

# What's New?

## **Prota**Structure<sup>®</sup> 2018



## **Prota**Details 2018



## **Prota**Steel<sup>®</sup> 2018



For support & training please contact  
Support : [asiasupport@protasoftware.com](mailto:asiasupport@protasoftware.com)  
Training : [asiasales@protasoftware.com](mailto:asiasales@protasoftware.com)

[www.protasoftware.com](http://www.protasoftware.com)

What's New in ProtaStructure 2018 – Quick Summary	3
What's New in ProtaDetails 2018	6
What's New in ProtaSteel 2018	7
1. User Defined maximum effective length factor for BS8110 (sp4)	8
2. Enhanced Wind Calculation Report for MS 1553 (sp4)	8
3. Automatic Wind Load Calculator (sp3)	8
4. Enhanced Revit Integration (sp3)	12
5. New EC8 Malaysia Annex (2017) Seismic Code (sp2)	15
6. Wind Load Import from csv file (sp1)	16
7. ProtaSteel	18
8. Steel Design to EC3, BS5950 & AISC360-10	18
9. Steel Trusses	18
10. Purlins, Girts & Braces	20
11. New Section Manager Interface & Enhanced Steel Database	23
12. New Material Definition, Selection & Assignment Interfaces	24
13. Merged Shear Walls – Core Walls	25
14. New Polyline Column Editor	26
15. Multiple Openings in Shear Walls	28
16. Infill / Brick Wall Insertion by picking two points	29
17. Partial Infill Walls	30
18. Multiple Openings in Infill Walls	31
19. Section Anchors	31
20. Selection Filter	32
21. Slab Drops of any shape	32
22. Inserting Multiple Slabs in one go	33
23. Enhancement to Cantilever Slab creation	34
24. Rigid and/or Semi-Rigid Diaphragms by Floor	34
25. Detail Drawings Preview	35
26. Enhanced Visual Interrogation	36
27. Enhancements in Revit Integration	36
28. Center of Rotation	37
29. Shortcut keys for 3D Display Mode & Camera Projection	37
30. User Defined Thumbnail	37
31. New Options Interface	38
32. New Dimension Styles Interface	38
33. Auto truncation of Beam Elevation Details	39
34. Column Grouping	40
35. New Containment detailing	40
36. Enhancement in Details Drawing Generation	41
37. Search a Member Detail or Text	44
38. Beam Detail: Separate Lap & Extension Dimension for Top & Bottom Bars	45
39. Cantilever Beam Detail – Rebar Extension length	45
40. Multiple selection of Tables for report generation	45
41. Pile Design	46
42. Miscellaneous Enhancement & Fixes in ProtaDetails	55
43. Beam to (RC) Wall/Column Connection (sp4)	56
44. Welded Steel Pipe (Circular Hollow Sections) Connection (sp4)	56
45. Embedded Steel Macro (sp4)	57

## What's New in ProtaStructure 2018 – Quick Summary

**ProtaStructure 2018** is a fantastic new release containing many new features and enhancements for you to enjoy. Key highlights include:

- [User Defined maximum effective length factor for BS8110 \(sp4\)](#)  
Maximum effective length factors for the design of columns and walls can be manually defined now for BS8110 template. This allows flexibility to limit the effective length for columns and walls in both major & minor axes.
- [Enhanced Wind Calculation Report for MS 1553 \(sp4\)](#)  
The wind load calculation & reporting has been enhanced to show the key multipliers and values. This allows easy verification of the final design wind pressures and wind loads.
- [Automatic Wind Load Calculator \(sp3\)](#)  
The wind load calculator automatically calculates the wind forces in accordance with the chosen wind code and applies it to the building structure. This greatly increases productivity as values need not be manually calculated & entered. The wind code of practices covered are BS6399-2(1997), MS1533(2002), EN1991-4(2005) & ASCE7(2010).
- [Enhanced Revit Integration \(sp3\)](#)  
Round-tripping from Revit to ProtaStructure is further enhanced. The Revit model can be changed & re-imported back to ProtaStructure. The Integration Status will color code which members are new, changed, unchanged or deleted. Details of integration, e.g. exactly which member size has changed, is clearly listed.
- [New EC8 Malaysia Annex \(2017\) Seismic Code \(sp2\)](#)  
New EC8 Malaysia Annex (2017) is implemented into ProtaStructure comprehensive seismic solutions based on the latest requirements stated in the EC8 Malaysia National Annex.
- [Wind Load Import from csv/excel file \(sp1\)](#)  
The wind loads can now be imported from a csv/excel file. This greatly increases productivity as values need not be entered manually.
- [ProtaSteel](#)  
ProtaSteel is our new powerful steel detailing module that works seamlessly with ProtaStructure. In ProtaSteel, you can auto create & design steel connections, insert ancillary steel members, fine tune member position & create final engineering drawings including connection details.
- [Steel Design to EC3, BS5950 & AISC360-10](#)  
Steel design is implemented for the following codes of practice: EuroCode 3, BS5950 (British) & AISC360-10 (American).
- [Steel Trusses](#)  
Auto-generated steel trusses of any shape can be modelled on the fly including Pratt, Warren, mono-pitched, duo-pitched truss arrangements. Trusses can be easily moved, edited and copied aiding modelling flexibility.

- [Purlins, Girts & Braces](#)

Setting up multiple purlins, girts & braces is very easy and fast. Purlins can be modeled automatically between multiple trusses and beams – they are intelligently batch generated based on simple inputs. Similarly, girts & braces can be automatically defined between multiple columns.
- [New Section Manager Interface & Enhanced Steel Database](#)

The Section Manager has been enhanced, making selection of members, especially steel profiles, with different material & shapes, easier and faster. The steel database has been enhanced & expanded to include many country specific sections.
- [New Material Definition, Selection & Assignment Interfaces](#)

The material manager and interfaces has been enhanced for better accessibility and ease of use. New material types such as prestressed rebar and soil has been added.
- [Significantly Enhanced Analysis Speeds](#)

Analysis speed significantly enhanced in PS2018 over previous releases with larger models being upwards of 5 times faster to run.
- [Merged Shear Walls – Core Walls](#)

Multiple shear wall panels can be selected and merged into a single core wall for integrated analysis, design, and detailing, significantly increasing design efficiency.
- [New Polyline Column Editor](#)

Define and design any steel reinforcement layout for irregular shaped columns and merged core walls. Take advantage of automatic containment or quickly position bars and links yourself.
- [Multiple Openings in Shear Walls](#)

Multiple openings can be defined for shear walls / core walls using the new **Wall Opening Editor**. Rectangular or square opening can be accurately positioned and meshed in the building analysis.
- [Infill / Brick Wall Insertion by Picking by 2 points](#)

User can pick two points on the screen and define an infill wall on beams and shear walls; greatly speeding up the modelling process.
- [Partial Infill Walls](#)

Infill walls can now be partially defined along beams, not only the full length of a beam. These wall loads are displayed in the 3D as well as 2D view for easy checking.
- [Multiple Openings in Infill Walls](#)

Multiple openings can be defined for infill walls using the new **Wall Opening Editor**. Rectangular or square openings can be accurately positioned to consider doors and windows and associated loading for more economic beam loading and design.
- [Section Anchors](#)

Column and beams positions can be anchored to a specific point on their sections. When the section size is changed, the member is automatically resized with reference to the anchor point; great for modifying arrangements against fixed architectural and layout positioning.
- [Selection Filter](#)

Select and filter elements quickly during modelling; very useful when managing and editing large models.

- **Slab Drops of any shape**  
Easily insert isolated slab drops (& openings) of any shape using the Slab Opening/Drop function anywhere in your project floor, handle toilet drops and architectural set downs with ease.
- **Inserting Multiple Slabs in one go**  
Multiple slabs can be inserted in one go, significantly speeding up repetitive slab modelling.
- **Enhancement to Cantilever Slab creation**  
Creation of cantilever slabs (Type 12) have been made easier & more intuitive. They are inserted by 3 clicks - clockwise or counter-clockwise direction does not matter anymore. In addition, cantilever slabs can now span more than one beam or wall.
- **Rigid and/or Semi-Rigid Diaphragms by Floor**  
Semi-Rigid (meshed) diaphragms or Rigid Diaphragms can be set separately for each storey to mimic designed slab behavior during building analysis.
- **Detail Drawings Preview**  
Beam and Column Elevations drawings can now be previewed in ProtaStructure without opening ProtaDetails. This enables user to quickly review details before exporting out to ProtaDetails for full drawing production.
- **Enhanced Visual Interrogation**  
Visual Interrogation has been enhanced to include the Steel Design Status and Steel Utilization Ratios for quickly reviewing design efficiency.
- **Enhancements in Revit Integration**  
ProtaStructure now supports bi-directional links with Revit Structure greatly enhancing project coordination and workflow. Model changes can be synchronized and tracked both in Revit and ProtaStructure. Take advantage of enhanced filtering commands to share parts of the model you like, co-ordinate a floor at a time or the whole project model, you choose.
- **Center of Rotation**  
A toolbar toggle button Center of Rotation is introduced to enable easy rotation of the 3D view about a point of interest. When activated the view will rotate about the cursor position.
- **Shortcut keys for 3D Display Mode & Camera Projection**  
Pressing F11 repeatedly now cycles through the display modes on a 3D View. Pressing F12 repeatedly will cycle through Camera Projection on a 3D view.
- **User Defined Thumbnail**  
You can set your own model thumbnail to be shown in the open project dialog whenever the project is selected. This helps you to easily identify & locate the project you want to open.

## What's New in ProtaDetails 2018

ProtaDetails 2018 contains several completely new features and significant enhancements:

- [New Options Interface](#)

Drawing and detailing options are now classified under a single interface. You can also search for a specific setting from this dialog.
- [New Dimension Styles Interface](#)

Dimension styles interface has been enhanced to easily manipulate both project and externally opened dwg dimension styles. All predefined detailing dimension styles can be edited and saved along with the project
- [Auto truncation of Beam Elevation Details](#)

Beam elevations details can be automatically truncated and organised to fit into your drawing sheet borders saving significant time creating your drawings.
- [Column Grouping](#)

Automatic column schedule grouping is performed for columns with the same size & reinforcement requirements, significantly reducing drafting requirements for large projects with repetitive column detailing.
- [New Containment detailing](#)

Full details are provided for new polyline columns and merged walls using the new containment design options.
- [Enhancement in Details Drawing Generation](#)

Many enhancements have been introduced in the details drawing manager to help with the auto-generation of drawings. In Form Plans, the ability to hatch columns/walls, show column loads & dimensions on plan view has been added. Additional key settings such as Scale are included in the same menu for easy accessibility.
- [Search a Member Detail or Text](#)

New quick search function to quickly locate a specific member detail, example beam 1B1. Very useful if you only want to draw out a particular detail. After drawing details or objects in the drawings, you can use the new text search function to locate it.
- [Beam Details: Separate Lap & Extension Dimension for Top & Bottom Bars](#)

The dimension options of lap & extension (or anchorage) are separated for top and bottom bars to cater for different users' preferences.
- [Cantilever Beam Detail – Rebar Extension length](#)

Cantilever top rebar extension lengths can be adjusted based on a user-defined factor multiplied by the cantilever length. This feature is implemented to cater for some user's preference for this manner of cantilever beam detailing.
- [Multiple selection of Tables for report generation](#)

Any table drawn in ProtaDetails can be converted to a report and generated via the Report Manager.

➤ [Pile Design](#)

The new pile design engineering macro calculates the structural and geotechnical capacity of axial and laterally loaded piles. Pile Working Load module calculates the allowable compression capacity based on soil investigation parameters. The Lateral Analysis module analyzes and designs the pile for additional shear & moments from applied lateral loads.

➤ [Miscellaneous Enhancement & Fixes in ProtaDetails](#)

There are numerous enhancements & fixes focused on producing better beam detailing (particularly rebar and slab profiles) & drawing standards, together with a revamped materials system & new dimension style interface which can be modified to suit your preferences and reused on your projects.

## What's New in ProtaSteel 2018

ProtaSteel 2018 contains several completely new features and significant enhancements:

➤ [Beam to \(RC\) Wall/Column Connection \(sp4\)](#)

This new connection macro creates an embedded steel connection between a steel beam and a concrete wall or column.

➤ [Welded Steel Pipe \(Circular Hollow Sections\) Connection \(sp4\)](#)

This new connection macro creates a welded steel tubes connection using welds. This macro is available from the Secondary Connections toolbar (as shown below).

➤ [Embedded Steel Macro \(sp4\)](#)

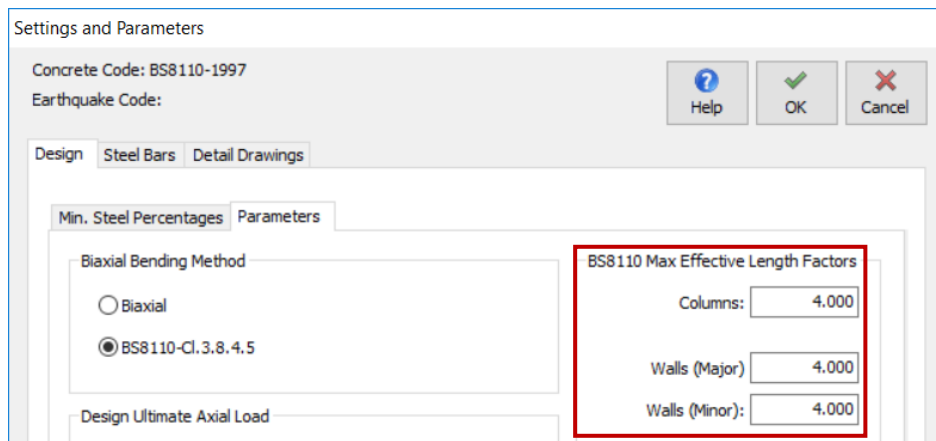
This new connection macro creates a generic embedded steel with anchor bars, which can then be used to connect other steel members to reinforced concrete members

**For detail list of enhancement & fixes of ProtaStructure Suite service packs, click on the links below :**

- [PS2018 Service Pack 1 \(sp1\) – What's New](#)
- [PS2018 Service Pack 2 \(sp2\) – What's New](#)
- [PS2018 Service Pack 3 \(sp3\) – What's New](#)
- [PS2018 Service Pack 4 \(sp4\) – What's New](#)

## 1. User Defined maximum effective length factor for BS8110 (sp4)

Maximum effective length factors for the design of columns and walls can be manually defined now for BS8110 template. This allows flexibility to limit the effective length for columns and walls in both major & minor axes. This new setting is accessible from Column Design Settings > Parameters.



## 2. Enhanced Wind Calculation Report for MS 1553 (sp4)

The wind load calculation & reporting has been enhanced to show the key multipliers and values. This allows easy verification of the final design wind pressures and wind loads.

Storey	Height (m)	Length (m)	Width (m)	Mz.Cat	C.dyn	C.fig	Vdes (m/s)	Pressures (kN/m <sup>2</sup> )	Loads (kN)	Cross Pressures (kN/m <sup>2</sup> )	Cross Loads (kN)
1	3.00	15.00	10.00	0.75	1.40	1.20	36.86	1.40	42.0	0.01	0.3
2	3.00	15.00	10.00	0.75	1.39	1.20	36.26	1.35	40.4	0.04	1.2
3	3.00	15.00	10.00	0.75	1.38	1.20	35.72	1.30	39.0	0.08	2.4

## 3. Automatic Wind Load Calculator (sp3)

The wind load calculator automatically calculates the wind forces in accordance with the chosen wind code and applies it to the building structure. This greatly increases productivity as values need not be manually calculated & entered. The wind code of practices covered are :

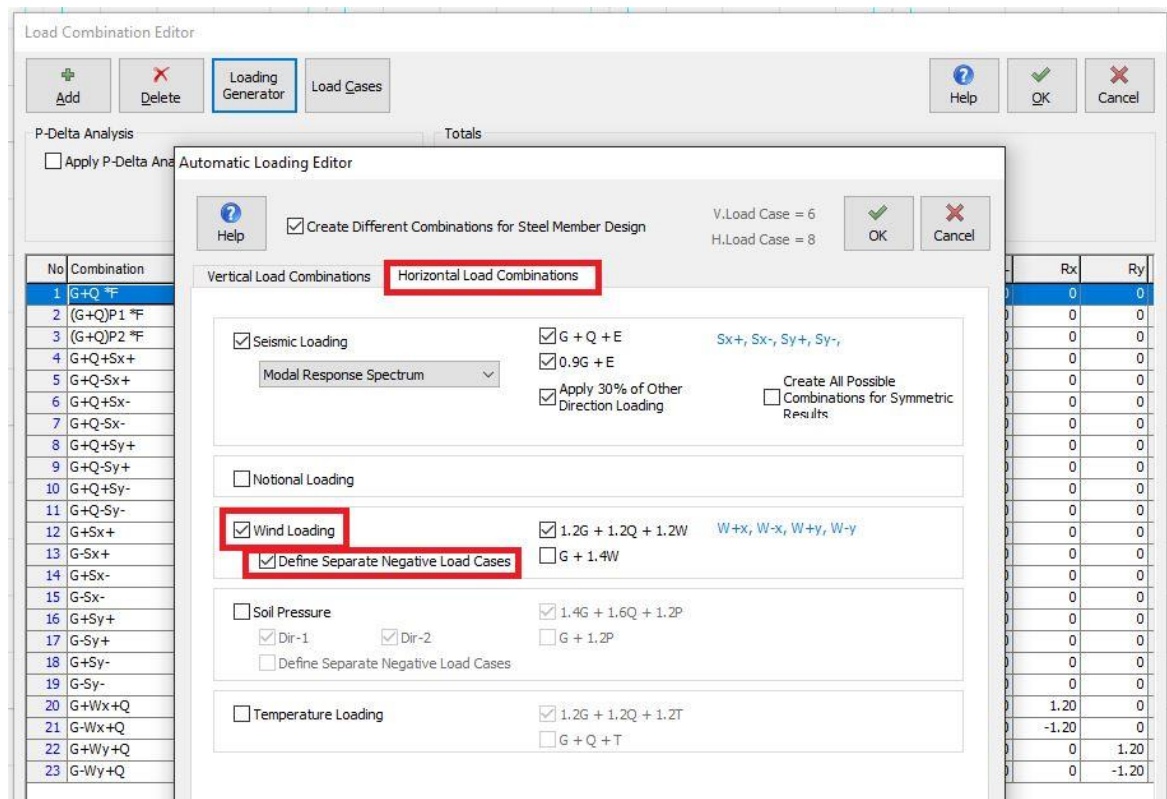
- BS6399-2(1997)
- MS1533(2002)
- EN1991-4(2005)
- ASCE7(2010)

### How to Use

#### Step 1: Define wind load combinations:

- Go to **Building Analysis** dialog via **Analysis** top drop down menu
- In **Pre-Analysis** tab, click **Load combinations** button
- In **Load Combination Editor**, click **Loading Generator** button
- In **Automatic Loading Editor**, go to **Horizontal Load Combinations** tab > Check **Wind Loading & Define Separate Negative Load Cases** (if necessary) as shown below

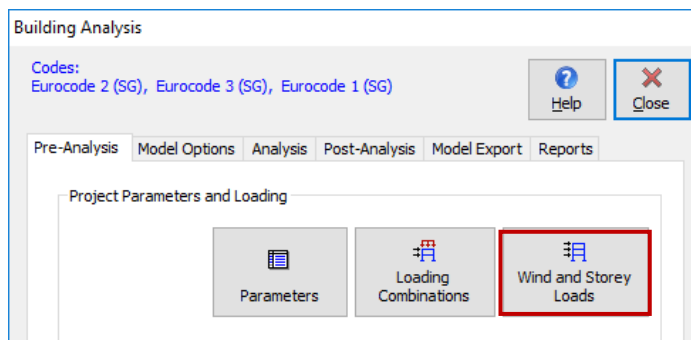




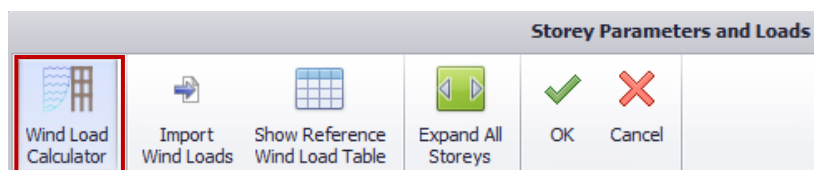
- Click **OK** to close Automatic Loading Editor dialog
- Click **OK** to close Load Combination Editor dialog

## Step 2: Calculate and Apply Wind Loads:

- In **Pre-Analysis** tab of Building Analysis menu, click **Wind and Storey Loads** button



- Click **Wind Load Calculator** in **Storey Parameters and Loads** dialog to launch the wind load calculator



Wind Load Calculator

General

Parameters

Report and Results

W+x (0°)  
W+y (90°)  
W-x (180°)  
W-y (270°)

**Basic Wind Properties**

Basic Wind Speed (m/s) : 33.5  
Min. Wind Pressure (kPa) : 0.65  
Primary Wind Angle (°) : 0.0

**Terrain and Surroundings**

H (m) : 100.0  
x (m) : 50.0  
Lu (m) : 150.0  
ns : 10.0  
Hs (m) : 200.0  
Bs (m) : 50.0  
Terrain Category : Category 4  
Slope Type : Escarpment

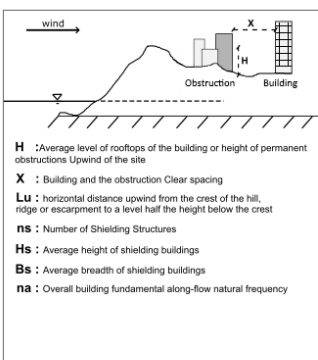
**Structure Properties**

Include Dynamic Factor (Cdyn) : ☒  
Roof Pitch (°) : 5.0  
na (Hz) : 0.2  
Structure Category : Category II  
Building Type 1 : Reinforced concrete  
Building Type 2 : Uniform cantilever  
Dominant Opening Area (m2) : 5.0  
Total Opening Area (m2) : 100.0  
Dominant Opening Side : Windward wall  
Internal Pressure Condition : Max positive  
Internal Pressure Case : No dominant openings  
Wall Permeable Type : Any wall except windward

Wind Code : MS1533 (2002)

Apply Cancel

**Diagram:**



**Legend:**

**H** : Average level of rooftops of the building or height of permanent obstructions Upwind of the site  
**X** : Building and the obstruction Clear spacing  
**Lu** : horizontal distance upwind from the crest of the hill, ridge or escarpment to a level half the height below the crest  
**ns** : Number of Shielding Structures  
**Hs** : Average height of shielding buildings  
**Bs** : Average breadth of shielding buildings  
**na** : Overall building fundamental along-flow natural frequency

➤ Change the **Wind Code** in the drop-down list below if required

➤ Define all parameters in the **Parameters** tab

#### Notes:

- Some parameters are illustrated and explained in the right figure in Parameters dialog
- Primary Wind Angle can be used to adjust the direction of the primary wind angle.  
The effective building width and length will be calculated accordingly. Direction factor will be applied where applicable.
- All other parameters are explained in the respective codes of practices
- Click the **wind load cases** in the **Report and Results** in the left pane to review the calculated wind pressures and wind loads (as shown below)

Wind Load Calculator

General

Parameters

Report and Results

W+x (0°)  
W+y (90°)  
W-x (180°)  
W-y (270°)

Storey	Height (m)	Length (m)	Width (m)	Pressures (kN/m2)	Loads (kN)
1	3.00	15.00	15.00	0.65	29.3
2	3.00	15.00	15.00	0.65	29.3

Create Report

Apply Cancel

➤ Click **Create Report** button at the bottom to generate the wind load summary report

➤ Click **Apply** at the bottom right to apply the wind loads to the model

- Review the applied wind loads in the **Storey Parameters and Loads** (as shown below)

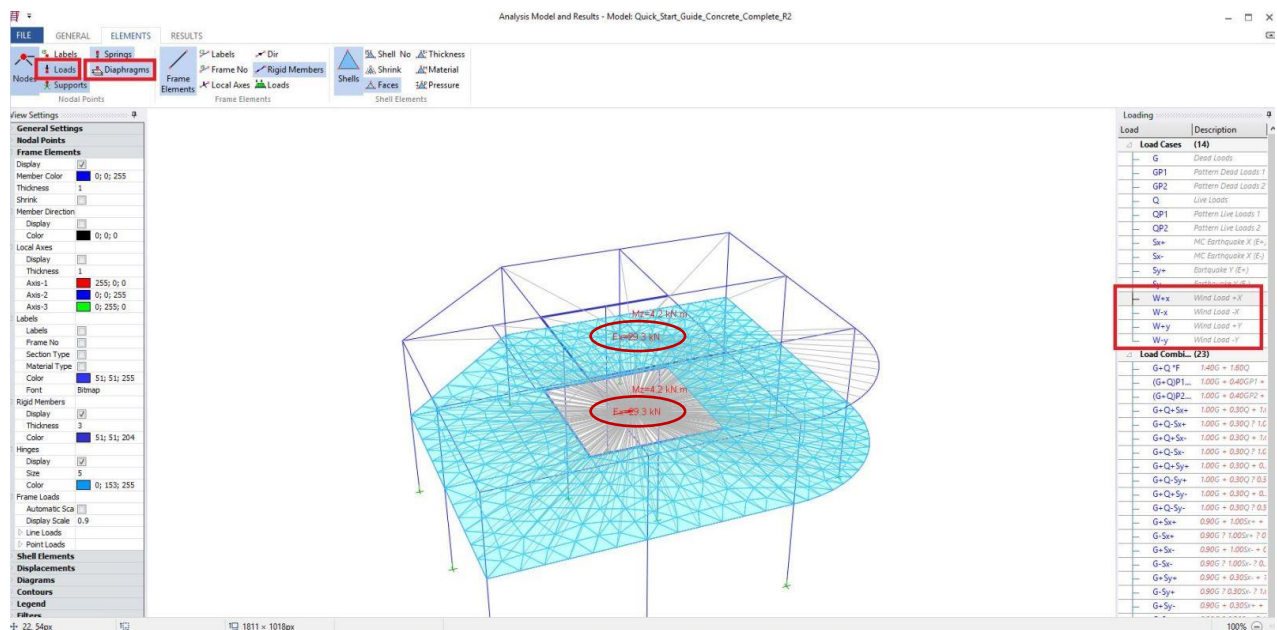
Storey Parameters and Loads					
Wind Load Calculator	Import Wind Loads	Show Reference Wind Load Table	Expand All Storeys	OK	Cancel
Storey		Mass (t)	X-g (m)	Y-g (m)	
St: 1 (+3.00m, Sim:2)		251.597	12.493	12.355	
	Fx (k N)	Fy (k N)	X (m)	Y (m)	
Sx+	0	0	0	12.493	13.105
Sx-	0	0	0	12.493	11.605
Sy+	0	0	0	13.243	12.355
Sy-	0	0	0	11.743	12.355
W+x	29.3	0	0	12.5	12.5
W-x	-29.2	0	0	12.5	12.5
W+y	0	29.3	29.3	12.5	12.5
W-y	0	-29.3	-29.3	12.5	12.5
St: 2 (+6.00m, Sim:1)		251.597	12.493	12.355	

**Note:** The apply wind loads are editable, you may click & edit the applied wind load

- Click **OK** to close **Storey Parameters and Loads** dialog

### Step 3: Run building analysis and review the results in Post-Analysis Result Display

- Go to **Building Analysis** tab, click **Start** to perform building analysis
- Go to **Post-Analysis** tab, click **Model and Analysis Results Display**
- Go to **Elements** tab, make sure the **Loads** and **Diaphragms** in the Nodal Points button group are activated (as shown below)



- Click any wind load case in the **Loading** tab (on the right panel).

