What's New?

For support & training please contact
Support : asiasupport@protasoftware.com
Training : asiasales@protasoftware.com

www.protasoftware.com

What's New in ProtaStructure 2019 – Quick Summary

ProtaStructure 2019 is a fantastic new release containing many new features and enhancements for you to enjoy. Key highlights include:

- **New Member Design Diagrams**

  The member design diagram is completely re-developed using the latest technology. The result is a very user-friendly interface with powerful interrogation functions to review forces including new member deflections plots. Diagrams shown are customizable and very accurate providing superior stationing for design, numerical integration and enveloping. New diagrams interface offers superior visualization with zoom, pan and stretch tools.

- **New 'Material and Section Effective Stiffness Factors'**

  The Section Stiffness Factors input table is enhanced to cover different section modifiers for different members in different degrees of freedom. This provides user with more refined control over the effective stiffness (cracked), including new settings for basement walls & coupling beams and separate in-plane and out of plane stiffness settings for shear walls and slabs.

- **Ability to Use Cracked Sections for Load Cases**

  Each load case can now be configured to use either cracked or uncracked sections for a given combination. This feature is necessary for advanced seismic analysis requirements (for most leading earthquake codes) as well as wind load or temperature loading which requires the assumption of cracked section properties. Two separate stiffness matrices are formulated for the analysis, run concurrently and enveloped for design.

- **New Member Type: Coupling Beam**

  A new type of beam is introduced: 'Wall Coupling Beam'. Coupling beams are used to link shear walls for increased stiffness to resist large lateral loads, e.g. seismic and wind loads. Checks are performed to validate this system to seismic codes of practice.

- **New Member Type: Basement Wall (Meshed)**

  A new type of wall is introduced: 'Basement Wall'. This enables special consideration for basement walls in seismic analysis & design, where the participation of the basements mass in the dynamic analysis is varied. These results are run concurrently and combined in accordance with the relevant seismic code of practice. This is a unique feature which reduces the number of analytical models which need to be considered for buildings with basements to just one model running all analytical variations in one analysis run.

- **Purlins and Girts with Sag rods**

  Sag rods can now be inserted in ProtaStructure. They are used to reduce bending moment in purlins and girts as well as acting as lateral supports against local buckling. Sag rods are always assumed to be pinned frames in building analysis.

- **Section Manager Interface & Enhanced Steel Database**

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

- **New Built-Up Steel Type: Cross, T & L Shapes**

  The 'Built Up Steel' type have been expanded to include cross (+), tee (T) & angle (L) shapes. With 'Built Up Steel', you can model non-standard steel sections not available in any of the country steel database. These built up steel column and beam can be exported to ProtaSteel as well.

- **New International & Country Specific Code of Practices**

  ProtaStructure now supports American codes of practice for steel and concreate design together with associated loadings codes. This includes comprehensive support for Seismic loading, analysis, design and detailing including sophisticated checks for complex topics like building irregularities, strong column and weak beam design philosophy, and beam column joint shear checks. Regional variations covering detailed loading and design rules for Indonesia and the Philippines are also introduced.

- **Steel Design Enhancement**

  Steel design is greatly enhanced and includes new design interfaces for purlin, girt & braces and new design diagrams which automatically detect restraint conditions and buckling lengths for more economical design. Steel members are checked for pass / failure status and the most critical Utilization Ratio (UR) reported. You can change the section sizes in the interactive design & new sections will be automatically checked.

- **Enhanced Visual Interrogation**

  Visual Interrogation has been further enhanced to include the Steel Design Status and Steel Utilization Ratios for quick review of design efficiency.

- **Transit columns/walls are displayed on all relevant plan views**

  Transit columns or shearwalls (Len Storey= 2 or more) are now displayed on all plan views of the storeys they are passing through. They are displayed in a lighter ghost color in the plan view of the passing storey. This greatly ease modeling speed and accuracy; it is a highly requested feature especially for steel modelling.

- **Automatic Wind Load Calculator**

  The new wind load calculator automatically determines wind forces in accordance with the chosen wind code and applies it 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).

- **New 64-Bit Advanced Solver**

  A new advanced 64-bit solver has been introduced which more efficiently solves both small and extremely large models. This also includes a new post analysis checks summary for quick validation of analysis results against code-based checks including building displacement, seismic and loading validation.

- **Enhanced Revit Integration**

  ProtaStructure supports bi-directional links with latest version Revit Structure 2019; greatly enhancing project coordination and workflow. Model changes can be synchronized and tracked both in Revit and ProtaStructure. Round-tripping between Revit to Protostructure is greatly enhanced. The Integration Status will color code which members are new, changed, unchanged or deleted. A detailed list explaining these changes is also provided.

- **IFC Export**

  ProtaStructure models can now be exported to IFC format using 'IFC2x3, Coordination View Version 2.0'. IFC File format is used by many other leading Building Information Modelling (BIM) platforms including Autodesk Revit, ArchiCAD, Bentley and Tekla Structures. This enhances Prota's standing as a leading collaborator and provider of advanced BIM technology.

What's New in ProtaDetails 2019

ProtaDetails 2019 contains many completely new features and significant enhancements:

- **New RC slab cross sections with reinforcement detailing**

  Reinforced slab cross section can be created with reinforcement detailing. In ProtaDetails, cut sections anywhere along the layout plan drawing and position it anywhere within the drawing sheet.

- **New Global Options Interface**

  Drawing and detailing options are now classified under a single interface; allowing easy global control on how the drawings are produced.

- **New Rebar Label Settings**

  Rebar label settings has been enhanced to allow for more user-defined options for arrow leaders (pointing to rebars) and dimensions arrows. This offers greater flexibility for users to produce high quality drawings to their company standards.

- **Enhanced Column & Beam Detailing Settings**

  Numerous new column and beam detailing options are introduced to enhance the quality of drawings. For column elevations drawings, longitudinal bars can be lapped in the mid-span of the column (requirement of some code of practice) & bending of bars (BOB) can be controlled. For beam elevation drawings, cantilever top rebar extension lengths can be set based on the cantilever length multiplied by a user-defined factor.

- **Automatic wall opening detailing**

  The reinforcement details are automatically created around the shear wall openings in the wall elevation details. The shear wall openings are displayed in both the sections as well as schedules.

- **Precision detailing for slanting RC columns, varying column section sizes & slanting RC beams**

  Column elevation detailing is greatly enhanced to include slanting (angled) columns and columns with varying section sizes. Longitudinal bar detailing including intelligent curtailment and bar cranking follows Eurocode and ACI recommendations for columns with changing dimensions. Any beams connected to the columns will be drawn in the elevation as well.

- **New Engineering Utilities for quick engineering design & calculation**

  A new handy **Engineering Utilities** is provided for quick engineering design. This includes a calculator for concrete cover, rebar anchorage & lap lengths; simple beam analysis tool to calculate maximum shear, moment and deflections; hydraulics calculator for uniform flow depth & gradually varied flow, etc.

- **Enhanced Retaining Wall Module**

  The retaining wall module is greatly enhanced and now supports the American ACI318 & ASCE7 code. In addition, Eurocode is expanded to cover EC2, EC7 and EC8 with all three design

approaches. There are numerous other improvements including a new design & detailing interface where rebars can be edited interactively.

- **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

- **Multiple selection of Tables for report generation**

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

What's New in ProtaSteel 2019

ProtaSteel 2019 contains major new features and significant enhancements:

- **Intelliconnect for fully automated steel connection design**

  IntelliConnect automatically assesses joints to your preferences, batch creates and designs steel connections. It covers a wide range of simple, moment, splice and weld connections. It considers constructability by intelligently arranging and configuring all the components of the connections without user intervention.

- **Connection Design Reports**

  ProtaSteel 2019 now creates detailed design check reports for the following connections in accordance to EC3, AISC (LRFD), AISC (ASD), and BS5950 for all beam to beam & beam to column connections and simple base plate connections. When a connection is inserted, first a guideline design is performed. Then, the rigorous design checks are done according to selected code clauses. The reports can be created on demand in rich text format (RTF) and can be added to ProtaStructure's report manager for compilation into final design documentation. Design Status colors can be seen on model view before report creation.

- **Connection Design Summary Table**

  A connection design summary table presents an overview of selected connection macros. This includes "Frame Id", profile types of macro dependents, connection type, capacity ratio of connection, design status according to capacity ratio and whether design report of connection is created.

- **Design Overrides**

  This feature allows you to impose your own company standards and preferences if these are higher than the auto-design results. For example, you can specify the minimum number of bolts and thickness of the plates for a given connection type.

- **Auto Presets**

  This feature enables the user to create relations between profile types and connection macro presets. This allows the program to automatically apply a saved "preset" connection to a particular profile section.

- **User-defined Design Capacity Ratios**

User can now input predefined minimum shear and axial capacity ratio's for connection design (design force / by maximum member capacity). These ratios will be used for design if actual shear or axial forces imported from ProtaStructure analysis are smaller. Specifying capacity ratios prevents unrealistic connection design, e.g. 1 no. of bolt when actual design forces are exceeding small.

- **New Truss Apex Gusset Connection**

  This connection macro creates & designs an apex truss connection with gusset plate & supports all profile types. This macro is available from the Secondary Connections toolbar.

- **New Truss End Plate With Gusset Plate**

  This macro creates and designs a connection between truss ends and steel columns using end and gusset plates. This macro is available from the Secondary Connections toolbar.

- **New Truss Apex Connection Macro**

  This macro creates a welded Apex connection without design using "I" profiles.

- **New Simple Base Plate Connection Design**

  This connection macro creates a simple column base plate connection.

- **New Beam to (RC) Wall/Column Connection**

  This connection macro creates and designs an embedded steel connection between a steel beam and a concrete wall or column. This macro is available from the Secondary Connections toolbar.

- **New Welded Steel Pipe (Circular Hollow Sections) Connection**

  This macro creates steel tubes connections using welds. This macro is available from the Secondary Connections toolbar.

- **New Embedded Steel Macro**

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

- **New Guardrail Macro**

  This connection macro creates hand rails on I, U & C beam profiles.

- **IFC File Export**

  ProtaSteel models can now be exported to IFC format. This facilitates collaboration with many other leading Building Information Modelling (BIM) platforms including Autodesk Revit, ArchiCAD, Bentley and Tekla Structure and reinforces Prota's standing as a leading provider of advanced BIM technology.

Miscellaneous Enhancement & Fixes

There are many other enhancements & fixes included in ProtaStructure Suite 2019. Click on the links below for more details

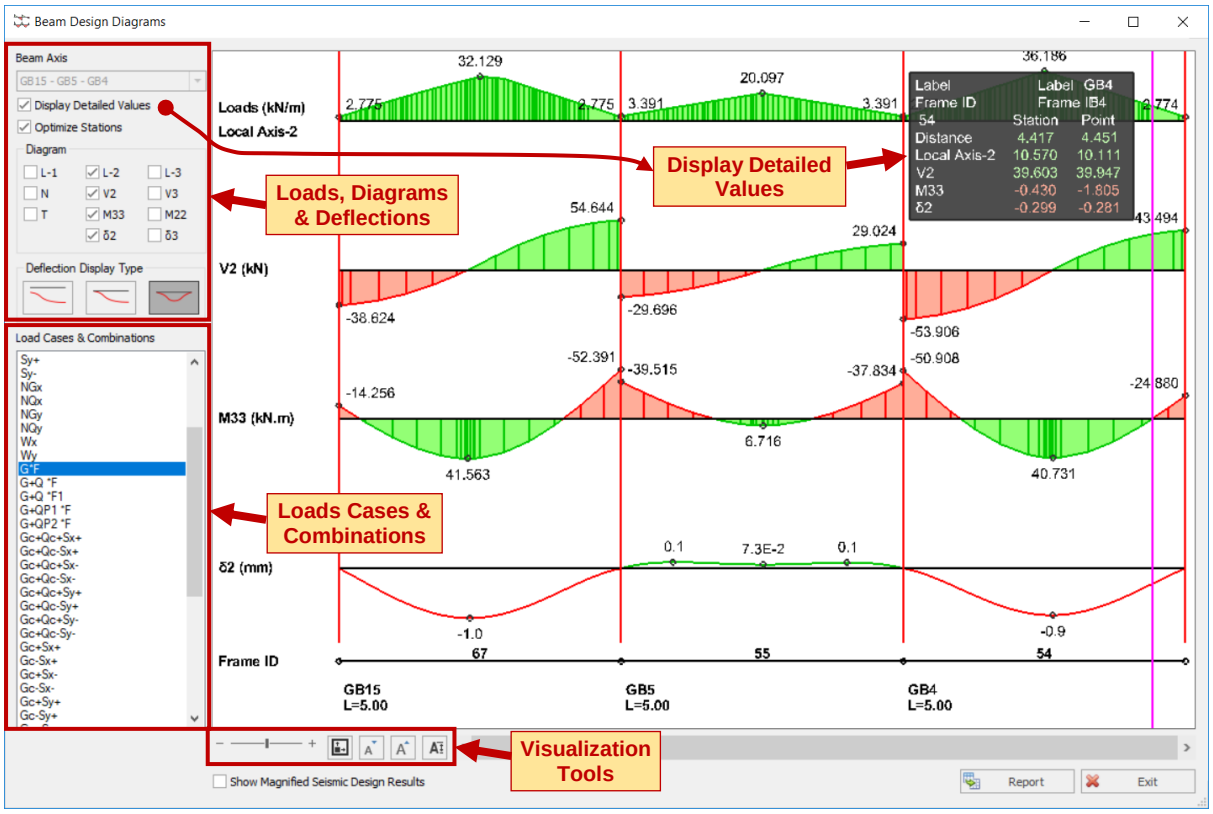
- ProtaStruture enhancement & fixes
- ProtaDetails enhancement & fixes
- ProtaSteel enhancement & fixes
- ProtaBIM enhancement & fixes

1. New Member Design Diagrams

The member design diagram is completely re-developed using the latest technology. The result is a very user-friendly interface with powerful interrogation functions to review forces including new member deflections plots. Diagrams shown are customizable and very accurate providing superior stationing for design, numerical integration and enveloping. New diagrams interface offers superior visualization with zoom, pan and stretch tools.

How to Use

- *Select a beam on plan view → Right-click →* ***Beam Analysis Results Diagram***
- *Alternatively, access the* ***Interactive Beam Design*** *→ Pick '****Diagrams****'*



Loads, Diagrams & Deflections

- **Display Detailed Values** :
  - Checked : A tracing window will appear showing the exact values of the diagrams, eg. shear & moment, when the mouse cursor is placed at a particular location along the member.
- **Optimize Stations** :
  - Unchecked : The diagrams are displayed using default maximum number of stations.
  - Checked : The number of stations will be reduced & optimized to maintain similar accuracy.
- **Diagrams** :
  - L1/ L2 / L3 : Check to show external slab loads decomposed & user-defined loads on beams
  - N = Axial force ; T = Torsion
  - V2 = major shear ; M33 = major moment ; δ2 = major deflection
  - V3 = minor shear; M22 = minor moment ; δ3 = minor deflection

➢ **Deflection Display Types** :

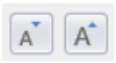
Deflections are calculated along the member span for each load case and combination. Deflections can be displayed in three different ways:

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

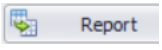
Visualization Tools & Report

➢ **Horizontal Scale**  → Increase or decrease horizontal scale of diagrams

➢ **Default Display Scale**  → click to reset to default scale

➢ **Increase / decrease font size** 

➢ **Default Font Size**  → Click to rest to default font size

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

The same new diagrams are used for columns & walls :

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

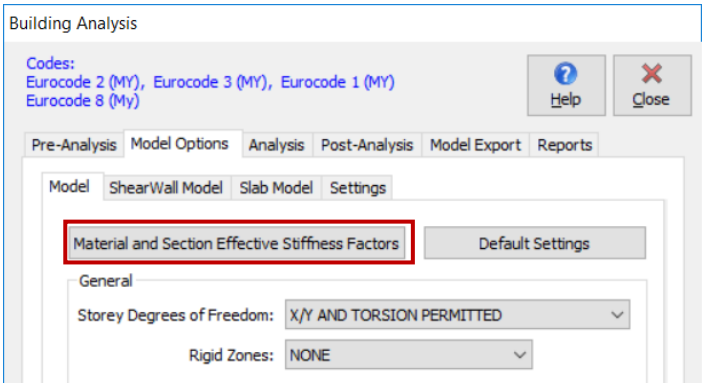
Note : The analysis post processor (Analysis Model and Results Display) is enhanced to utilize the new diagram engine. As such, the occasional "RHS Moment Mismatch" warning & failure during beam design is completely eliminated.

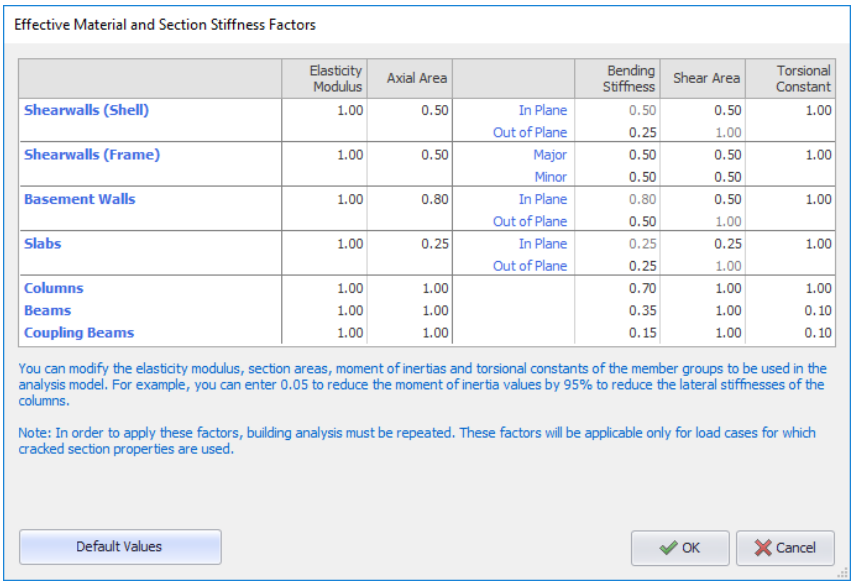
2. New 'Material and Section Effective Stiffness Factors'

The **Section Stiffness Factors** input table is enhanced to cover different section modifiers for different members in different degrees of freedom. This provides user with more refined control over the effective stiffness (cracked), including new settings for basement walls & coupling beams and separate in-plane and out of plane stiffness settings for shear walls and slabs.

How to Use

- ➢ *The* ***Material & Section Effective Stiffness Factors*** *can be accessed via* ***Building Analysis → Model Options → Model*** *tab.*





The table provides refined control over the effective stiffness (cracked) properties :

- **Shearwalls** stiffness can be adjusted differently for FE Shell & Frame (Mid-Pier) assumption.
- **In-Plane** & **Out of Plane** stiffness can be set separately for shell assumption of shearwalls, basement walls and slabs
- **Basement Walls** & **Coupling Beams** stiffness is added due to respective new member type. These members type is applicable to code-specific seismic analysis, e.g. Turkish code.

Important Notes *:*

- The stiffnesses table will only be applied for load cases with '**Used Cracked Sections'** checked in the respective **Load Case Editor**, except for seismic load cases where cracked sections are always assumed.
- Each seismic code has its own set of default values. If an entered value is different than the default, then editor color will turn to **orange** to inform the user.
- Building analysis must be repeated each time the effective stiffness factors are changed.

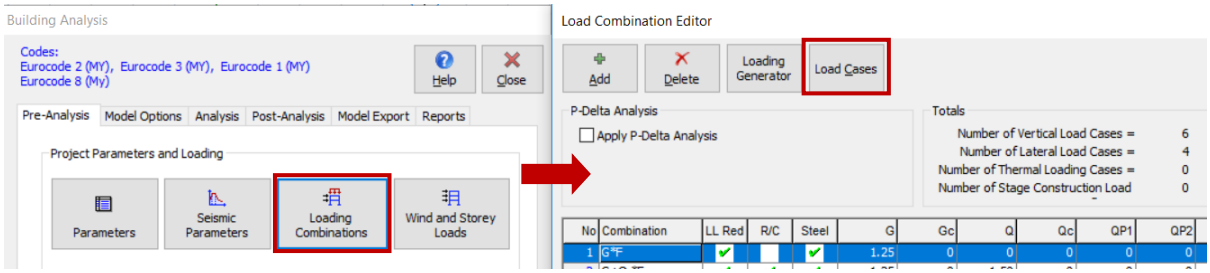
3. Ability to Use Cracked Sections for Load Cases

Each load case can now be configured to use either cracked or uncracked sections for a given combination. This feature is necessary for advanced seismic analysis requirements (for most leading earthquake codes) as well as wind load or temperature loading which requires the assumption of cracked section properties. Two separate stiffness matrices are formulated for the analysis, run concurrently and enveloped for design.

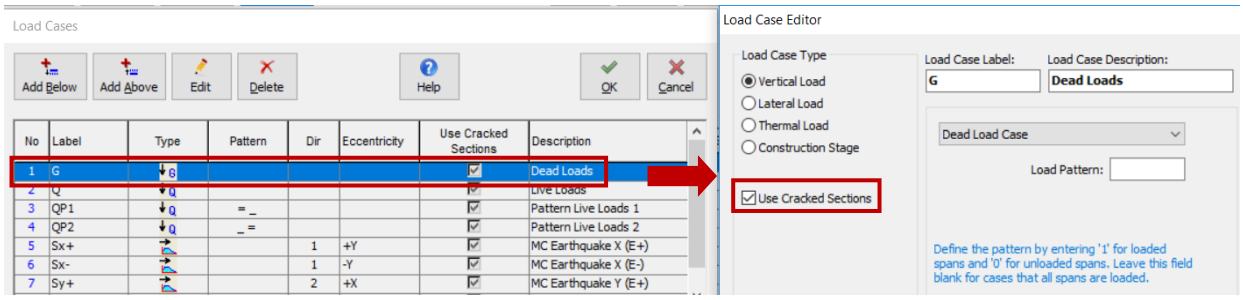
How to Use

For existing project with load cases and combinations already generated :

- *Go to **Building Analysis** menu → **Load Combinations** → **Load Cases***



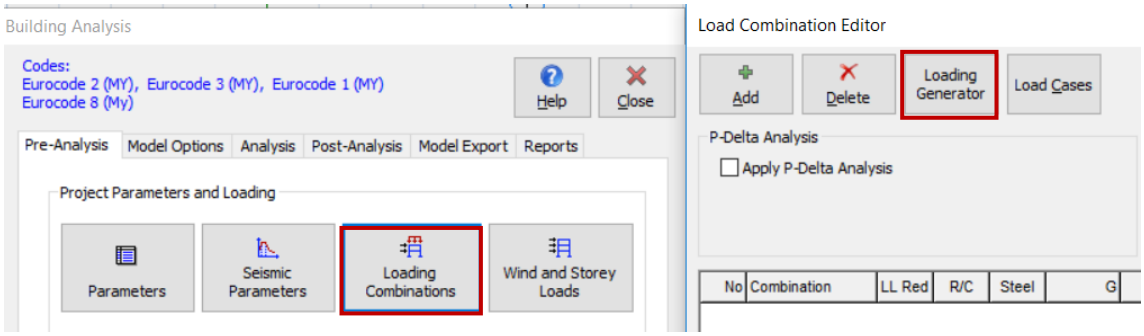
- *Double-click on a **load case** to access **Load Case Editor***

