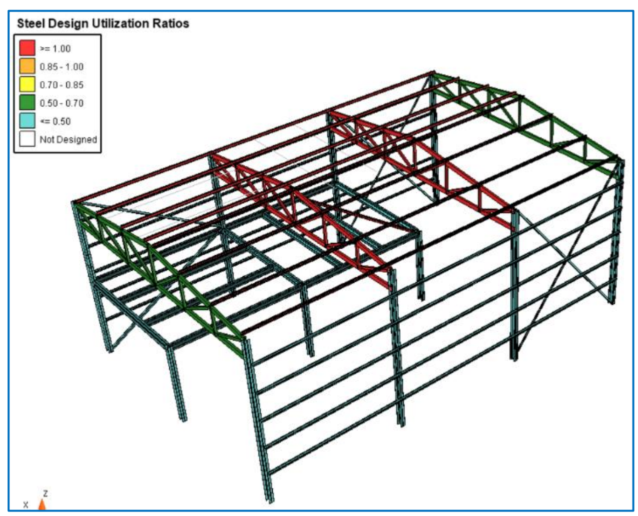
The design status can be displayed graphically for in plan and/or 3D window

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



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





You may want to check the design of the rest of the steel members such as steel beams & truss. The design interface is similar to that of the column design.

46. Closing Summary

Congratulations! You have created your first model in **ProtaStructure**.

In this Quick Start Guide, we have created, analyzed and designed a simple small model. This will give you the necessary knowledge to proceed to a real project.

For more help and guidance, please refer to **ProtaStructure** help reference accessible from Help menu. We recommend you read the **What's New** document for details of new features & enhancements.

Alternatively, you can attend our training courses to obtain a more in-depth knowledge of the usage of the software. Please visit our website www.protasoftware.com for more information.

The detail drawings of all the members and the general arrangement drawings can be automatically produced and managed in **ProtaDetails** (concrete) & **ProtaSteel** (steel). We recommend you proceed to read the **ProtaDetails** & **ProtaSteel** Quick Start Guide as the next step.