The applied wind loads will be shown as nodal point loads in the model. These wind loads will be automatically distributed to the floor diaphragm(s) and isolated member(s) where applicable.

### Assumptions & Limitations

1. Irregular buildings on plan (such as C shape, U shape) will be converted to an equivalent rectangle.
2. Wind loads will only be calculated for windward and leeward faces to obtain the net wind pressures/loads on each elevation.
3. This wind load will not, at this stage, support wind calculations of low rise industrial buildings where side and roof wind pressure calculations are required.
4. The number, distance and dimensions of adjacent buildings in different wind directions will be taken as an average input and shall be manually calculated by the user.

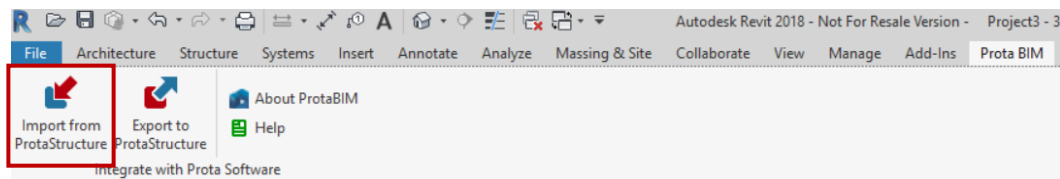
For building height exceeding 150m to 200m, the calculated wind loads might not be applicable. Please refer to the respective code of practice as requirements are different. A wind tunnel test should be conducted if necessary.

## 4. Enhanced Revit Integration (sp3)

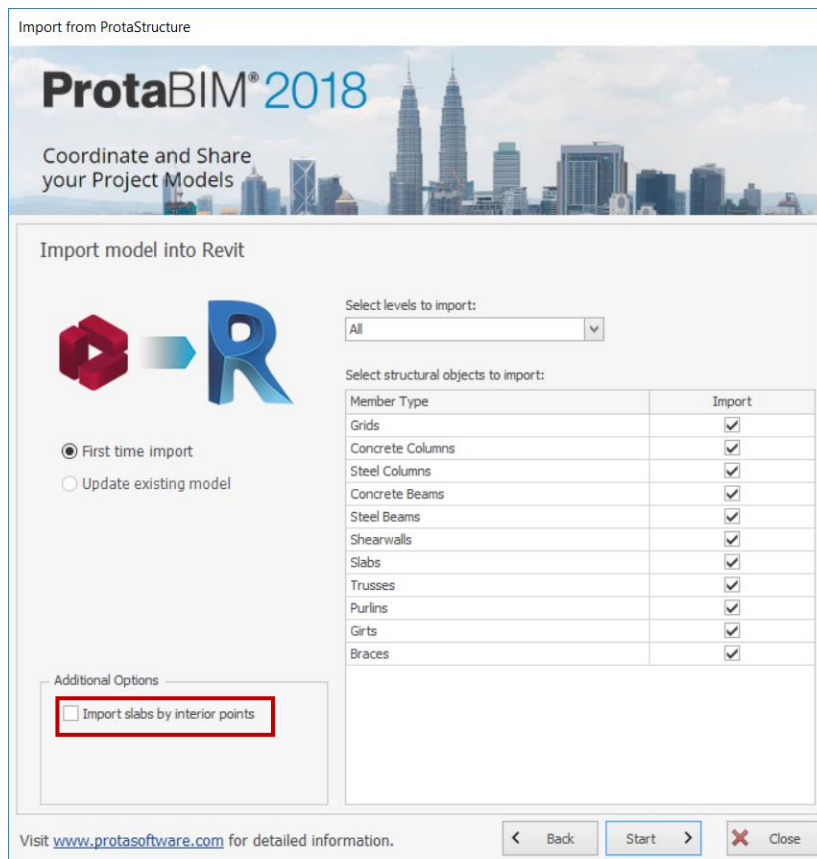
Round-tripping from Revit to Protostructure is further enhanced. When the Revit model is changed & re-imported back into Protostructure, the import dialog box allows you to decide whether to automatically remove the deleted members in the Revit model. The Integration Status will color-code which members are new, changed, unchanged or deleted.

### How to Use

- *In Revit, use ProtaBIM to import a new ProtaStructure model*

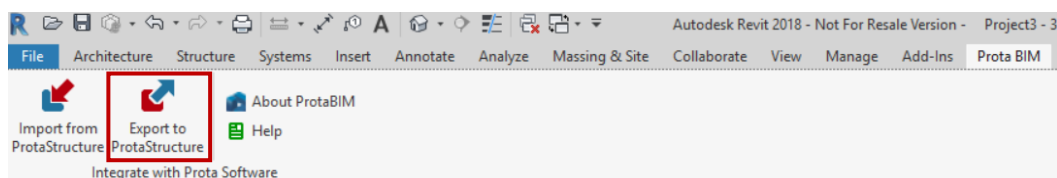


- *Choose the desired options of import.*

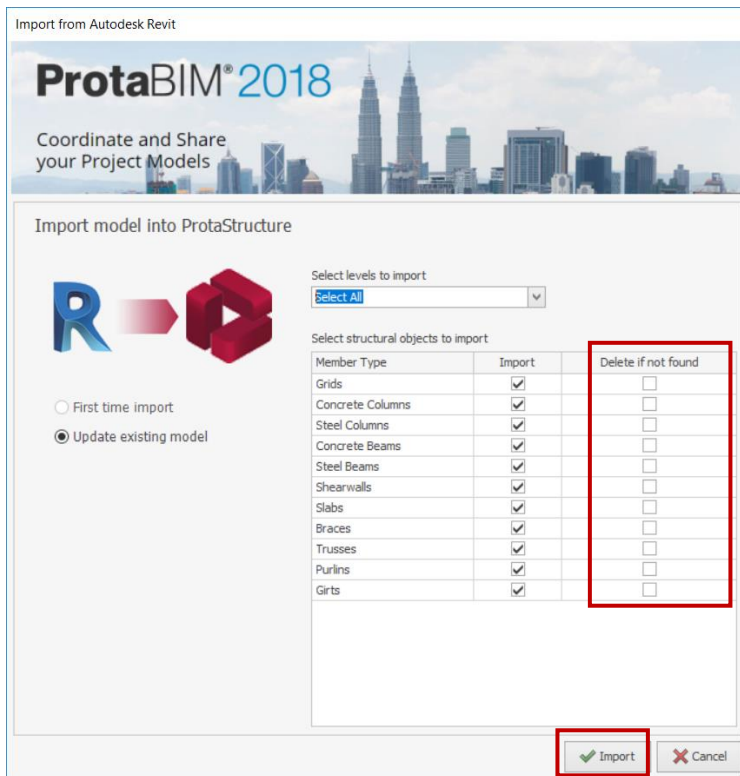


**Note :** Check the option **Import slabs by interior points** will avoid slab trimming and overlapping with other member issues.

- Once imported, make changes to the Revit model, example add, delete, change members.
- Choose **Export to ProtaStructure** in ProtaBIM tab



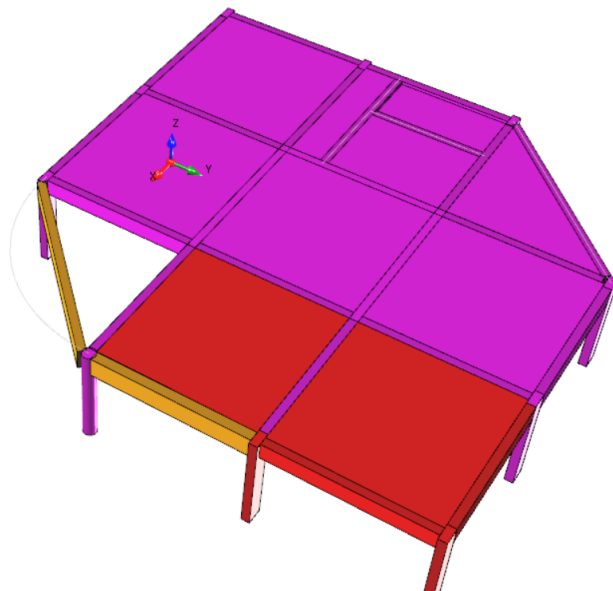
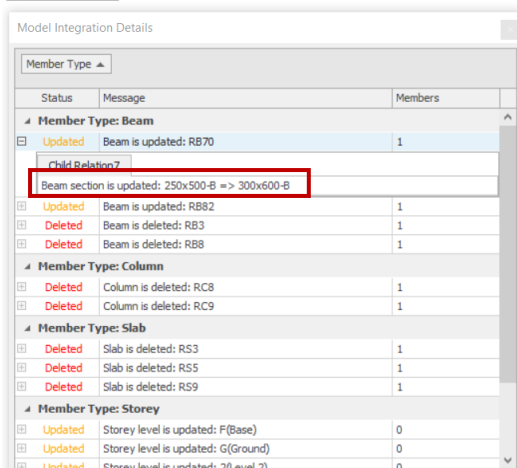
- **Select** the same ProtaStructure project in Open Project dialog.  
A "RevitData" folder will be created under selected project folder.
- **Launch** ProtaStructure or bring it to front if already loaded.
- Ensure that the project you will update is open.
- Go to **File Menu > Mode/File import > pick Import From Revit Structure**
- In Import dialog, choose the desired options > Pick **Import** (as shown below)



**Note :** Check the option “Delete if not found” if you want members that are deleted in Revit to be completely removed in ProtaStructure.

After a successful import, the **Build Model Check** dialog will appear. If you wish to run it, pick **Start** then **Close**. The view will automatically switch to a 3D view with the **Integration Status** color coding which member(s) is **new**, **changed**, **deleted** & **unchanged**.

#### Integration Status



The **Model Integration Details** dialog lists more details of the integration.

- Click the “+” sign to reveal more details, example the exact change of member sizes

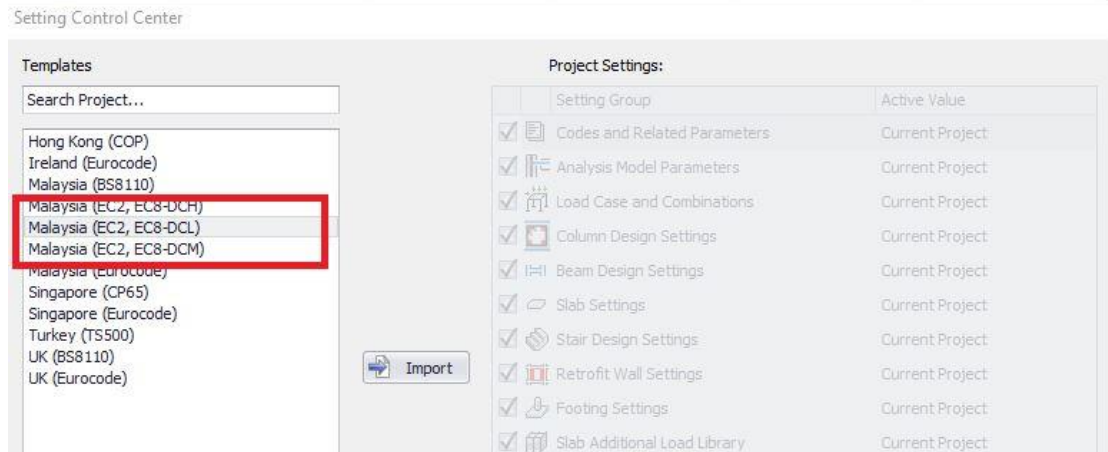


## 5. New EC8 Malaysia Annex (2017) Seismic Code (sp2)

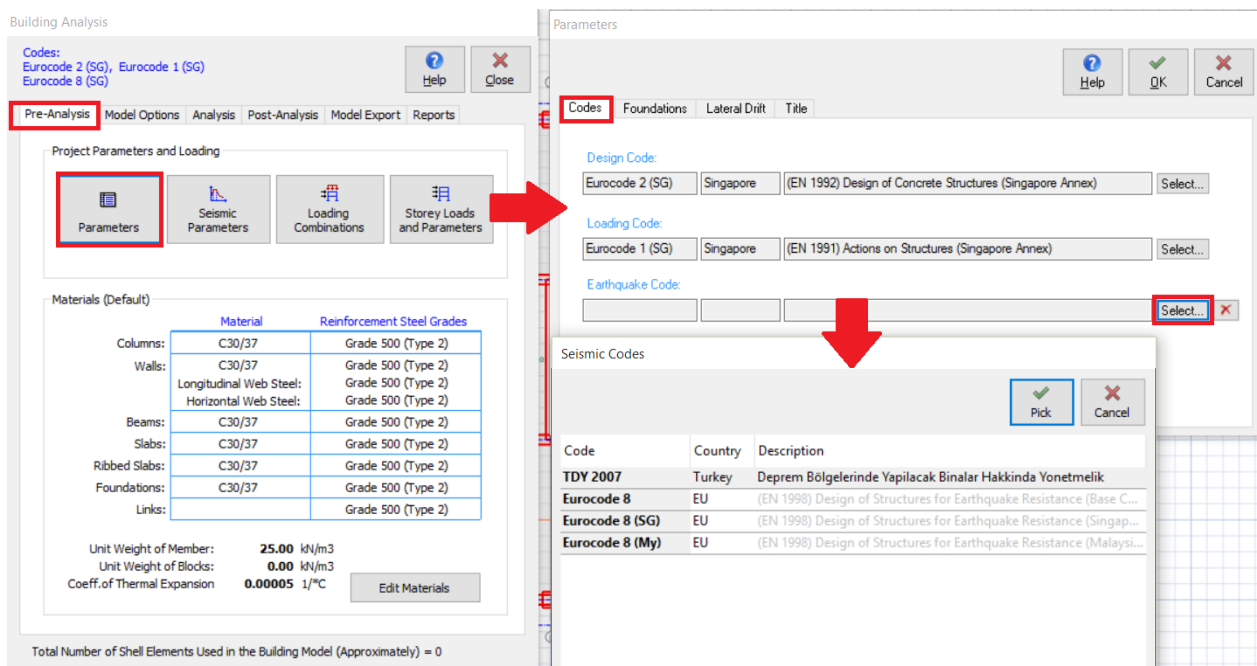
New EC8 Malaysia Annex (2017) is implemented into ProtaStructure comprehensive seismic solutions based on the latest requirements stated in the EC8 Malaysia National Annex.

### How to Use

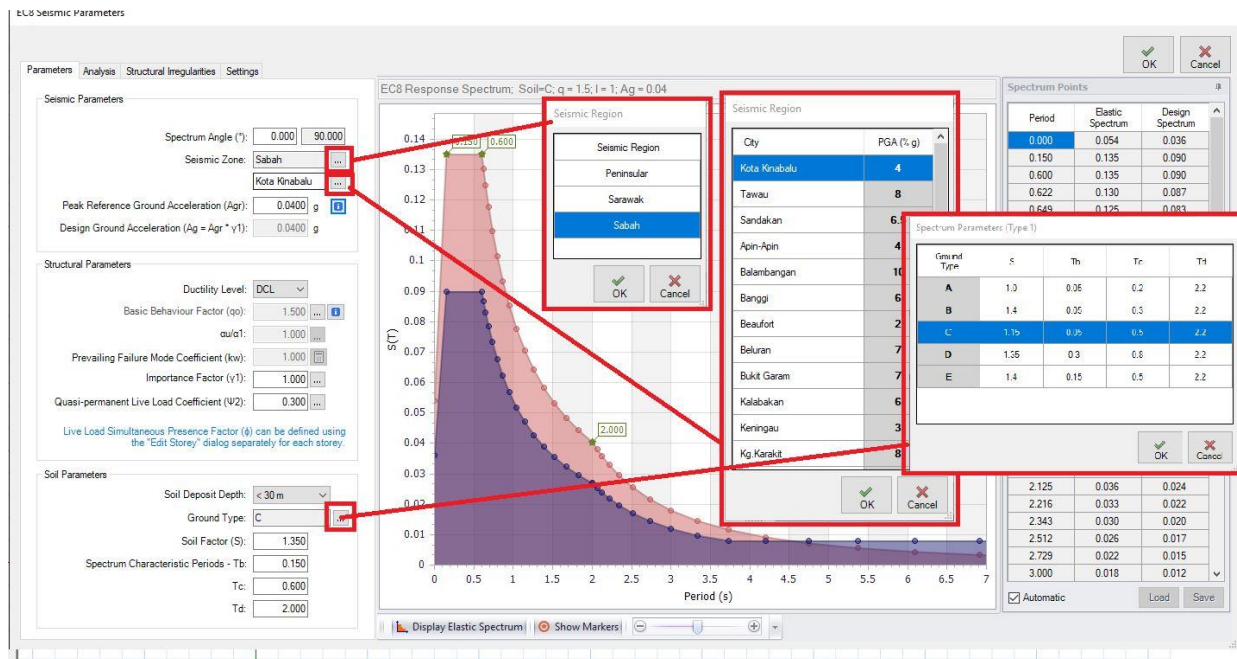
- Create a new model by choosing **Malaysia EC2, EC8 - DCL/DCM/DCH** template



- Ensure the **Eurocode (My)** is selected in **Building Analysis > Pre-analysis tab > Click Parameters > Codes tab** (as shown below)



- Click **Pick** to close the Seismic code dialog
- Click **OK** to close the Parameters dialog
- Define the various seismic parameters in the **Building analysis > Seismic Parameters** (see image below)



**Note :** For further guidance refer to this online help article : [Seismic Load - Basic Guide](#).

## 6. Wind Load Import from csv file (sp1)

The wind loads can now be imported from a csv / excel file. This greatly increases productivity as values need not be entered manually.

The sample csv file can be found in the default installation folder :

**C:\Users\User Name\Documents\ProtaLib\Samples**

- **WindPointLoad\_SampleImport.csv** is used if "Define Separate Negative Wind Load Case" is unchecked in Load Combination Editor (see below).
- **WindPointLoadSeperateNegativeLoadCases\_SampleImport.csv** is used if "Define Separate Negative Wind Load Case" is checked in Load Combination Editor (see below).

### How to Use

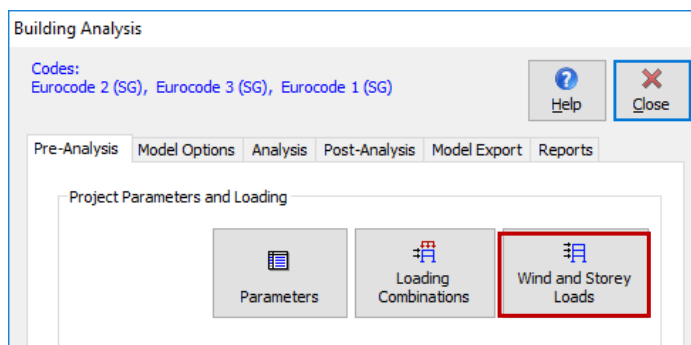
- Open the sample file *WindLoadImportSample.xlsx* or *csv* file in Microsoft Excel

kN	m							
	Direction X				Direction Y			
Storey	Fx (kN)	Fy (kN)	X (m)	Y (m)	Fx (kN)	Fy (kN)	X (m)	Y (m)
1	100	0	12.3	12.5	0	130	12.5	13.5
2	101	0	12.3	12.5	0	131	12.5	13.5
3	102	0	12.3	12.5	0	132	12.5	13.5
4	103	0	12.3	12.5	0	133	12.5	13.5

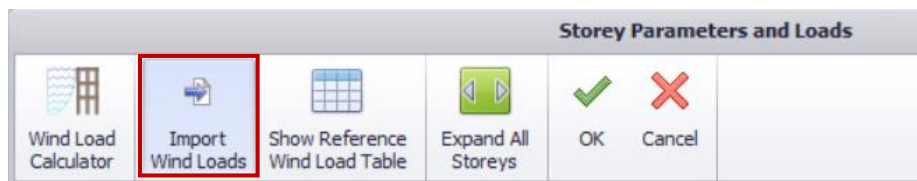
- Enter the values of the wind force **Fx** & **Fy** the corresponding coordinates **X** & **Y** for point of application, i.e. figures in red as shown above
- Save file as **.csv** format
- In Building Analysis menu, use **Loading Generator** to generate the wind load cases & combination



- In Building Analysis menu > Pre-Analysis Tab > click on **Wind & Storey Loads** button



- Pick **Import Wind Loads** in Storey Parameter and Loads dialog



- Select the csv file previously set-up
- Expand the storey loads information by click the "+" icon & check the values imported (as shown below)

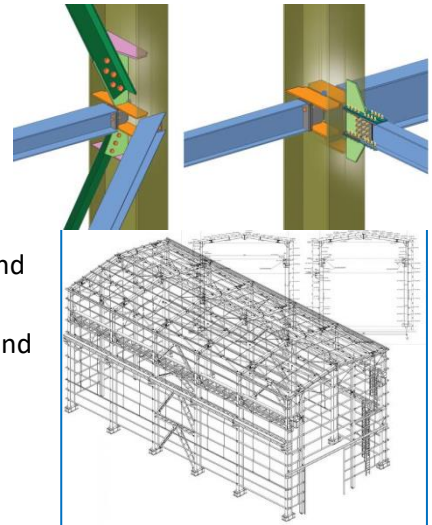
Storey	Mass (t)	X-g (m)	Y-g (m)
St: 1 (Ground, Sim:2,3)	165.62	12.4073	12.4841
	Fx (k N)	Fy (k N)	X (m)
			Y (m)
NGx	0	0	12.4073
NQx	0	0	12.4073
NGy	0	0	12.4073
NQy	0	0	12.4073
W+x	47.69	29.25	12.5
W-x	-47.69	-29.25	12.5
W+y	-29.25	47.69	12.5
W-y	29.25	-47.69	12.5
St: 2 (+6.00m, Sim:1,3)	165.62	12.4073	12.4841
St: 3 (+9.00m, Sim:2,1)	165.62	12.4073	12.4841
St: 4 (Roof)	178.35	12.4337	12.5054
St: 5 (+15.00m)	6.52	12.4998	12.4999

**Tip :** You can also click on **Expand All Storeys** icon to expand all the storeys information in one go.

## 7. ProtaSteel

ProtaSteel is a new powerful steel detailing module to produce engineering drawings, general arrangement drawings & connection details. The main steel model is transferred seamlessly from ProtaStructure to ProtaSteel. In ProtaSteel, you can :

- Create steel connections. Most of the standard connections will be auto-designed based on material strength.
- Insert ancillary & secondary such as sag rods, doors & windows opening, steel staircase, etc.
- Perform minor adjustment & fine tuning of member position.
- Create engineering drawings, general arrangement drawings and connection details.
- Additional software configurations are available for assembly and shop detailing – speak to us if this is of interest



### How To Use :

- Model, analyse & design the steel members in ProtaStructure
- Go to **Steel Design** (top menu) → **Load ProtaSteel**

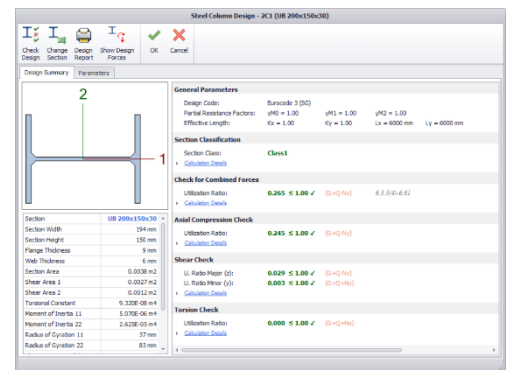
Please refer to the Quick Start Guide in ProtaSteel for further guidance.