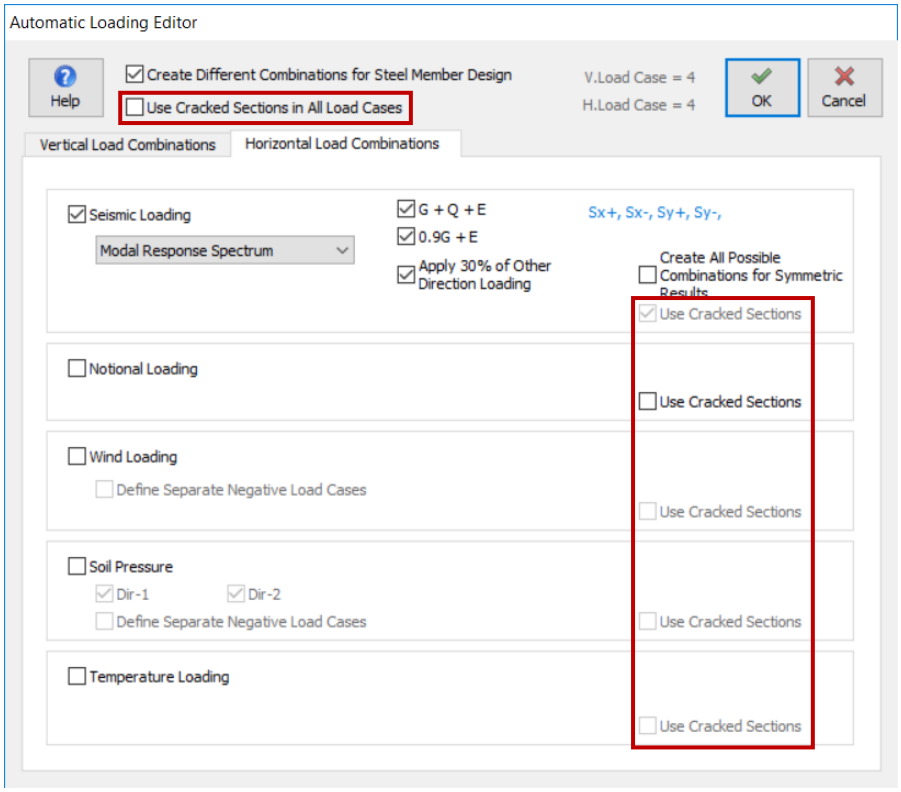


- *Check / uncheck **Use Cracked Sections***

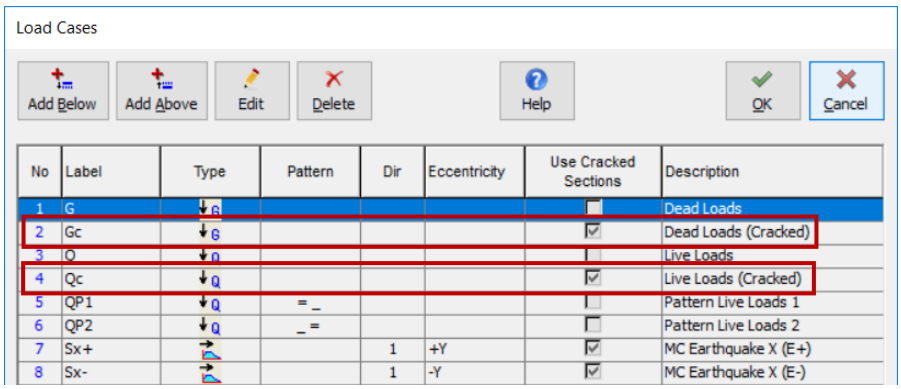
For new project, it is highly recommended to use the automatic **Loading Generator** to generate the load cases & combinations :

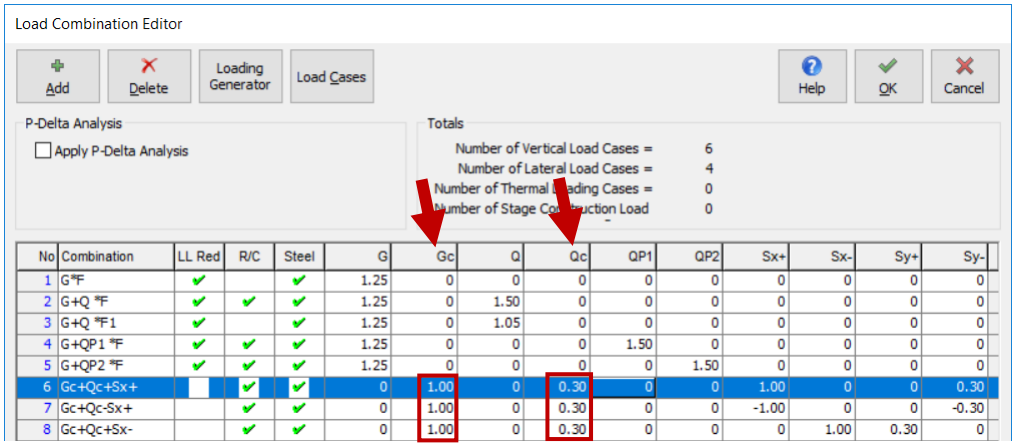
- *Go to **Building Analysis** menu → **Load Combinations** → **Loading Generator***





- For load cases with cracked sections assigned, stiffness adjustments in **Effective Material & Sections Stiffness Factors** table will be applied; else 100% default stiffness (uncrack) is assumed.
- '**Use Cracked Sections**' property is always checked for seismic loading because it is always necessary to use effective stiffness factors for seismic design.
- Notional Loading, Wind Loading, Soil Pressure and Temperature loading can be selectively configured to use cracked sections.
- If '**Use Cracked Sections in All Load Cases**' option is checked then all load horizontal load cases will use cracked section properties. If unchecked, then additional **Gc** and **Qc** load cases will be created and used in seismic design combinations.





No	Combination	LL Red	R/C	Steel	G	Gc	Q	Qc	QP1	QP2	Sx+	Sx-	Sy+	Sy-
1	G*F	✔		✔	1.25	0	0	0	0	0	0	0	0	0
2	G+Q *F	✔	✔	✔	1.25	0	1.50	0	0	0	0	0	0	0
3	G+Q *F1	✔		✔	1.25	0	1.05	0	0	0	0	0	0	0
4	G+QP1 *F	✔	✔	✔	1.25	0	0	0	1.50	0	0	0	0	0
5	G+QP2 *F	✔	✔	✔	1.25	0	0	0	0	1.50	0	0	0	0
6	Gc+Qc+Sx+		✔	✔	0	1.00	0	0.30	0	0	1.00	0	0	0.30
7	Gc+Qc-Sx+		✔	✔	0	1.00	0	0.30	0	0	-1.00	0	0	-0.30
8	Gc+Qc+Sx-		✔	✔	0	1.00	0	0.30	0	0	0	1.00	0.30	0

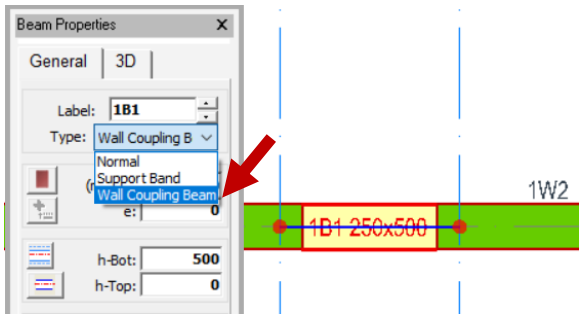
Cracked section properties table is included in the '**Post Analysis Design Checks**' report.

4. New Member Type: Coupling Beam

A new type of beam is introduced: '**Wall Coupling Beam'**. Coupling beams are used to link shear walls for increased stiffness to resist large lateral loads, e.g. seismic and wind loads. Checks are performed to validate this system to seismic codes of practice. This consequently decrease overturning effect and improve overall stiffness of a building system.

How to Use

- *Create a RC beam between two shear walls*
- *In the **Beam Properties** → **Type** dropdown → pick **Wall Coupling Beam***



This new beam type enables **Prota**Structure to:

- Use different cracked section properties as dictated by different seismic codes
- Assess and check the connected wall panels to see if they act as a coupled wall or not.
- Utilize different design and detailing approach toward these beams
- Details are shown in the '**Post Analysis Checks Report' → Wall / Frame Effects Checks**

WALL/FRAME EFFECT CHECKS:

(TBEC 2018 - Cl. 4.3.4.5 and 4.3.4.6)

EARTHQUAKE DIRECTION: 1 (Angle From X 0.00 Degrees)

(Sub levels surrounded by walls are ignored.)

Warning:
Coupling coefficent (Ω) of some coupled walls exceed 2/3 limit. (Cl 4.5.4.5)
It is recommended to edit the geometry in order to lower the transferred shear to the coupled walls.

Member	Angle	Dx+ (ES Deprem X (E+))			Dx- (ES Deprem X (E-))		
		M_{33} (t)	M_{22} (t)	M_{proj} (t)	M_{33} (t)	M_{22} (t)	$M_{projection}$ (t)
S1	0.00	3.79	0.41	3.79	2.18	-0.37	2.18
S2	0.00	4.41	-0.03	4.41	2.56	-0.24	2.56
S11	0.00	1.99	-0.49	1.99	1.99	0.49	1.99
S12	0.00	2.18	-0.37	2.18	3.79	0.41	3.79
P1 - P2	0.00	1161.18	-0.43	1161.18	1161.18	0.43	(P) 1161.18
Total Overturning Moment (Mo):				1191.041			1191.041
Wall Overturning Moment (MDev):				1161.183			1161.183

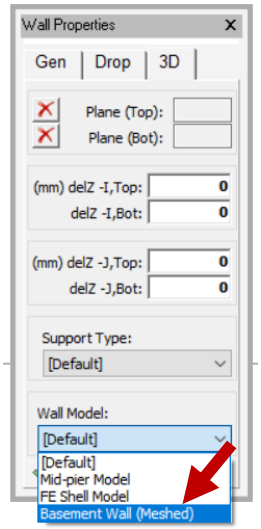
Member	Load Case	M_{Dev} (t.m)	cN_v (t)	Bağ Katsayısı $\Omega = M_{Dev} / cN_v$	
P1 - P2	Dx+	1161.18	938.093	1.238	> 2/3 X
	Dx-	1161.18	938.093	1.238	> 2/3 X

$\alpha_M = M_{Dev} / M_b =$ (E+) = 0.97 > 0.75 / (E-) = 0.97 > 0.75

Warning: Dir 1... Load Bearing System cannot be considered as High Ductile Wall + Normal Frames
Structural System Type A12 or A13 should selected and analysis should be repeated.

5. New Member Type: Basement Wall (Meshed)

A new type of wall is introduced: 'Basement Wall'. This enables special consideration for basement walls in seismic analysis & design, where the participation of the basements mass in the dynamic analysis is varied. These results are run concurrently and combined in accordance with the relevant seismic code of practice. This is a unique feature which reduces the number of analytical models which need to be considered for buildings with basements to just one model running all analytical variations in one analysis run.



How to Use

- *In the Wall Properties, go to **3D** tab*
- *Wall Model drop down → Pick '**Basement Wall (Meshed)**'*

This new wall type enables **Prota**Structure to:

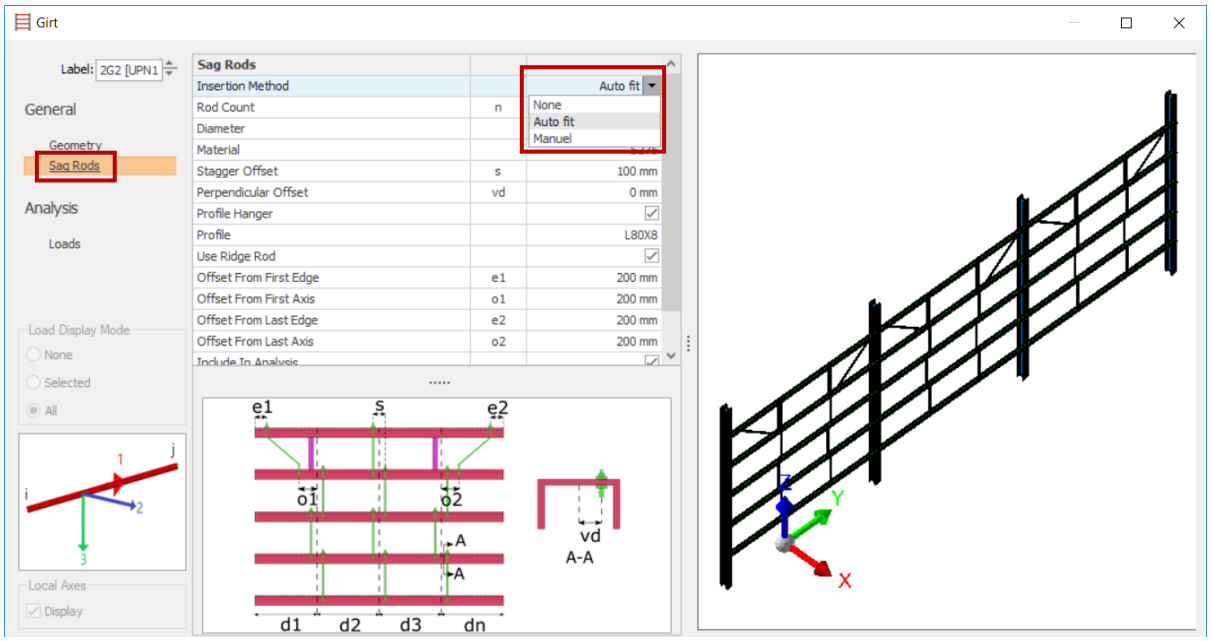
- Use different cracked section properties as dictated by different seismic codes
- Always use FE Shell elements for these members in the building analysis.
- Utilize different design and detailing approach toward these walls

6. Purlins and Girts with Sag rods

Sag rods can now be inserted in ProtaStructure. They are used to reduce bending moment in purlins and girts as well as acting as lateral supports against local buckling. Sag rods are always assumed to be pinned frames in building analysis.

How to Use

- *In Girt Properties → **Geometry** → Set up the girts*



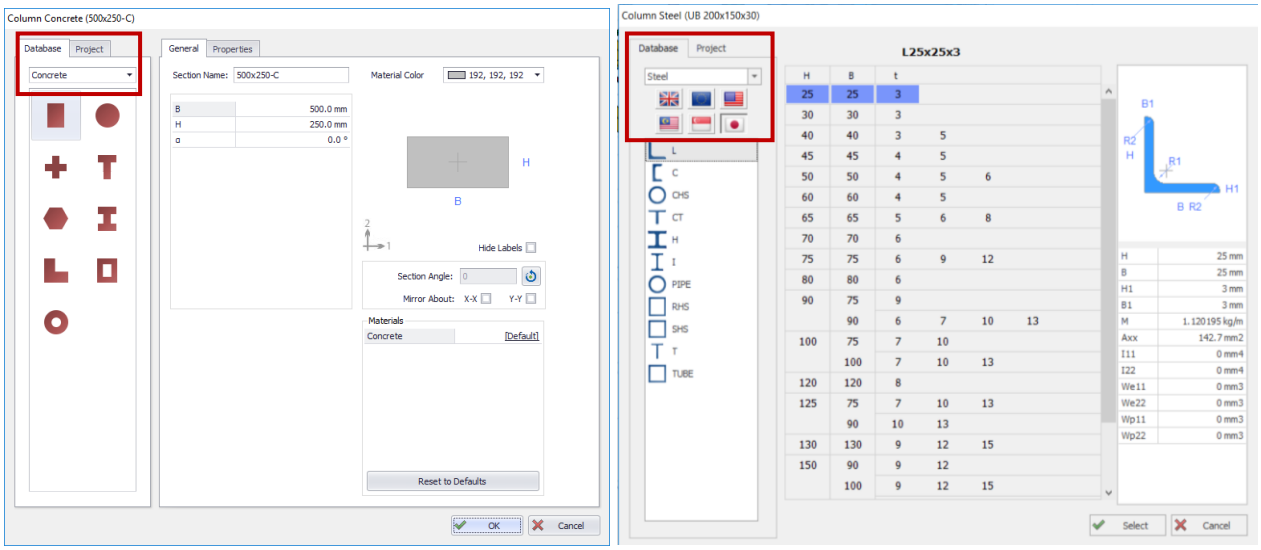
- *Pick '**Sab Rods**' → Click **Insertion Method** drop down & pick the options :*
  - ***Auto fit** : Enter the number of sagrods and the spacing will be divided equally.*
  - ***Manual** : Enter irregular spacings separated by comma.*
- *Enter all other parameters with reference to the help diagram at the bottom.*

TIPS :

- The right-hand diagram refreshes automatically to show the final setup. Scroll the mouse wheel to zoom in & out. Right-click and drag to move.
- Although, physical members of the sag rods are placed in an alternated pattern, the analytical model perfectly connects to column/beam lines and insertion points.

7. Section Manager Interface & Enhanced Steel Database

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



How To Use:

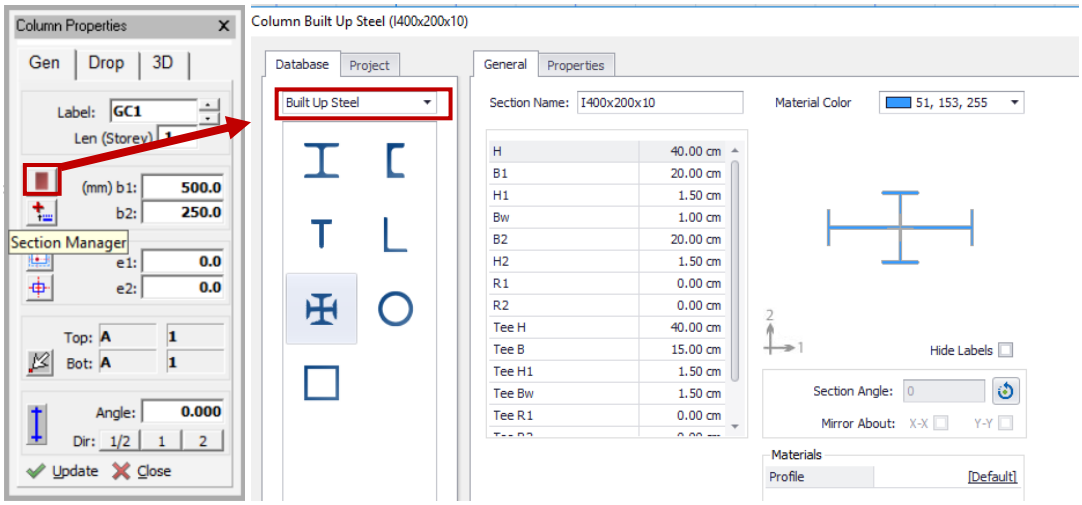
- *Section Manager is accessible from various places of member creation :*
  - *Beam & Column properties → **Section Manager** icon* 
  - *Select a column or beam → Right-click → **Edit Section / Material***
  - ***Steel Truss** / **Purlin** / **Girt** / **Brace** creation dialog*
- *In the Database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

8. New Built-Up Steel Type: Cross, T & L Shapes

The '**Built Up Steel**' type have been expanded to include cross (+), tee (T) & angle (L) shapes. With 'Built Up Steel', you can model non-standard steel sections not available in any of the country steel database. These built up steel column and beam can be exported to ProtaSteel as well.

How to Use

- *In Steel Column or Steel Beam Properties → Pick '**Section Manager**'*
- *Pick '**Built Up Steel**' under Database → Pick a shape icon*



➢ *Enter all the required parameters such as section name & dimensions.*

Note : Cross shape can only be used for columns.

9. New International & Country Specific Code of Practices

ProtaStructure now supports American codes of practice for steel and concreate design together with associated loadings codes. This includes comprehensive support for Seismic loading, analysis, design and detailing including sophisticated checks for complex topics like building irregularities, strong column and weak beam design philosophy, and beam column joint shear checks. Regional variations covering detailed loading and design rules for Indonesia and the Philippines are also introduced.

International loadings & seismic code are implemented as follows :

- UBC & IBC (American) *New!*
- SNI 1726-2012 (Indonesia) *New!*
- NSCP 2015 (Philippines) *New!*
- Eurocode 1 & Eurocode 8 (Singapore, Malaysia & UK Annexes)
- BS6399 (British)

Concrete design is implemented for the following code of practices :

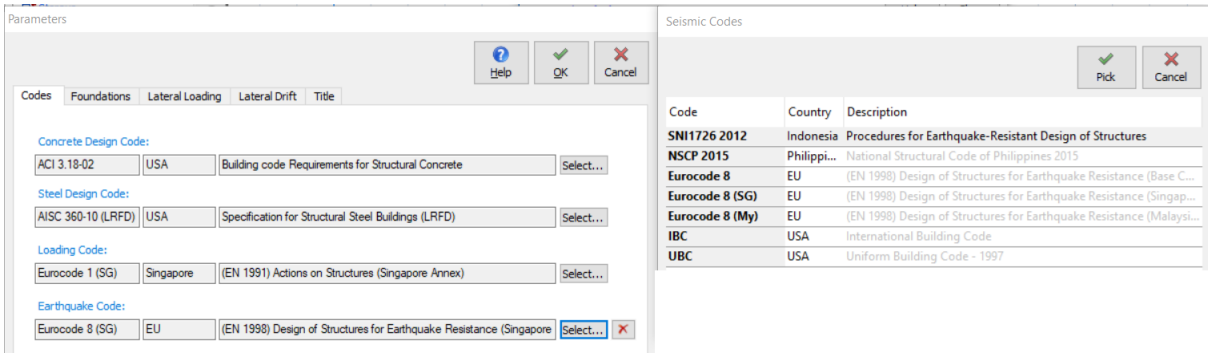
- ACI 318-2 (American) *New!*
- EuroCode 3 with Singapore, Malaysia & UK Annexes
- BS8110-1997 (British)
- TS500-2000 (Turkey)

Steel design is implemented for the following code of practices:

- ✓ AISC360-10 (LFRD) (American) *New!*
- ✓ AISC360-10 (ASD) (American) *New!*
- ✓ EuroCode 3 with Singapore, Malaysia & UK Annexes
- ✓ BS5950 (British)
- ✓ TSC 2016 (LRFD & ASD) (Turkey)

How to Use

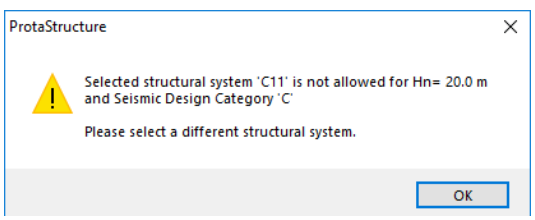
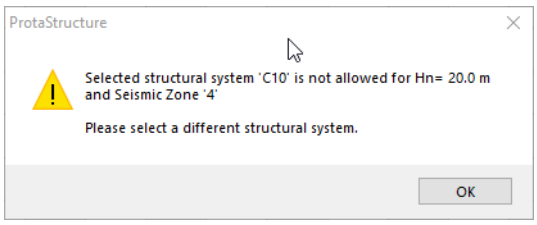
➢ *Go to Analysis → Building Analysis dialog → Parameters*



➢ *Under **Earthquake Code** → Pick **Select** → Choose required seismic code*

Tips & Notes :

Following items are supported for Indonesia and Philippines Seismic Codes:

Indonesia SNI1726:2012, IBC	**Philippines NSCP2015, UBC97**
User needs to pick Spectral Accelerations at **short period, Ss** and **1 sec period, S1** values from hazard map using the exact coordinate of the project site.	User needs to select **Seismic Zone, Seismic Source Type, Closest Distance to Source**. Near source factors are automatically calculated.
User decides on the '**Risk Category**' of the building and selects the appropriate values on the UI.	User decides on the **Occupancy Category**. Importance Factor is automatically fetched from the code.
ProtaStructure automatically determines the "**Seismic Design Category**" depending on selected Risk Category and acceleration values.	There is no Seismic Design Category concept in NSCP and UBC. Seismic Zone serves the same purpose.
User also selects the proper '**System Category**' and '**Structural System Type**' for his building. Behaviour Factor (R), Overstrength Factor (Ω_0) and Displacement Scale Factor (C_d) are automatically fetched from tables in the seismic code.	User also selects the proper '**System Category**' and '**Structural System Type**' for his building. Behaviour Factor (R) and Overstrength Factor (Ω_0) are automatically fetched from tables in the seismic code.
If a selected 'Structural System Type' is not permitted for the current **Design Category** and **Building Height**, then **Prota**Structure will warn the user with a message. 	If a selected '**Structural System Type**' is not permitted for a given **Seismic Zone** and **Building Height**, then **Prota**Structure will warn the user with a message. 

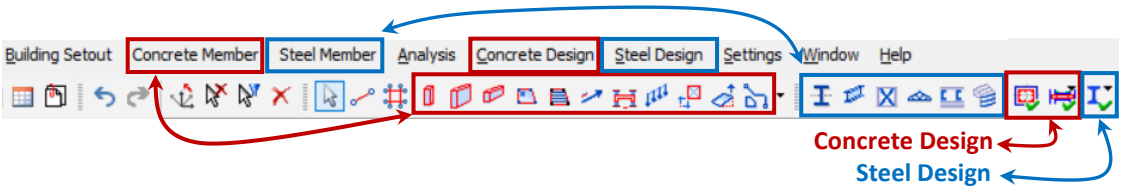
The following are the scope and assumption of the seismic analysis and design :

- Elastic and Design spectra are automatically calculated.
- Same EQ Static and Mode-superposition analysis methods are applied.
- Maximum period of vibration checks are automatically made.
- Following irregularity checks are automatically made:
  - Torsional Irregularity
  - Extreme Torsional Irregularity (*Not needed for NSCP and UBC)*
  - Weak Storey
  - Extreme Weak Storey (*Not needed for NSCP and UBC)*
  - Soft Storey
  - Extreme Soft Storey (*Not needed for NSCP and UBC)*
  - Nonuniform Mass Distribution
- Applicability of Equivalent Static Method is automatically checked.
- Overstrength factors are automatically applied to design forces.
- If there is a basement in the building, a two-stage approach is used to analyse the building. Basement and superstructure are analyzed separately in different load cases and the results are combined for the design.
- Results for cracked and uncracked section analysis can be combined in same design combination.

10. Steel Design Enhancement

Steel design is greatly enhanced and includes new design interfaces for purlin, girt & braces and new design diagrams which automatically detect restraint conditions and buckling lengths for more economical design. Steel members are checked for pass / failure status and the most critical Utilization Ratio (UR) reported. You can change the section sizes in the interactive design & new sections will be automatically checked.

