

ProtaSteel[®] 2021

powered by **comosys** 



Quick Start Guide

For support & training please contact

Support : asiasupport@protasoftware.com
globalsupport@protasoftware.com

Training : asiasales@protasoftware.com
globalsales@protasoftware.com

www.protasoftware.com

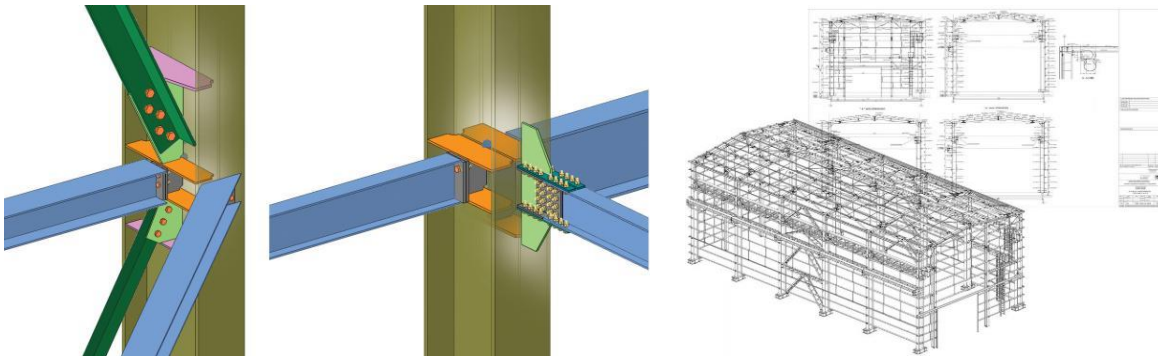
1. Introduction	4
2. Exporting model to ProtaSteel	4
3. User Interface	5
4. Zoom & Pan Methods	6
5. Selection Methods	6
6. Views & View Settings	6
7. Creating new Views	7
8. Visibility & Selectability	8
9. Macros	9
10. Insert Sag rods	10
11. Adjustment of members by creating points	12
12. Adjustment of members orientation & coordinates	13
13. Steel Connections	15
14. Base Plate Connection	17
15. Fin Plate Connection	18
16. Beam to Beam End Plate Connection	20
17. Haunch Connection	20
18. Stiffened End Plate Connection	21
19. Corner Bolted Gusset Plate Connection	21
20. Bolted Gusset Plate Connection	22
21. Splice Connection	23
22. Truss End Plate With Gusset Plate	24
23. Welded Pipe Connection (info only)	25
24. Beam to (RC) Wall/Column Connection	25
25. Intelliconnect For Fully Automated Steel Connection Design	26
26. Manual Connection	36
27. Connection Design Reports	43
28. Connection Design Summary Table	45
29. Collision / Clash Checks	46
30. Audit Model	47
31. Detail Item Manager	47
32. Preparing Engineering Drawings	50
33. Drawing Module	50
34. Creating Drawings via Drawing Manager	50
35. Visibility and Selectability in Drawing Module	54
36. Profile Annotation	56
37. Dimension	57
38. Creating Connection Detail	58

39. Numbering Objects	61
40. Creation of Parts and Assembly Drawings	63
41. Import Titleblock using DXF Import	64
42. DXF Export	65
43. IFC File Export	66
44. Material Take off	67
45. Closing Summary	69

1. Introduction

Thank you for choosing **ProtaSteel** – Steel detailing & connection creation module of the ProtaStructure suite. The main steel model must be created in ProtaStructure. The model can be exported & opened in ProtaSteel seamlessly. In ProtaSteel, you can :

- Model ancillary/secondary steel members such as roof purlins, girts, sag rods, staircase, etc.
- Adjust members to final positions for accurate detailing
- Create and design steel connections
- Create any steel connections manually, save them macro library and re-use them at similar nodes.
- Produce engineering drawings of the steel structure and connections
- Sharing the 3D model created with other BIM platform such as Tekla Structures.



Before you proceed, it is highly recommended you go through **ProtaStructure Quick Start Guide** as knowledge of the fundamentals of ProtaStructure is required.

This Quick Start Guide aims to get you up and running quickly. You should be able to learn the fundamentals of ProtaSteel in approximately 2 hours by following this simple guide.