## 8. Steel Design to EC3, BS5950 & AISC360-10

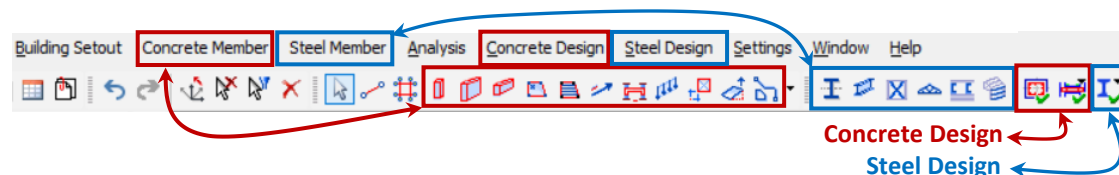
Steel design is implemented for the following code of practices:

- EuroCode 3 with Singapore, Malaysia & UK Annexes
- BS5950 (British)
- AISC360-10 (LFRD) (American)
- AISC360-10 (ASD) (American)

Steel members will be checked for pass / failure and the most critical Utilization Ratio (UR) reported. You can change the section sizes in the interactive design & new section will be automatically checked.



In line with the expanded scope of steel members, the user interface has been enhanced to group concrete & steel members modeling & design separately.



## 9. Steel Trusses

Auto-generated steel trusses of any shape can be modelled on the fly, e.g. Pratt, Warren, mono-pitched, duo-pitched truss. Trusses can be easily moved, edited and copied aiding modelling flexibility.

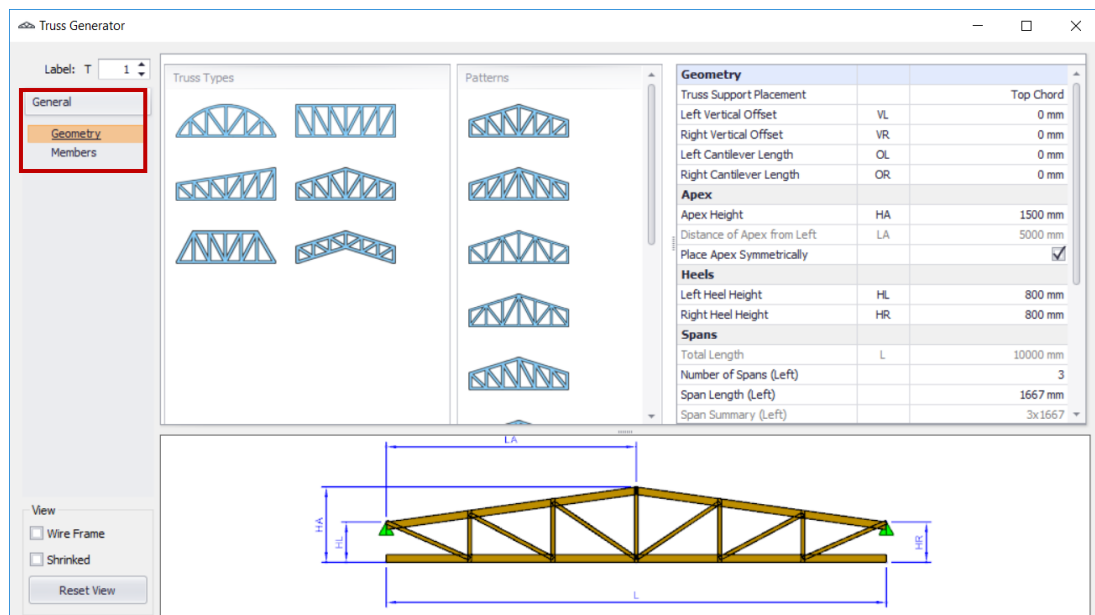
### How To Use:

- Click on **"Truss"** toolbar button. 
- Pick a **grid intersection** to define the start end of the truss (in the 3D or 2D view)

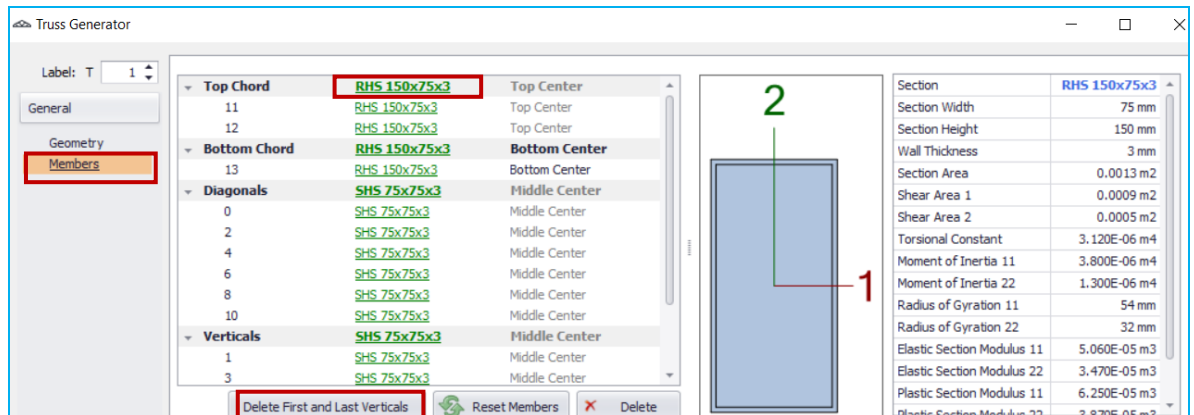
A dynamic 'rubber band' will appear with the default truss type if no truss is defined before.

- Pick another **grid intersections** to define the end of the truss.

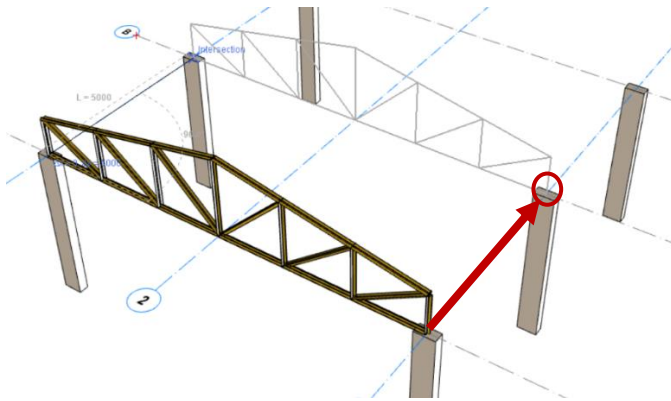
After you pick the second point, the truss dialog will appear.



- In the **Geometry** dialog, specify the following :
  - Type of the truss.
  - Geometry of the truss, i.e. dimensions, support offsets etc.
- In the **Members** dialog, specify the following :



- Assign **Sections** to the top & bottom chord, Diagonals & Verticals
- **Delete First & Last Verticals** – will ensure the end verticals are not overlapping with the column if the truss bottom & top chord connects directly to the column.
- Press **OK** → The Truss will be inserted and **COPY** mode will be activated.
- You can continue to insert more trusses of the same type by picking insertion points.



- Press **ESC** or **Right-Click** will end the copy operation.


### Copying/Moving an existing Truss:

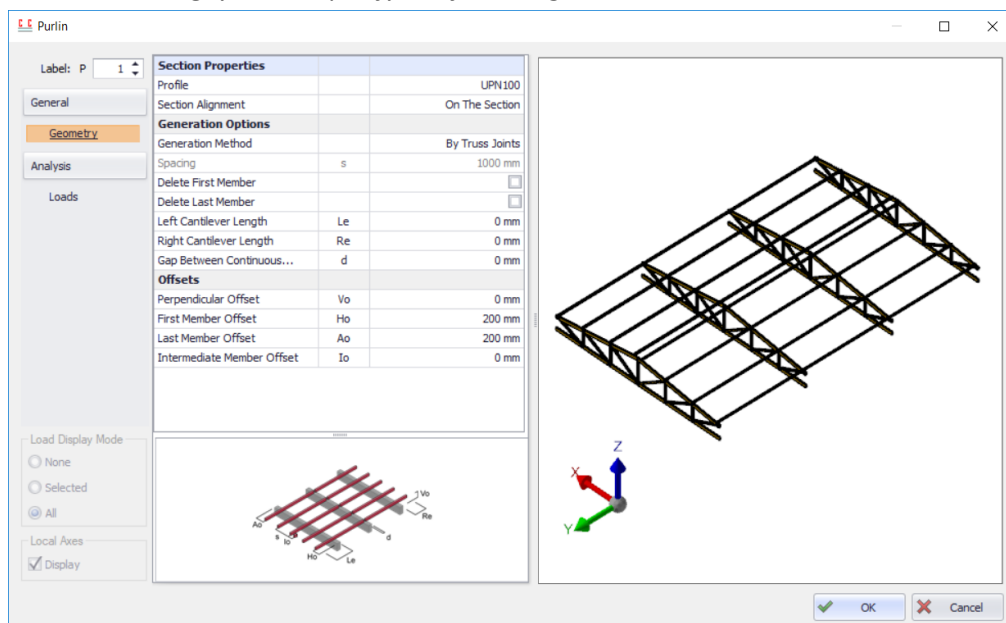
- Select the truss → Pick **Copy** or **Move** command.
- Select the **base-point** → Select the destination point.
- The **COPY** mode will continue until you press **ESC** or **Right-click**

## 10. Purlins, Girts & Braces

Setting up multiple purlins, girts & braces is very easy and fast. Purlins can be modeled automatically between multiple trusses; they are intelligently batch generated based on simple inputs. Similarly, girts & braces can be automatically defined between multiple columns or beams. Loads can be easily applied.

### How To Create Purlins:

- Click on “**Purlin**” toolbar button 
- Select the **first truss** → Select the **last truss** for purlins to be inserted  
Intermediate trusses will be automatically found.
- On **Purlin Dialog**, you can specify the following:

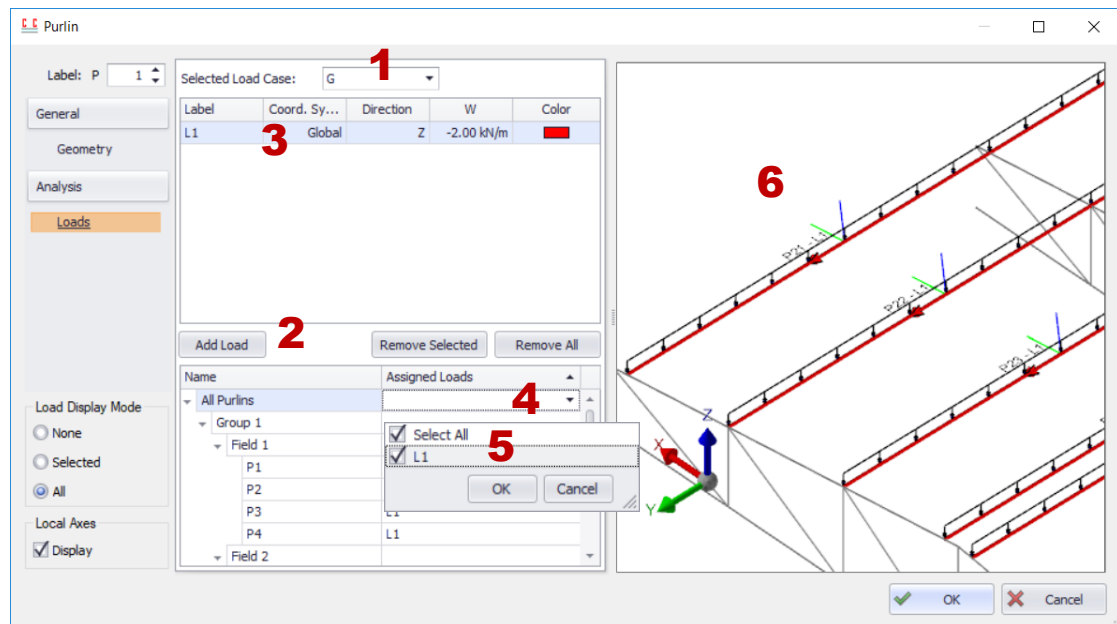


- In the **Geometry** dialog, specify the following :
  - Profile / Section of the for the purlin

- Section Alignment: On the Section / Under the Section / Center etc.
- Specify the purlins to be placed on truss joints only or by spacing
- Specify Left & Right Cantilever Length
- Set firsts & last purlin offsets


The diagram will automatically update in accordance with the geometry inputs. You can zoom in/out the diagram by scrolling the mouse wheel. To rotate the view, right-click & drag.

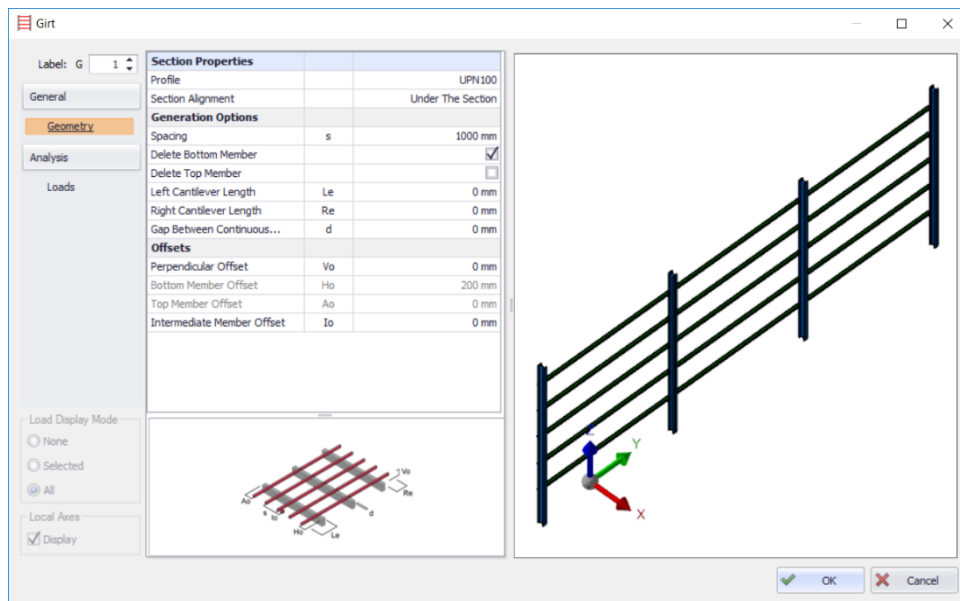
In the **Loads** dialog, specify the following :



- 1 Select the **Load Case** to insert the loading
- 2 Click **Add Load** → Load Label **L1** will be created  
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 individual **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

### How To Create Girts:

- Click on **Girt** toolbar button 
- Select the **first column** → Select the **last column**  
Intermediate columns will be automatically found.
- On **Girt** dialog, you can specify the following:




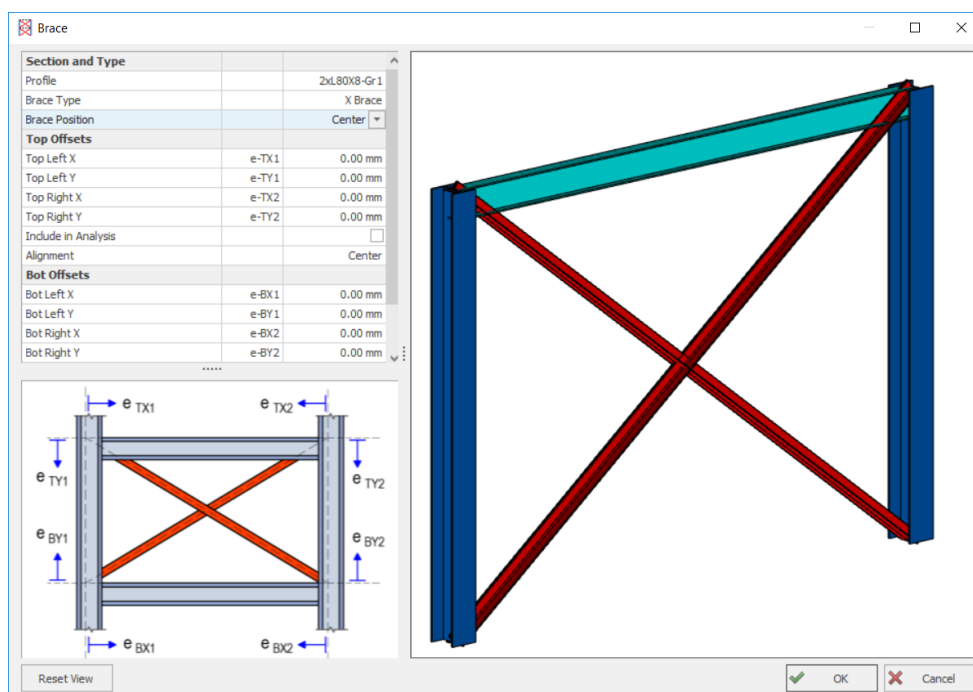
- Profile / Section of the girt
- Section Alignment : Under The Section / On The Section
- Spacing, Delete Bottom / Top Member, Left / Right Cantilever, Offsets

The diagram will automatically update in accordance with the geometry inputs. You can zoom in/out the diagram by scrolling the mouse wheel. To rotate the view, right-click & drag.

- In the **Loads dialog** you can assign loadings to girts in the same way as purlins.

### How To Create Brace:

- Click on the **Brace** toolbar icon 
- Pick **two adjacent** columns or two beams
- On **Brace** dialog, you can specify the following:



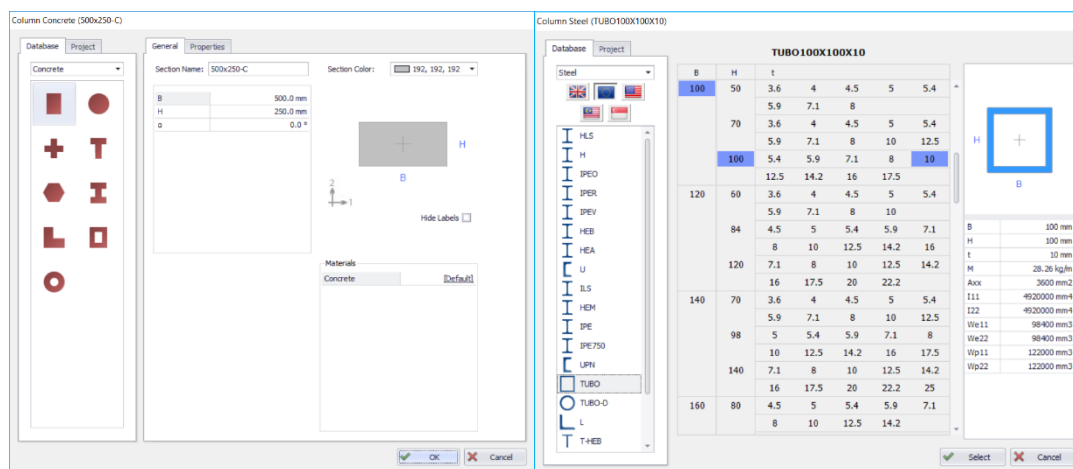
- Profile / Section of the for the brace

- Brace Type : Diagonal, X Brace, Y Brace or K Brace
- Out of Plane / In Plane Alignment
- Divide : If checked, the braces will be aligned in the exact same vertical plane & be divided into separate segments in analysis.
- Top & Bottom Offsets


The diagram will automatically update in accordance with the geometry inputs. You can zoom in/out the diagram by scrolling the mouse wheel.

## 11. New Section Manager Interface & Enhanced Steel Database

The Section Manager has been enhanced, making selection of members, especially steel profiles, with different material & shapes, easier and faster. The steel database has been enhanced & expanded to include many country specific sections.



### How To Use:

- Section Manager is accessible from various places of member creation :
  - Beam & Column properties → **Section Manager** icon 
  - Select a column or beam → Right-click → **Edit Section / Material**
  - **Steel Truss / Purlin / Girt / Brace** creation dialog
- In the Database tab, choose the **Material**, e.g. Concrete / Steel
- **Concrete** : Pick the concrete **Shape** → Input the dimensions
- **Steel** : Pick the **Country** Database → Pick Steel Type & Sections

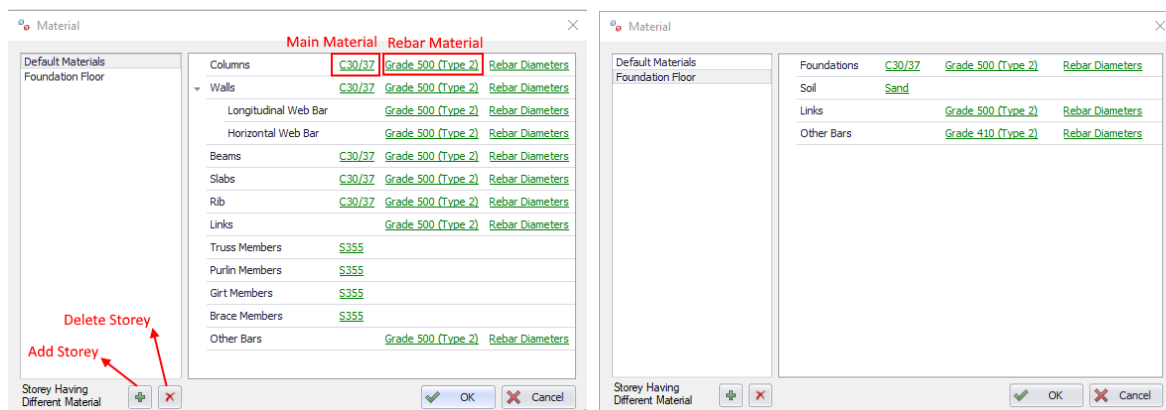


## 12. New Material Definition, Selection & Assignment Interfaces

Default material assignment and individual material definition/selection interfaces has been enhanced. All member types including new steel members can be listed on a single page and default materials are assigned to them via this interface.

### How To Use Default Material Assignment Interface:

- Default Materials Manager can be accessed through **“Building Analysis”** menu in ProtaStructure and under **“Settings”** menu in ProtaDetails
- Add/Delete storeys for exclusive material definition. “Default Materials” and “Foundation Floor” cannot be deleted
- Click on the material label to select a different default material for that member type
- Truss/Purlin/Girt/Brace members only allow to select a steel material. There is no “Rebar Material” defined for them
- Shearwall mesh steel material can be selected using “Longitudinal Web Bar” material under “Walls” branch. In this case “Horizontal Web Bar” row will disappear
- The rebar material on the “Walls” row only represents the corner/headzone rebars
- Other Bars branch is used to define a default material for the stray rebars which is not related with any concrete member such as column/beam/slab etc. in ProtaDetails
- Pad Footing, Pile Cap, Strip Footing, Raft Foundation and Pile default materials are all set from “Foundations” row under “Foundation Floor”.
- Soil materials can be edited from “Soil” branch. These are currently used in **“Pile Design and Detailing”** macro

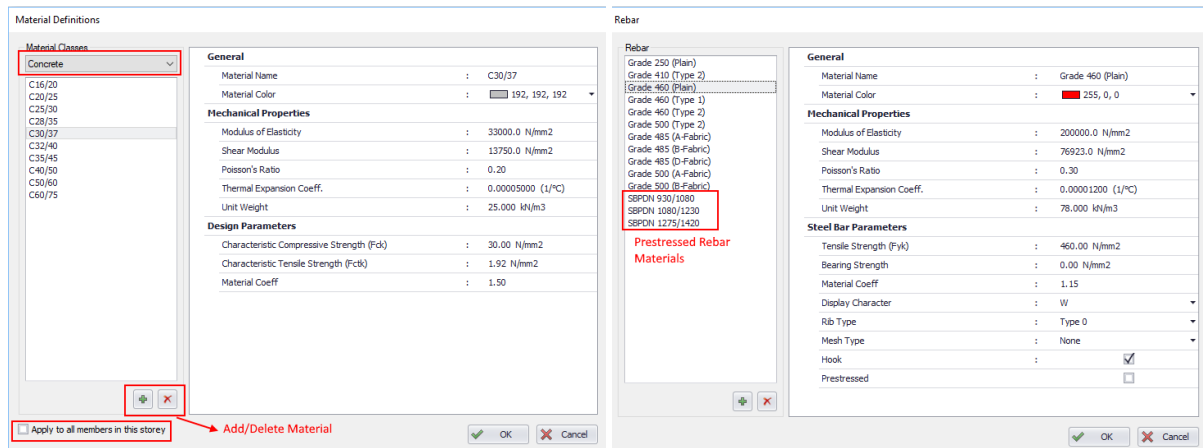


### How To Use Material Definition/Selection Interface:

- Material definition interface opens up when you click on material labels in **“Default Material Assignment”** interface. You can also reach it from material selection buttons under almost all macros for exclusive material selection
- New materials can be defined and selected at the same time from this UI.
  - Select the material class from the top combo which opens with a relevant material class normally
  - Select the material from the Left Hand Side list if it is available there
  - If you will define a new material, click on “+” icon at the bottom. The selected material will be cloned and ready to edit as a new material
  - Use “x” button to delete a material. If this material is not used by any member delete command will be accepted. Otherwise a warning will be prompted and material will not be deleted



- Interface is presented in a notepad-like style. You can click and edit all the fields at the right hand side. The units are displayed automatically.
- “Apply to All Members” is visible only if the interface is reached from “Default Material Assignment” interface.