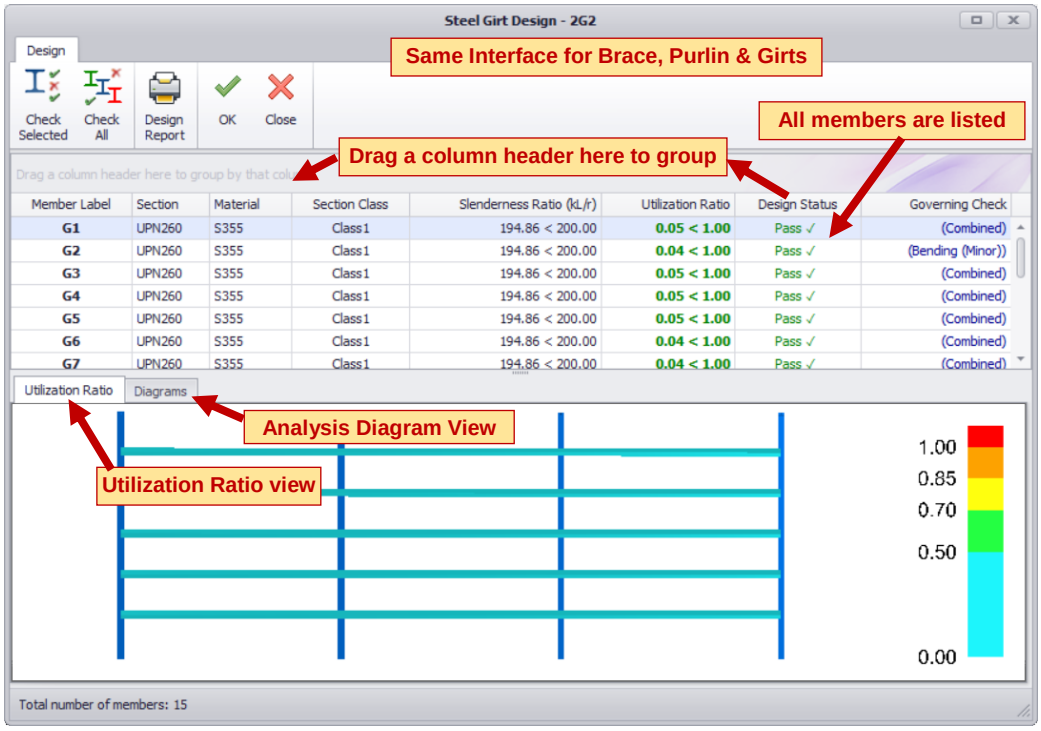
In line with the expanded scope of steel members, the user interface is enhanced to group concrete & steel members modeling & design separately.



How to Use

- *Go to Steel Design (top menu)* → ***Brace / Purlin / Girt Design***
- *Alternatively, select a brace, purlin on girt* → ***Right-click*** → ***Steel Member Design***

The design dialog lists down all the members of the group (as shown below) :

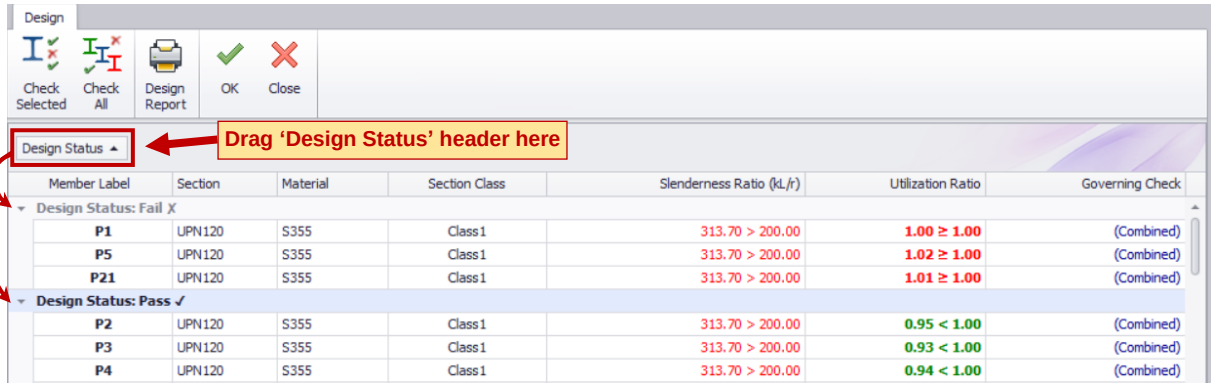


➢ *Pick **Check All** to preform design check for all members*

The result such as Design Status, Utilization Ratio & Governing Check will be updated in both the table and diagram view.

Design results can be grouped or categorized for easy viewing. For example, if there are combination of pass and Fail members, grouping by design status is helpful :

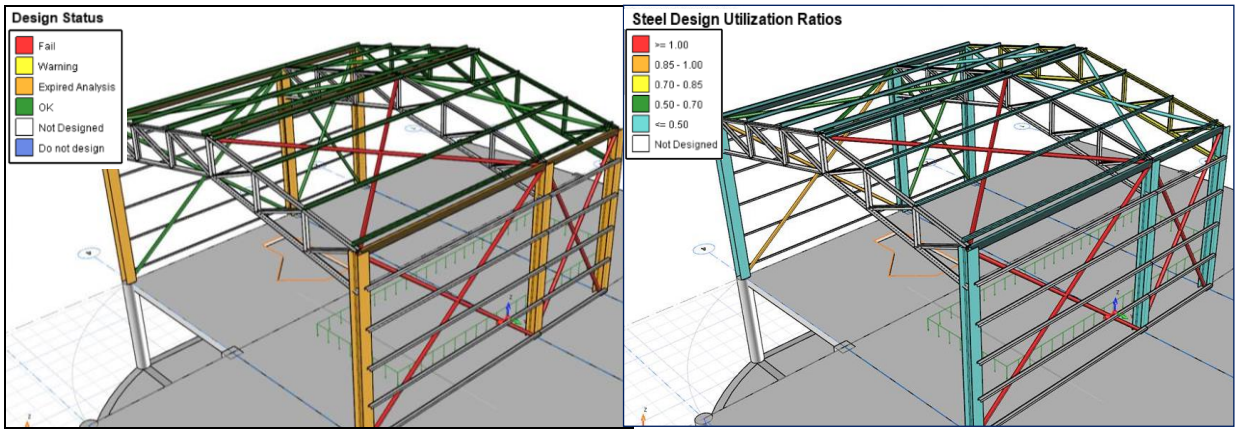
➢ *Click & drag **Design Status** header to the designated row at the top (as shown below)*



➢ *Double-click on specific member to access the detail interactive steel design.*

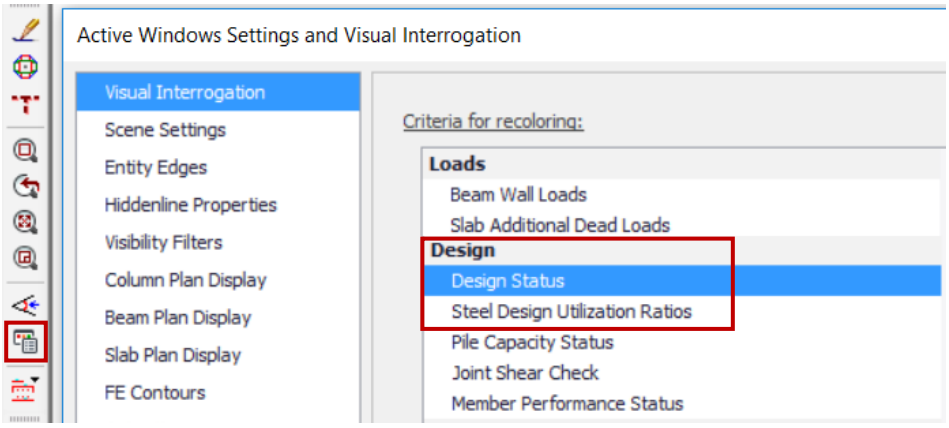
11. Enhanced Visual Interrogation

Visual Interrogation has been enhanced to include the **Steel Design Status** and **Steel Utilization Ratios** for quick review of design efficiency.



How to Use

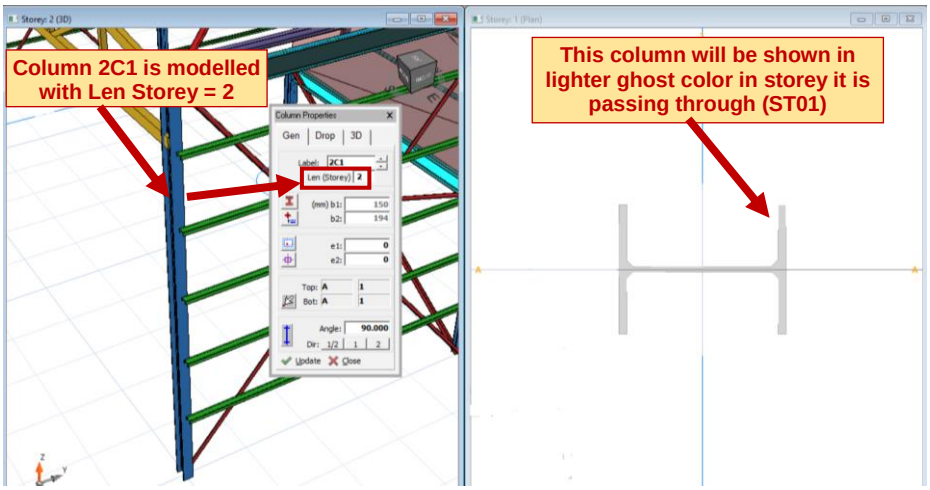
- *Go to Visual Interrogation* → ***Criteria of Recoloring*** → *Pick* ***Design Status / Steel Design Utilization Ratios***



12. Transit columns/walls are displayed on all relevant plan views

Transit columns or shearwalls (Len Storey= 2 or more) are now displayed on all plan views of the storeys they are passing through. They are displayed in a lighter ghost color in the plan view of the passing storey. This greatly ease modeling speed and accuracy; it is a highly requested feature especially for steel modelling.

In the example *Quick Start Guide Steel*, the column is modelled in ST02 with spanning two storeys (Len Storey=2) to accurately capture the design effective length. This column will be displayed in lighter grey color in ST01 (as shown below).



13. Automatic Wind Load Calculator

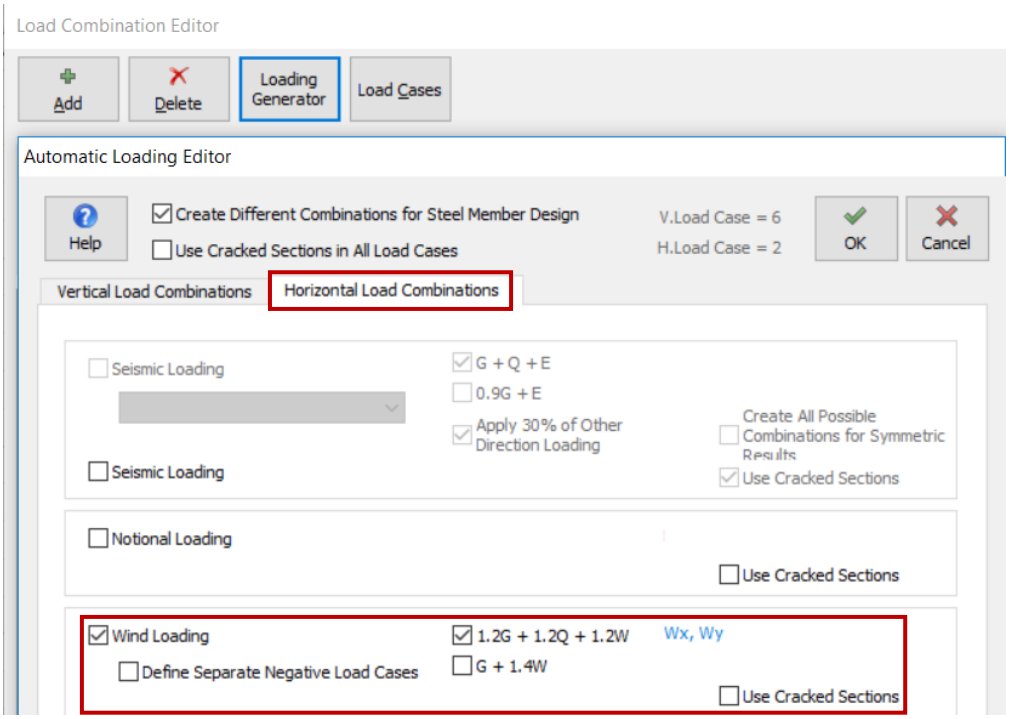
The new wind load calculator automatically determines wind forces in accordance with the chosen wind code and applies it the building structure. This greatly increases productivity as values need not be manually calculated & entered. The wind code of practices covered are :

- ✓ 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 **Loading combinations** button
- *In Load Combination Editor, click **Loading Generator** button*
- *In Automatic Loading Editor, go to **Horizontal Load Combinations** tab > Check **Wind Loading** as shown below*

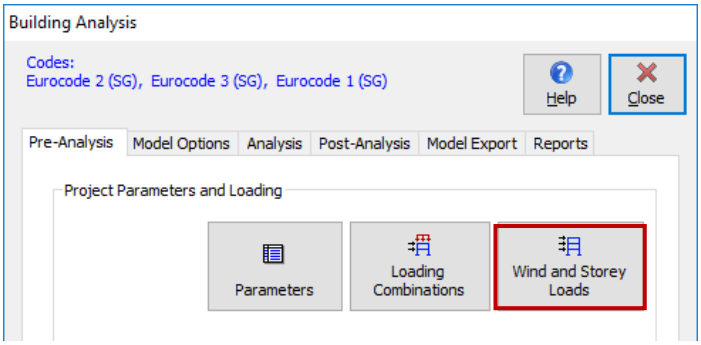


Note : Check "**Define Separate Negative Load Cases**" if the wind acting from left to right & right to left is expected to be different, e.g. irregular shaped building. Additional wind load cases Wx+, Wx-, Wy+ & Wy+ will be created.

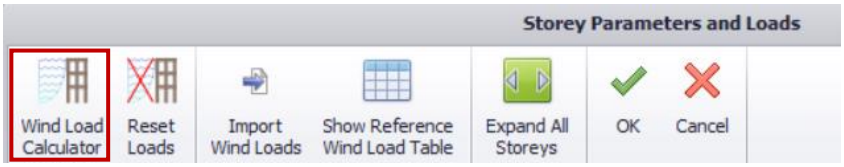
- *Click **OK** to close Automatic Loading Editor dialog*
- *Click **OK** to close Load Combination Editor dialog*

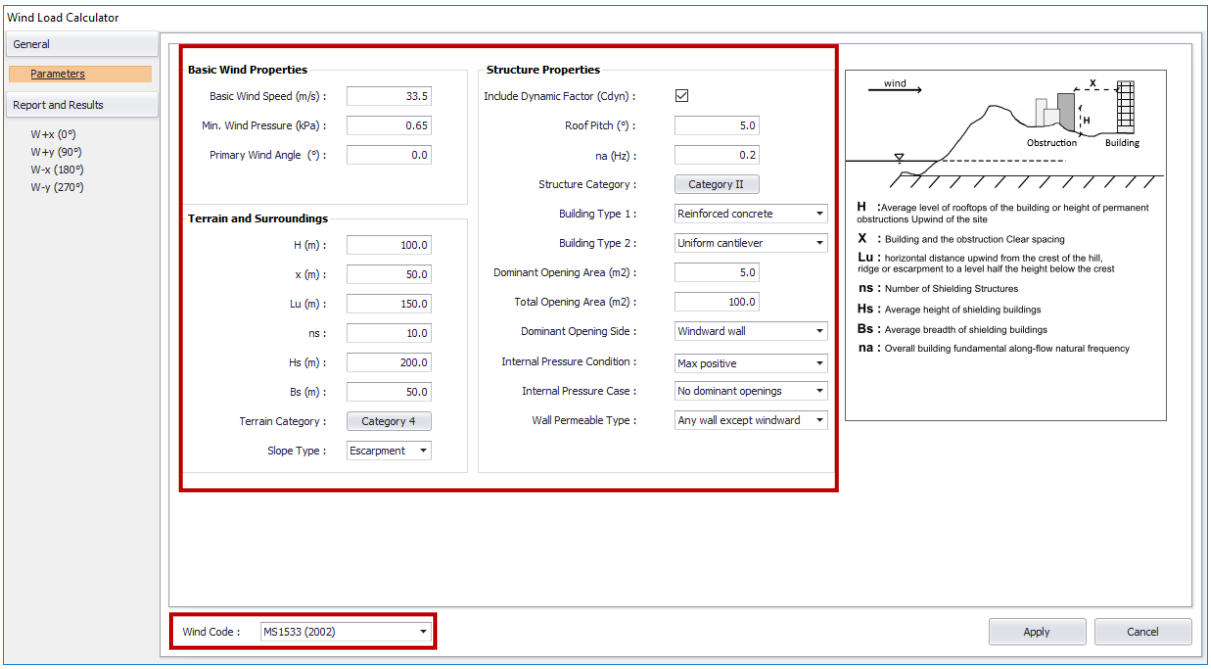
Step 2: Calculate and Apply Wind Loads:

- *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*





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

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

Notes:

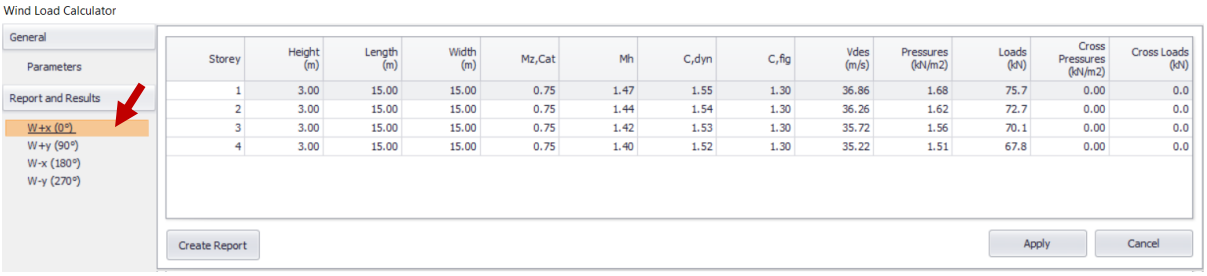
✓ Most parameters are illustrated and explained in the right figure.

✓ '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.

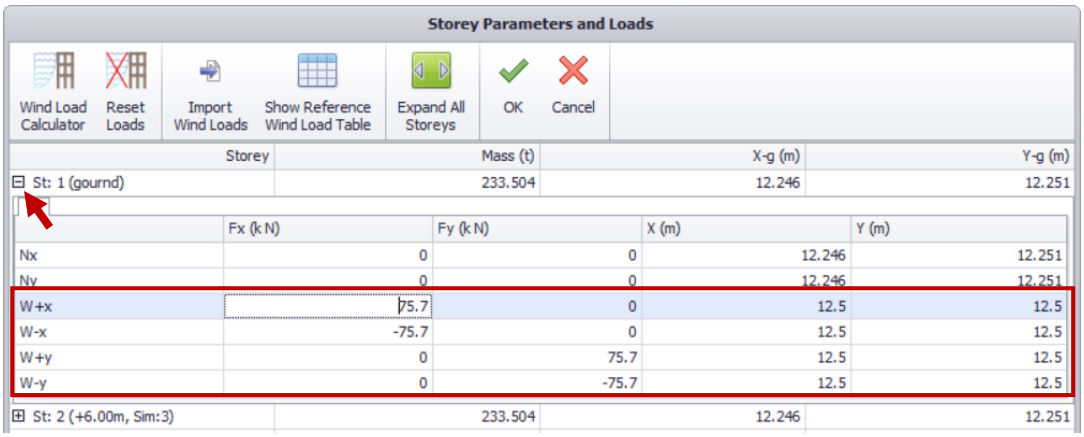
✓ All other parameters are explained in the respective codes of practices

✓ If there are basements that are not subjected to wind loads, it can be excluded via **Edit Storey** dialog (**Building Setout** top menu) → Input **No. of Rigid Basements**

➢ *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)*



- *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)*

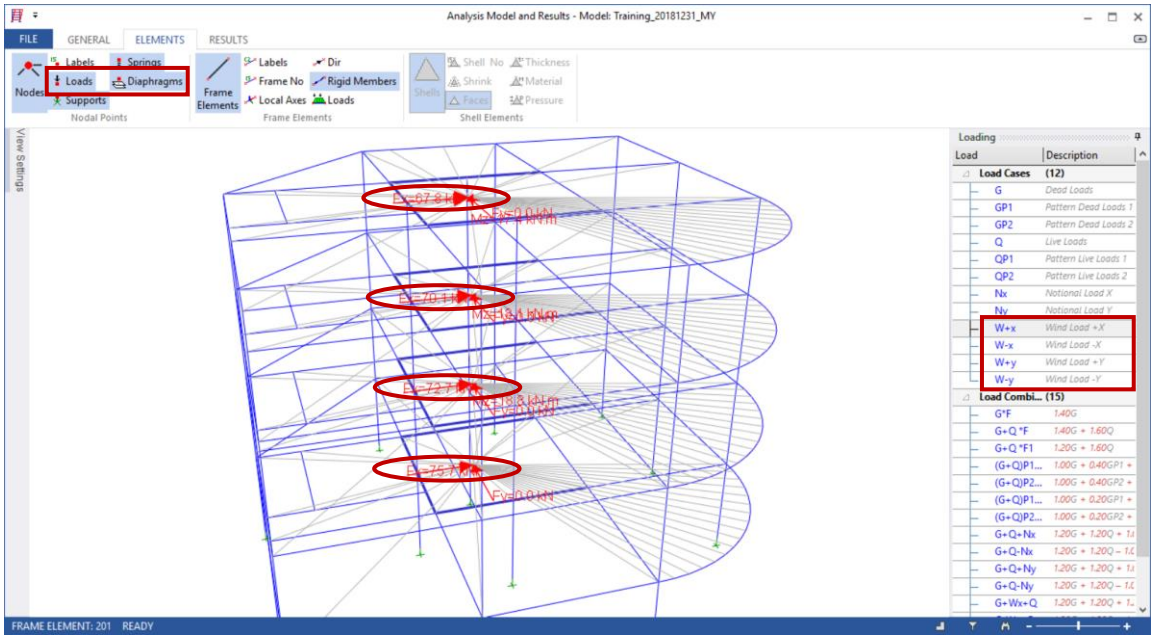


Note: The apply wind loads are editable, you may click & edit the applied wind load

- *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 that are automatically distributed to the center of every floor diaphragm and isolated member(s) where applicable. Mz is the twisting moment due the wind load & is automatically calculated based on the eccentricity of the center of diaphragm & center of rotation multiplied by magnitude of the wind load case.

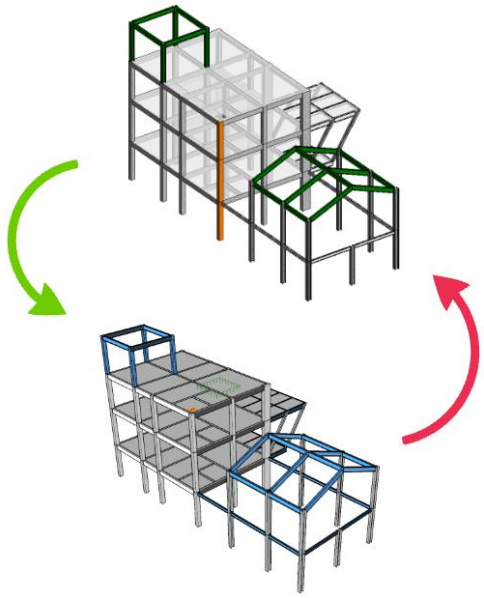
Assumptions & Limitations

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.
5. 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.

14. Enhancements in Revit Integration

ProtaStructure supports bi-directional links with latest version Revit Structure 2019; greatly enhancing project coordination and workflow. Model changes can be synchronized and tracked both in Revit and ProtaStructure. Round-tripping between Revit to Protostructure is greatly enhanced. The Integration Status will color code which members are new, changed, unchanged or deleted. A detailed list explaining these changes is also provided.

- 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 is 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).



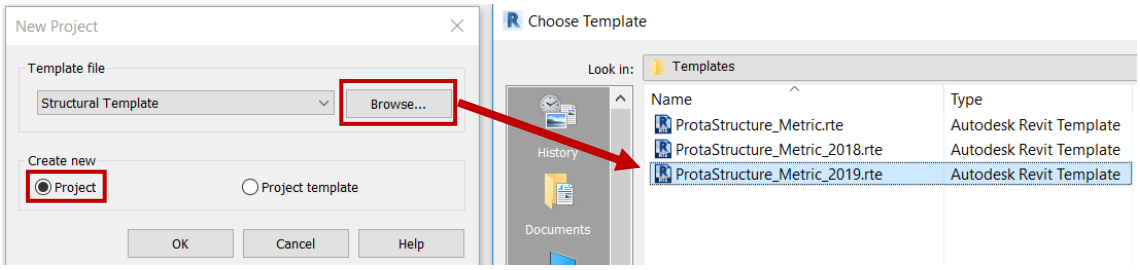
Please note that **ProtBIM** addon must be installed in the same computer where Revit Structure is installed. Please download from online help : http://help.protasoftware.com.

Round-tripping between 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.

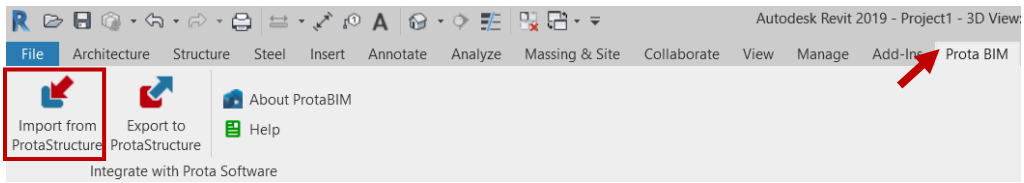
How to Use

1. Import a new ProtaStructure to Revit

➢ *In Revit, start a new project using **ProtaStructure_Metric_2019.rte** template*

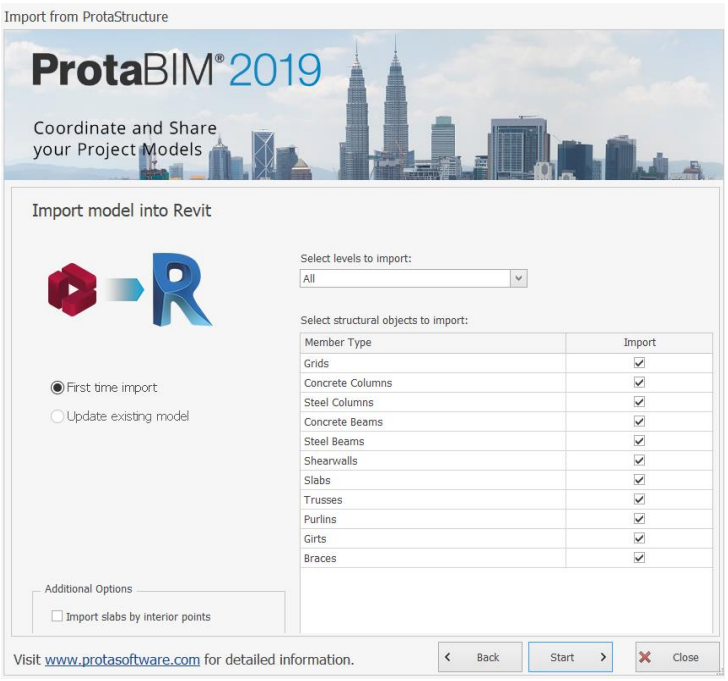


➢ *In Revit, use **Prota BIM** tab to import a new ProtaStructure model*



➢ *Browse & select the ProtaStructure model file with extension **.Prota***

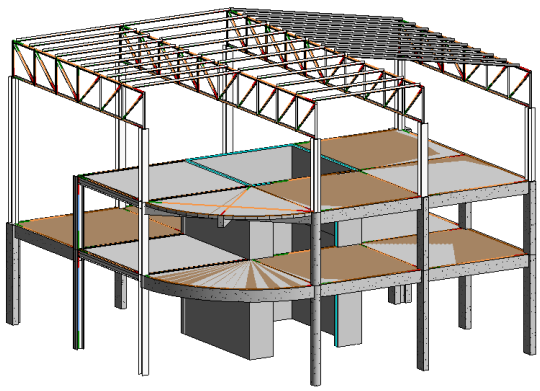
- *Pick Import options as desired.*



Note : '**First Time Import**' will be checked and will be the only option if it is first import to Revit. Check the option **Import slabs by interior points** will avoid slab trimming and overlapping with other member issues.

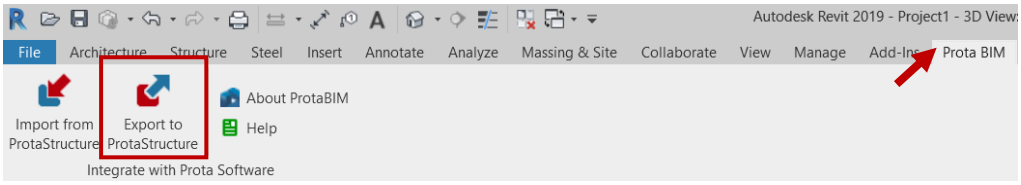
- *Pick Import options as desired → Next → Close*

A model import summary will be shown with a successful import.

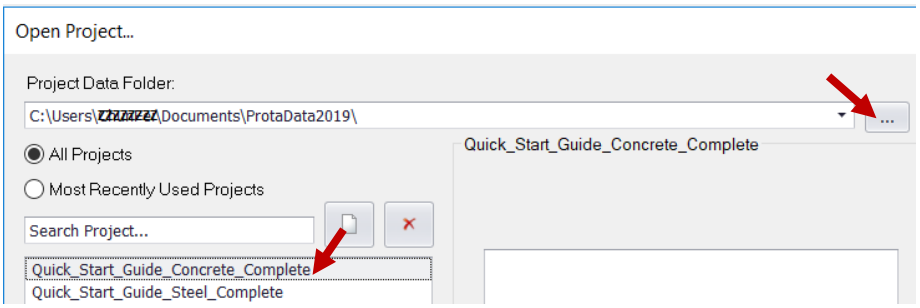


2. Export to ProtaStructure (after changes in Revit)

- *After successful import, make changes to the model in Revit*
- *Select **Export to ProtaStructure** in **ProtaBIM** ribbon*

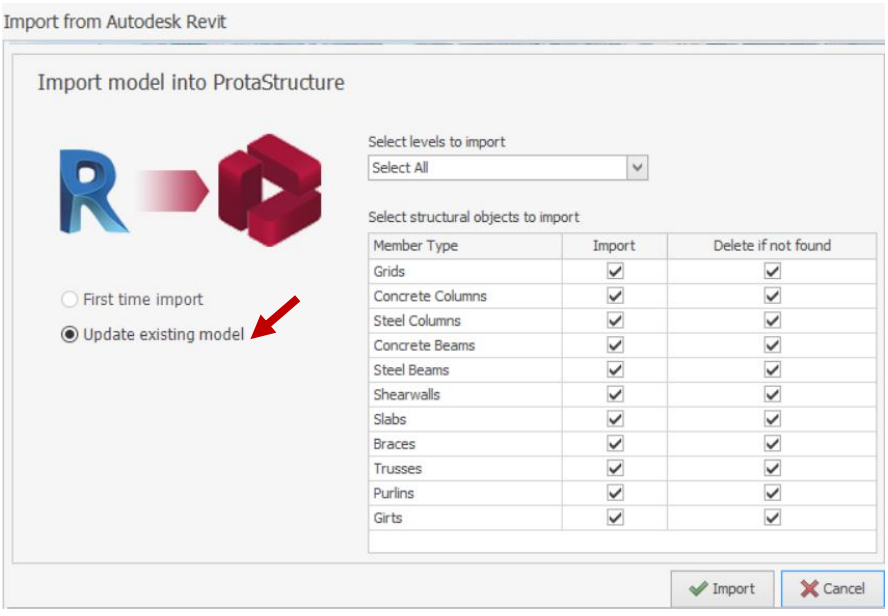


- *Select the **same** ProtaStructure project in **Open Project** dialog (as item 1).*



A "RevitData" sub-folder will be created under selected project folder with new project files incorporating the changes.

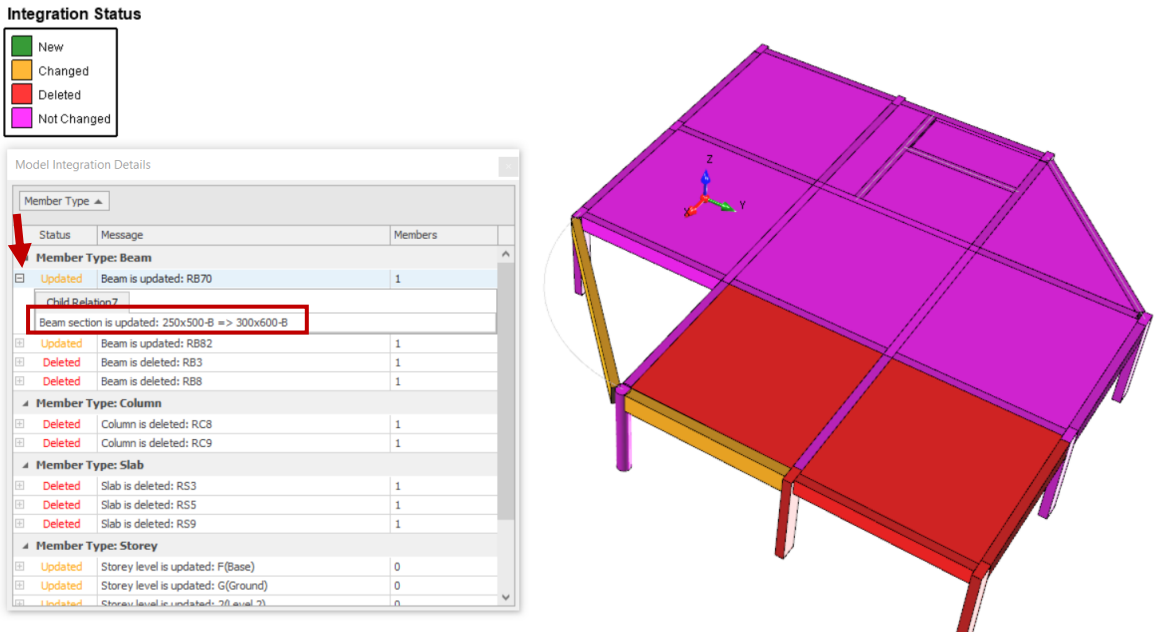
- *Launch ProtaStructure or bring it to front if it is already loaded.*
- *Ensure the same project is opened (project name should not be changed)*
- *Go to **File** Menu → **Mode/File import** → pick **Import From Revit Structure***
- *In Import dialog, choose the desired options → Pick **Import***



Note : 'Update existing model' will be automatically checked. Check the option 'Delete if not found' if you want members that are deleted in Revit to be completly removed in ProtaStructure.

- *Building Model Check will be triggered. Take note of any errors & **Close***

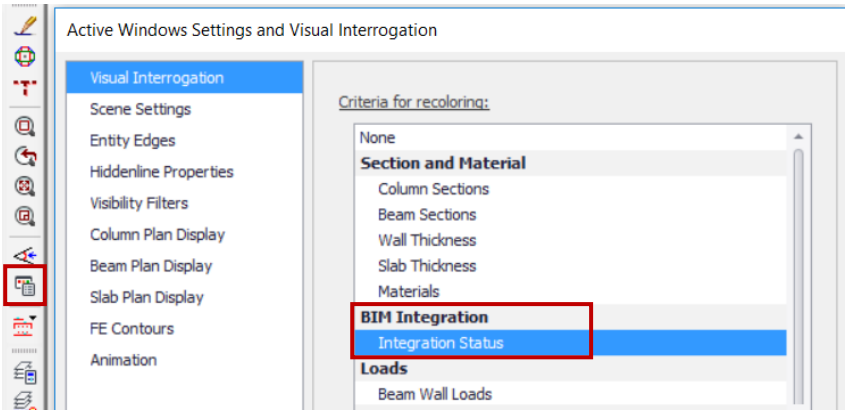
The view will automatically switch to a 3D view with the **Integration Status** color coding showing whether member(s) is **new**, **changed**, **deleted** & **unchanged**.



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*

The Integration Status view can re-opened via **Visual Interrogation → Integration Status**



3. Re-Import (Update) a ProtaStructure model to Revit

- *Make changes to the model in ProtaStructure. Once complete **Save** the project*
- *Go back to Revit Structure & ensure the same Revit model is opened.*
- *Go **Prota BIM** tab → pick **Import from ProtaStructure***
- *Browse & select the ProtaStructure model file with extension **.Prota***

In the Import dialog, "**Update exiting Model**" will be automatically be checked

- *Review the import Options → **Import***

The round tripping of "update" import and export to & from Revit & ProtaStructure can be repeated as many times as you wish.

15. IFC Export

ProtaStructure model can be exported to IFC format using '**IFC2x3, Coordination View Version 2.0**'. IFC File format is implemented by most design software on the market including leading Building Information Model (BIM) platforms like Autodesk Revit, Bentley and Tekla Structures. The main purpose is to allow sharing of building information models between various BIM platforms.

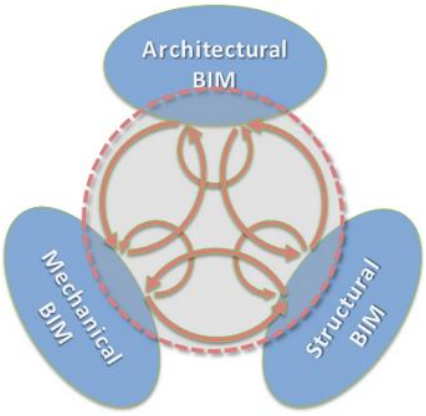
How to Use

- *Go to **File → Model/File Export → Export IFC File**" menu.*

Tips & Notes :

Coordination View Version 2.0

The Coordination View targets the coordination between the architectural, mechanical and structural engineering tasks during the design phase.



Purpose of Coordination View Version 2.0

The Coordination View has been the first view definition developed by buildingSMART International and is currently the most implemented view of the IFC schema.

The main purpose of the Coordination View is to allow sharing of building information models between the major disciplines of architecture, structural engineering, and building services. It contains definitions of spatial structure, building, and building service elements that are needed for coordinating design information among these disciplines.
The support of round-trip scenarios is excluded from the support of the coordination view.

How can you use IFC Files?

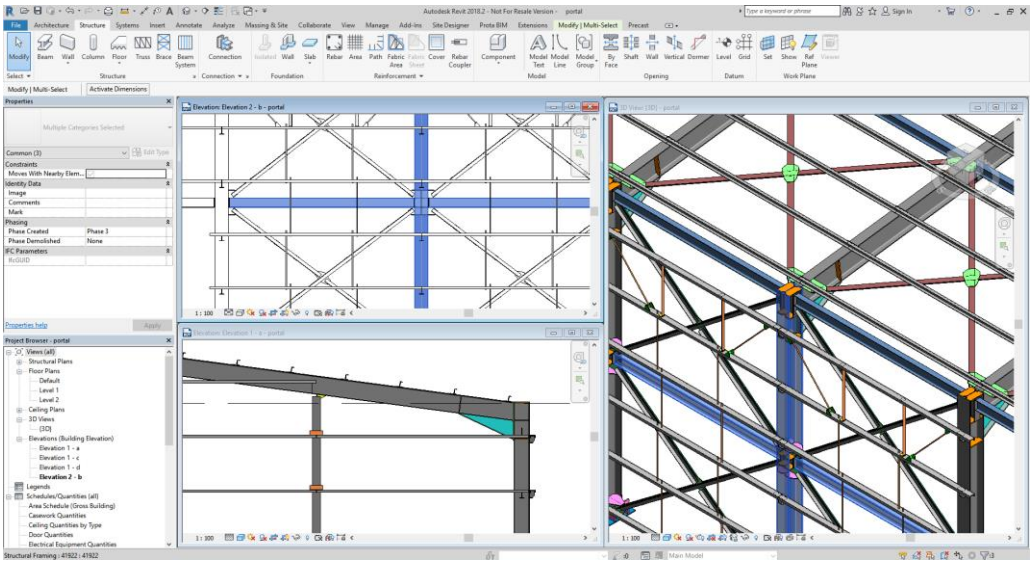
IFC File format is implemented by most design software on the market including leading BIM platforms like Autodesk Revit, Bentley and Tekla Structures.

IFC communication is not and should not be expected to be as smart as the current bi-directional communication between Revit and **Prota**Structure. Revit and **Prota**Structure communication is a high level integration specifically developed with programming interfaces.

In IFC integration, it is up to the destination software to decide on how to interpret the exported model. **Prota**Structure has no control on the model once the model is exported to IFC.

There are some free IFC model viewer such as 'Solibri Model Viewer' to view and coordinate IFC files : https://www.solibri.com/solibri-model-viewer

ProtaSteel is also capable of creating IFC files. IFC files created by ProtaSteel includes all the connections create. An example screenshot of ProtaSteel IFC imported into Revit is shown below.

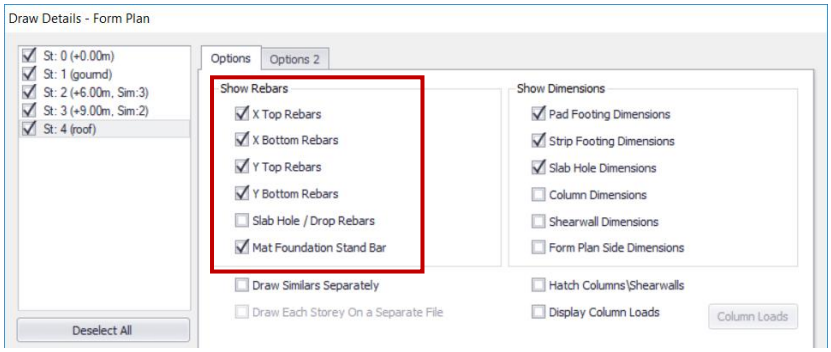


16. New RC slab cross sections with reinforcement detailing

Reinforced slab cross section can be created with reinforcement detailing. In ProtaDetails, cut sections anywhere along the layout plan drawing and position it anywhere within the drawing sheet.

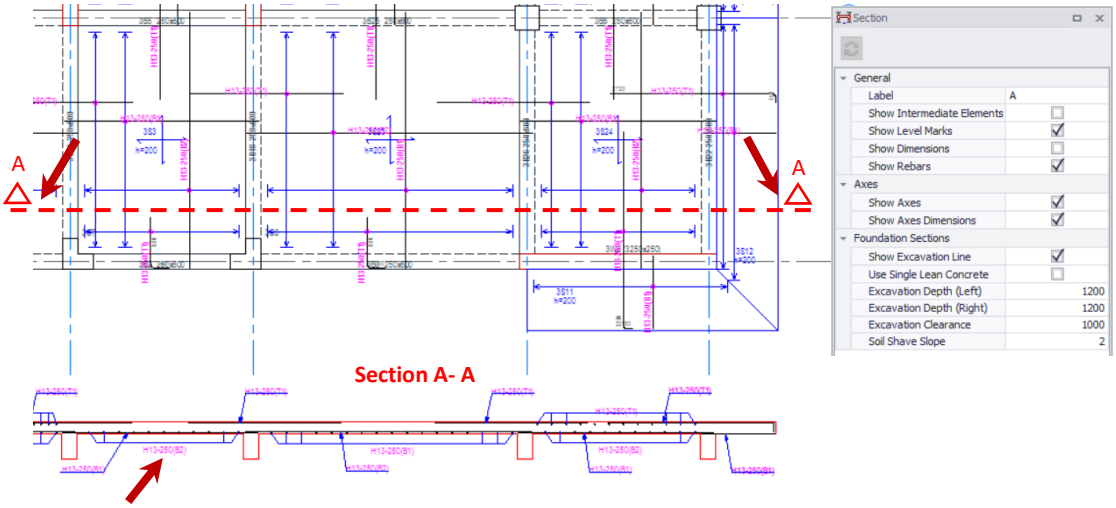
How to Use

- *In ProtaStructure, ensure that all the slabs reinforcement has been designed using slab strips.*
- *Go to **Concrete Design** → **Load ProtaDetails***
- *Generate the **Form Plan** drawing ensuring options to **Show Rebars** are checked*



Tip : More slab detailing options are available tin the **Options 2** tab.

- *Go to **Detail Library** (top menu) → Pick **Plan Sections***
- *Alternatively, in the **Command** line at the bottom, type "**FormSection**"*
- *In the **Section** dialog, input / select the preferred options*
- *Draw a "section cut" line by clicking on 2 points on the plan view*

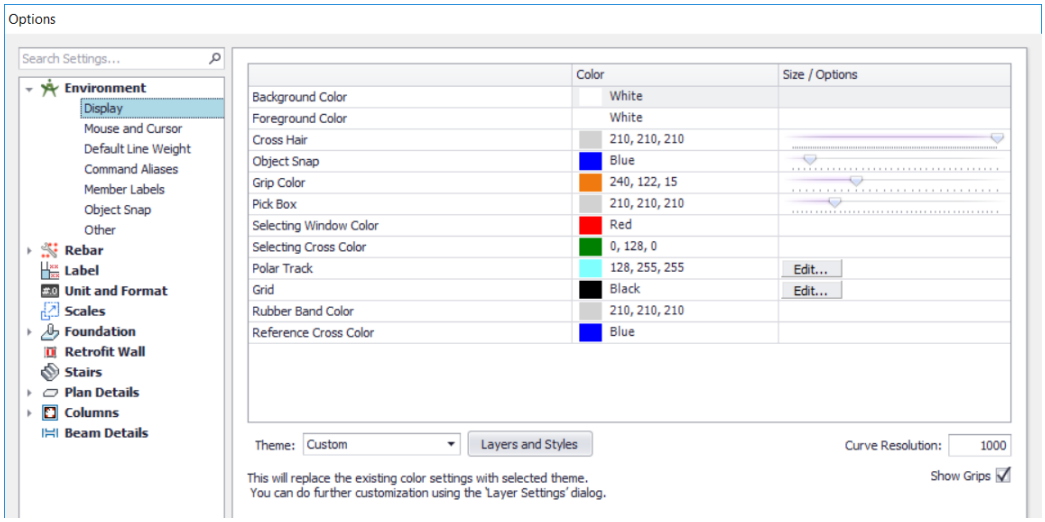


- *Click on the desired location to place the section detail*

Tip : Click ORTHO (orthogonal) icon right at the bottom to ensure the line is exactly horizontal/vertical.

17. New Global Options Interface

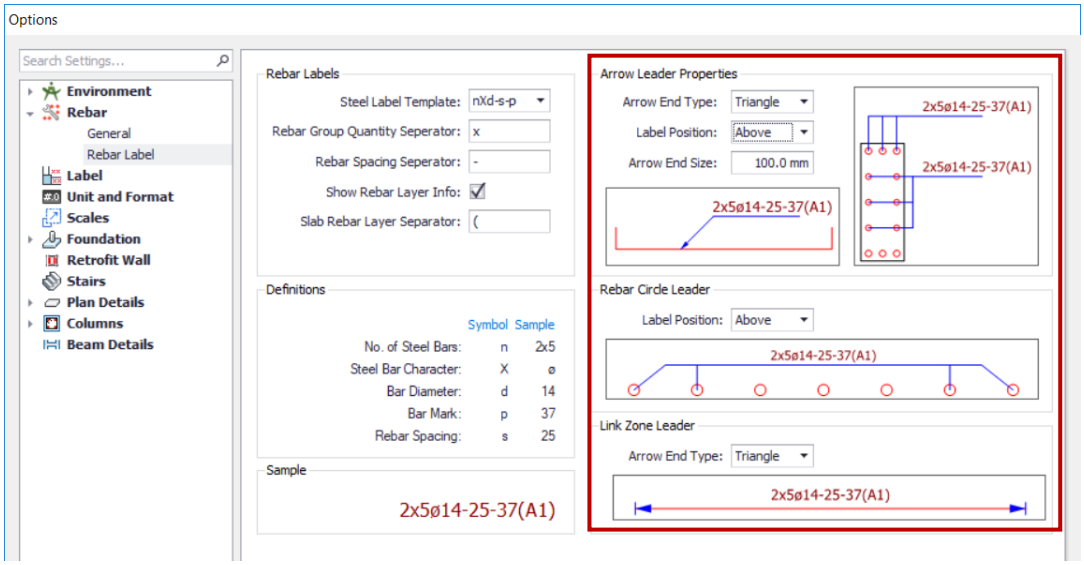
Drawing and detailing options are now classified under a single interface; allowing easy global control on how the drawings are produced. This is accessible from **Settings** (top menu dropdown) → **Options**.



You can navigate from the left side panel by drilling down to each setting. Further, you can quickly search for any specific settings by typing in the "**Search Settings…**" text box. The navigation panel will be filtered out for the results as you type.

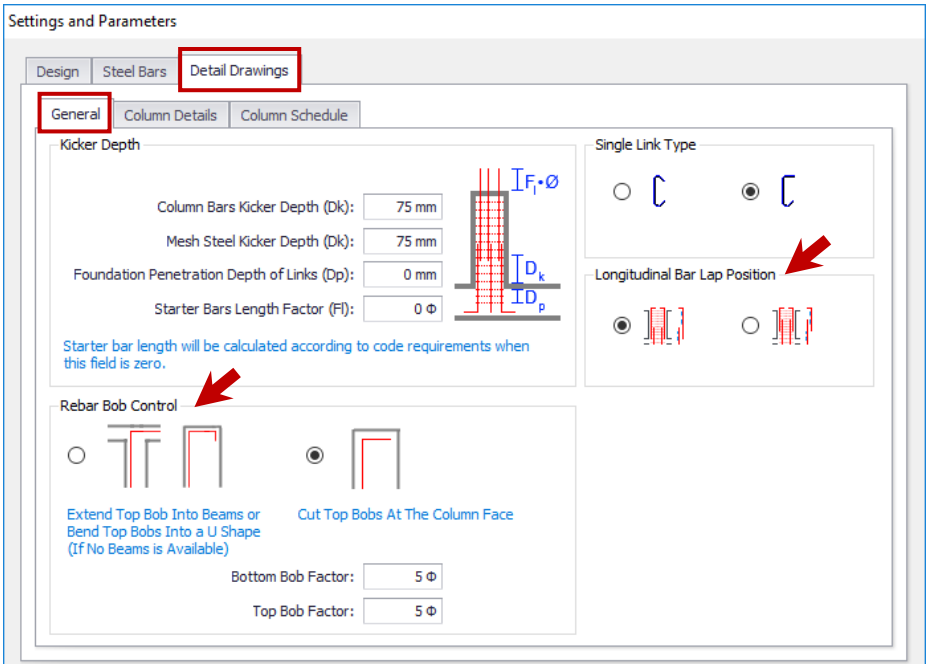
18. New Rebar Label Settings

Rebar label settings has been enhanced to allow for more user-defined options for arrow leaders (pointing to rebars) and dimensions arrows. This offers greater flexibility for users to produce high quality drawings to their company standards. This is are accessible to **Settings** (top menu dropdown) → **Rebar** → **Rebar Label**.