### 13. Merged Shear Walls – Core Walls

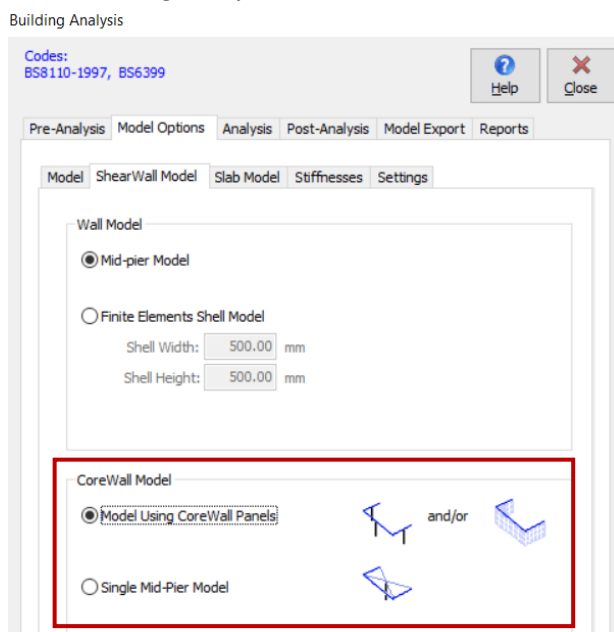
Multiple shear wall panels can be selected and merged into a single core wall for integrated analysis, design, and detailing, significantly increasing design efficiency.

#### How To Use:

- **Multiple select connecting shearwalls → Right-click → Merge Vertical Members**
  - Separate wall panels will be merged into a single core wall.
  - All the panels are also kept in the wall records so walls can be unmerged.
  - If there are any openings on panels, they will be reflected on the core wall as well.
  - The label of the longest wall will be used.

The merged core walls can be analyzed as **separate panels** or as **single mid-pier** :

- **Go to Building Analysis menu → ShearWall Model → CoreWall Model**



➤ **Pick Model Using Corewall Panels**

Analysis will create separate mid-pier for each wall panel. The mid-piers will still act as integrated core-wall as they are connected by their rigid arms. The core-wall will still be designed as a combined entity.

➤ **OR pick Single Mid-Pier Model**

The analysis will combine all walls as single mid-pier so analysis forces can be viewed as single entity. The core-wall will be designed as a combined entity.

Resultant design forces are automatically calculated with respect to geometric center of the merged core wall.

During design, the rebar sizes are automatically selected with the pre-defined numbers & position (in polyline column editor). In short, rebar size will change; but rebar numbers and positions are fixed.

You cannot assign openings, span loads or nodal loads on a merged core-wall. You must explode it first. To unmerge the walls :

- **Select the merged wall → Right-click → Explode Merged Members**

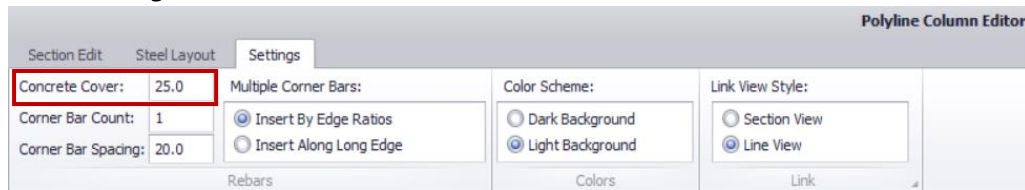
## 14. New Polyline Column Editor

Define and design any steel reinforcement layout for irregular shaped columns and merged core walls. Take advantage of automatic containment or quickly position bars and links yourself.

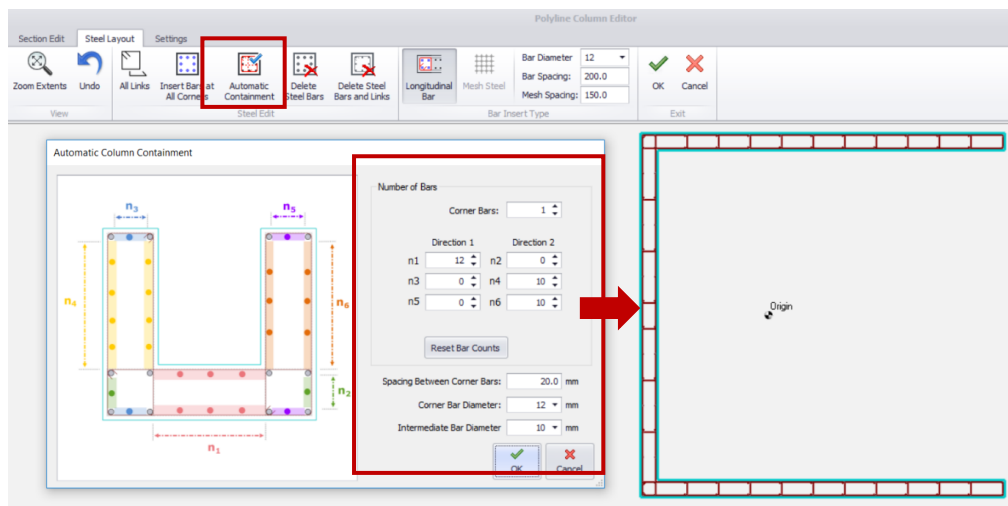
### How To Use Automatic Containment for merged core-walls :

Rebars for the standard core wall shapes like L, U, T can be automatically laid out. In the Polyline Column Editor menu :

- **Select the merged core wall → Right-click → Polyline Column Editor**  
 ➤ **Go to Settings tab → Check Concrete Cover**

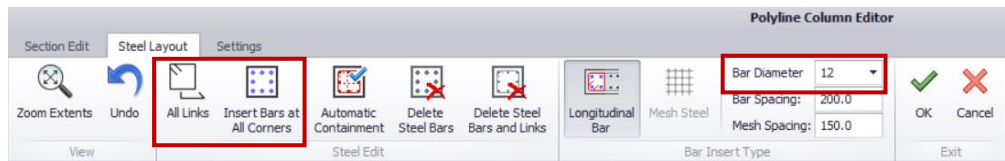


- **Go to Steel Layout tab → Pick Automatic Containment → Input the parameters → OK**



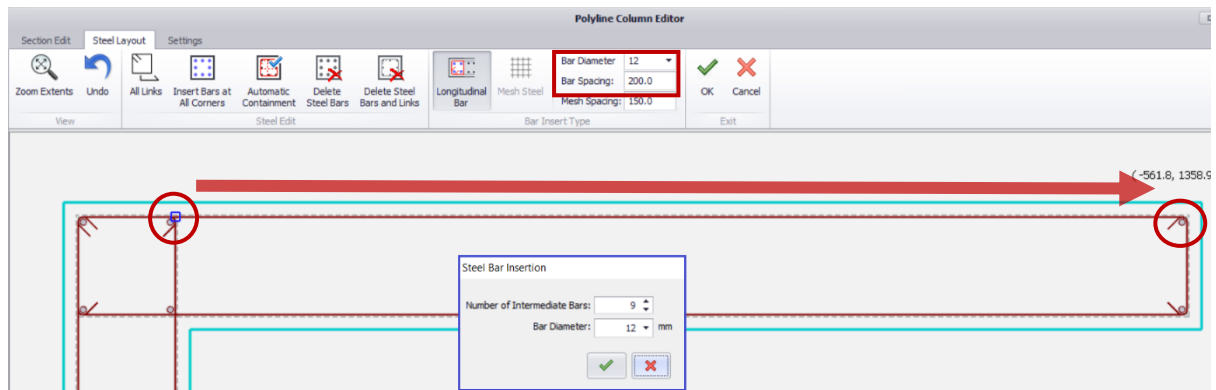
## How To Insert Reinforcement individually :

For non-standard shapes of merged corewalls, where the automatic containment may not be possible, you can define the reinforcement positioning manually. To insert the main rebars :

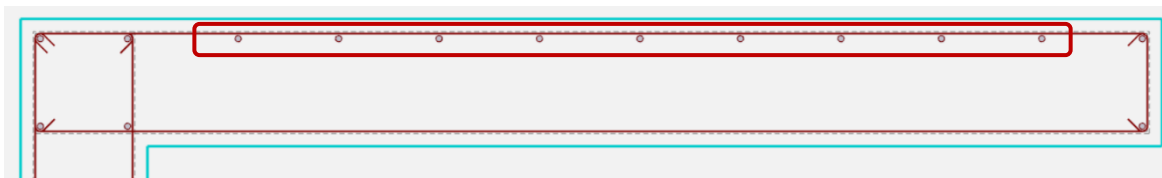


- Input **Bar Diameter** to set default rebar
- Pick **All Links** → Links will automatically be places around the perimeter of the walls
- Pick **Insert Bars at All Corners** → Rebars will be added at corners of links.

To insert multiple bars along the edge of the wall:

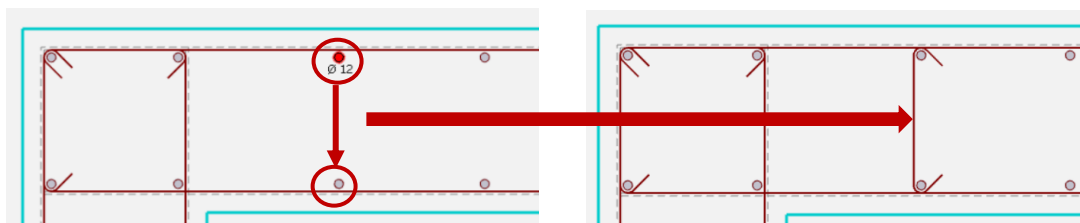


- Input **Bar Diameter & Bar Spacing** to set default rebar & spacing
- Place cursor on an **edge** of intersection of grey grid → a **square grip** will appear
- **Left-click & drag** to the next intersection of grey grid → ensure you see a **square grip**
- Confirm the **Number of Intermediate Bars & Bar Diameter** → OK



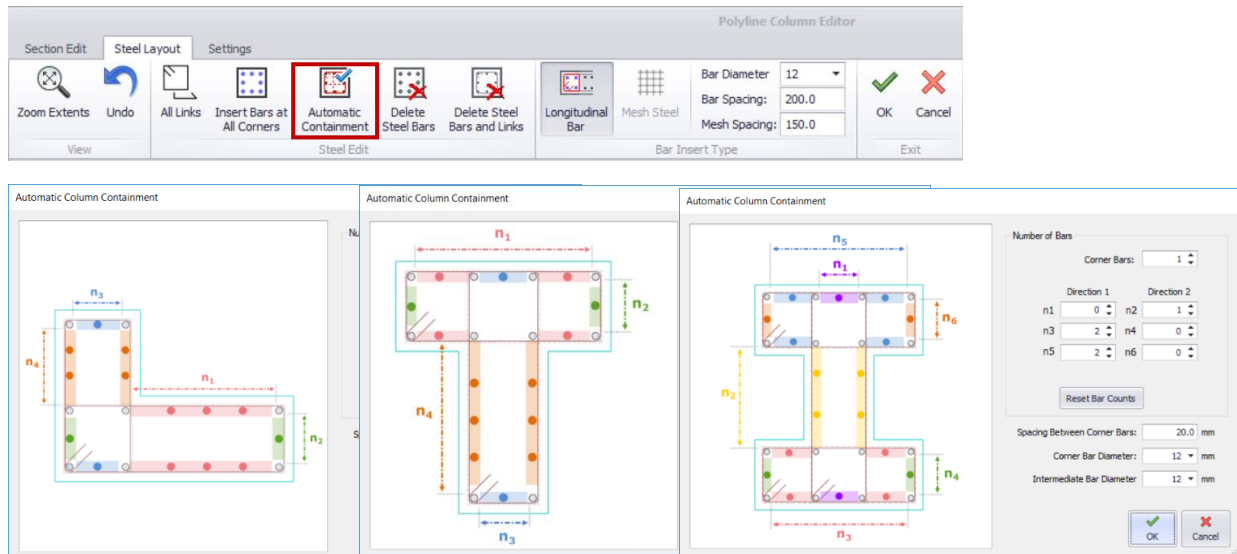
To insert links between rebars :

- Insert required rebars at **both edges** of the wall
- Move cursor over an **existing rebar** → a **black circle** will appear
- **Left-click & drag** to the next rebar → a **link** will be created



## How To Use for columns :

Automatic Containment will work for standard column shapes such as L, C, T, I, + shapes.



Use Automatic Containment as far as possible. Any additional rebars can be entered manually.

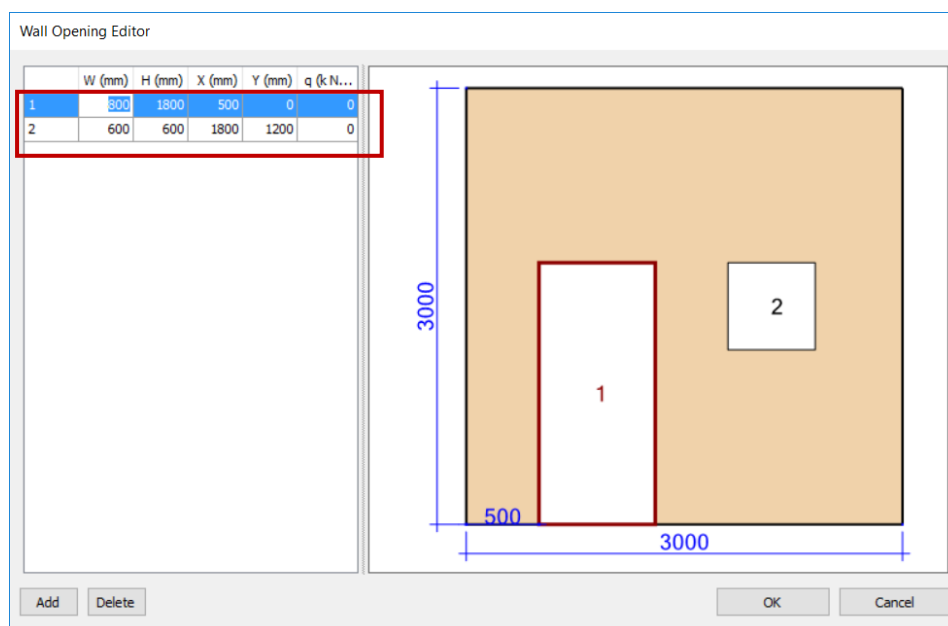
The same method used for merged core-wall can be used to insert bars & links manually for irregular (polyline) shaped columns. This can handle multiple bar layers, bundled bars and ties. All arrangements can then be designed and bar sections made in the column section design once the analysis has been performed

## 15. Multiple Openings in Shear Walls

Multiple openings can be defined for shear walls / core walls using the new **Wall Opening Editor**. Rectangular or square opening can be accurately positioned and meshed in the building analysis.

### How To Use:

- **Select a shearwall → Right-click → Wall Opening Editor**



- Click **Add / Delete** buttons to insert / delete openings.
- You can **select** the openings on the drawing or on the table.
- You can change **Width, Height and Position** of the openings on the table.
- Before analysis, ensure that the wall model is set to **FE Shell Model** :
  - **Select** a wall → Right-click → Properties → 3D tab → Wall Model Type → Pick **FE Shell Model** (only this wall will be meshed) **OR**
  - Go to Building Analysis menu → Model Options → ShearWall Model → check **Finite Element Shell Model** (All walls in the model will be meshed)

The shearwall openings cannot be closer than 200mm to each other or to the shearwall edges to ensure proper meshing. This is automatically adjusted in the meshing of the wall during analysis.

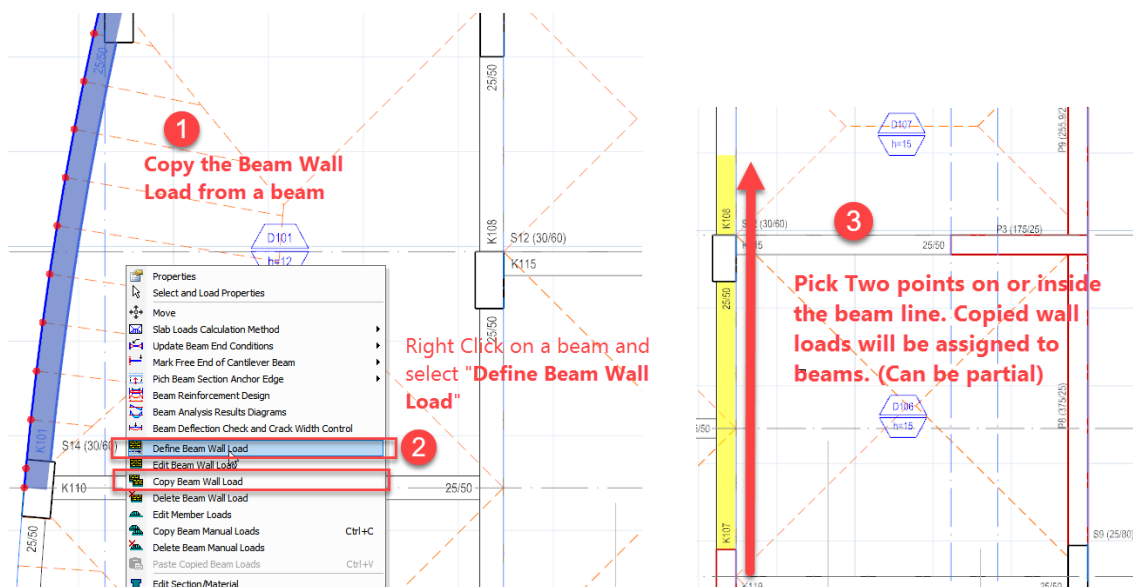
## 16. Infill / Brick Wall Insertion by picking two points

User can pick two points on the screen and define an infill wall on beams and shear walls; greatly speeding up the modelling process.

- If the infill wall spans different beams and walls, it is partially assigned to related members.
- Infill wall will be applied to the edge of the columns, so loading is accurate.

### How To Use:

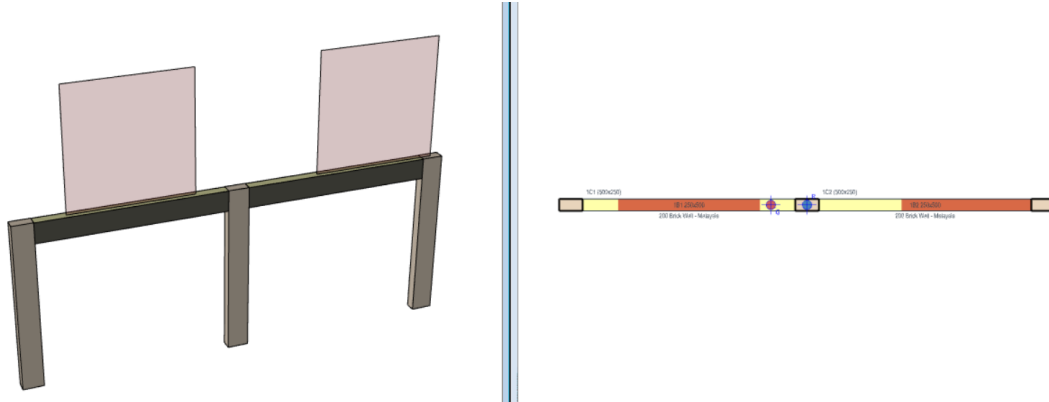
- 1** Right-click on a beam with wall load applied → select **Copy Beam Wall Load**
- 2** Right-click on a beam → **Define Beam Wall Load**
- 3** Left-click **two points** on other beams and walls to copy that load on the beam/walls.



- If the drawn line does not span the entire length of beam, a partial wall load will be defined.

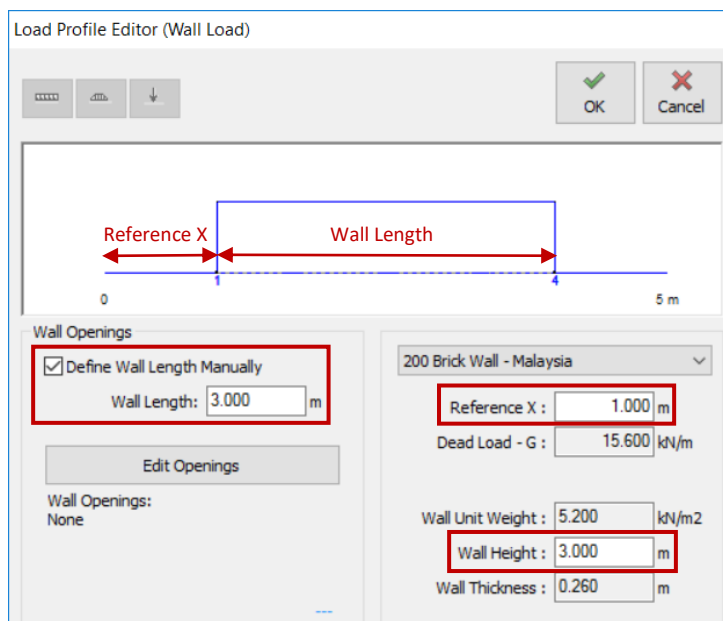
## 17. Partial Infill Walls

Infill walls can now be partially defined on beams, not only the full length of the beam. These wall loads are displayed in the 3D as well as 2D view for easy checking.



### How To Use:

- **Right-click on a beam** → select **Edit Beam Wall Load**



- Check **Define Wall Length Manually** → Input **Wall Length**
- Pick **Wall Type** from the drop down → Input **Reference X**
- Input **Wall Height (m)**

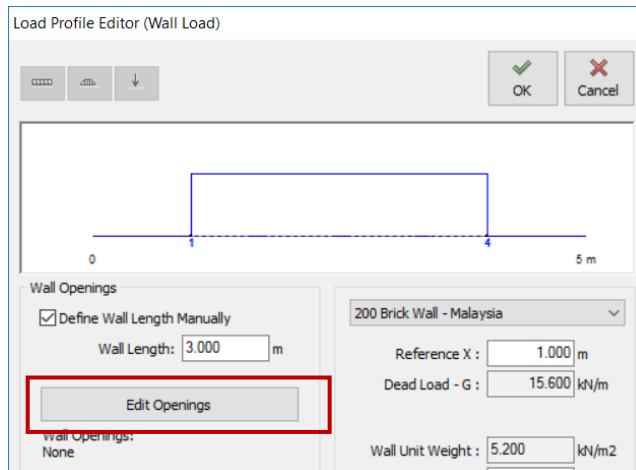
If **Define Wall Length Manually** option is unchecked, it is assumed that the wall will be applied to the entire span of the beam. The load will be auto-calculated based on total beam length minus the column sizes at the upper storey.

## 18. Multiple Openings in Infill Walls

Multiple openings can be defined for infill walls using the new **Wall Opening Editor**. Rectangular or square openings can be accurately positioned to consider doors and windows and associated loading for more economic beam loading and design.

### How To Use:

- **Right-click** on a beam and select **Edit Beam Wall Load**



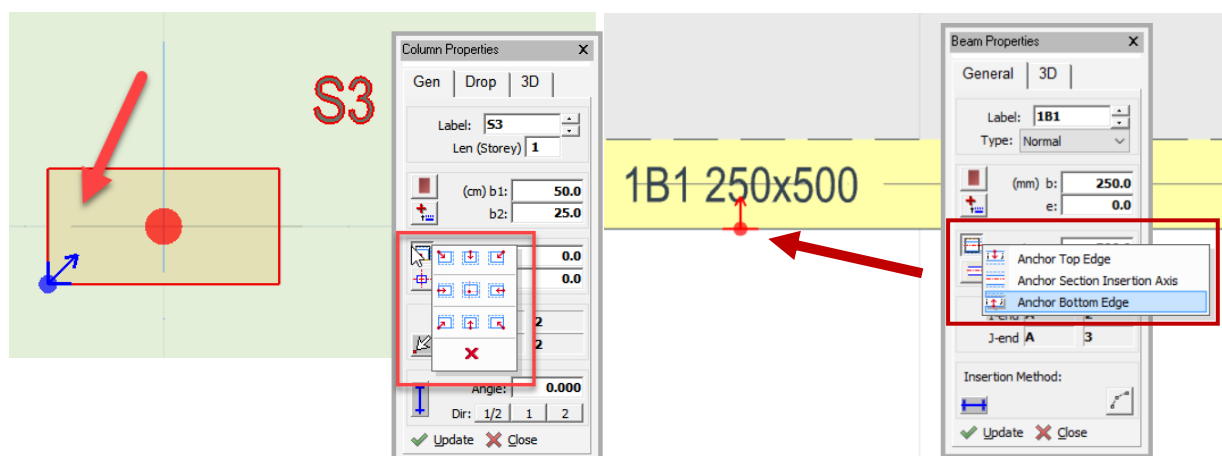
- Click **Edit Openings** button → “**Wall Opening Editor**” dialog will appear
- The method of insertion is the same as **shear wall opening** (refer to previous section)

## 19. Section Anchors

Column and beams positions can be anchored to a specific point on their sections. When the section size is changed, the member is automatically resized with reference to the anchor point; great for modifying arrangements against fixed architectural and layout positioning.