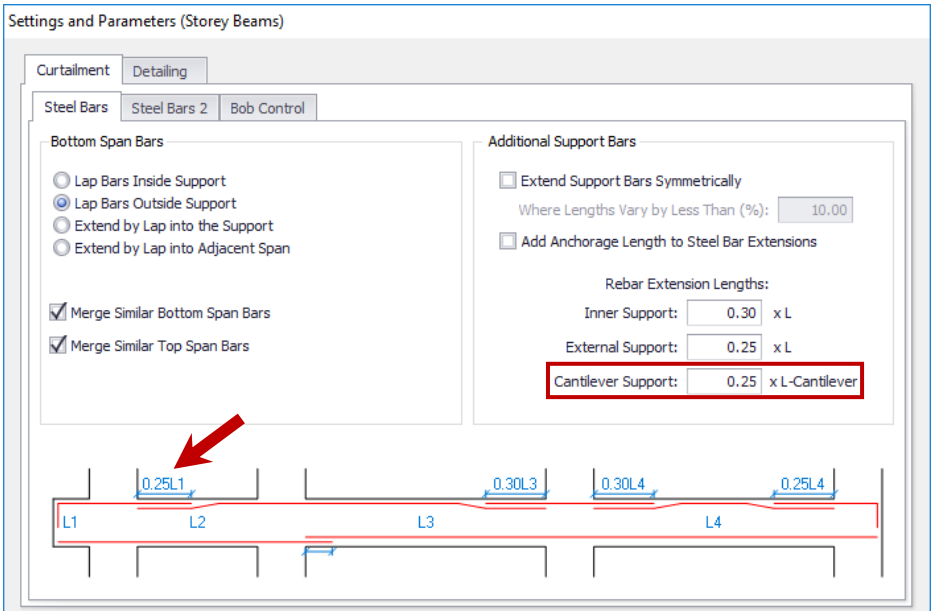


19. Enhanced Column & Beam Detailing Settings & Drawings

Numerous new column and beam detailing options are introduced to enhance the quality of drawings. For column elevations drawings, longitudinal bars can be lapped in the mid-span of the column (requirement of some code of practice) & bending of bars (BOB) can be controlled.
These new options are accessible to **Settings** (top menu dropdown) → **Column Settings** → **Details Drawings** → **General** tab.

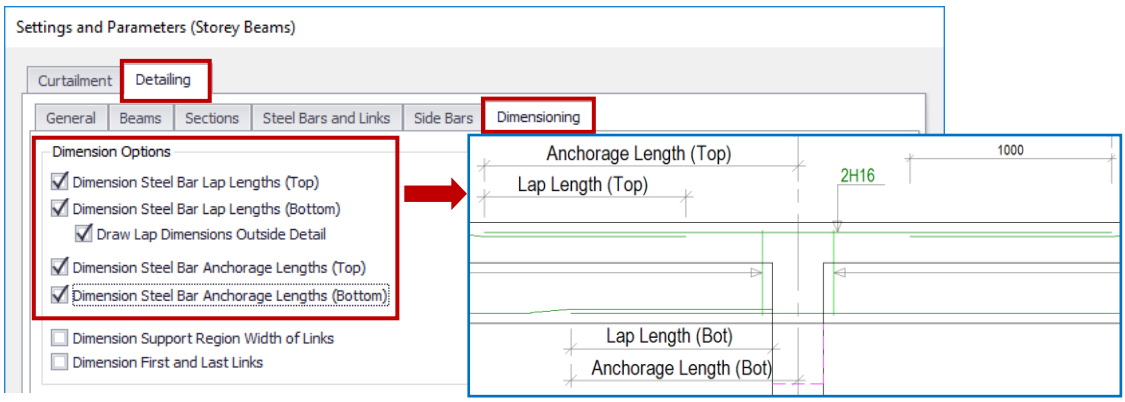


The beam elevation detail is also enhanced significantly. Cantilever top rebar extension lengths can be adjusted based on a user-defined factor multiplied by the cantilever length. This feature is implemented to cater for some user's preference for this manner of cantilever beam detailing.
This feature accessible via **Settings** (top menu dropdown) → **Beam Settings** → **Steel Bars** tab.



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

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 via the **Detailing → Dimensioning** tab.

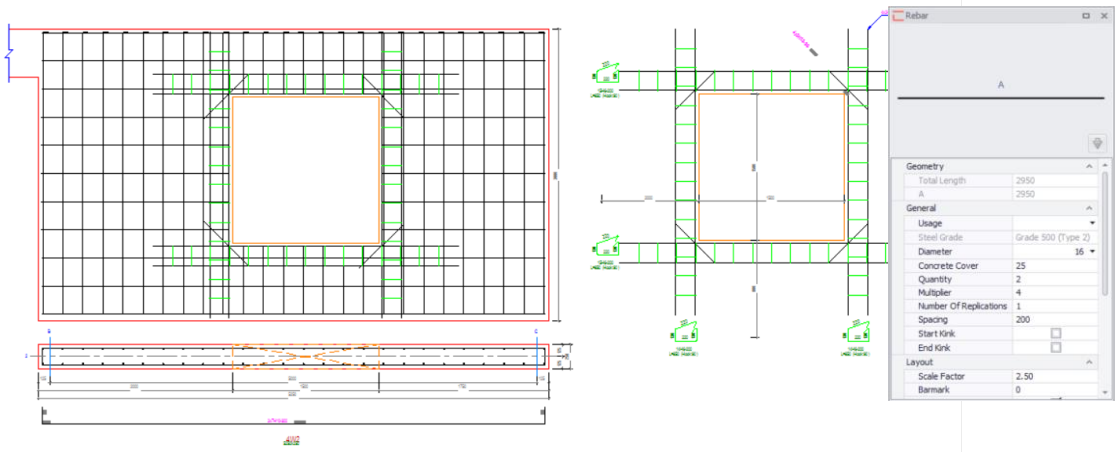


Important Notes :

- The column & beam detailing settings are directly inherited from ProtaStructure where the exact same settings exist.
- Any changes to the settings in ProtaDetails will only affect new details; existing beam details remains unchanged.
- Changes in settings in ProtaDetails cannot be saved; it will be discarded when ProtaDetails is closed.
- If you want to save changes, please update the same settings in ProtaStructure.
- We strongly recommend you review these settings before producing any drawings.

20. Automatic wall opening detailing

The reinforcement details are automatically created around the shear wall openings in the wall elevation details. The shear wall openings are displayed in both the sections as well as schedules.

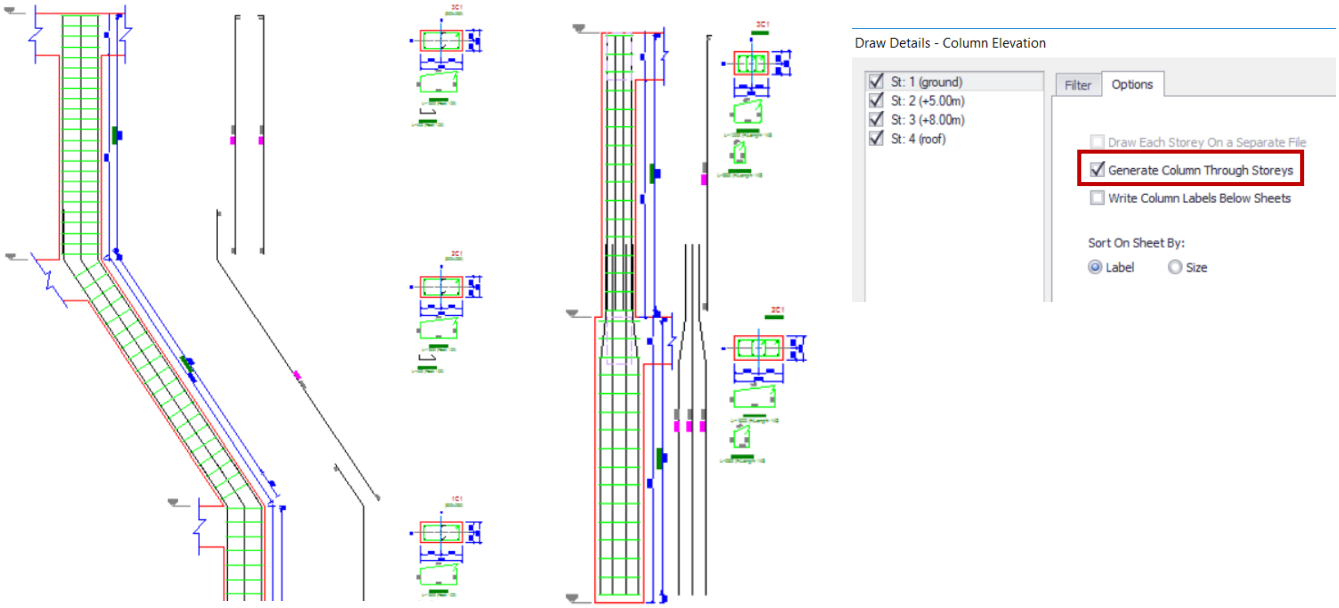


The reinforcement detailing around the opening are only suggested rebars; no actual design is performed by the program.

- *The details can easily be amended by selecting any rebar → the Rebar dialog will appear where changes to size, length & shape can be done.*

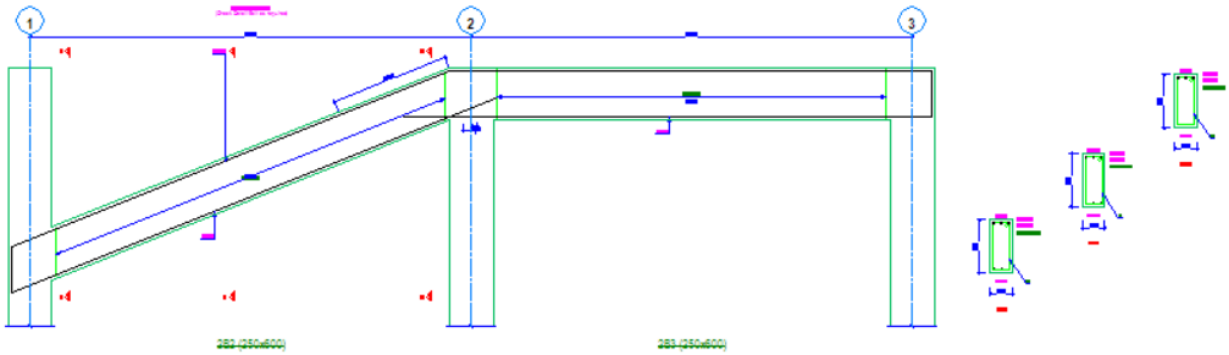
21. Precision detailing for slanting RC columns, varying column section sizes & slanting RC beams

Column elevation detailing is greatly enhanced to include slanting (angled) columns and columns with varying section sizes. Longitudinal bar detailing including intelligent curtailment and bar cranking follows Eurocode and ACI recommendations for columns with changing dimensions. Any beams connected to the columns will be drawn in the elevation as well.



- *Ensure to tick "**General Column Through Storeys**" in **Options** tab of the **Column Elevation** dialog box.*

There is significant enhancement in slanting beam detailing; details are more precise and accurate in beam elevation drawings.

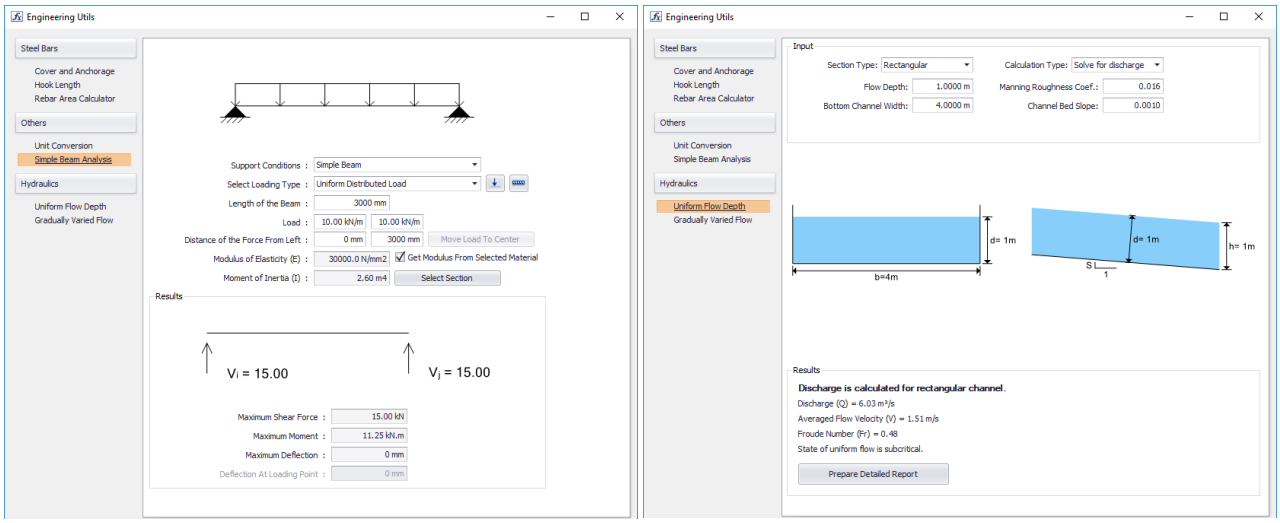


22. New Engineering Utilities for quick engineering design & calculation

A new handy **Engineering Utilities** is provided for quick engineering design. This includes a calculator for concrete cover, rebar anchorage & lap lengths; simple beam analysis tool to calculate maximum shear, moment and deflections; hydraulics calculator for uniform flow depth & gradually varied flow, etc.

How to Use

- *Go to **Design Library** (top dropdown menu) → **Engineering Utilities***



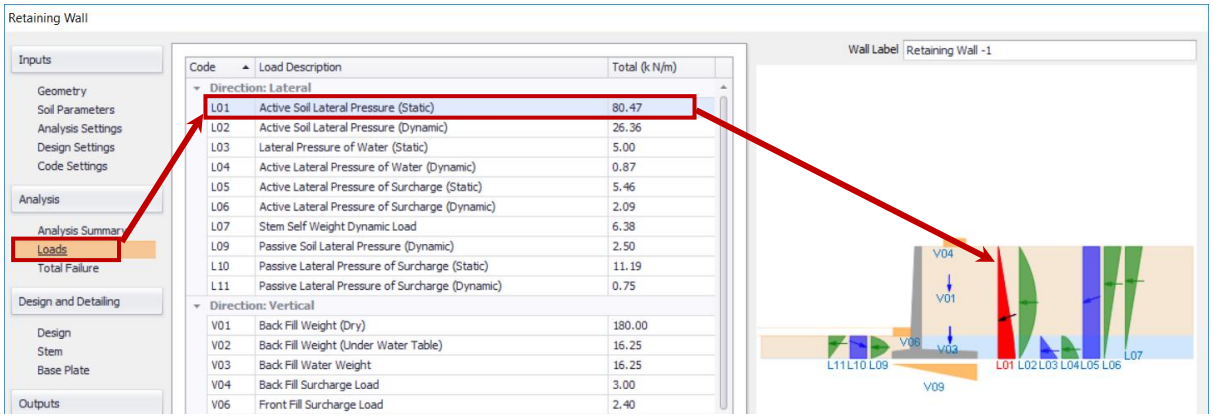
The following is the complete list of the modules and their functions :

- **Cover and Anchorage Calculator**
  - *Calculates concrete cover, anchorage and lap lengths*
- **Hook Length Calculator**
  - *Calculates rebar bending, extension & total hook length*
- **Rebar Area Calculator**
  - *Calculates provided steel area per meter (e.g. 10H16 @ 100mm spacing) based on required steal area*
- **Unit Conversion**
  - *Converts common engineering units, eg. Pascal (Pa) → Kilopound Per Square Inch (ksi)*
- **Simple Beam Analysis**
  - *Calculates maximum shear, moment & deflection for single span beam with various support condition such as simple, cantilever & fixed end*
- **Uniform Flow Depth (Hydraulics)**
  - *Solves for discharge, flow depth, channel width or channel slope for rectangular, trapezoidal or circular channels under 'uniform flow depth' condition*
- **Gradually Varied Flow (Hydraulics)**
  - *Solves for discharge, flow depth, channel width or channel slope for rectangular, trapezoidal or circular channels under 'gradually varied flow' condition*

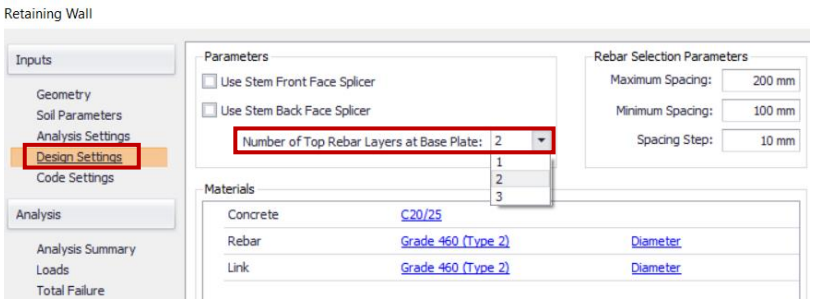
23. Enhanced Retaining Wall Module

The retaining wall module is greatly enhanced and now supports the American ACI318 & ASCE7 code. In addition, Eurocode is expanded to cover EC2, EC7 and EC8 with all three design approaches. There are numerous other improvements :

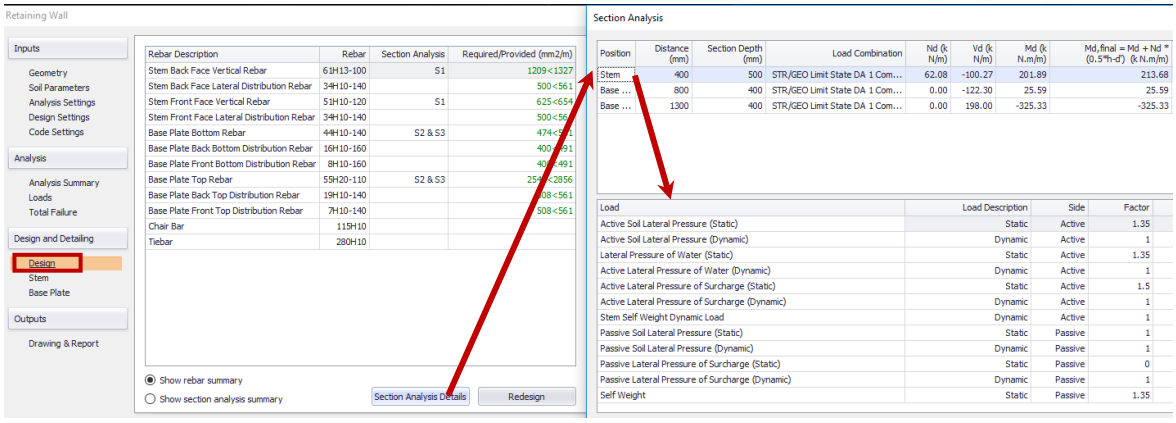
- Generated loads can be inspected separately for all parts of the wall for long term and short term analysis.
  - *Go to **Loads** → Select a **Load** → selected load will be colored in red*



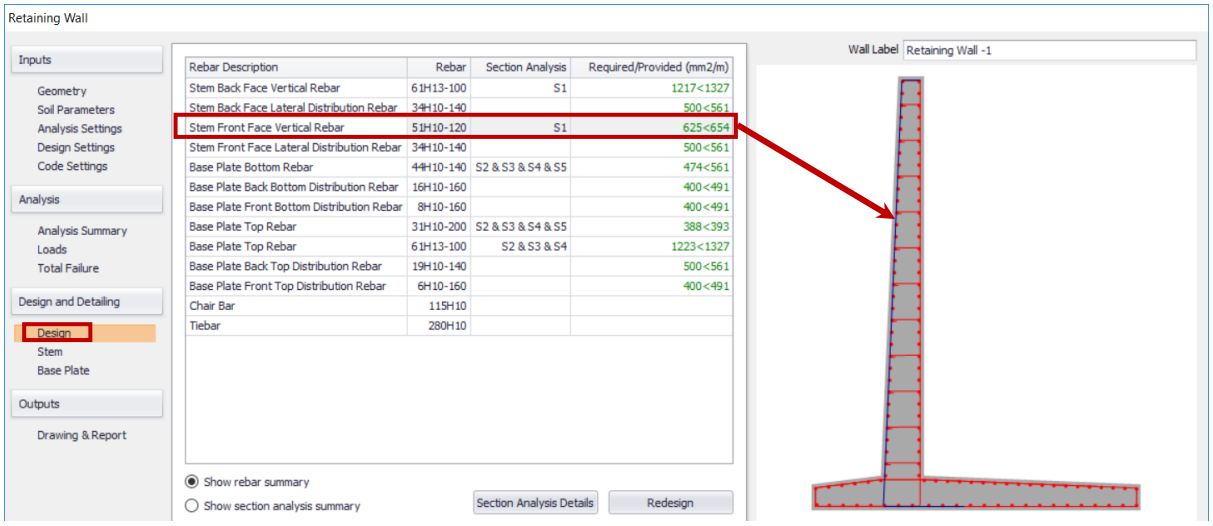
- Effect of shelves on lateral earth pressure and fill weights are now applied and is shown in detailed loading scheme
- Two different approaches to total failure are introduced; namely Fellenius and Bishop's methods
  - *Go to **Total Failure → Analysis Method** → Select **Fellenius / Bishop's Method***
- New option to have multiple layers of rebars at base plate
  - *Go to **Design Settings → Number of Top Rebar Layers at Base Plate** → Select **1/2/3***



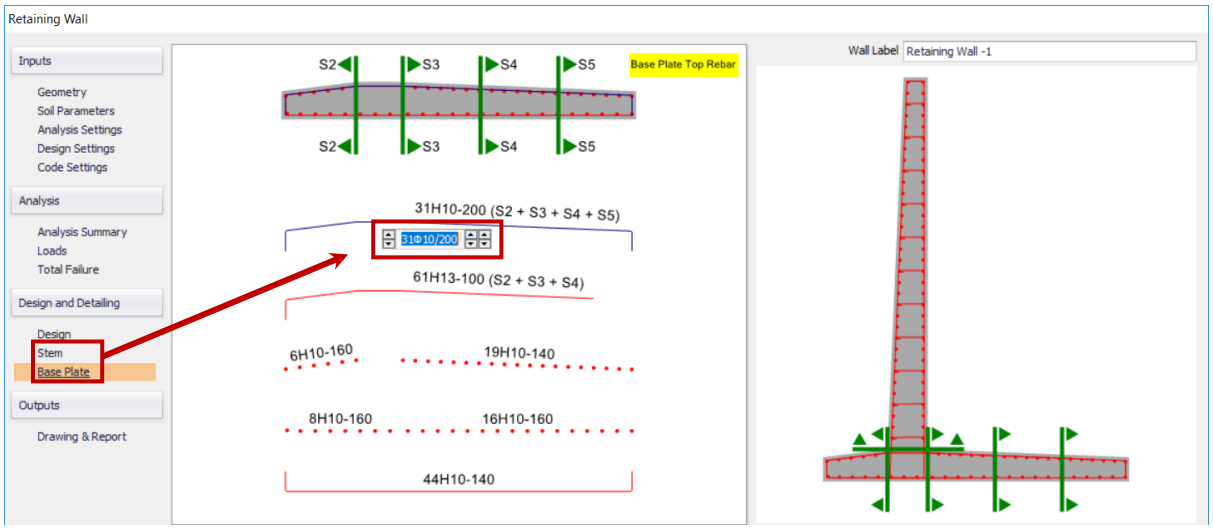
- Design approach is changed to be accurately based on reaction/forces at specified sections
- Detailed loadings and design forces at each section can be further inspected
  - *Go to **Design → Pick Section Analysis Detail** → Select **Position** to reveal detail checks*



- New Design interface allows easy visual identification of various rebars
  - *Go to **Design** → Pick a **Rebar** → the rebar will be highlighted in **blue** in the diagram*



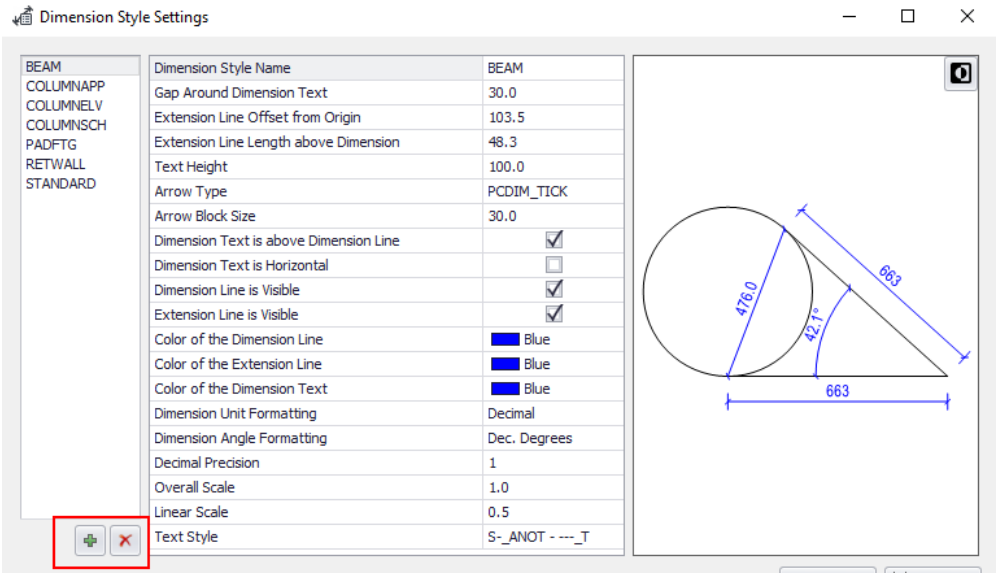
- New Detailing interface allows easy visualization and edit of rebars interactively.
  - *Go to **Stem / Base Plate** → Pick a Reinforcement → Edit number / diameter / spacing*



24. 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. There is no need to save project or drawing explicitly.



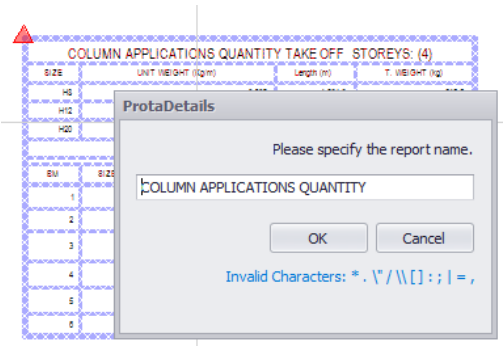
25. Multiple selection of Tables for report generation

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

How To Use:

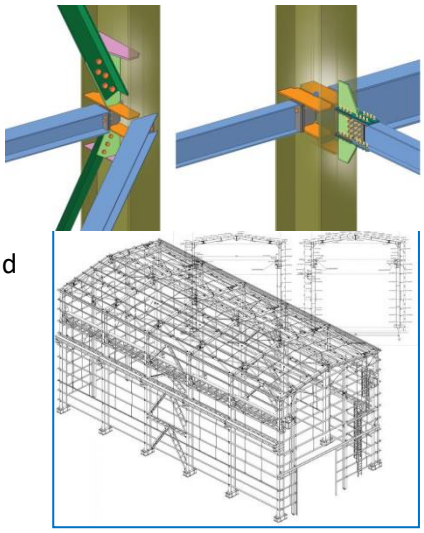
- ***Select** a table or multiple select tables (hold down **CTRL**)*
- *Right-click → **Add to Report***
- *Give it a title → **OK***
- *The report will be created*

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



26. ProtaSteel Introduction

ProtaSteel is a 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.

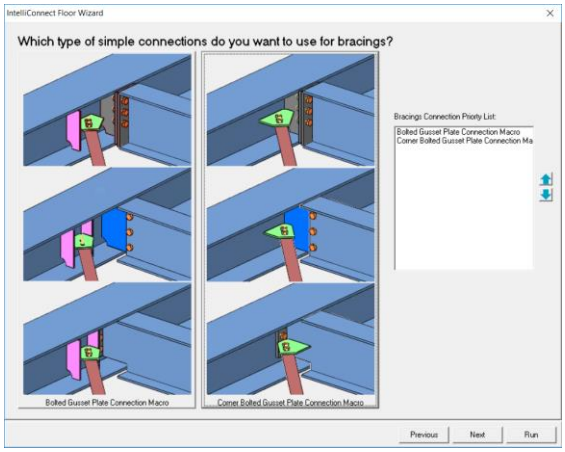
How To Use :

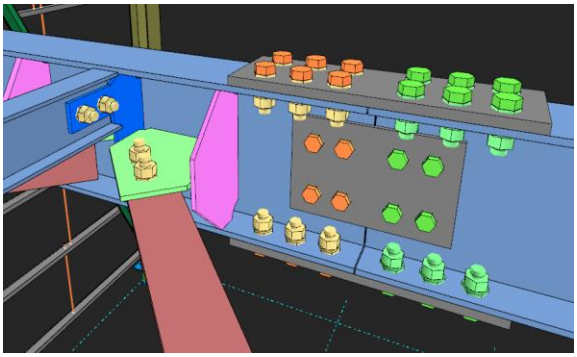
- ➢ *Model, analyze & design the steel structure in ProtaStruture*
- ➢ *Go to **Steel Design** (top menu) → **Load ProtaSteel***

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

27. Intelliconnect for fully automated steel connection design

IntelliConnect automatically assesses joints to your preferences, batch creates and designs steel connections. It covers a wide range of simple, moment, splice and weld connections. It considers constructability by intelligently arranging and configuring all the components of the connections without user intervention. This results in huge improvement in productivity as it minimizes time required to manually create & correct these connections.





Why use IntelliConnect?

Advanced steel detailing software typically create connections automatically between two elements by applying certain rules, i.e. “Rule based joints”. These rules normally include items such as names of the profiles, the existence of moment releases or lack thereof, the loads in the elements and approach conditions (whether the secondary is approaching the main from the flange side or the web side) etc.

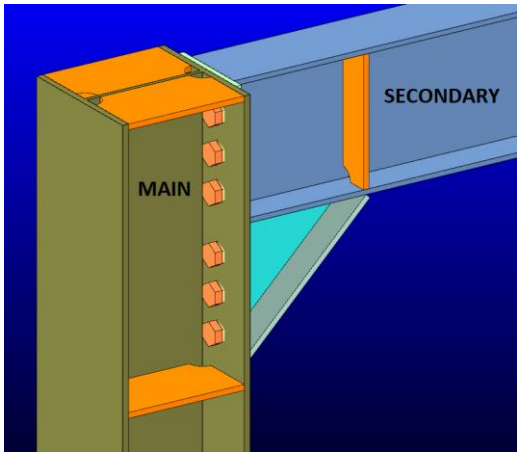


Figure 1

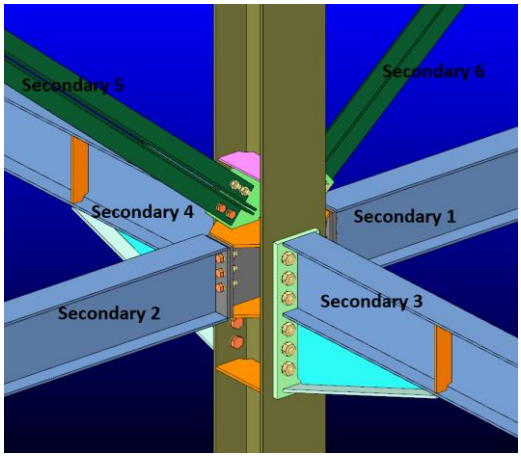
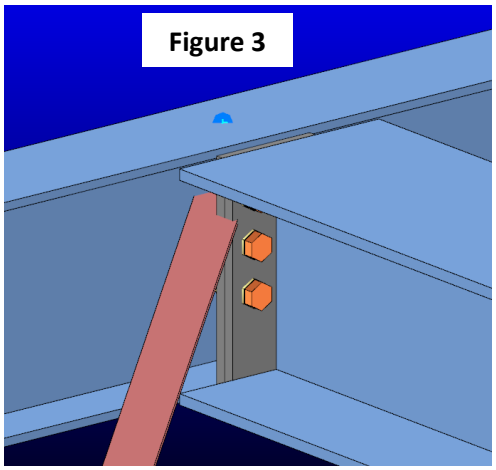


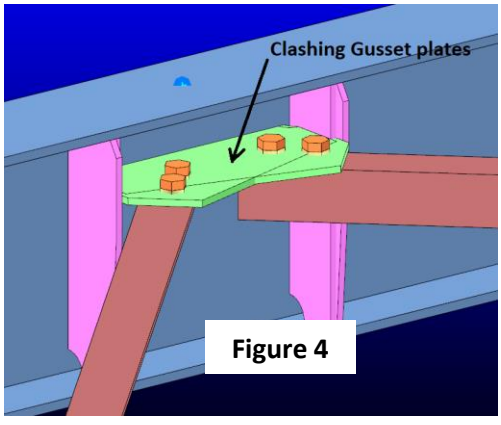
Figure 2

While this feature of "Rule based joints" is beneficial in simple cases where the main and the secondary (incoming profile) are the only two profiles present at a joint (see Figure 1 above), it fails to deliver for complex joints when more than two elements meet at the same joint (see Figure 2) :

- For complex joints, applying a blind "Rule based joint" would create a huge number of clashes, resulting in wasted & unproductive time in manual amendments.



- "Rule based joints" will not have the intelligence to "auto-shift" the secondary member slightly to facilitate a connection. Such requirements are commonplace in steel detailing when incoming members need to be shifted slightly to avoid clashes and make erection possible.
- A typical example (see Figure 3) would be the need to shift the horizontal bracing in flooring systems away from the beam-beam intersection points to avoid clashes with connection plate.



- "Rule based Joints" typically do not combine incoming elements to create a single common connection if the situation demands it.
- For example (see Figure 4), two horizontal braces will connect to the same point separately, creating a clash between the gusset plates, rather than using a single common gusset plate.

IntelliConnect is an advanced technology which attempts to address these issues in certain cases which are common in building detailing. These includes "simple cases", e.g. where there are only 2 primary elements and "complex cases" where more elements meet at the same node.

Cases covered

The following "**Simple Cases**" are covered by ***IntelliConnect*** :

1. Splice connection (Supports whatever profiles are supported by the Splice macro)
2. Column Base plate with no incoming vertical bracing. (The column needs to be an I section)
3. A single beam framing into a main beam. The Beams can be I or channel profiles. The type of beam to beam connection can be any one of the following 1) The Stiffend end plate connection. 2) The Fin plate connection 3) The beam to beam end plate connection.
4. A single beam framing into a column. The profiles may be I or channel profiles. The type of beam to beam connection can be any one of the following 1) The Stiffend end plate connection. 2) The Fin plate connection 3) The beam to beam end plate connection.
5. A single horizontal brace framing into a beam. The horizontal brace may be any profile supported by the Bolted gusset or welded gusset macros)
6. A single vertical brace framing into a column. The vertical brace may be any profile supported by the bolted gusset or welded gusset macros.
7. A single purlin sitting on a beam or truss element. The type of connection may be any connection supported by the Purlin/Girt macro
8. A single girt framing into a column. The type of connection may be any connection supported by the Purlin/Girt macro
9. Creation of batten plates on twin profiles.
10. Connection of a single purlin passing continuously over a truss top chord or roof beam

The following **complex cases** are covered by ***IntelliConnect*** :

1. Two horizontal braces framing into a beam (from any side – web or flange)
2. Two vertical braces framing into a column (from any side – web or flange)
3. Truss connecting with more than one truss secondary framing into the truss top or bottom chord.
4. Two purlins sitting on a truss top chord or a roof beam
5. Two girts framing into a column (either colinear or one from each orthogonal direction)
6. Two floor beams framing into a main beam from opposite sides
7. Combinations of floor beams and horizontal braces framing into each other at the same point from both sides
8. Up to 4 beams framing into a column (two from flange side and two from the web side)
9. Combination of beams and vertical braces framing into a column

As far as limitations on profile types and connection types are concerned, the same limitations given in the simple cases apply.

How to use

The following is the summary of how to ***Intelliconnect*** :

- The user will first select a single or multiple nodes which holds the information of all elements meeting at a single point.
- After which the user simply right clicks and goes into the ***IntelliConnect*** context sensitive menu and select the type of connections to create.
- For some ***IntelliConnect*** connections however the user needs to select frames and not nodes (e.g. simple cases 9 and 10 stated previously)
- ***Intelliconnect*** will its "internal classifying algorithm" to connect as many of the nodes selected by the user as possible.

- However, if it encounters a case which is not supported, it will either partially connect the node or leave it completely unconnected.

- After the process is finished the user can browse through these partially connected or unconnected nodes and connect/correct them manually.
- User can browse through these nodes one by one through a navigator.

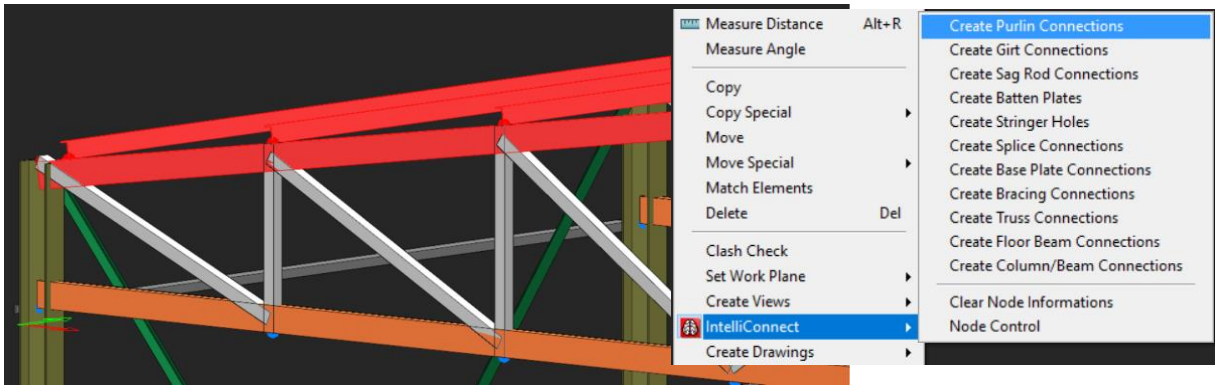
Purlin Example

- *Select the nodes (joints) of the steel structure where you wish to create connection*

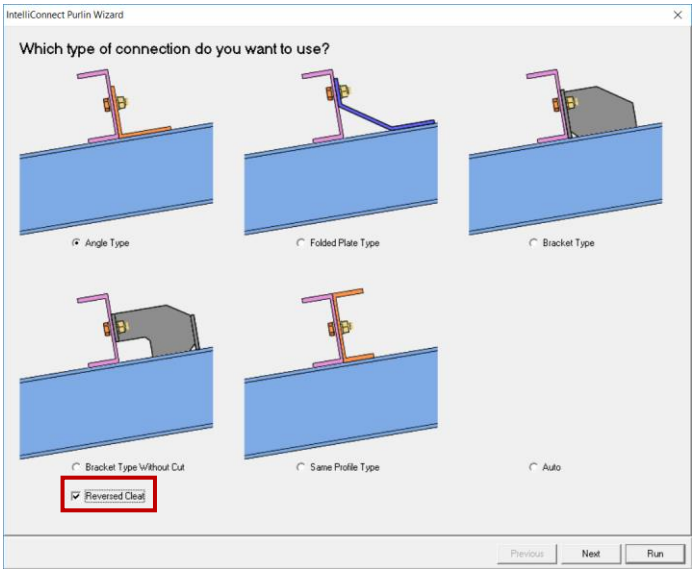
Joints are shown as blue dots at the end of the member. For simplicity, you can click and drag a box to select all members which will then also select the nodes.

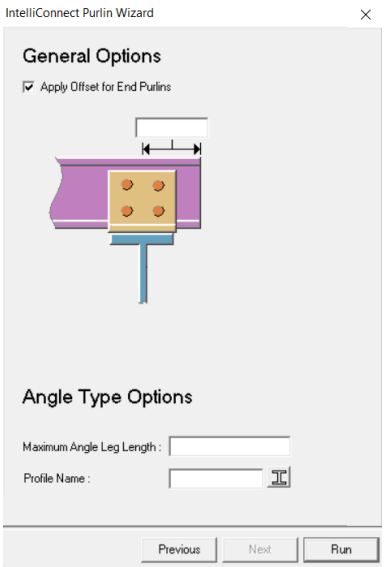
For illustration purpose, we will create the purlin connections based on the Quick Start Guide Steel model.

- *Select all the truss members and purlins by left-click a drag a box around them*

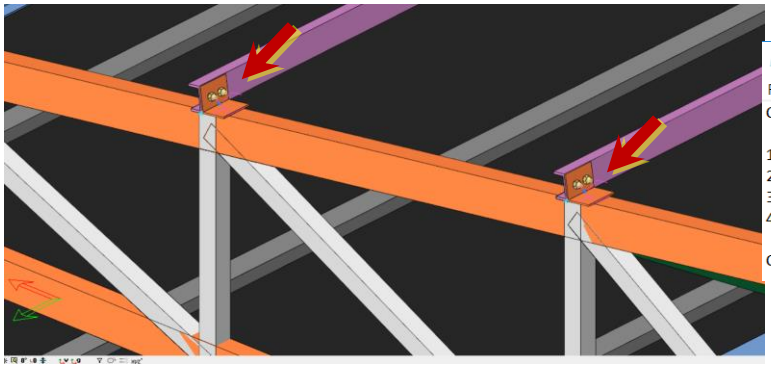


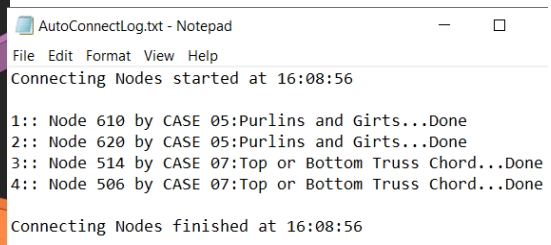
- *Right-click → **IntelliConnect** → Pick **Create Purlin Connection***





- *In the Purlin Wizard, choose the preferred type of connection*
- *For this example, choose "Angle Type" & "Reversed Cleat"*
- *Input further parameters in the next wizard dialog → **Run***
- *AutoConnectLog text file will open → Review its contents as it lists all the connections that are successfully / unsuccessfully created*

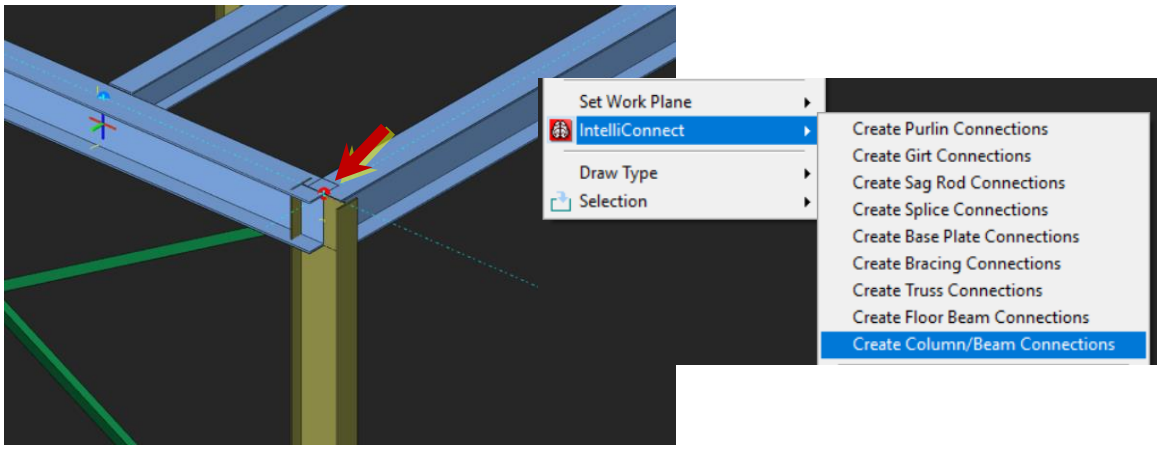




How To Use (Beam – Column – Brace Example)

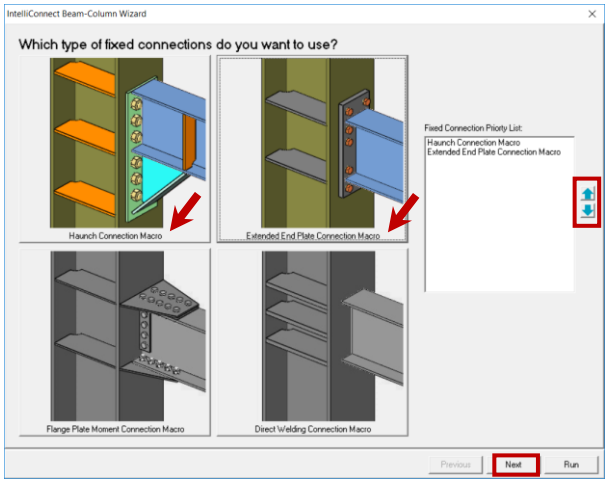
For illustration purpose, we will auto create a beam – column – brace connection based on the Quick Start Guide Steel model.

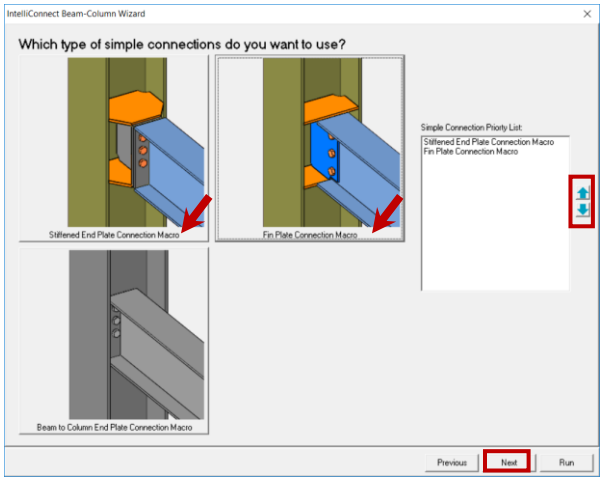
- *Select the joint where beam, column and brace meets*



- *Right-click →* ***IntelliConnect*** *→ Pick* ***Create Column/Beam Connection***

Where there is multiple possible connection configuration, the subsequent dialog allows you to choose which connection type to consider and it's priority.

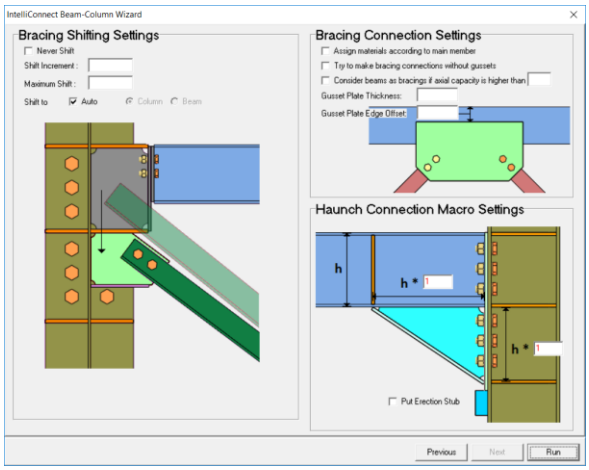


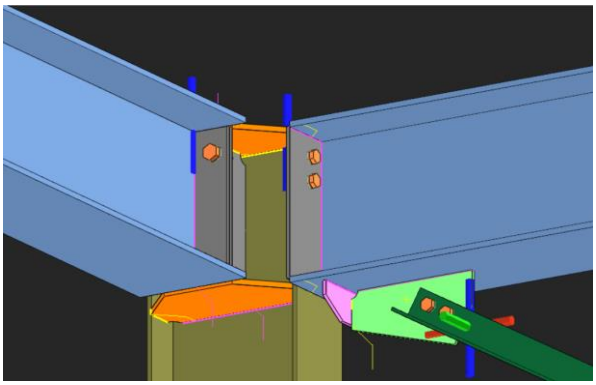


- *Pick the connection types to consider → Adjust the priority using up/down arrow →* ***Next***

The relevant bracing shifting, bracing connection & haunch connecting settings will appear next

- *Make desired adjustment →* ***Run*** *→ Connection will be created*



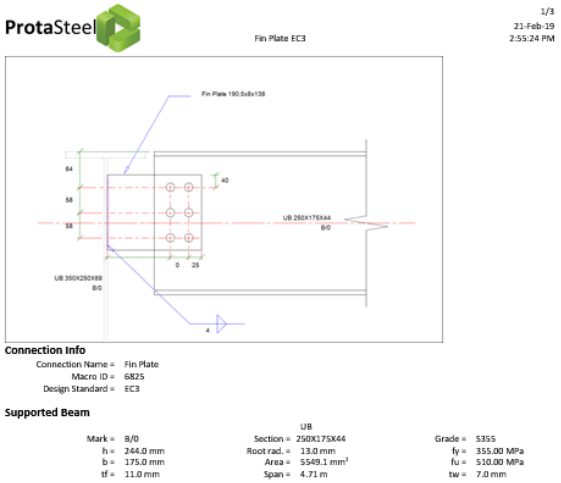


28. Connection Design Reports

ProtaSteel 2019 now creates detailed design check reports for the following connections in accordance to EC3, AISC (LRFD), AISC (ASD), and BS5950 :

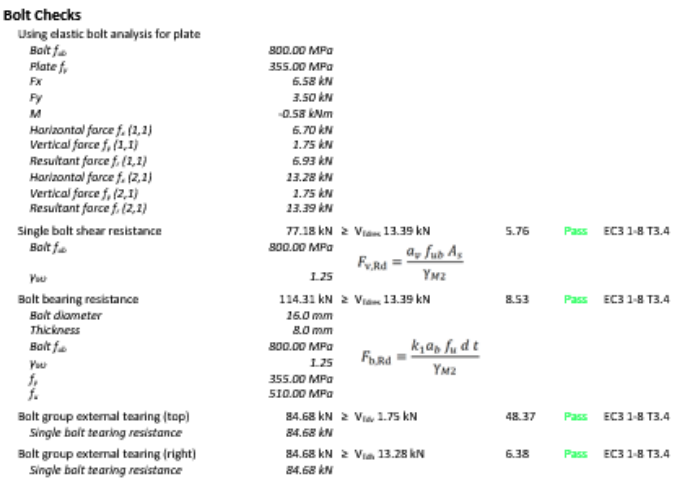
- Beam to beam Connections : End Plate, Stiffened End Plate, Fin Plate
- Beam to Column Connections : End Plate, Stiffened End Plate, Fin Plate
- Simple base plate connections

When a connection is inserted, first a guideline design is performed. Then, the rigorous design checks are done according to selected code clauses. Reports include both geometric checks and capacity checks. In addition, intermediate calculations, references to the appropriate clauses of the design code, and corresponding equation are displayed for easy checking.