### Set Anchor points via Properties dialog :

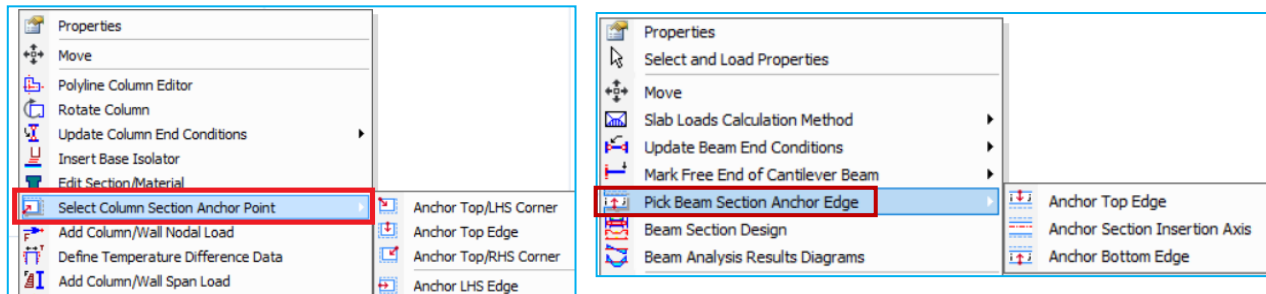
- **Select** a column or beam
- **Right-click** → **Properties**
- Click on the **Anchor** icon & choose the desired anchor points



The default for column is “**Anchor Section Insertion Point**”. The beam default is “**Anchor Section Insertion Axis**”.

### Set Anchor points via right click dialog :

- **Select** a column or beam
- **Right-click** → **Select Column Section Anchor Point** → Choose the anchor **OR**
- **Right-click** → **Pick Beam Section Anchor Edge** → Choose the anchor



Alternatively, select the column or beam (without opening the member properties dialog)

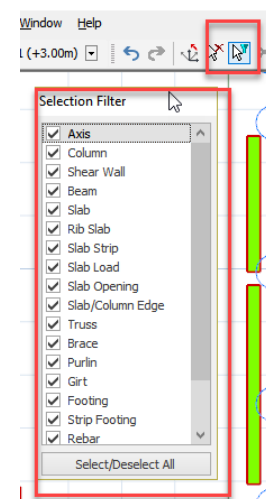
- Hold **CTRL + SHIFT** → Press **Arrow Keys** to set anchors to corners automatically
- Hold **CTRL** → Press **Arrow Keys** to flush the member to grids

## 20. Selection Filter

Select and filter elements quickly during modelling; very useful when managing and editing large models.

### How To Use:


- Click and toggle **Selection Filter** on top toolbar.
- **Check** the member types you want to select.
- When you select members by window method, only the specified member types will be selected.
- You can continue to select any member using the structure tree even if the selection filter is activated.
- The selection will be de-activated when you **untoggle** the **Selection Filter** toolbar button.



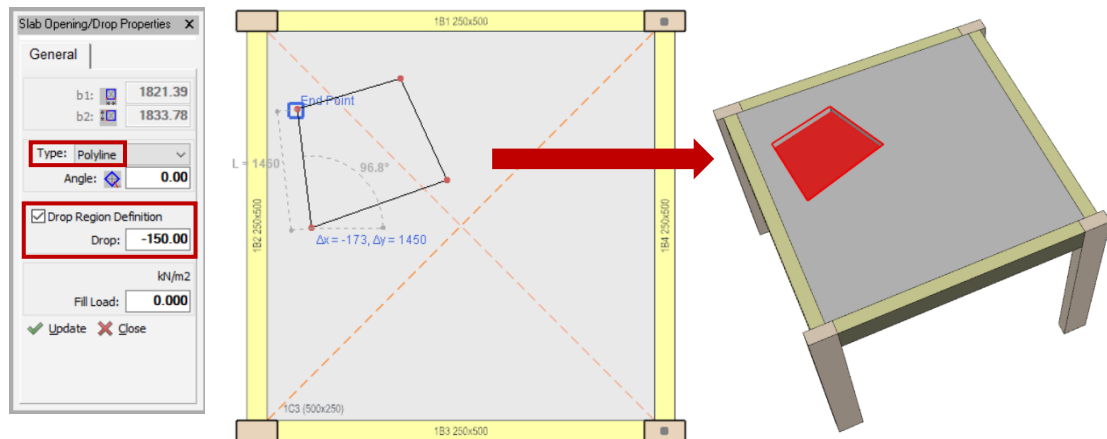
## 21. Slab Drops of any shape

Easily insert isolated slab drops (& openings) of any shape using the Slab Opening/Drop function anywhere in your project floor, handle toilet drops and architectural set downs with ease.

### How To Use:

- Click on **Slab Opening** icon 
- Under Type → Choose **Polyline** (the other options are Rectangular / Circular)
- Click successively to define the **corners** of the drop → last click to close the loop





- Check **Drop Region Definition** → Input Drop in mm

Negative value is a drop & positive value is a raise. Example:

- If the slab thickness is 150mm & a -50mm drop is entered, then the slab thickness at the drop is 100mm.
- If a positive 50mm is entered, the slab will be thickened & raised with final depth of 200mm.

## 22. Inserting Multiple Slabs in one go

Multiple slabs can be inserted in one go, significantly speeding up repetitive slab modelling.

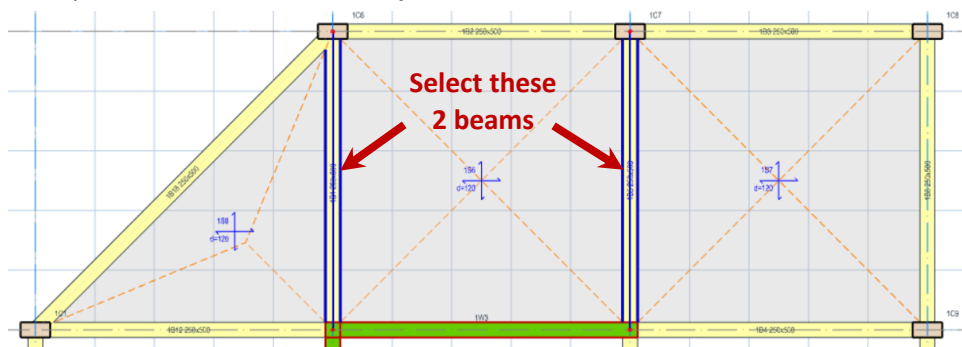
### How To Use:

- **Multiple select** any beams at the boundary of intended slabs

Example : You only need to choose an internal beam to insert 2 slabs on each side of the beam.

- Click on **Slab Properties** icon & input the required data
- **Concrete Member** (top menu) → **Insert Slabs Automatically**

Multiple slabs will be inserted adjacent to all the selected beams.

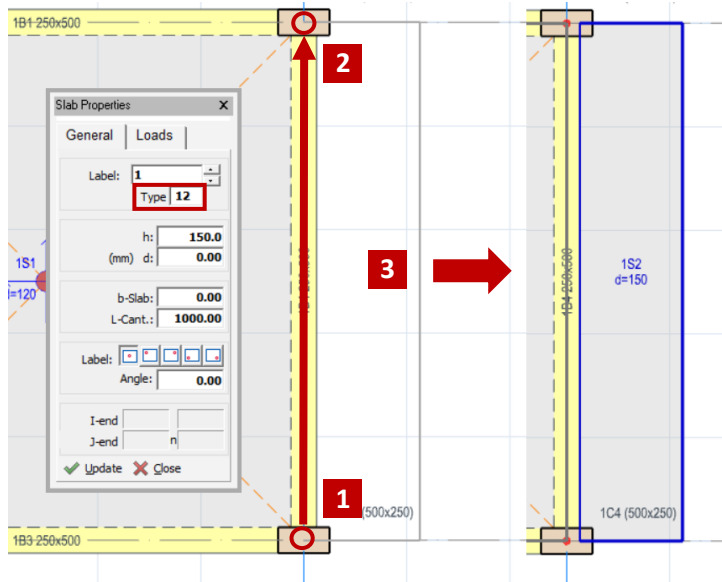


## 23. Enhancement to Cantilever Slab creation

Creation of cantilever slabs (Type 12) have been made easier & more intuitive. They are inserted by 3 clicks - clockwise or counter-clockwise direction does not matter anymore. Cantilever slabs can now span more than one beam or wall.

### How To Use:

- Click on **Slab** icon & choose **Type 12**



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

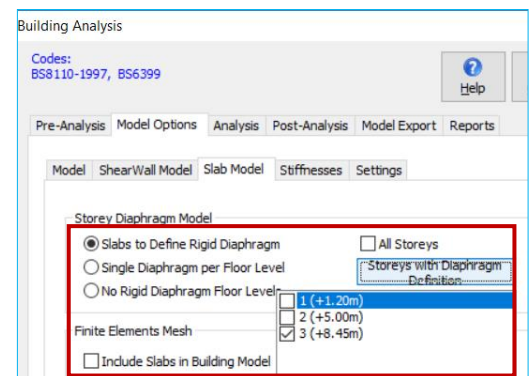
## 24. Rigid and/or Semi-Rigid Diaphragms by Floor

On the Building Analysis Menu, each storey can be assigned different diaphragm behavior. For example, you can mesh a floor using semi-rigid diaphragm and at the same time assign other storeys to use rigid diaphragm.

### How To Use:

- Go to **Building Analysis** → **Model Options** → **Slab Model**
- **Uncheck All Storeys**
- **Pick Storeys with Diaphragm Definition**
- **Tick only Storeys to consider diaphragm**

The chosen diaphragm modelling will be assigned only to selected storeys. Unselected storeys will have **no diaphragm or semi-rigid diaphragm** if the floors are meshed for that storey.

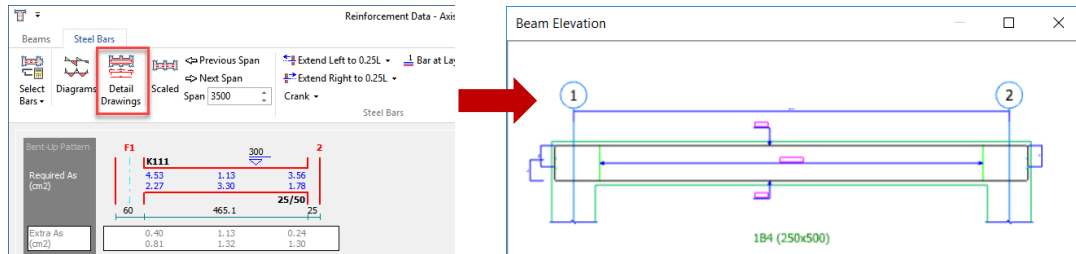


## 25. Detail Drawings Preview

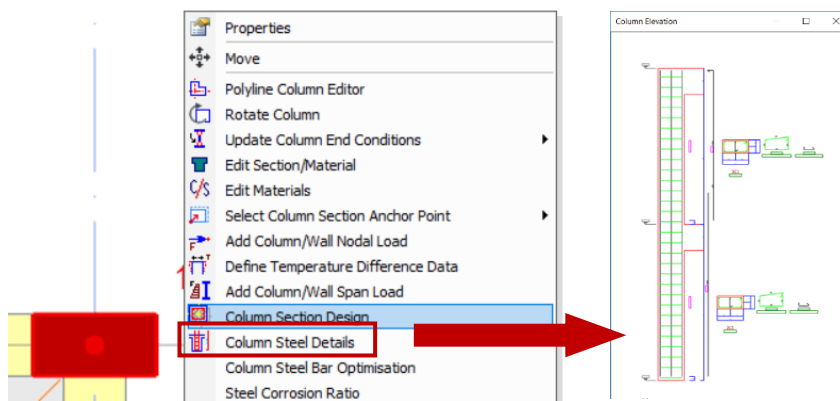
Beam and Column Elevations drawings can now be previewed in ProtaStructure without opening ProtaDetails. This enables user to quickly review details before exporting out to ProtaDetails for full drawing production.

### How To Use:

- For Beam Details: On beam reinforcement editor → Select **Detail Drawings**

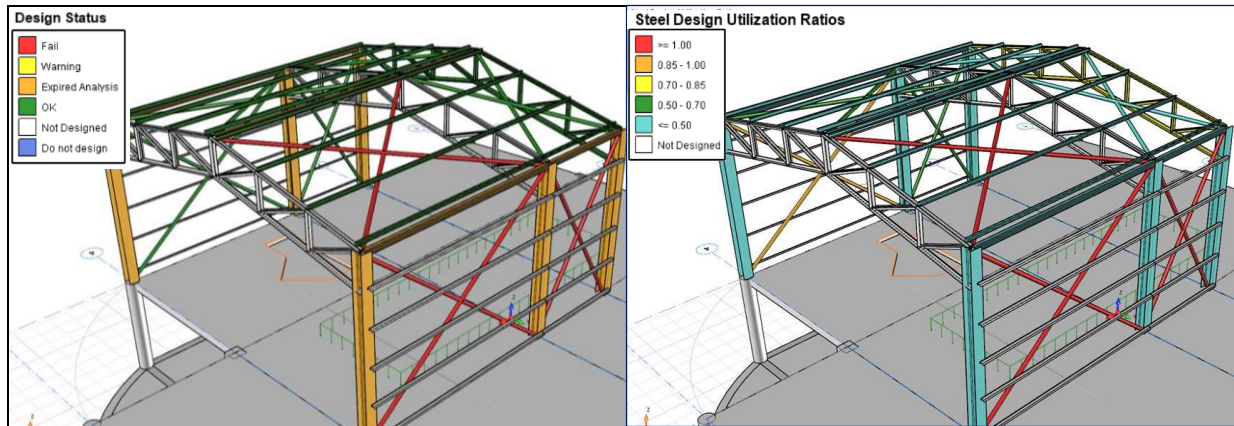


- For Column Details: Right-click on a selected column → select **Column Steel Details**.



## 26. Enhanced Visual Interrogation

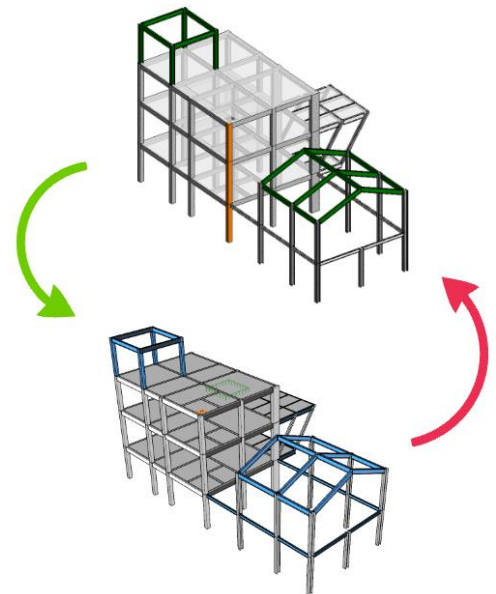
Visual Interrogation has been enhanced to include the **Steel Design Status** and **Steel Utilization Ratios** for quickly reviewing design efficiency.



## 27. Enhancements in Revit Integration


ProtaStructure now supports bi-directional links with Revit Structure greatly enhancing project coordination and workflow. Model changes can be synchronized and tracked both in Revit and ProtaStructure. Take advantage of enhanced filtering commands to share parts of the model you like, co-ordinate a floor at a time or the whole project model, you choose.

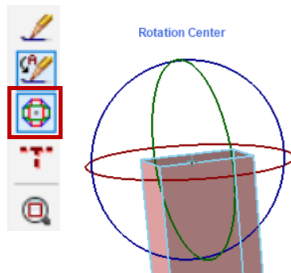
- You can import a ProtaStructure model into Revit. It is fast & simple because the model is opened directly in Revit; there is no intermediary file.
- The link supports multi-material, e.g. concrete & steel.
- There is no limitation in geometry, e.g. includes sloping slabs.
- It is flexible because you can pick and choose what you want to import, e.g. particular storeys or member type, i.e. beams or columns or slabs.
- Changes in Revit model can then be exported back to ProtaStructure. The ProtaStructure model will be updated (e.g. change of member size) or merged (e.g. addition/deletion of members).



Please note that **ProtBIM** add-on must be installed in the same computer where Revit Structure is installed. Please download from online help : <http://help.protasoftware.com>.

## 28. Center of Rotation

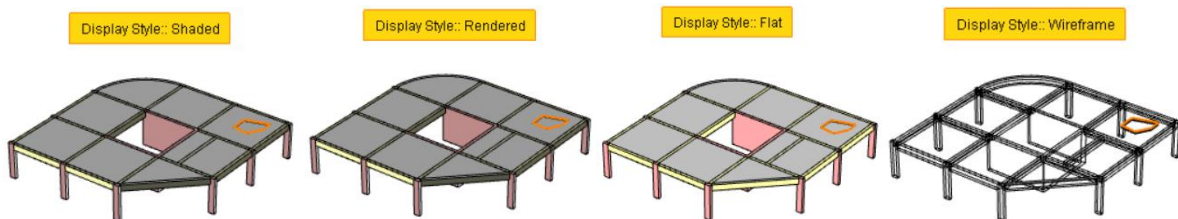
A toolbar toggle button **Center of Rotation**  is introduced to enable easy rotation of the 3D view about a point of interest.



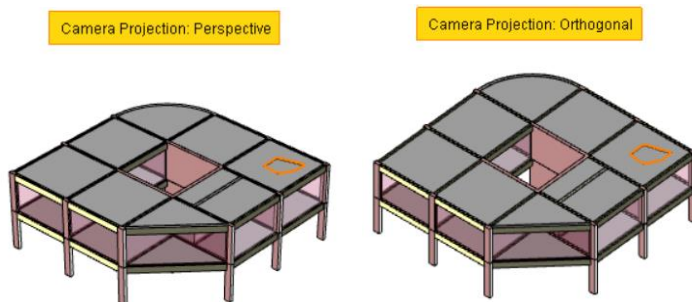
- When **activated (On)**, place the mouse cursor on the point of interest, e.g. a column/beam joint.
- **Right-click & drag** & the view will rotate about the cursor position (globe will appear).
- If **deactivated (Off)**, the view will rotate about viewport center.
- Shortcut key is **F9** to toggle on/off.

## 29. Shortcut keys for 3D Display Mode & Camera Projection

Pressing **F11** repeatedly now cycles through the **display modes** on a 3D View. The mode is shown at the top of the screen (**Shaded, Rendered, Flat, Hidden Lines, Wireframe**).



Pressing **F12** repeatedly will cycle through **Camera Projection** on a 3D view. The active name is shown at the top of the screen (**Perspective & Orthogonal Projection**).



## 30. User Defined Thumbnail

You can set your own model thumbnail to be shown in the open project dialog whenever the project is selected. This helps you to easily identify & locate the project you want to open.

The thumbnail is automatically created using the latest active view camera position during each save operation.

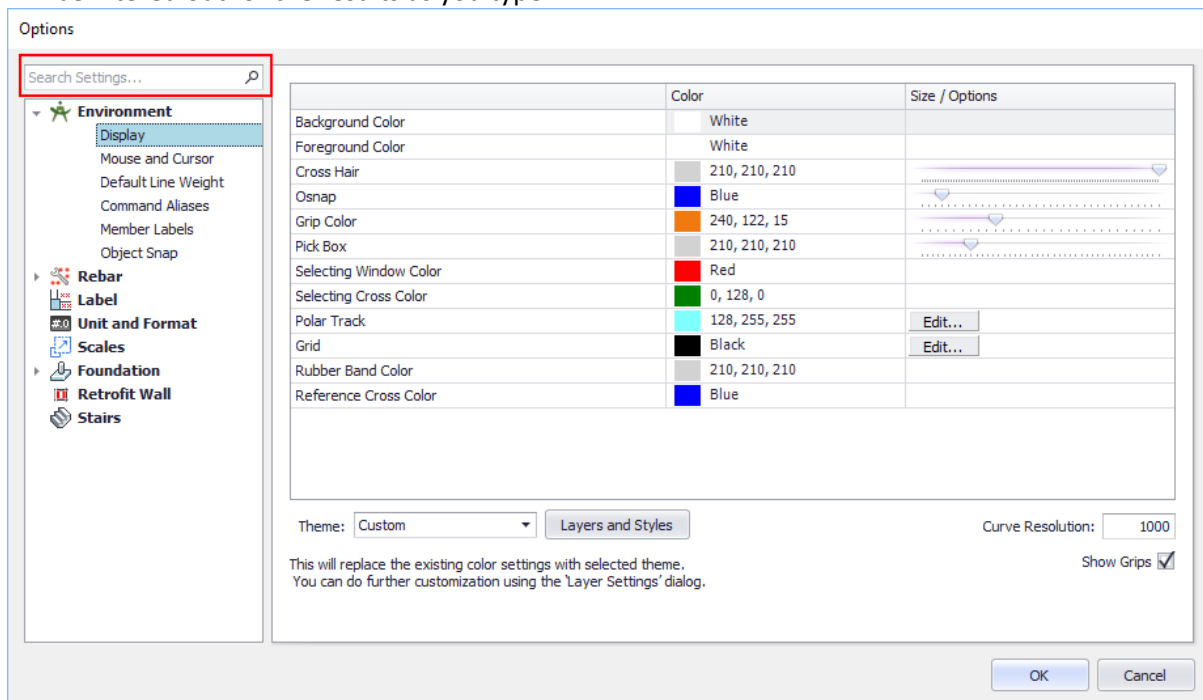
### How To Use:

- Go to the desired view & camera position
- Go **File** → **Save Active View As Thumbnail Image**

The file name is **thumbnailMain.jpg** & it is stored in the project folder.

## 31. New Options Interface

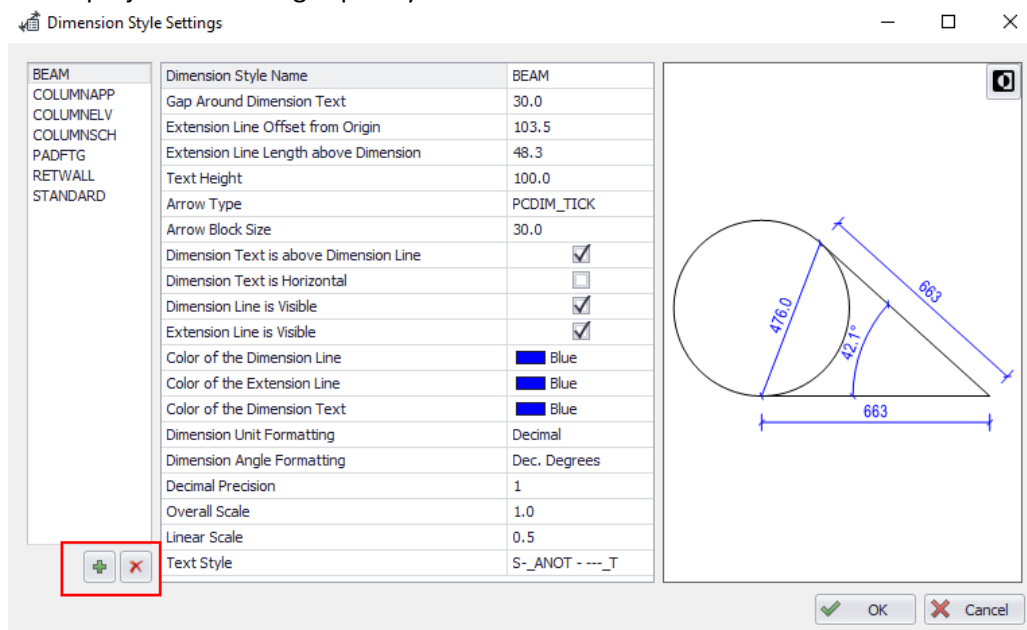
Drawing and detailing options are now classified under a single interface. You can navigate from the Left Hand Side panel and search for any specific option on the search text box. The navigation panel will be filtered out for the results as you type.



## 32. New Dimension Styles Interface

Dimension styles can be edited using this interface and can be saved along with the project

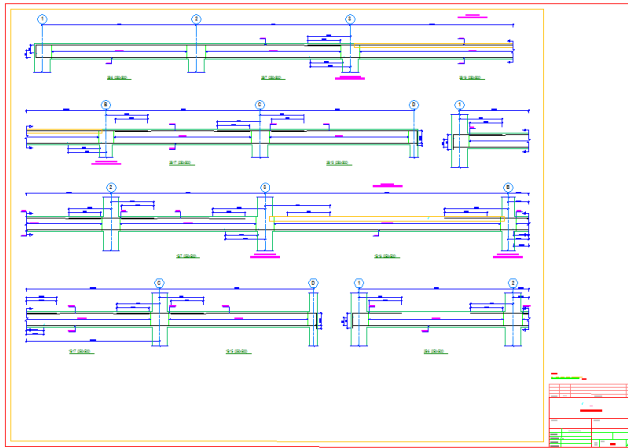
- You can find it under **“Formats > Dimension Styles”**
- All the necessary dimension styles for detail generation will come predefined
- Use **“+”** and **“x”** buttons to add/remove dimension styles
- Edit them as you wish. When you click **“Ok”** they will be saved in the project data. No need to save project or drawing explicitly





### 33. Auto truncation of Beam Elevation Details

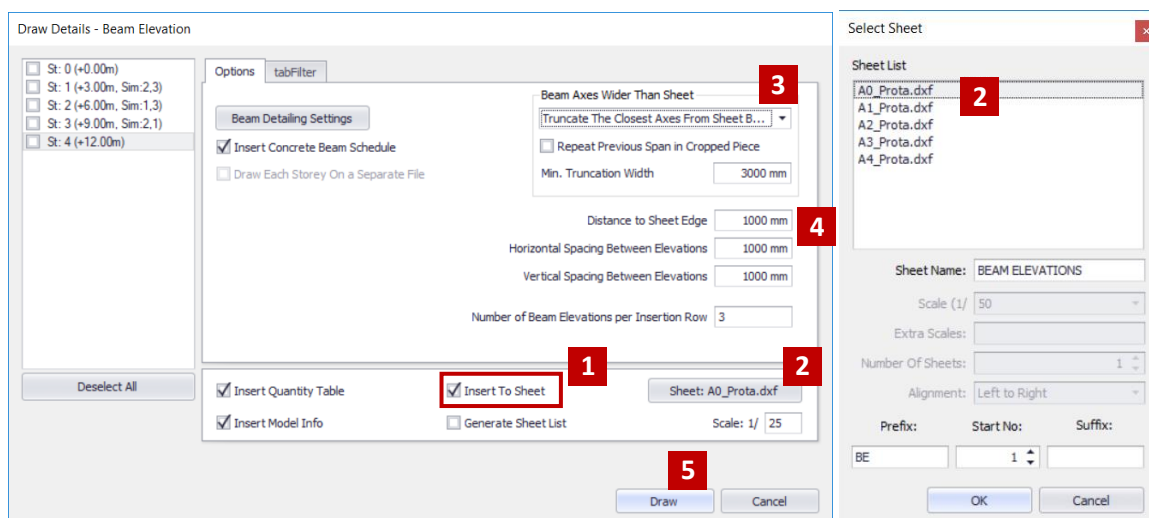
In **ProtaDetails**, long beam elevations can be automatically truncated to fit in any sheet border. This translates to significant increase in productivity as these time-consuming editing is no longer required.



#### How To Use:

- Click **Auto Generate Details / Details Drawings Manager** in **Start-up** page
- Or **Right-Click** on **Storey Beam Elevation Drawings** → **Draw all Beam Elevations**

In **Draw Details – Beam Elevations**, follow these steps :



- 1 Check **Insert to Sheet**
- 2 Click **Sheet** → Choose the **sheet** in Sheet list
  - You can save your own sheet in C:\Users\[User]\Documents\ProtaLib\Sheets
- 3 Pick how to truncate under **Beam Axes Wider Than Sheet** by select from drop down :
  - **Don't Truncate** even if beams details are out of sheet border
  - **Truncate from Sheet Border**
  - **Truncate The Closest Axes from Sheet Border** – will ensure truncation only at axes
- 4 Input the various criteria of truncation & detailing
- 5 Click **Draw**

### 34. Column Grouping

Automatic column schedule grouping is performed for columns with the same size & reinforcement requirements, significantly reducing drafting requirements for large projects with repetitive column detailing.

- Check option to **Group Similar Column** in **Options** tab of **Draw Details – Column Schedule**
- **Draw the Column Schedule**

Draw Details - Column Schedule

Filter Options

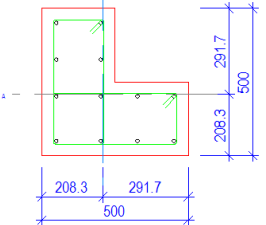
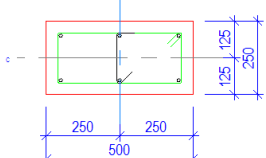
☒ St: 1 (+3.00m, Sim:2,3)  
☒ St: 2 (+6.00m, Sim:1,3)  
☒ St: 3 (+9.00m, Sim:2,1)  
☒ St: 4 (+12.00m)

☒ Draw Column Insertion Axes  
☒ Dimension Column Insertion Axes  
☒ Write Axes Labels On Section  
☒ Dimension Column Section  
☒ Center Column Labels and Texts in Cell  
☐ Draw Separator Lines Between Label Texts  
☒ Print Column Insertion Axis Labels  
☒ Draw Column Vertical Separator Lines  
☒ Draw Outline of Change in Column Size

☒ Print Column Labels Below the Table  
☒ Print Storey Labels Above the Column  
☒ Group Similar Columns  
☒ Create Separate Tables for Columns and Shearwalls  
☐ Show Link Details

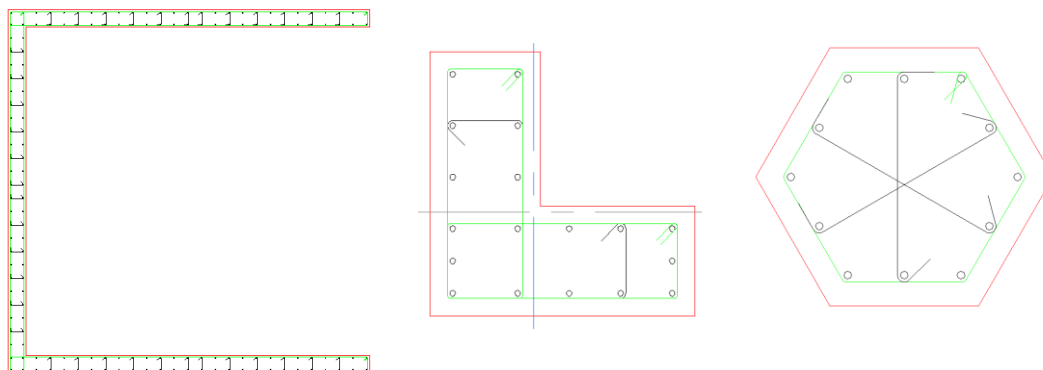
Similar Storey Note: AS BELOW

Link Count: Show per Section

St: 1 (+3.00m)	AS BELOW	AS BELOW
		
	(500x500)	(500x250)
	12H12	6H12
	Link H8-240+H8-240	Link H8-240
		Tie H8-240
	GC1,GC2,1C1,1C2,2C1,2C2,RC1,RC2	GC5,GC6,GC7,GC8,GC9,1C5,1C6,1C7,1C8,1C9,2C5,2C6,2C7,2C8,2C9,RC5,RC6,RC7,RC8,RC9

### 35. New Containment detailing

Full details are provided for new polyline columns and merged walls using the new containment design options.

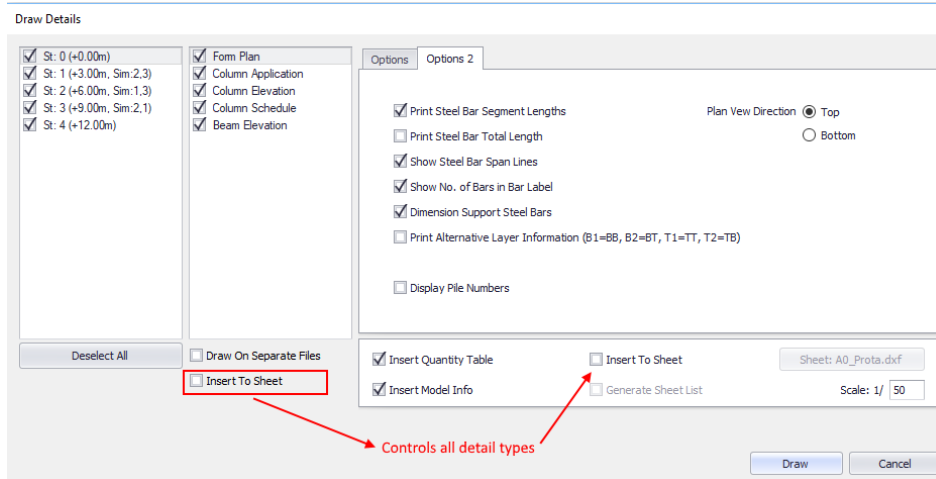




### 36. Enhancement in Details Drawing Generation

There are many enhancements in the details drawing manager to help with the auto-generation of drawings :

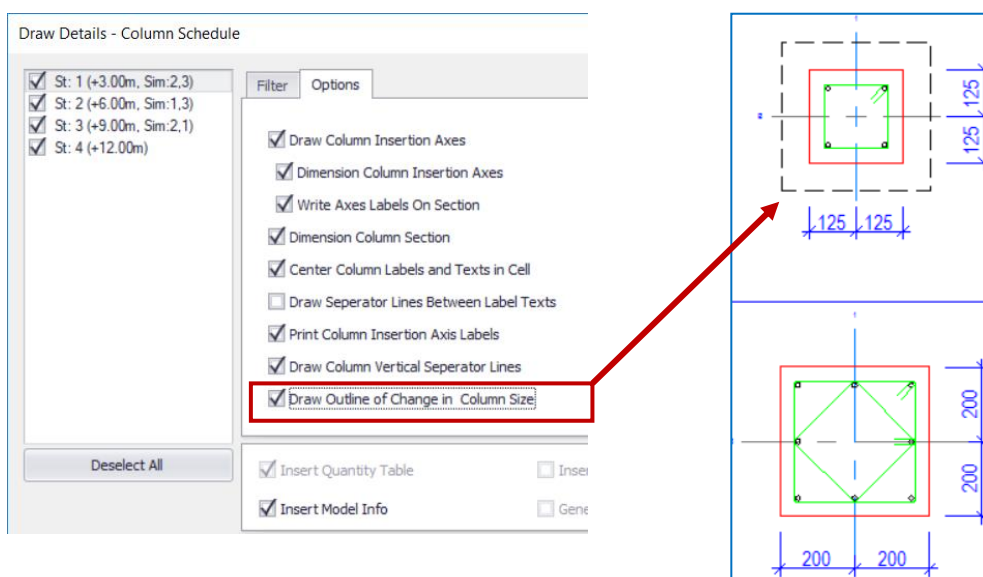
- In Beam Elevation, new filtering options to choose beams to drawn out.
- In Form Plan, ability to hatch columns/walls, show column loads & dimensions on plan view.
- Additional 'Options' tab to control the drawing generation & presentation
- Additional key settings such as **Scale** are included in the same menu for easy accessibility
- New master check box to check **"Insert to Sheet"** on all detail types at once



### Show outline of lower storey columns sections in Column Schedule

You can show the lower storey column sections outline overlayed & drawn in Column Schedules.

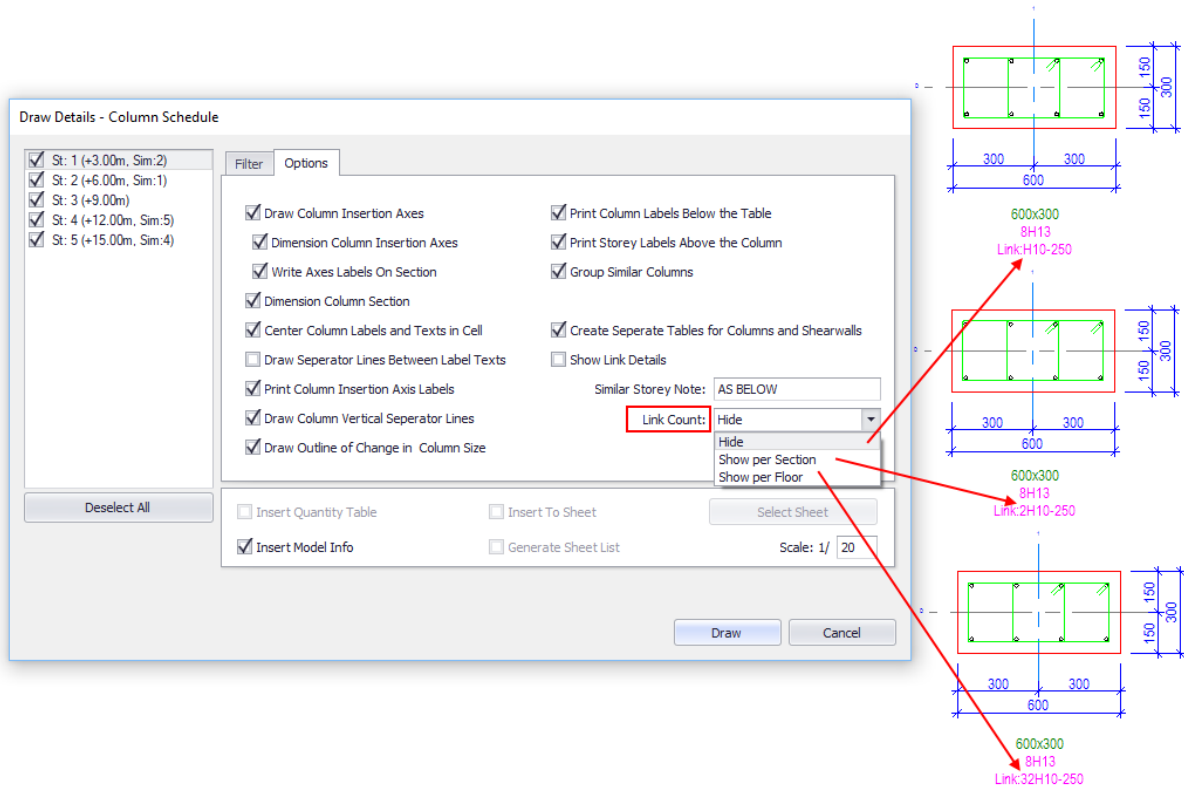
- Under Options tab, check **"Draw Outline of Change in column size"**



## Three options for link count in Column Schedule

You can choose to hide the number of links or display it per section of the column or per total number in the floor.

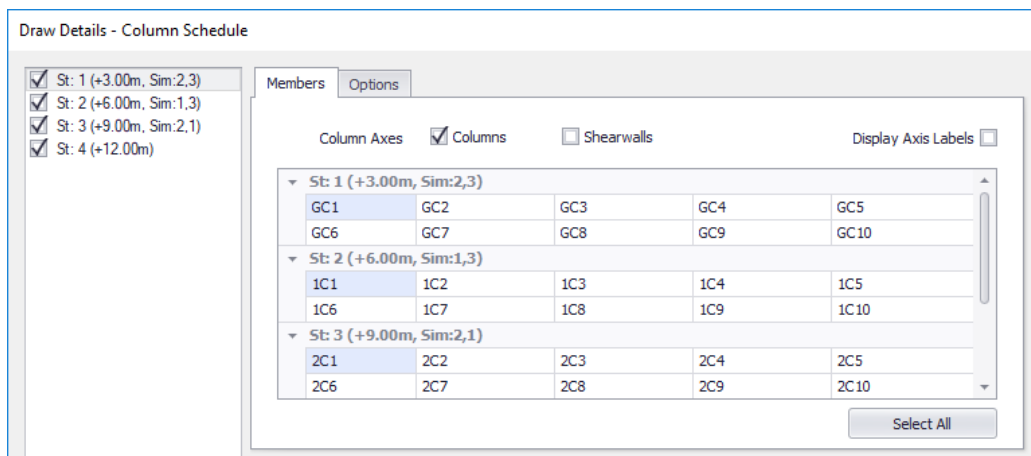
- Under options tab use **“Link Count”** to specify the displayed link quantity



## Filtering interface for Column Schedules & Elevations.

There is a new filtering interface for Column schedule and elevations.

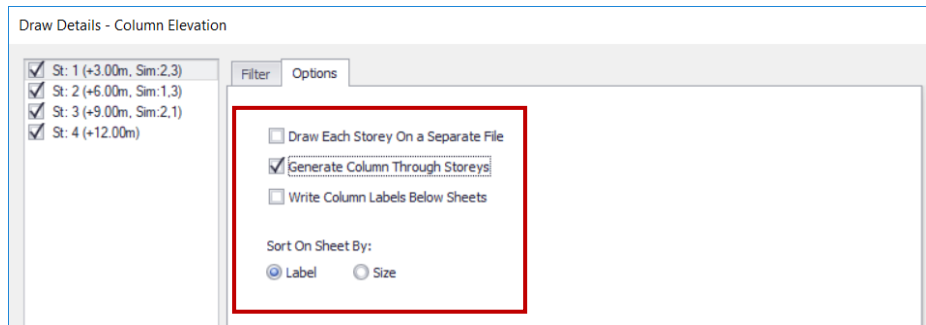
- All columns are displayed ordered by their labels and grouped by their storeys.
- In column schedule when you select a column, the columns at the same position will be automatically selected for selected storeys.



## Column Elevation Options

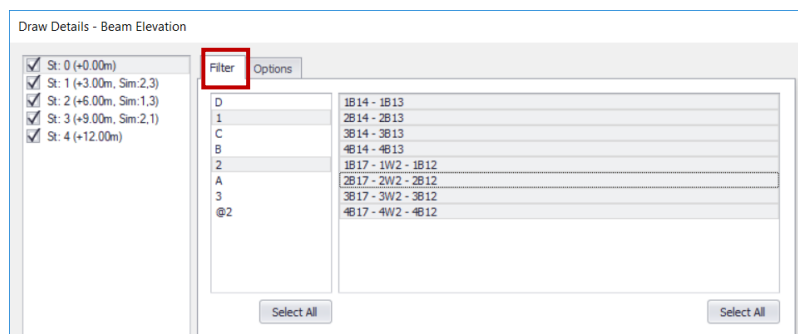
There are additional settings in the **Options** tab :

- Generate Column Through Storeys
- Sorting of column elevations inserting into sheets. You can now order them by size or by label. Label ordering is the default.



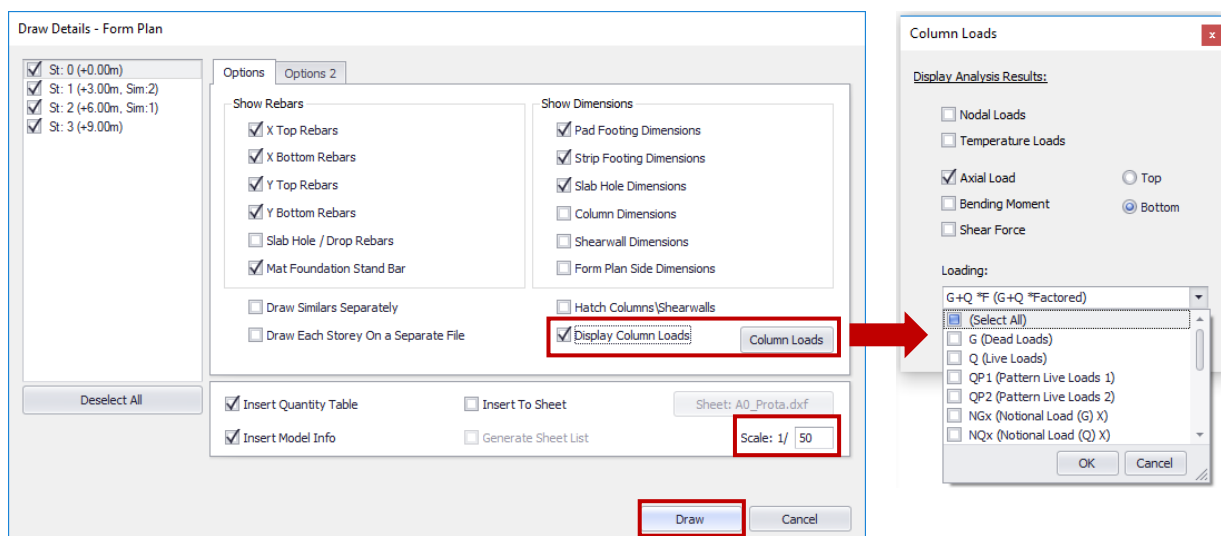
## Beam Elevation Filter

Beam Elevation filter is introduced to easily choose which beams drawing to prepare. User can filter beam elevations to be drawn by storey axis & beam label.



Hold down **CTRL** key to multiple select. Hold down **SHIFT** key → Click on the first item → Click on the last item & everything in between will be selected. Alternatively, click and drag mouse cursor on the list and all the touched items will be selected.

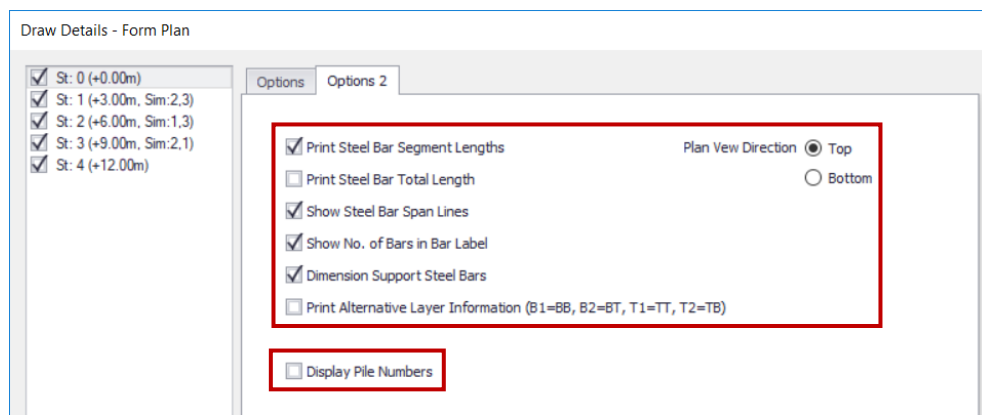
## How to show Column Loads on Form Plan :



- Check **Display Column Loads** → **Column Loads**
- In **Column Loads** dialog → Check **Axial Load / Bending / Shear**
- Check the **Load Cases** and/or **Load Combinations** → **OK**
- Confirm / change **Scale** → **Draw**

## Form Plan Options :

There are additional settings in **Options 2** tab which are mainly for slab reinforcements detail on the form plan.



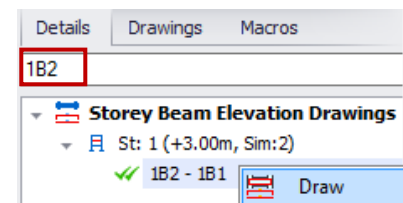
Option **Display Pile Numbers** will label each pile with a number to identify its position.

## 37. Search a Member Detail or Text

The new search functions quickly find, filter & locate a member detail or text, example beam 1B1.

### How To Use Search in Treeview :

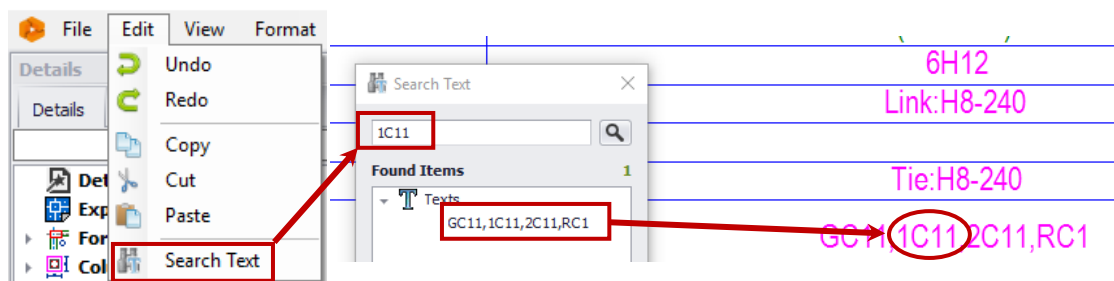
- Go the **Details** tab
- Input a **member label**, e.g. 1B2 in the box
- Only the details with the exact name match will be located & displayed. You can then draw out this detail only.
- Right-click → **Draw**



This is very useful if you only want to draw out a particular detail, not all of them.

After drawing details or objects in the drawing, you can use the new text search function to locate it.

### How To Use Text Search :

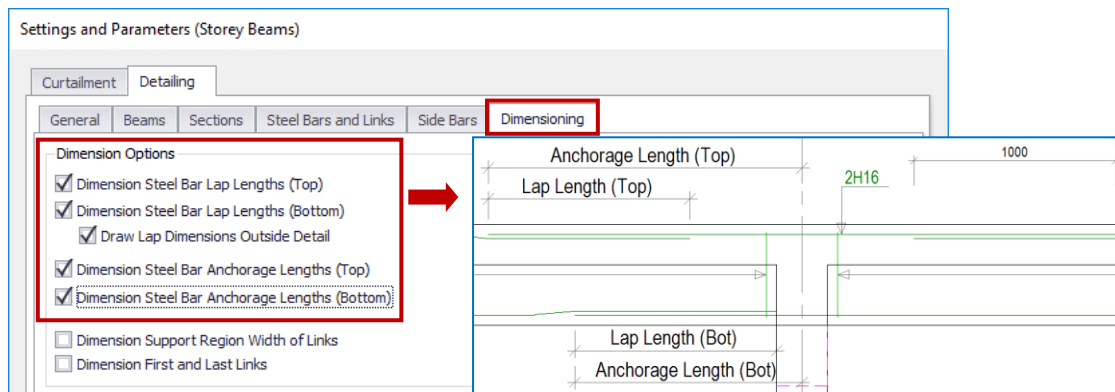


- Go to **Edit** → **Search Text** (CommandLine: SearchText)
- In the **Search Text** dialog, type the **text**, eg. 1C11
- **Double-click** on the found items & it will be located & zoomed into focus

You can use this function to search for all texts in primitive entities and intelligent objects as well.