ProtaSteel Fin Plate EC3 1/3 21-Feb-19 2:55:24 PM

Connection Info
Connection Name = Fin Plate
Macro ID = 6825
Design Standard = EC3

Supported Beam
Mark = B/0 | UB | Grade = S355
h = 244.0 mm | Section = 250X175X44 | fy = 355.00 MPa
b = 175.0 mm | Root rad. = 13.0 mm | fu = 510.00 MPa
tf = 11.0 mm | Area = 5549.1 mm² | tw = 7.0 mm
Span = 4.71 m


Bolt Checks
Using elastic bolt analysis for plate

Bolt f_{ub}	800.00 MPa					
Plate f_y	355.00 MPa					
Fx	6.58 kN					
Fy	3.50 kN					
M	-0.58 kNm					
Horizontal force f_x (1,1)	6.70 kN					
Vertical force f_y (1,1)	1.75 kN					
Resultant force f_r (1,1)	6.93 kN					
Horizontal force f_x (2,1)	13.28 kN					
Vertical force f_y (2,1)	1.75 kN					
Resultant force f_r (2,1)	13.39 kN					
Single bolt shear resistance	77.18 kN	≥	V_{Edv} 13.39 kN	5.76	Pass	EC3 1-8 T3.4
Bolt f_{ub}	800.00 MPa	$F_{v,Rd} = \frac{\alpha_v f_{ub} A_s}{\gamma_{M2}}$				
γ_{M2}	1.25					
Bolt bearing resistance	114.31 kN	≥	V_{Edv} 13.39 kN	8.53	Pass	EC3 1-8 T3.4
Bolt diameter	16.0 mm					
Thickness	8.0 mm					
Bolt f_{ub}	800.00 MPa	$F_{b,Rd} = \frac{k_1 a_b f_u d t}{\gamma_{M2}}$				
γ_{M2}	1.25					
f_y	355.00 MPa					
f_u	510.00 MPa					
Bolt group external tearing (top)	84.68 kN	≥	V_{Edv} 1.75 kN	48.37	Pass	EC3 1-8 T3.4
Single bolt tearing resistance	84.68 kN					
Bolt group external tearing (right)	84.68 kN	≥	V_{Edh} 13.28 kN	6.38	Pass	EC3 1-8 T3.4
Single bolt tearing resistance	84.68 kN					

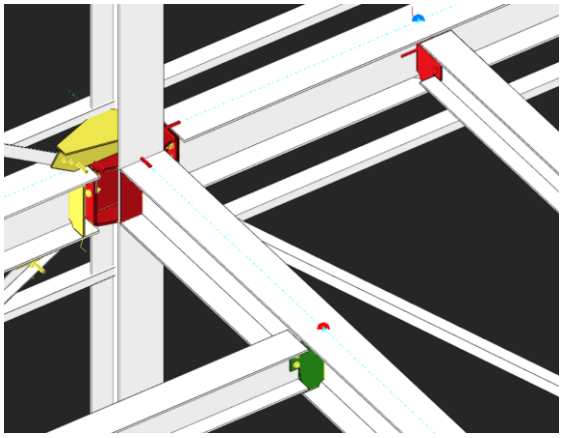
The reports can be created on demand in rich text format (RTF) and can be added to ProtaStructure's report manager for compilation into final design documentation. Design Status colors can be seen on model view before report creation.

How To Use

To show the utilization ratio of the connection visually:

- *Go to **View** (top menu)* → ***Connection Design Ratio Coloring***

The connection will color coded : **Yellow** = Not checked/designed; **Green** = Pass; **Red** = Fail.



To check the connection & generate the report :

➢ *Select the connection by any of these methods :*

- *Select the connection **macro sign***
- *Select any component of the connection (eg. a plate or a bot) → then press "**M**"*
- *Left click and drag a window enclosing the entire connection*

TIP : Hold down **CTRL** to multiple select

➢ ***Right-Click** → Connection Design Reports (or **Go to Drawings & Report** (top menu) → Connection Reports) → Select the desired report :*

- **Show Connection Report** → Show connection report of selected macro (s)
- **Default Connection Report** → Create connection report according to project preference
- **EC3 / BS5950 / AISC_LFRD /AISC_ASD Connection Report** → Create connection report for the selected code

29. Connection Design Summary Table

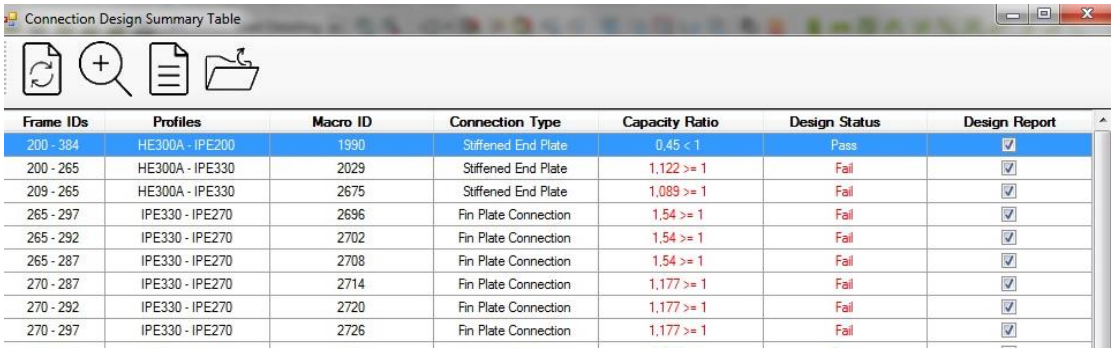
A connection design summary table presents an overview of selected connection macros. This includes "Frame Id", profile types of macro dependents, connection type, capacity ratio of connection, design status according to capacity ratio and whether design report of connection is created.

How To Use

➢ *Select one or more connection macro*

If no macro is selected, the table will show all supported macros in the model

➢ *Go to **Drawings & Reports** (top menu) → **Connection Reports** → **Connection Design Summary Table***



Frame IDs	Profiles	Macro ID	Connection Type	Capacity Ratio	Design Status	Design Report
200 - 384	HE300A - IPE200	1990	Stiffened End Plate	0,45 < 1	Pass	☑
200 - 265	HE300A - IPE330	2029	Stiffened End Plate	1,122 >= 1	Fail	☑
209 - 265	HE300A - IPE330	2675	Stiffened End Plate	1,089 >= 1	Fail	☑
265 - 297	IPE330 - IPE270	2696	Fin Plate Connection	1,54 >= 1	Fail	☑
265 - 292	IPE330 - IPE270	2702	Fin Plate Connection	1,54 >= 1	Fail	☑
265 - 287	IPE330 - IPE270	2708	Fin Plate Connection	1,54 >= 1	Fail	☑
270 - 287	IPE330 - IPE270	2714	Fin Plate Connection	1,177 >= 1	Fail	☑
270 - 292	IPE330 - IPE270	2720	Fin Plate Connection	1,177 >= 1	Fail	☑
270 - 297	IPE330 - IPE270	2726	Fin Plate Connection	1,177 >= 1	Fail	☑

The top icons from left to right :

- **Update** → Refreshes the table
- **Show in Model** → Locate & zoom in the selected macro in the model view. Double-clicking on any row will also perform the same function.
- **Create Report** → Create the connection design report of the selected macro. Hold down CTRL key to multiple select.
- **Open Report** → Open the selected design report

30. Design Overrides

This feature allows users to impose minimum parameters to a connection if these standards are higher than auto-design results. For example, minimum number of bolts can be specified. This will result in practical & constructible connections; saving many hours of design review and amendments.

- In many cases the design shear in simple steel beams are quite low (since the beam size is usually governed by span moments). The auto-design of the connection will likely result in very few numbers of bolts, very small plate thicknesses and / or very small weld sizes as these still passed recommended design formulas.
- Obviously, such design is not acceptable. Typically, the steel detailer will increase the number of bolts or plate thickness or weld size to a reasonable number. However, such manual intervention is unproductive and defeats the purpose of automated design.
- To resolve this issue, the ProtaSteel provides an override file to specify minimum parameters such as number of bolts and plate thickness.
- This feature applies to the following connection macros :
  1. Stiffened End plate macro (Both for beam to beam and beam to column connections)
  2. Fin plate macro (Both for beam to beam and beam to column connections)
  3. End plate macro (Both for beam to beam and beam to column connections)

How To Use

The design overrides are defined in text files in the preference folder :

- *Click on '**Open Preferences Folder**' (top menu)* 
- *Open the file "Simple Beam Connections Design Overrides.txt"*

The structure of this file is as shown below.

Simple Beam Connections Design Overrides - Notepad

File Edit Format View Help

```
#Version=1.1
#! EP:End Plate Macro, SEP:Stiffened End Plate Macro, FP:Fin plate macro
#! BD: Bolt diameter, BN: Number of Bolt
#! PT:Plate Thickness,PT:Plate Thickness of Stiffener
#! WSW: Weld Size on Web, WSF: Weld Size on Flange
#!

Profil  EP_BD  EP_BN  EP_PT  EP_WSW  SEP_BD  SEP_BN  SEP_PT  SEP_PTS  SEP_WSW  FP_BD  FP_BN  FP_PT  FP_WSW
IPE80   16     2      8      3       16      2       8       8        3        16     2      8      3
IPE100  16     2      8      3       16      2       8       8        3        16     2      8      3
IPE120  16     2      8      3       16      2       8       8        3        16     2      8      3
IPE140  20     2      10     3       20      2       10      8        3        16     2      8      3
IPE160  16     4      10     3       16      4       10      8        3        16     2      10     3
IPE180  16     4      10     3       16      4       10      10       3        20     2      10     3
IPE200  16     4      10     3       16      4       10      10       3        20     2      10     3
IPE220  20     4      12     3       20      4       12      10       3        20     2      10     3
IPE240  20     4      12     3       20      4       12      10       3        16     3      12     3
IPE270  20     6      15     3       20      6       15      10       3        20     3      12     3
IPE300  20     6      15     4       20      6       15      12       4        20     4      12     3
IPE330  20     6      15     4       20      6       15      12       4        20     4      12     3
IPE360  20     6      15     4       20      6       15      12       4        20     4      12     4
```

Legend for parameter names

End Plate parameters

Stiffened End plate parameters

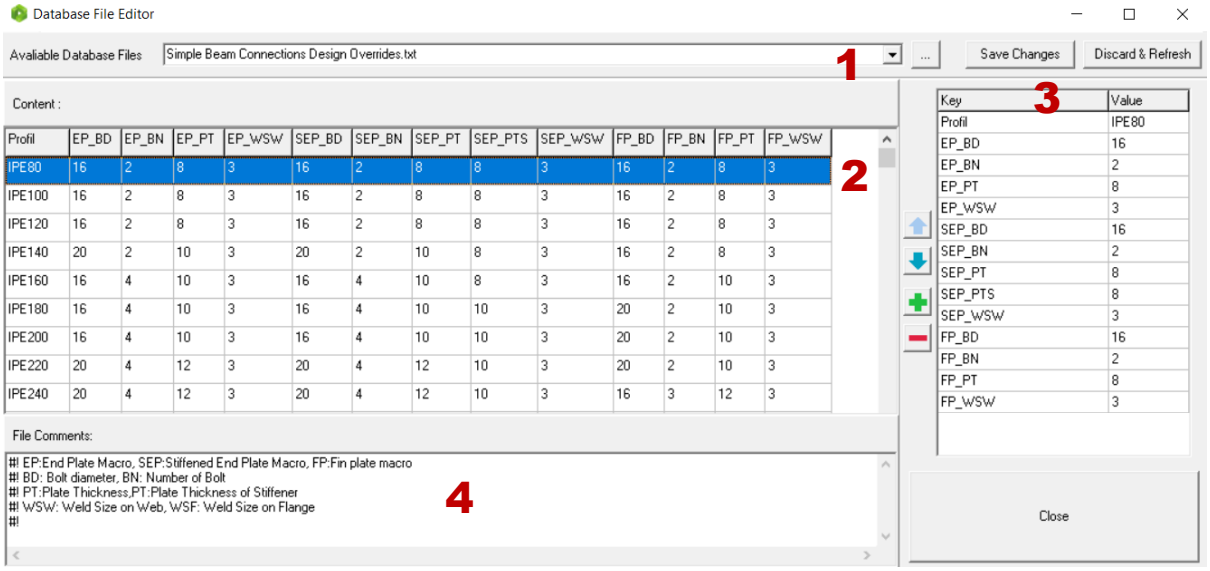
Fin Plate Parameters

- ***Modify** & **Save** this text file to force the program to use "higher" values of selected design parameters than those suggested by the design formulas.*

The program will only use these values if they are "higher" than the that proposed by the code calculations.

The **Database Editor** can also be used to edit the text file in ProtaSteel :

➢ *Go to **Tools → Databases → Profile Database Editor***



1. Database Files

- *Select the file to view or edit → 'Simple Beam Connection Design Overrides'*

2. Content

- *Select the row to edit*
- *To add new row, pick the*  *sign / To delete a row, pick the* *sign*

3. Key / Value

- *View or change the values of the key*
- *Saves Changes or Discard & Refresh*

4. File Comments

- *Read the notes to understand key abbreviations & how to use it*

31. Auto Presets

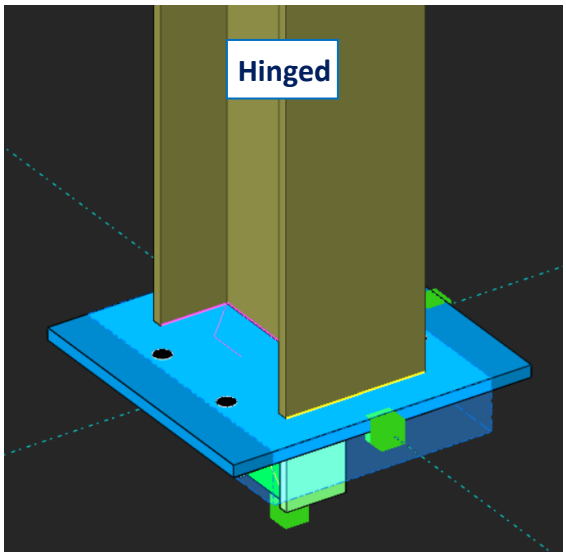
This feature enables the user to create relations between profile types and connection macro presets. This allows the program to automatically apply a saved "preset" connection to a particular profile section.

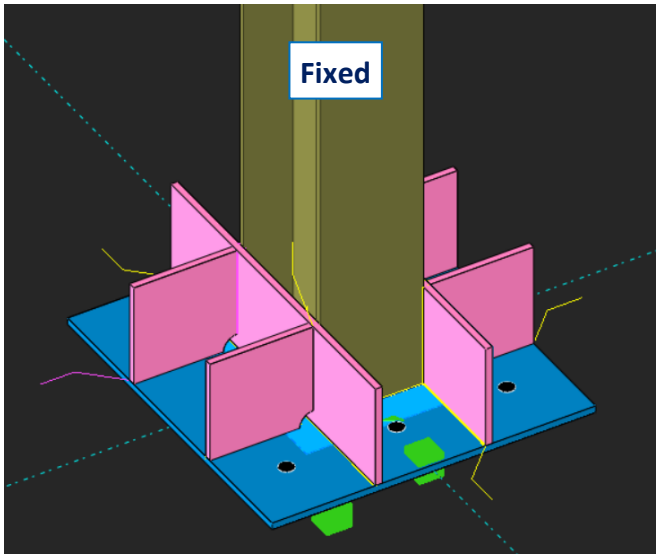
- ❖ All connections macro supports "Macro Presets" where user can save a particular connection parameters & configuration and reuse it elsewhere without having to enter all the values again.
- ❖ For example, for the haunch connection, the user may want to save different parameters for each profile that is commonly used, example for UC152x152, UC203x203 & UC254x254.
- ❖ However, as the number of presets increases, it becomes increasing difficult & cumbersome to manage as the user still need to recall the correct preset to apply with every connection macro.

To resolve this issue, the "Auto-Preset" feature is developed make application of these pre-sets automated; thus increasing productivity and eliminating errors.

How To Use

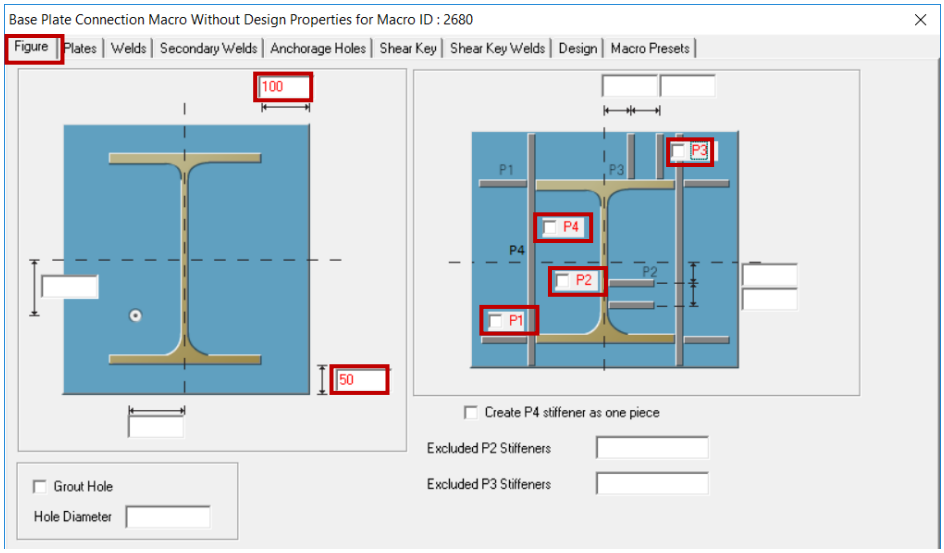
For illustration, we will use the connection macro '**Column base plate without design**'. The objective is set up a fixed column base plate for a fixed base and hinged column base plate for a hinged base.

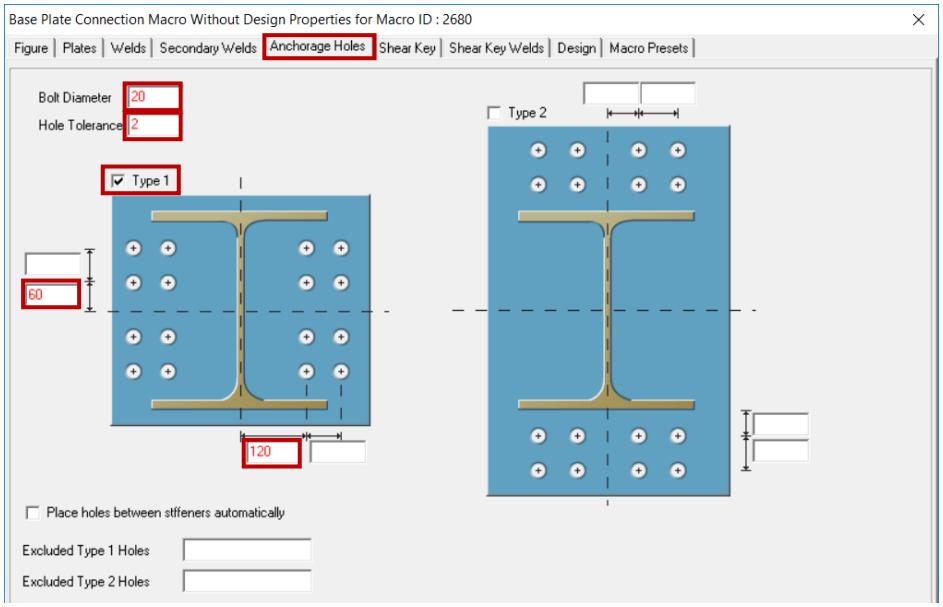




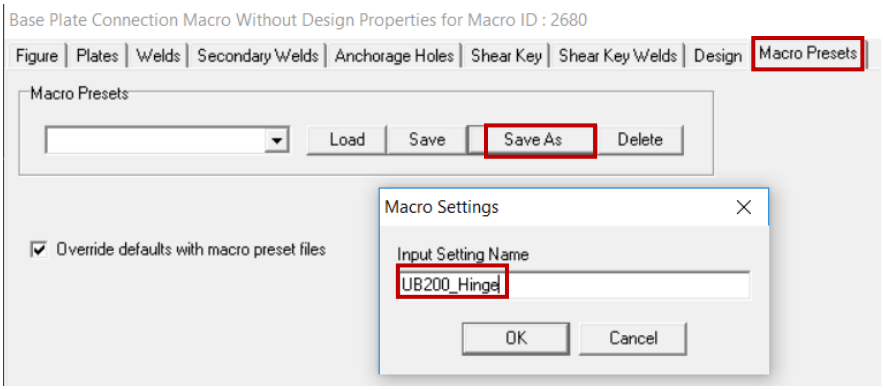
Note : This connection macro does not perform any design – it's is entirely up to the user to set up this connection.