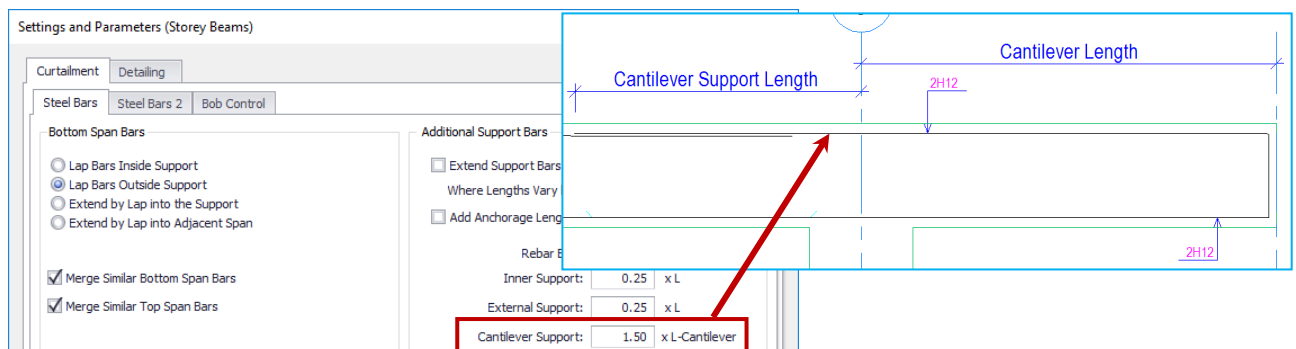
### 38. Beam Detail: Separate Lap & Extension Dimension for Top & Bottom Bars

The dimension options of lap & extension (or anchorage) are separated for top and bottom bars to cater for different users' preference. These options are accessible from the **Beam Design Settings**.



### 39. Cantilever Beam Detail – Rebar Extension length

Cantilever top rebar extension lengths can be adjusted based on a user-defined factor multiplied by the cantilever length. This feature is implemented to cater for some user's preference for this manner of cantilever beam detailing. This option is accessible from the **Beam Design Settings** → **Steel Bars**.



*Note : A minimum length of Inner & External support will be applied if the resultant cantilever support length is less than the former.*

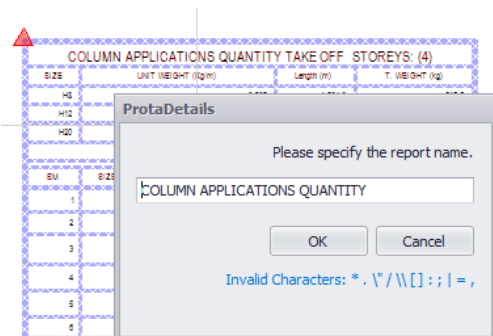
### 40. Multiple selection of Tables for report generation

Any table drawn in ProtaDetails can be converted to a report and generated via the Report Manager.

#### How To Use:

- **Select** a table or multiple select tables (hold down **CTRL**)
- **Right-click** → **Add to Report**
- Give it a title → **OK**
- The report will be created

This particular report will also be added in the report repository **Report Manager** in ProtaStructure (under Other Structural Members)



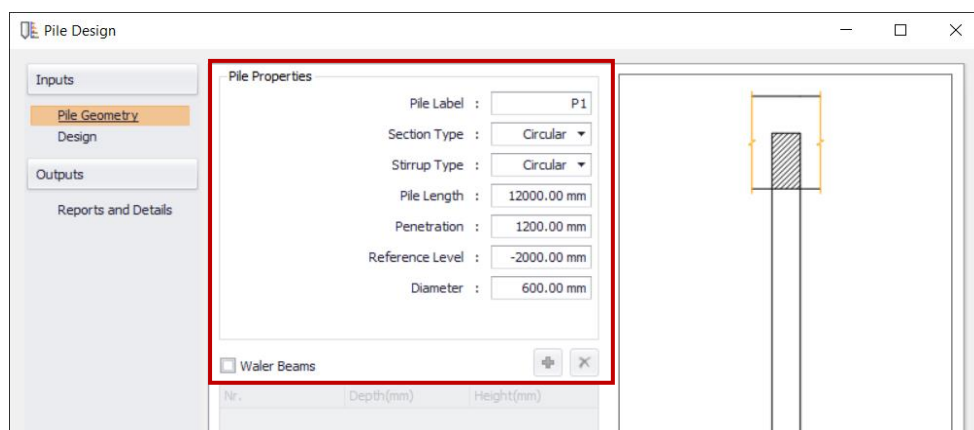
## 41. Pile Design

The new pile design engineering macros calculates the structural and geotechnical capacity of the pile. It consists of three modules:

- **Pile Detailing** → Generates the pile details
- **Working Load** → Calculates the allowable pile structural compression and geotechnical capacities
- **Lateral Analysis** → Analyze and design the pile for combined axial, shear and bending actions

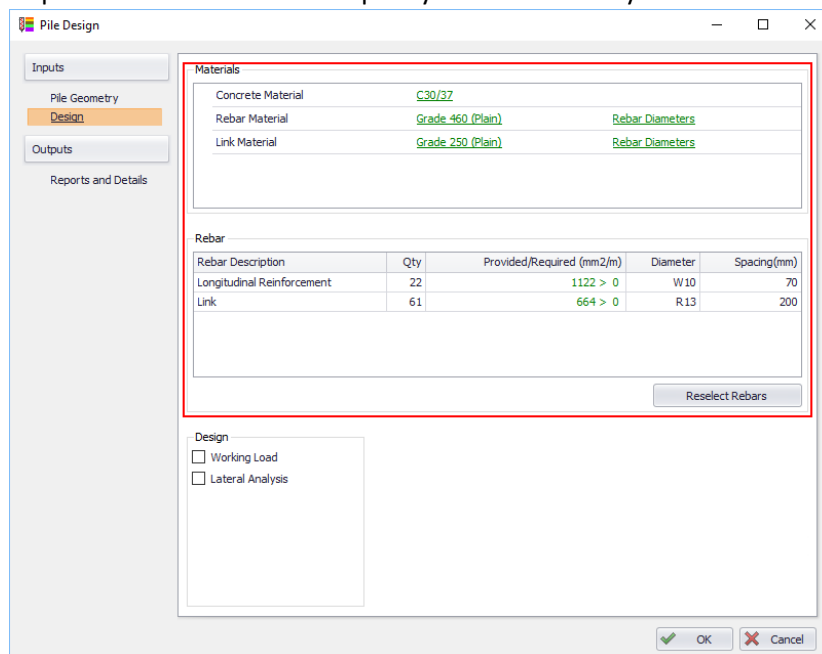
### Pile Detailing :

- Go to **Design Library** (top menu) → **Pile Design**
- In **Pile Geometry**, input all parameters such as Circular / Square / Rectangular, size, length, etc



- In **Design**, select concrete grade, rebar and link grades & diameters.
- The **Main reinforcement & Stirrup (links)** can be defined.

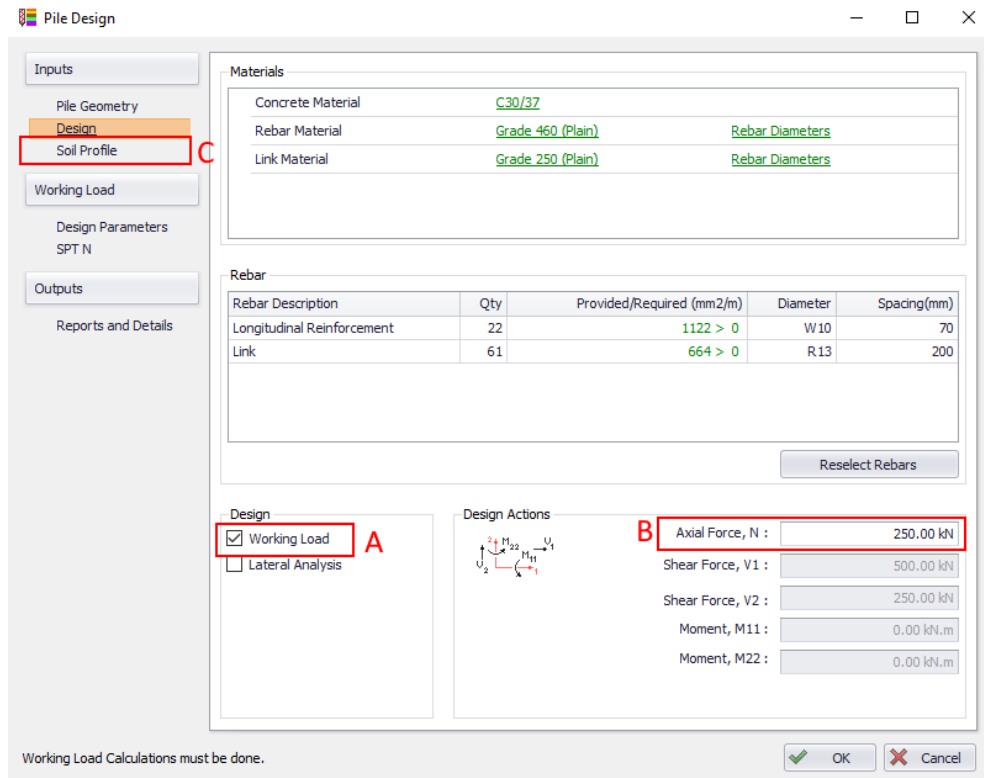
Note: The defined materials and rebars will be used for generation of Pile Details, calculations of pile allowable structural capacity and lateral analysis.



Rebar Description	Qty	Provided/Required (mm2/m)	Diameter	Spacing (mm)
Longitudinal Reinforcement	22	1122 > 0	W10	70
Link	61	664 > 0	R13	200

➤ Click **OK** (on the right bottom of the **Pile Design** dialog) > Pick an insertion point on the drawing to insert the Pile Details.

## Working Load (Allowable compression capacity):



**Inputs**

- Pile Geometry
- Design**
- Soil Profile
- Working Load
- Design Parameters
- SPT N
- Outputs
- Reports and Details

**Materials**

Concrete Material	C30/37	
Rebar Material	Grade 460 (Plain)	Rebar Diameters
Link Material	Grade 250 (Plain)	Rebar Diameters

**Rebar**

Rebar Description	Qty	Provided/Required (mm <sup>2</sup> /m)	Diameter	Spacing(mm)
Longitudinal Reinforcement	22	1122 > 0	W10	70
Link	61	664 > 0	R13	200

**Design**

☒ Working Load

☐ Lateral Analysis

**Design Actions**

Axial Force, N : 250.00 kN

Shear Force, V1 : 500.00 kN

Shear Force, V2 : 250.00 kN

Moment, M11 : 0.00 kN.m

Moment, M22 : 0.00 kN.m

Working Load Calculations must be done.

OK Cancel

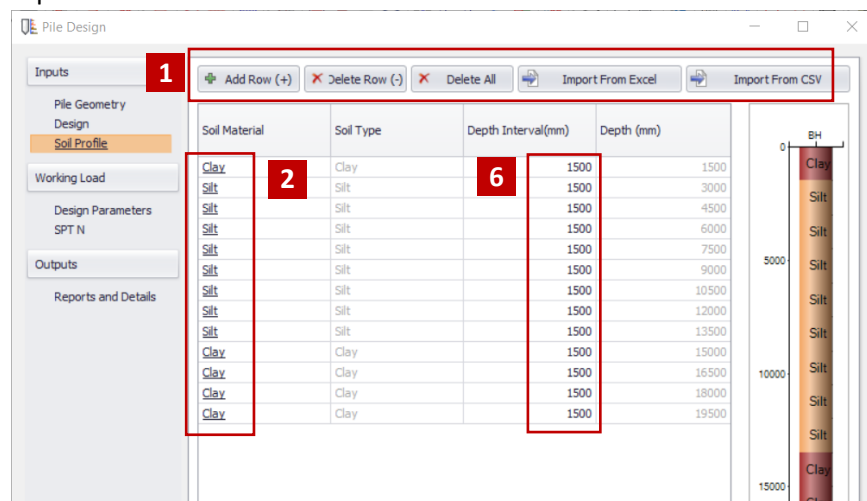
### A. Check the Working Load in the Pile Design > Design tab

The **Soil Profile** and **Working Load** tabs will be visible on the left panel. Also **Design Actions** group box will be visible at the bottom-right corner of the form.

In **Working Load** tab, there are two sub-tabs: **Design Parameters** and **SPT N**

### B. Enter Axial Force value.

### C. In Soil Profile tab, update the soil profile based on the bore log in the soil investigation report.



**Inputs**

- Pile Geometry
- Design**
- Soil Profile**
- Working Load
- Design Parameters
- SPT N
- Outputs
- Reports and Details

**Soil Profile**

Soil Material	Soil Type	Depth Interval(mm)	Depth (mm)
Clay	Clay	1500	1500
Silt	Silt	1500	3000
Silt	Silt	1500	4500
Silt	Silt	1500	6000
Silt	Silt	1500	7500
Silt	Silt	1500	9000
Silt	Silt	1500	10500
Silt	Silt	1500	12000
Silt	Silt	1500	13500
Clay	Clay	1500	15000
Clay	Clay	1500	16500
Clay	Clay	1500	18000
Clay	Clay	1500	19500

1 Add Row (+) 2 Delete Row (-) 3 Delete All 4 Import From Excel 5 Import From CSV

6

7 BH

8 Clay

9 Silt

10 Silt

11 Silt

12 Silt

13 Silt

14 Silt

15 Silt

16 Silt

17 Silt

18 Clay

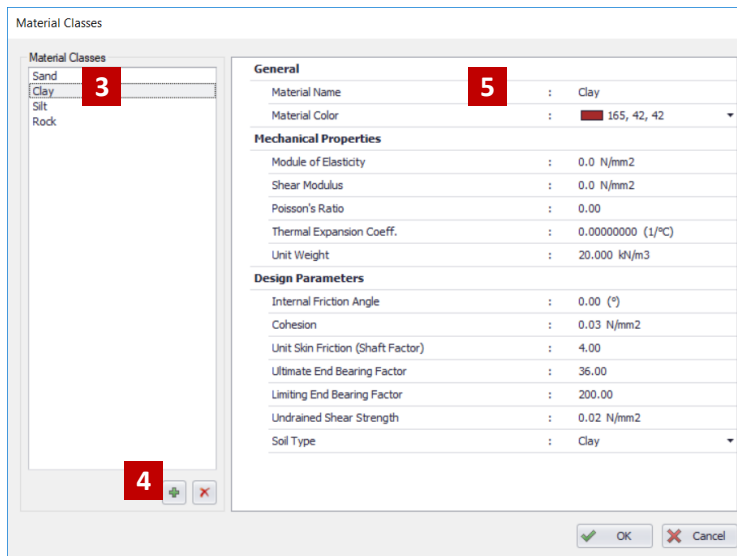
19 Clay

20 Clay



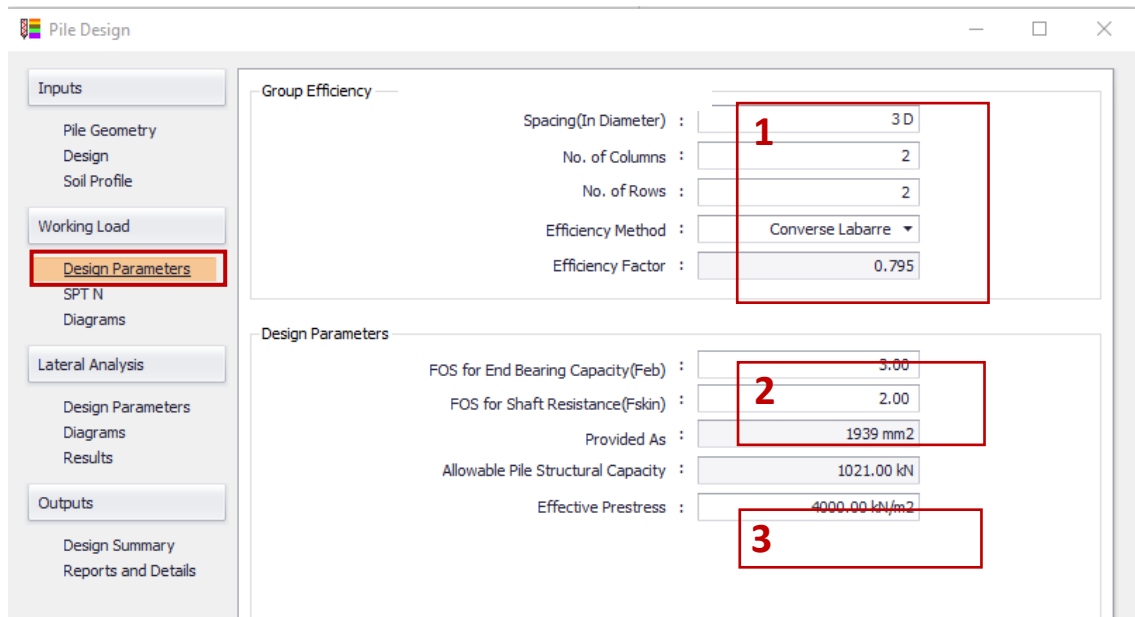
1. The soil profile can be edited using the **Add Row, Delete Row, Delete All and Import from CSV** buttons
  - **Add Row** → adds a new row above the selected row
  - **Delete Row** → deletes the selected row
  - **Delete All** → removes all rows
  - **Import from Excel / CSV** → import Soil Profile from an **Excel / CSV** file

Sample data import files for pile design & pile cap design macros are available under [\\ProtaLib\Samples](#) folder
2. Click the **label of soil materials** in the first column to define new soil materials or edit the soil materials. A **Soil Materials** dialog will be shown.



3. Select a different soil material from the **Soil Materials** dialog → Click **OK** to change the soil material of the selected row
4. Click “+” button to add a new soil material. Click “x” button to delete a selected soil material.
5. Materials name and color, mechanical properties, and design parameters of the soil materials can be edited.
6. The thickness of the soil layers can be edited in the **Depth Interval** column

➤ The **Design Parameters** tab allows you to enter the following parameters:



1. Input / confirm parameters of **Group Efficiency**. The purpose is to determine the Efficiency Factor to be applied to the pile working load. Set the **Efficiency Method** to **Do Not Apply Efficiency Method** if the efficiency factor is not required.
2. Input / confirm the Factor of safety for end bearing & shaft resistance
3. Input / confirm the **Effective Prestress** (For RC spun pile ONLY)  
The **provided As** (Area of Provided Main Rebars) and **Allowable Pile Structural Capacity** are automatically calculated from the pile properties and rebars in the **Pile Geometry** and **Design** tabs.

The **SPT N** tab allows you to enter the SPT N (Blow Counts of Standard Penetration Test) of each soil layers.

Soil Ma...	SPT N	Av... SPT N	Depth Int...	De...	Unit Skin Fric...	Ult... Skin Fric...	Cu... Skin Fric...	Fac... Skin Fric...	Ult... End Be...	Fac... End Be...	Wo...	Negat... Skin Friction
Clay	15	7.5	3000	3000	32	181	181	72	76	25	97	<input type="checkbox"/>
Clay	8	11.5	1500	4500	48	136	317	126	117	39	165	<input type="checkbox"/>
Silt	12	10	1500	6000	40	113	430	171	102	34	205	<input type="checkbox"/>
Silt	15	13.5	1500	7500	56	158	588	234	137	46	280	<input type="checkbox"/>
Silt	11	13	1500	9000	52	147	735	292	132	44	336	<input type="checkbox"/>
Silt	11	11	1500	10...	44	124	859	341	112	37	378	<input type="checkbox"/>
Silt	13	12	1500	12...	48	136	995	396	122	41	437	<input type="checkbox"/>
Silt	15	14	1500	13...	56	158	1153	458	143	48	506	<input type="checkbox"/>
Silt	22	18.5	1500	15...	72	204	1357	539	188	63	602	<input type="checkbox"/>
Silt	25	23.5	1500	16...	96	271	1628	647	239	80	727	<input type="checkbox"/>
Clay	40	32.5	1500	18...	128	362	1990	791	331	110	901	<input type="checkbox"/>
Clay	50	45	1500	19...	180	509	2499	993	458	153	1146	<input type="checkbox"/>
Clay	50	50	1500	21...	200	566	3065	1218	509	170	1388	<input type="checkbox"/>
Clay	50	50	1500	22...	200	566	3631	1443	509	170	1613	<input type="checkbox"/>

Calculate Working Load **3**

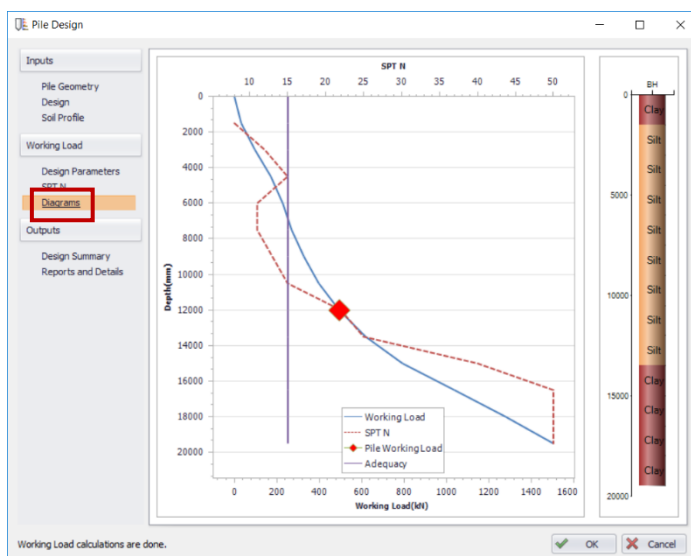
Calculate Working Load Working Load **0.00 kN**

Note: If Soil Type is Rock, please insert SPT N as Unconfined Rock Compressive Strength in kPa

Working Load Calculations must be done.

OK Cancel

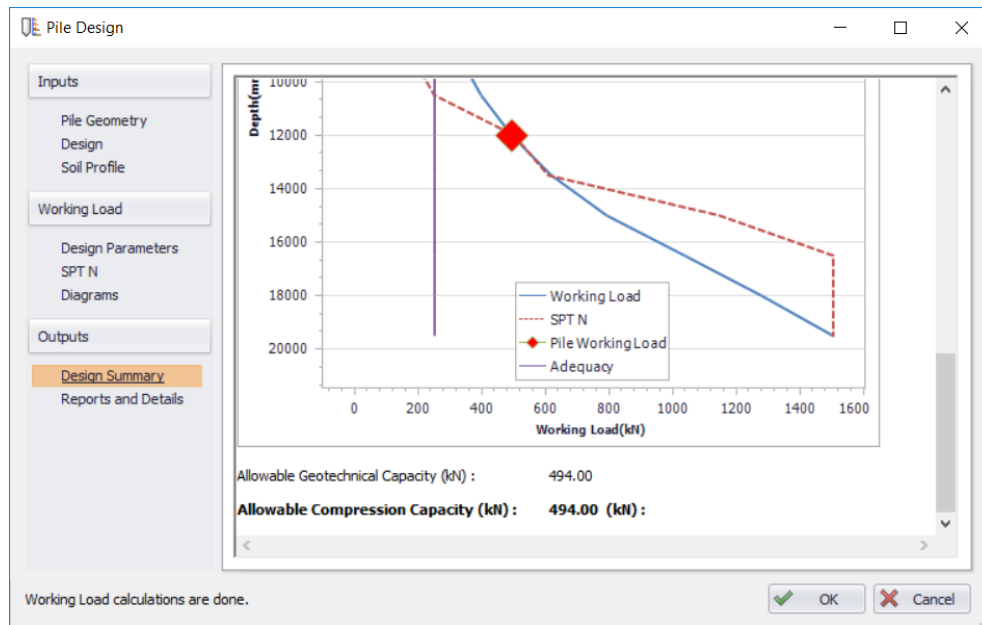
1. Input the SPT N for each soil layer
  2. The skin friction can be set to negative by checking the checkbox in the last column *Note: For Rock type, instead of entering SPT N, please enter **Unconfined Compressive Strength** (in kPa) in the SPT N column.*
  3. Click **Calculate Working Load** button to calculate the working load at the pile toe
- The **Diagrams** tab will be visible if **Calculate Working Load** button is clicked.



You can review the **SPT N vs Depth** and **Working Load vs Depth** plots in the diagram.

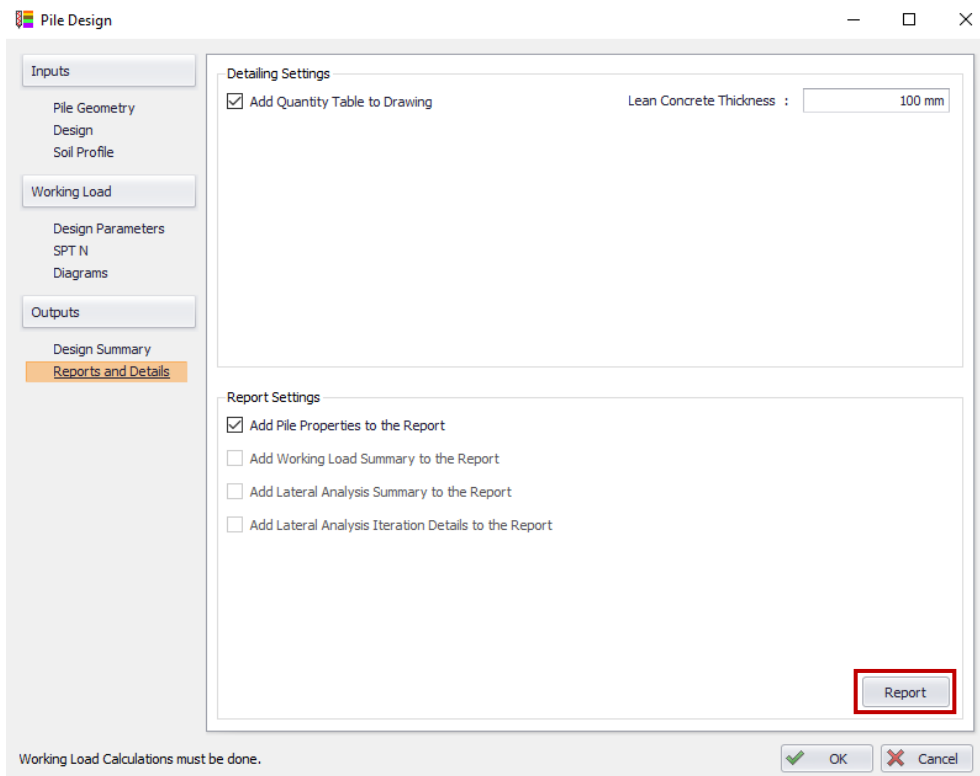
If the **Red Diamond** marker (i.e. Pile Working Load) is on the right-hand side of the **Adequacy** line, it indicates that the pile working load is sufficient.

The **Design Summary** tab shows the design summary for the **Working Load**.



The final **Allowable Compression Capacity** of the pile is the lesser of the structure & geotechnical capacity.

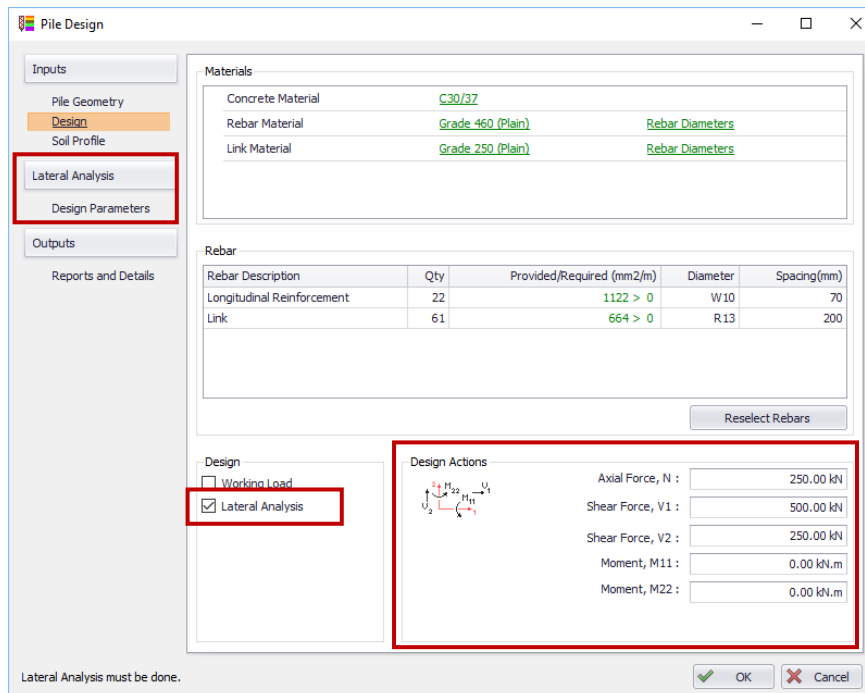
The **Results and Details** tab allows you to generate the design report.



Working Load Calculations must be done.

➤ Click **Report** button to create the report.

## Lateral Analysis (Design for pile combined axial and lateral shear or bending actions):



**Materials**

Concrete Material	C30/37	
Rebar Material	Grade 460 (Plain)	Rebar Diameters
Link Material	Grade 250 (Plain)	Rebar Diameters

**Rebar**

Rebar Description	Qty	Provided/Required (mm2/m)	Diameter	Spacing(mm)
Longitudinal Reinforcement	22	1122 > 0	W10	70
Link	61	664 > 0	R13	200

**Design**

☐ Working Load  
☒ Lateral Analysis

**Design Actions**

Axial Force, N : 250.00 kN  
 Shear Force, V1 : 500.00 kN  
 Shear Force, V2 : 250.00 kN  
 Moment, M11 : 0.00 kN.m  
 Moment, M22 : 0.00 kN.m

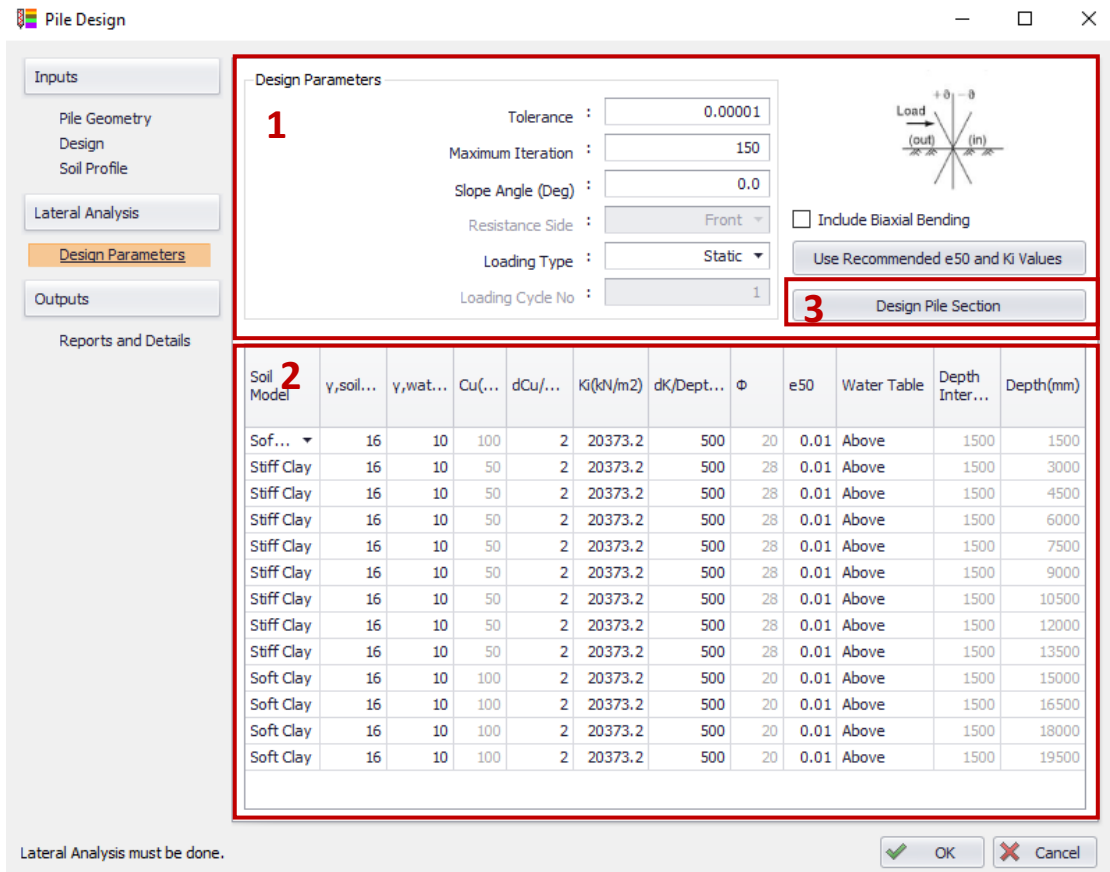
Lateral Analysis must be done.

➤ **Check the Lateral Analysis in the Pile Design > Design tab.**

The **Lateral Analysis** tabs will be visible on the left panel.

**Design Actions** group box will be visible on the bottom-right corner.

**Design Parameters** tab in the Lateral Analysis tab allows you to enter the following parameters:



**Design Parameters**

1

Tolerance : 0.00001  
 Maximum Iteration : 150  
 Slope Angle (Deg) : 0.0  
 Resistance Side : Front  
 Loading Type : Static  
 Loading Cycle No : 1

☐ Include Biaxial Bending  
 Use Recommended e50 and Ki Values

3 Design Pile Section

Soil Model	$\gamma_{soil}$	$\gamma_{wat}$	Cu	dCu	Ki(kN/m2)	dK/Dept	$\phi$	e50	Water Table	Depth Inter...	Depth(mm)
Sof...	16	10	100	2	20373.2	500	20	0.01	Above	1500	1500
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	3000
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	4500
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	6000
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	7500
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	9000
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	10500
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	12000
Stiff Clay	16	10	50	2	20373.2	500	28	0.01	Above	1500	13500
Soft Clay	16	10	100	2	20373.2	500	20	0.01	Above	1500	15000
Soft Clay	16	10	100	2	20373.2	500	20	0.01	Above	1500	16500
Soft Clay	16	10	100	2	20373.2	500	20	0.01	Above	1500	18000
Soft Clay	16	10	100	2	20373.2	500	20	0.01	Above	1500	19500

Lateral Analysis must be done.

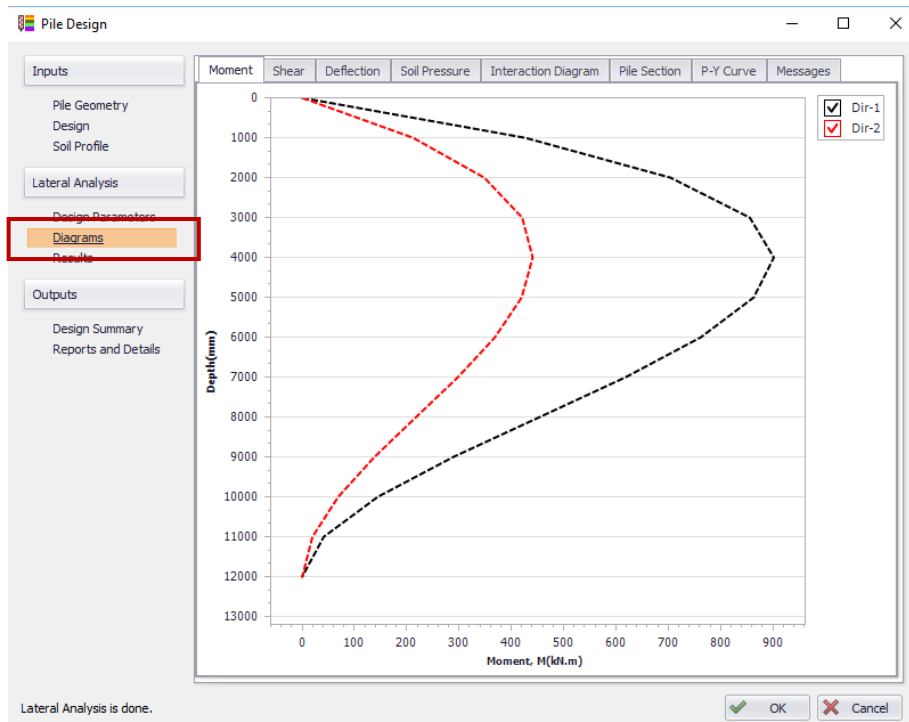
## 1. Design Parameters

- **Tolerance:** Allowable error of analyzed results
- **Maximum Iteration:** Analysis will stop at maximum iteration even though the error is greater than the tolerance
- **Slope Angle:** If the ground is sloping or the pile is raking, please enter the relative angle in degree.
  - Pile rotates anticlockwise with respect to ground surface, the slope angle shall be in positive sign
  - Pile rotates clockwise with respect to ground surface, the slope angle shall be in negative sign
  - Slope (Ground surface) rotates clockwise with respect to pile, the slope angle shall be in positive sign
  - Slope (Ground surface) rotates anticlockwise with respect to pile, the slope angle shall be in negative sign
- **Resistance Side (only applicable if slope angle is not zero):** If the soil is in passive on right-hand side of pile, set this to **Front**. If the soil is in passive on left-hand side of pile, set this to **Back**.
- **Loading Type:** If this is set to static, it indicates that the applied loading is static. If this is set to cyclic, the loading will be applied for number of cycles as defined in **Loading Cycle No.**
- **Loading Cycle No. (only applicable if Loading Type is set to Cyclic):** Number of cycles for the cyclic loading.

## 2. Soil Model

- **Soil Model:** You can choose various soil models for different type of soil namely, (i) Soft Clay; (ii) Stiff Clay; (iii) Sand; (iv) Void; (v) Linear Elastic (E Constant) or (vi) Linear Elastic ("E" Incremental)
- $\gamma'_{\text{soil}}$ : Effective soil unit weight;  $\gamma_{\text{water}}$ : Water Unit weight
- **Cu (Can be edited in Soil Materials in Soil Profile tab):** Soil Undrained shear strength
- **dCu/dZ:** Increment of Undrained Shear Strength against depth
- **Ki:** Initial soil stiffness; **dK/dZ:** Increment of soil stiffness against depth
- **$\Phi$  (Can be edited in Soil Materials in Soil Profile tab):** Soil Friction angle
- **e50:** Strain corresponding to one-half the maximum principal stress difference
- **Water Table: Above** – water table above this soil layer; **Below** - Water table below this soil layer
- **Depth Interval (Can be changed in Soil Profile tab):** Soil layer thickness
- **Depth (Auto calculated):** Bottom depth of this soil layer

3. Click **Design Pile Section** in **Design Parameters** tab to perform the lateral pile analysis. Then, the **Diagrams** and **Results** tabs will be visible. The design bending moment, shear, deflection and soil pressure diagrams can be reviewed in the **Diagrams** tab for both directions 1 and 2



➤ The tabulated analysis results can be reviewed in **Results** tab for both directions 1 and 2

Direction 1*		Direction 2*				
Element No	Depth(,...	Limiting Pressure(kN/m2)	Pressure(kN/m2)	Deflection(mm)	Shear(kN)	Moment(kN.m)
0	0	36	0.19	0	500	0
1	1000	10908	56.46	0	368.99	446.14
2	2000	21708	108.05	0	225.66	737.98
3	3000	32508	154.79	0	104.29	897.47
4	4000	43308	195.94	0	4.88	946.57
5	5000	54108	230.43	0	-72.58	907.23
6	6000	64908	256.73	0	-128.08	801.41
7	7000	75708	272.27	0	-161.62	651.07
8	8000	86508	272.2	0	-173.21	478.16
9	9000	97308	244.12	0	-162.84	304.64
10	10000	108108	81.38	0	-130.52	152.47
11	11000	118908	292.84	0	-76.24	43.61
12	12000	129708	404.37	0	-21.8	0

➤ Click **Design Summary** tab to review the design summary for the lateral analysis

### Lateral Pile Design Summary

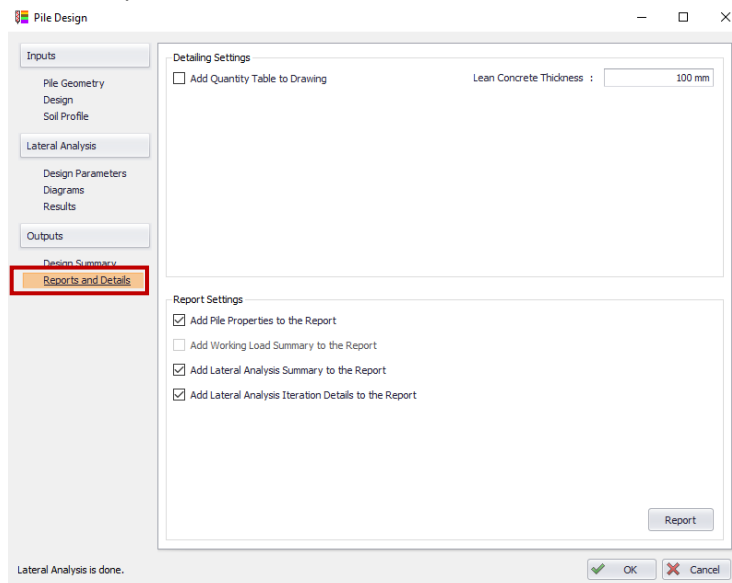
Design Shear, V1 (kN) : 500.00  
 Design Moment, M1 (kN.m) : 0.00  
 Design Shear, V2 (kN) : 250.00  
 Design Moment, M2 (kN.m) : 250.00  
 Design Axial, N (kN) : 250.00  
 Pile Flexural Rigidity, EI (Dir 1) (kN.m2) : 16540.48  
 Pile Flexural Rigidity, EI (Dir 2) (kN.m2) : 16540.48  
 Loading Cycle Number : 1  
 Loading Type : Static  
 Slope Angle (Deg) : 0  
 Resistance Side : Front  
 Tolerance : 1E-05  
 Max Iterations : 150

Soil Model	$\gamma_{sat}$	$\gamma_{sub}$	$C_u$	$dC_u$	$k_{log}/...$	$k_{log}/...$	$\phi$	$\alpha$	$\alpha_{50}$	Water Table	Depth Inter...	Depth...
Stiff Clay	35	10	30	2	20373.2	500	0	0.01	Above	1500	1500	
Stiff Clay	35	10	30	2	20373.2	500	0	0.01	Above	1500	3000	
Stiff Clay	35	10	30	2	20373.2	500	0	0.01	Above	1500	4500	
Stiff Clay	35	10	30	2	20373.2	500	0	0.01	Above	1500	6000	

Notes:  
 1= Soft Clay; 2= Stiff Clay; 3= Sand; 4= Void; 5= Linear Elastic (E Constant); 6= Linear Elastic (E Incremental)  
 $\gamma_{sat}$  (kN/m3) : Effective Soil Unit Weight  
 $\gamma_{sub}$  (kN/m3) : Water Unit Weight  
 $C_u$  (kN/m2) : Soil Undrained Shear Strength  
 $dC_u/dz$  (kN/m2/m) : Increment of Undrained Shear Strength against Depth  
 $k_{log}$  (kN/m2) : Initial Soil Stiffness  
 $\alpha/k_{log}$  (kN/m2/m) : Increment of Soil Stiffness against Depth  
 $\phi$  (deg) : Soil Friction Angle  
 $\alpha_{50}$  : Strain corresponding to one-half the maximum principal stress difference  
 Water Table : Above: Water Table Above this soil layer, Below: Water Table Below this soil layer



- Go to **Reports and Details** tab → Check **Lateral Analysis** → Click **Report** button to generate the report.



## 42. Miscellaneous Enhancement & Fixes in ProtaDetails

### Beam & Strip Footing Details

- Known issues on "Beam Elevation Front and Back Intermediate Elements" are fixed & enhanced by new generation algorithm.
- Enhanced presentation of link zone leaders is introduced.
- Strip Footing Tapers were not shown on the section ever. Fixed
- Missing bar extension dimensions in some cases. Fixed

### Pilecaps

- Several design and detailing enhancements on pile caps.
- Rebar weights were calculated less due to smaller hook length. Fixed
- Option to show original pile locations on pilecap detail plan view.

### Retaining Walls

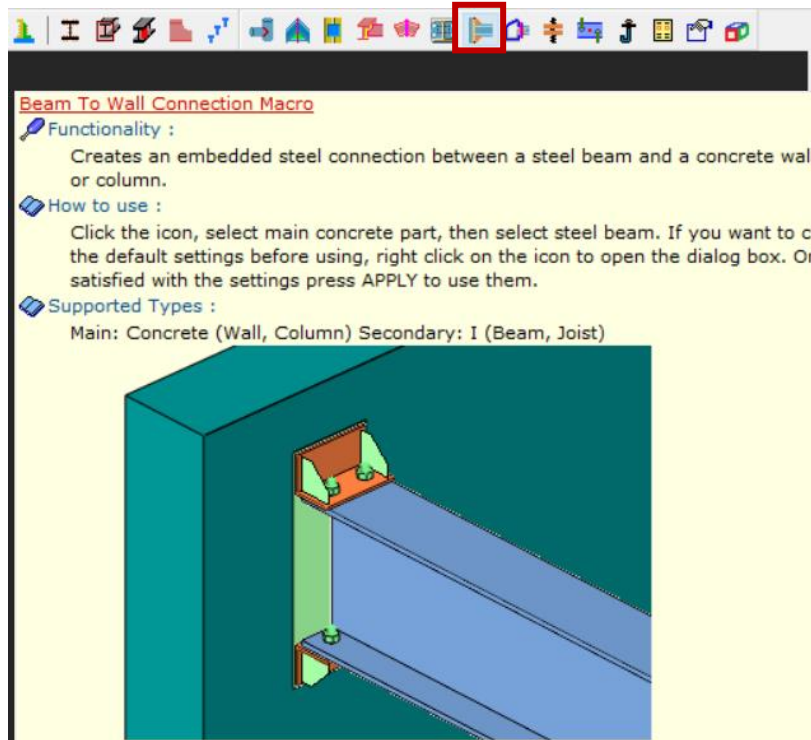
- Retaining wall rebars resets when opened in edit mode. Fixed
- There is no need to calculate total failure if both front and back fill is zero.
- Numerous detailing issues fixed.

**NEW Order by date or project name options in Project Open dialog** : allow you to easily find the project you want to open in both ProtaStructure & ProtaDetails.

**Sample data import files for pile and pile cap design macros** : added under **\\ProtaLib\Samples** folder upon installation. Relevant **Import** button opens up this folder.

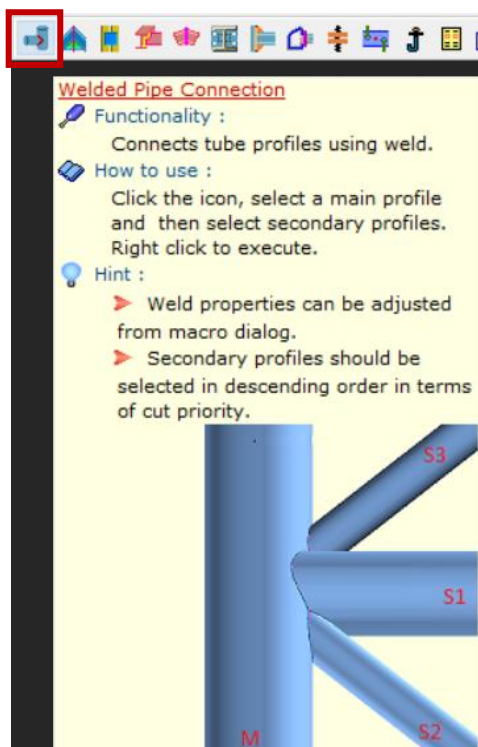
## 43. Beam to (RC) Wall/Column Connection (sp4)

This new connection macro creates an embedded steel connection between a steel beam and a concrete wall or column. This macro is available from the Secondary Connections toolbar (as shown below).



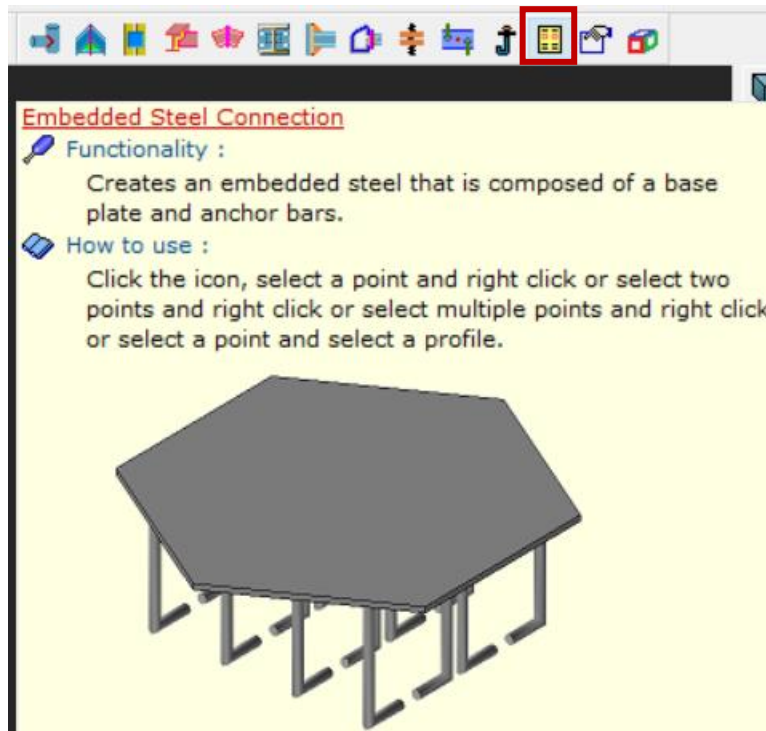
## 44. Welded Steel Pipe (Circular Hollow Sections) Connection (sp4)

This new connection macro creates a welded steel tubes connection using welds. This macro is available from the Secondary Connections toolbar (as shown below).



## 45. Embedded Steel Macro (sp4)

This new connection macro creates a generic embedded steel with anchor bars, which can then be used to connect other steel members to reinforced concrete members. This macro is available from the Secondary Connections toolbar (as shown below).



----- End of document -----