➢ ***Right-Click** on '**Column Base Plate Macro Without Design**'* → *Input/change as below*



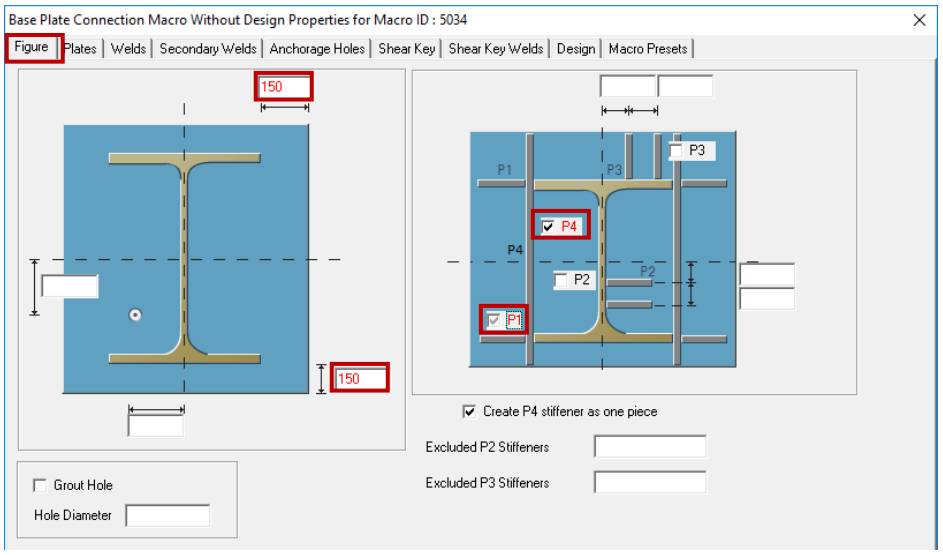


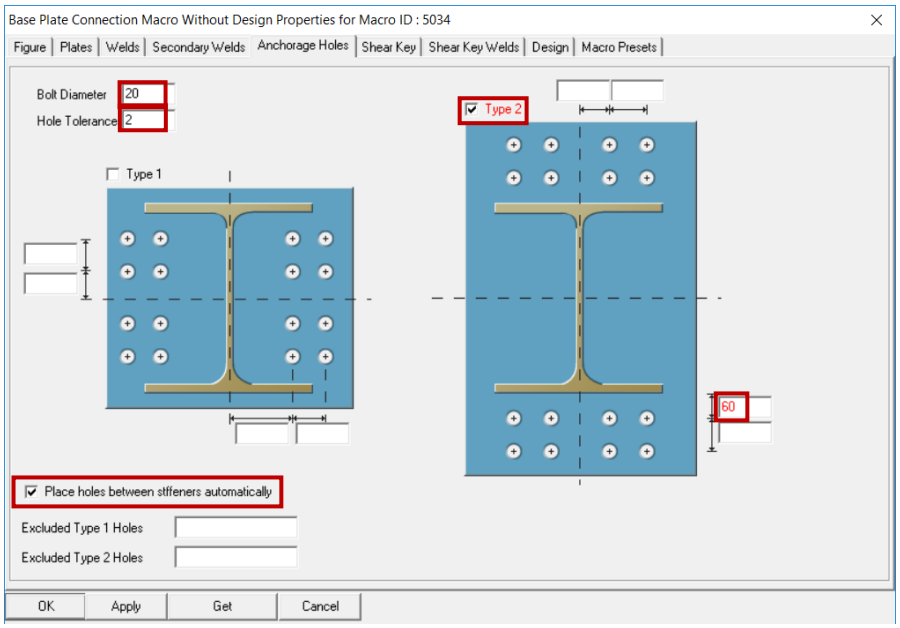
➢ *Go to Maco Preset tab →* ***Save As →Type "UC_Hinge"*** *→ Close the dialog*



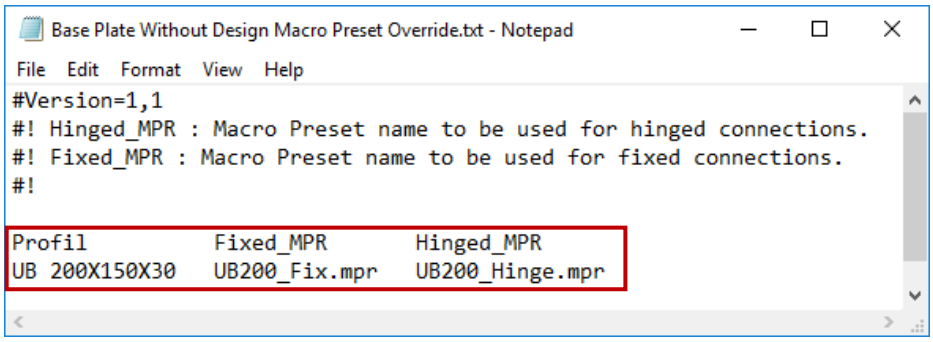
The intention is this macro connection will be automatically used for column with this UB profile and hinged at base.

➢ ***Right-Click*** *on 'Column Base Plate Macro Without Design' → Input / change as shown*





- *Go to **Macro Preset** tab → **Save As** → **Type "UB_Fixed"** → Close the dialog*
- *Click '**Open Preferences Folder**' (top menu)*
- *Double-click on file '**Base Plate Without Design Macro Preset Override.txt**' to open it*
- *Ensure the highlighted part is typed as shown below. Use "tab" to as separator.*



The above means that whenever there is a UB200x150x30, use macro preset "UB200_Fix" if the base is fixed and use "UB200_Hinge" if the base is hinged.

- *Go back the 3D view → Double-click on one of the column base joint*

In the Joint dialog, it shows that the joint in restrained in both translation (U1,U2,U3) as well as rotation (R1, R2, R3).

The joint restraint is directly transferred from ProtaStructure.

To change the joint to hinge :

- *Uncheck "R1", "R2" & "R3"*
- *Close the Joint dialog*

- ***Click** 'Column Base Plate Macro Without Design' → Select the column above*

The hinge column base connection will be created.

- ***Click*** *'Column Base Plate Macro Without Design'* → *Select another column with default fixed restraint.*

The fixed column base connection will be created.

TIP : The preset files can also be edited via ***Tools → Databases → Profile Database Editor***

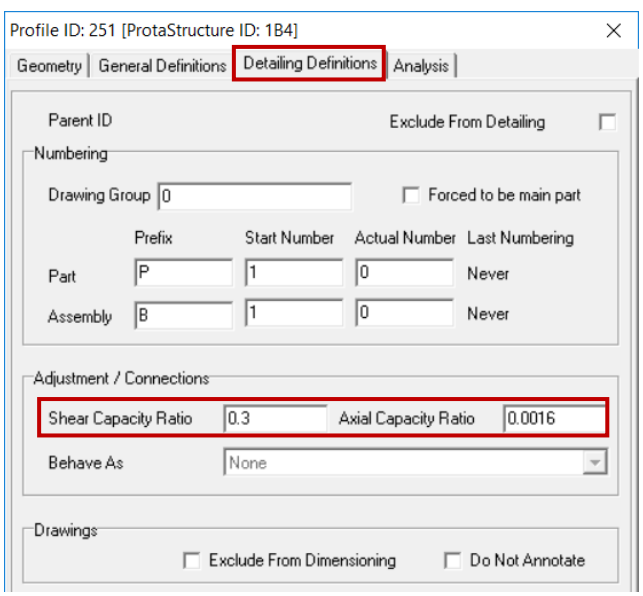
Each macro preset is saved with an extension of "mpr". Each time a preset is saved, a file with an extension of mpr is created in the relevant directory.

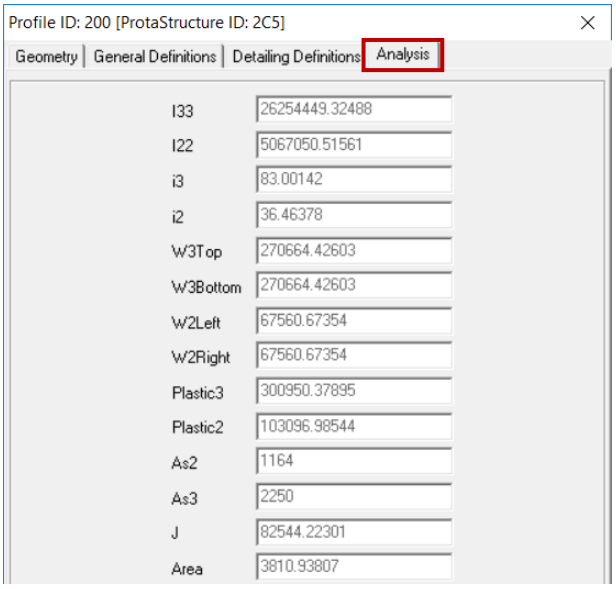
32. User-defined Design Capacity Ratios

User can now input shear and axial capacity ratio for connection design (design force / by maximum member capacity). These ratios will be used for design if actual shear or axial forces imported from ProtaStructure analysis are smaller. Specifying capacity ratios prevents unrealistic connection design, e.g. 1 no. of bolt when actual design forces are exceeding small.

How to use

- ***Double-click*** *the member is critical to the connection design*
- *Go to* ***Detailing Definitions*** *tab in Profile dialog → Input Shear / Axial Capacity Ratio*

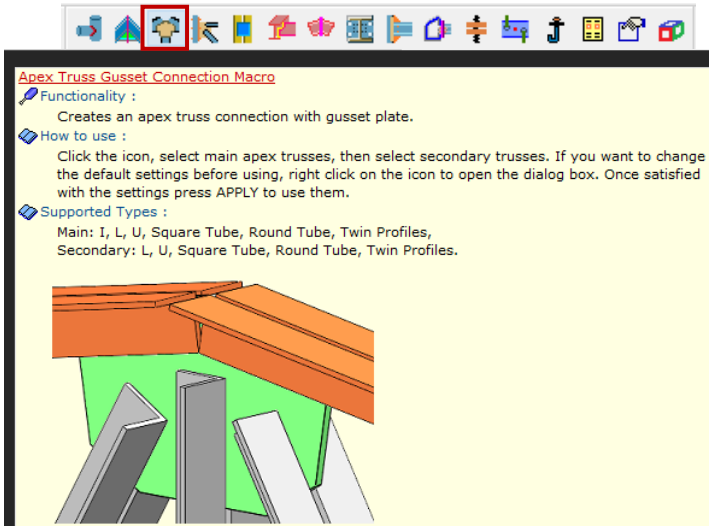




- *Go to* ***Analysis*** *tab to check the geometric properties from ProtaStructure*

33. Truss Apex Gusset Connection

Apex Truss Gusset Connection macro creates & designs a connection between two top chords and secondary truss elements with gusset plate. This macro is available from the Secondary Connections toolbar.



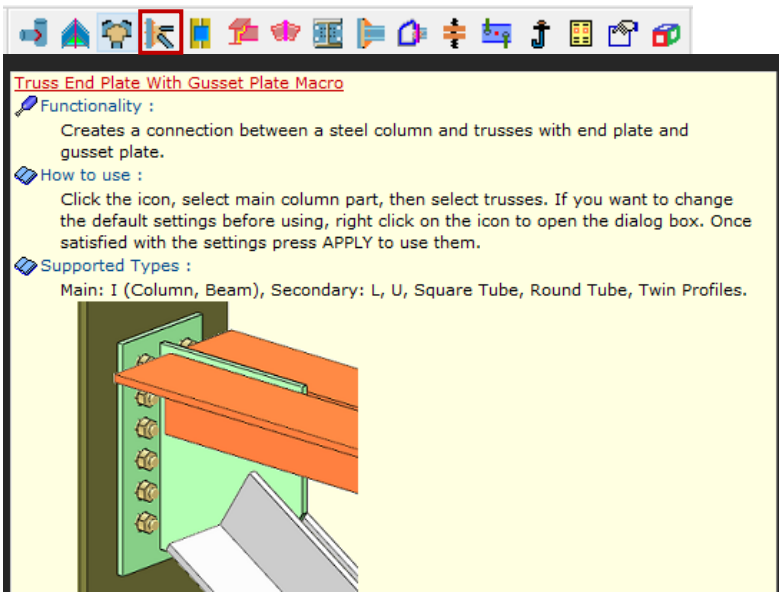
This macro supports connection with 1 to 3 secondary profiles. These secondary profiles can be welded or bolted, and their profiles and orientations can be different. However, profile type and orientation of top chord elements need to be the same. Selection order of secondary profiles is also important such that it determines which elements are pulled back and in what order.

Cases Covered

Apex Truss Gusset Connection Macro supports to cases that connected by several apex and secondary profile types. For secondary trusses; L, U, square tube, round tube and twin profiles are supported. For Main apex trusses; in addition to secondary profile types I profile is supported.

34. Truss End Plate With Gusset Plate

This macro creates and designs a connection between truss ends and steel columns using end and gusset plates. This macro is available from the Secondary Connections toolbar.



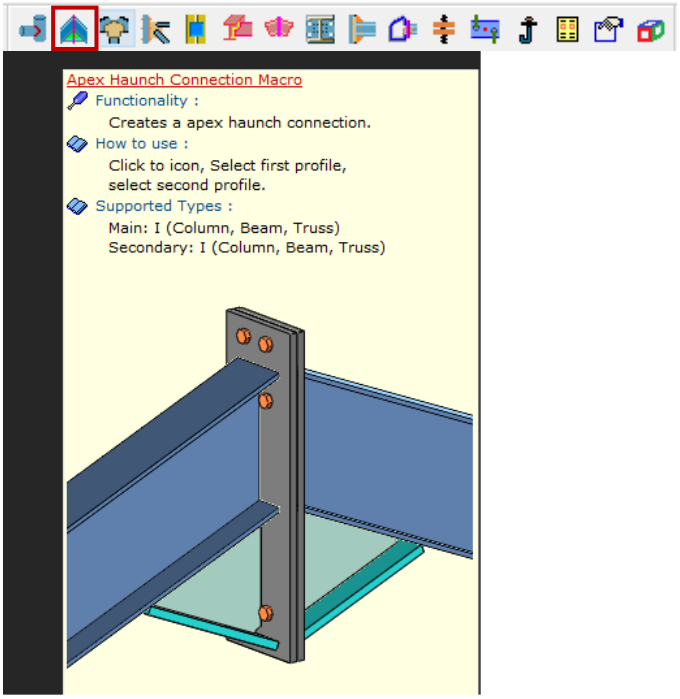
Secondary profiles are connected to gusset plate with welded connection and their section type and orientation can be different. Main column profile and end plate are connected with bolted connection. Selection order of secondary profiles is imported. Upper secondary profile needs to be selected first.

Cases Covered

Truss end plate with gusset plate macro supports to cases that connected by several secondary profile types. For secondary trusses; L, U, square tube, round tube and twin profiles are supported. For Main column; I profile is supported.

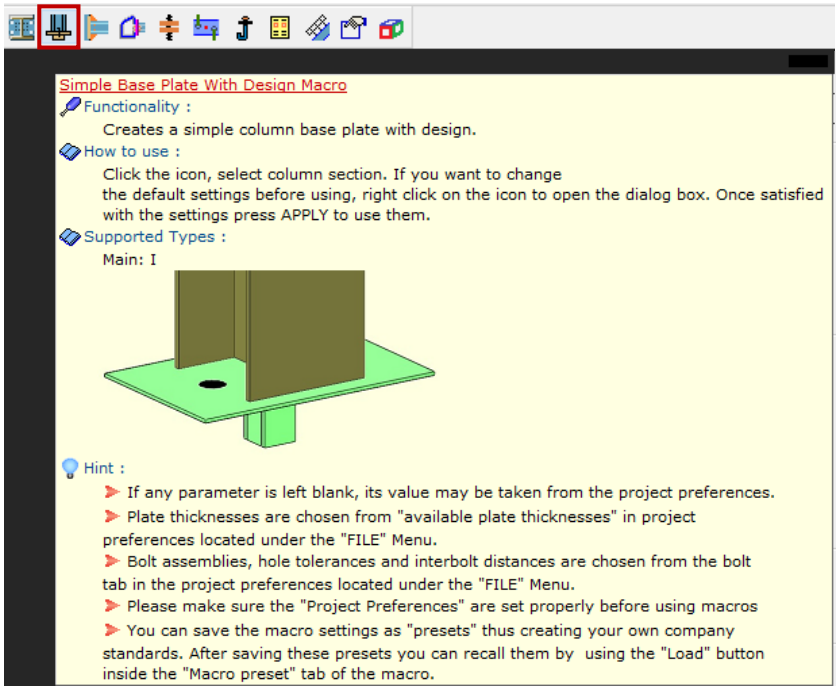
35. Truss Apex Connection Macro

This macro creates a welded Apex connection without design using “I” profiles. This macro is available from the Secondary Connections toolbar.



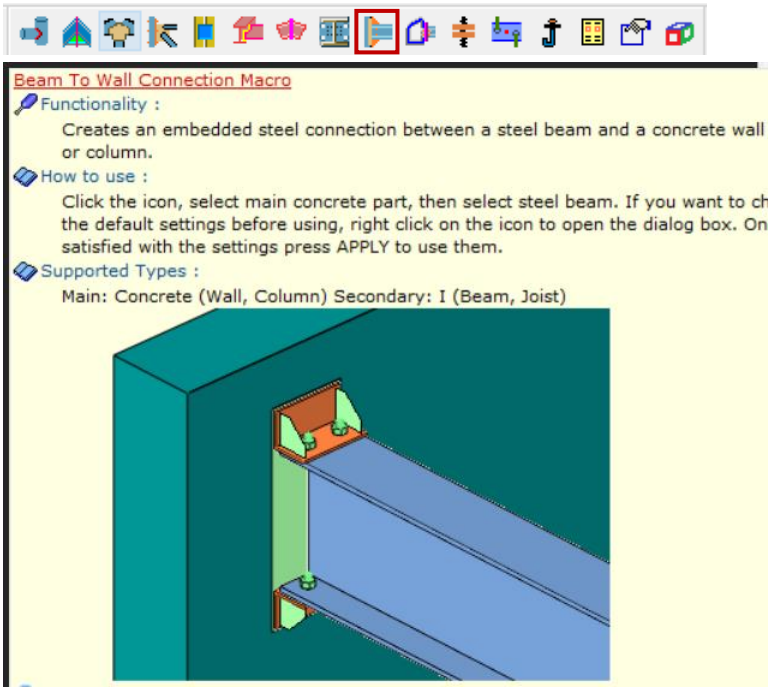
36. Simple Base Plate Connection

This connection macro creates a simple column base plate connection. This macro is available from the Secondary Connections toolbar.



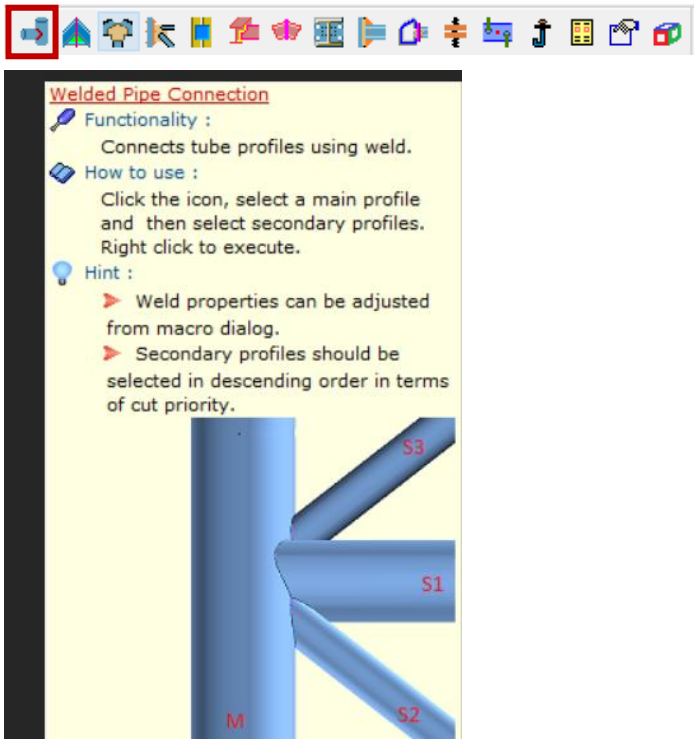
37. Beam to (RC) Wall/Column Connection

This connection macro creates & designs an embedded steel connection between a steel beam and a concrete wall or column. This macro is available from the Secondary Connections toolbar.



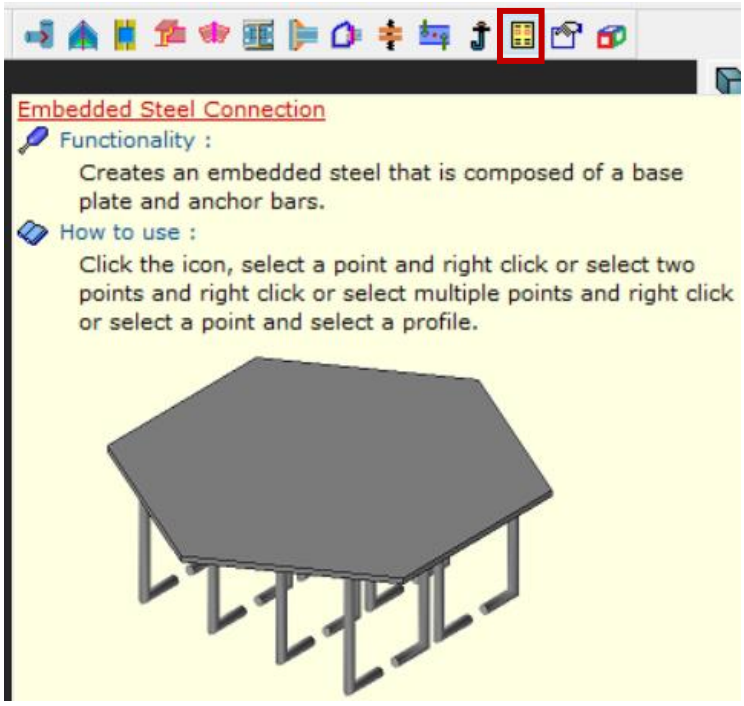
38. Welded Steel Pipe (Circular Hollow Sections) Connection

This macro creates steel tubes connections using welds. This macro is available from the Secondary Connections toolbar.



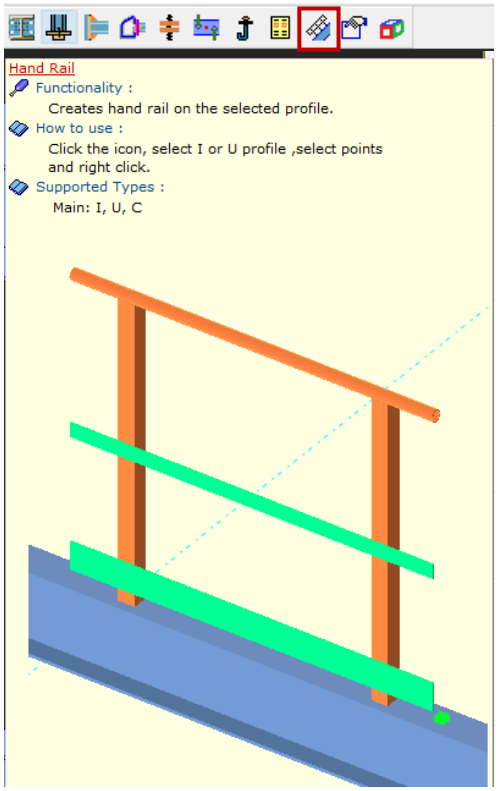
39. Embedded Steel Macro

This 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).



40. Hand Rail Macro

This connection macro creates hand rails on I, U & C beam profiles. This macro is available from the Secondary Connections toolbar.



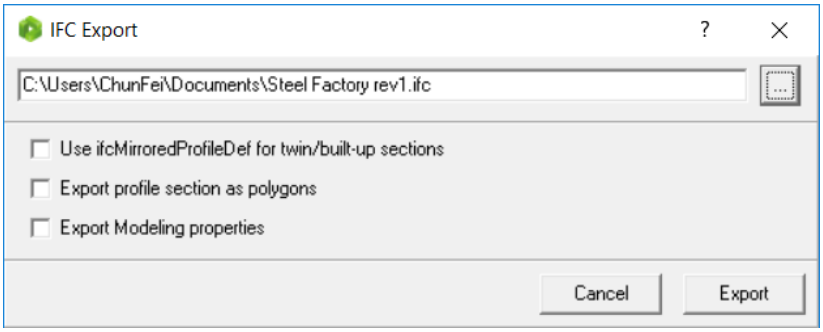
41. IFC File Export

ProtaSteel models can now be exported to IFC format. This facilitates collaboration with many other leading Building Information Modelling (BIM) platforms including Autodesk Revit, ArchiCAD, Bentley and Tekla Structure and reinforces Prota's standing as a leading provider of advanced BIM technology.

How to use

- ***Select objects*** *to be exported as IFC file format.*

  If no object is selected, then all objects will be exported.
- *The* ***File*** *→* ***Export*** *→* ***IFC Export***



- *Click '**...**' to specfy the file name and location to export*
- *Check the desired export options →* ***Export***

42. ProtaStructure Enhancement & Fixes

- ✓ Resolved out of memory issue for large models.
- ✓ Various enhancement of pilecap design to arrive at an economical solution
- ✓ Steel member design is enhanced to overcome some reported issues of crashes & inconsistencies
- ✓ “D” square mesh (fabric) added with all commonly used bar diameters
- ✓ Cannot insert beams when axis of insertion is exactly at global origin
- ✓ Rigid link erroneously generated at the bottom of discontinuous mesh walls – Fixed

43. ProtaDetails Enhancement & Fixes

- ✓ Undo & redo functionality introduced for all Cad and Intelligent Cad operations
- ✓ Drawing sheets with horizontal title blocks at the bottom added
- ✓ Leader size and presentation added in Options > Rebar Labels
- ✓ Slab design settings does not transfer consistently to ProtaDetail - Fixed
- ✓ Similar beams listed wrongly in concrete rebar schedules – Fixed
- ✓ Beam Quantity table not scaled with drawing scale – Fixed

44. ProtaSteel Enhancement & Fixes

- ✓ Member labels are now exported in “User Defined 3” field in CDFX file. These labels are used in Connection Report generation for better user experience. These fields can be annotated (in the model) using "Edit > Create Annotation" field.
- ✓ Splice connection is now working for truss elements as well.
- ✓ ProtaStructure element ID numbers are now displayed in Frame Element Property Editor title bar and in “General Definitions” tab. This ID is also used in Connection Design Reports to refer to frame elements.
- ✓ DXF files can now be imported into model
- ✓ Slanted columns imported with wrong rotation - Fixed
- ✓ Geometry of object is not valid error during import of ProtaStructure model – Fixed
- ✓ Anchor bolt, shear key, and rag bolt types are added to Embedded Steel Connection macro.
- ✓ After the first export from ProtaStructure to ProtaSteel, when an object is selected, the view was zooming out to model extents - Fixed

45. ProtaBIM Enhancement & Fixes

- ✓ ProtaBIM now recognizes and uses built-in Revit families for standard shapes. (Rectangular Concrete Column, Beam, UC Column, UB Beam, CHS, RHS, SHS etc.)
- ✓ Family subtypes are extended in the template. Users can see more types under a certain famiy (only for built-in)
- ✓ Abandoned ProtaStructure families are also recognized, in case an existing model is using them.
- ✓ Usage of standard families offers flexibility, so that users do not have to stick with ProtaStructure families if the model is started from Revit.

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