2. Exporting model to ProtaSteel

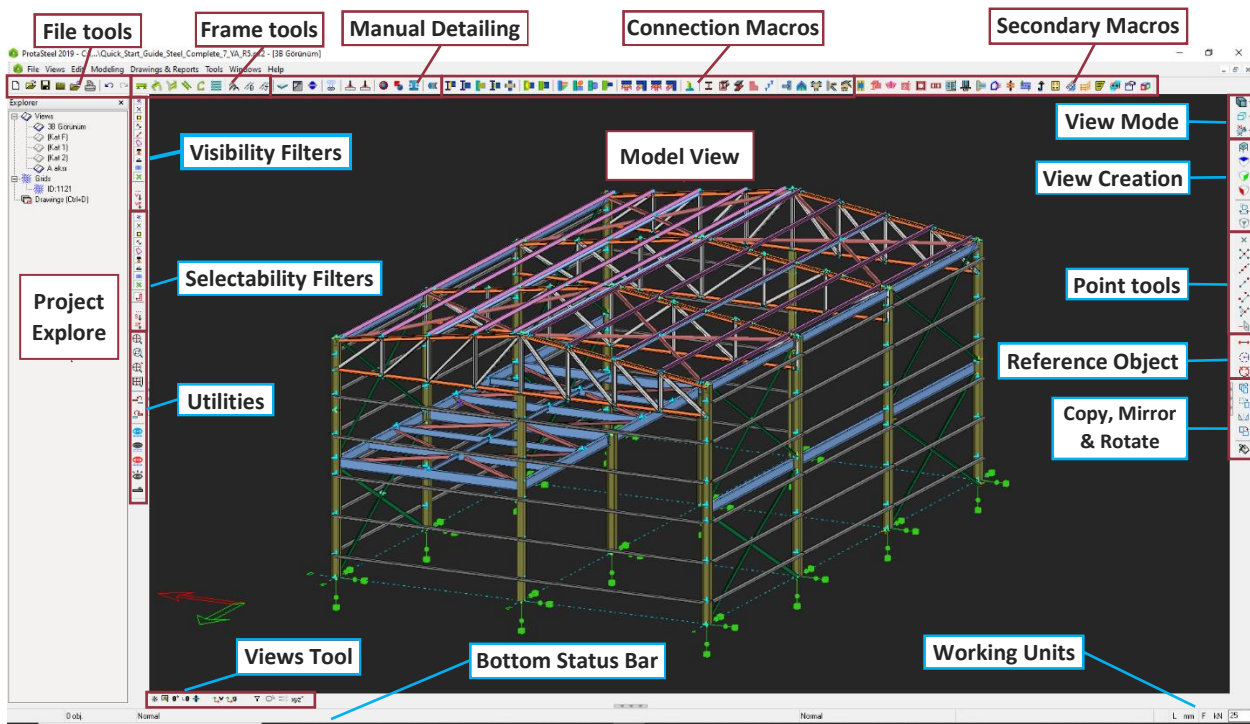
A sample steel model is already created & installed in default installation folder of ProtaStructure. We will now open the model, run analysis & export to ProtaSteel :

- Start **ProtaStructure** and the **Open Project** dialog will appear.
- Select & open project **“Quick_Start_Steel_Complete”**
- Go to **Analysis** → **Building Analysis**
- Go to **Analysis tab** → Tick **Building Analysis** → **Start**
- Close the **Building Analysis** menu
- Go to **Steel Design (top menu)** → Load **ProtaSteel**

ProtaSteel will start and load the project model. ProtaSteel model is created in subfolder “ProtaSteel” in the project folder.

3. User Interface

The various components of **ProtaSteel** user interface are as shown below:



Default views are created under the Views folder in the Explorer pane in the left pane. By the default the views created are 3D view & plan view of each stories. **Double-click** on the desired view to access it.

The graphical editor supports multiple windows; you can open as many windows as you desire. Modelling can be done both on the plan view as well as the 3D View.

Frame Creation Tool



In ProtaSteel, you can only create joist, purlin, girt, stair member, eaves beam & sag rods. Main members such as beam, column & brace must be created in ProtaStructure.

Manual Detailing Tool



These are secondary detailing tools such as weld creation, section plate creation, element cutting & trimming & profile adjustments.

Connections Macros



These are all the steel connections creation tools, eg. gusset plate, fin plate, haunch, splice etc.

Tip : If you hover the mouse cursor over any icons, a tooltip will appear with detail explanation on how to use it. By right-clicking on the macro icon, you can set the default parameters that will be used each time it is run.

4. Zoom & Pan Methods

The view manipulation is similar to ProtaStructure. You will find it easiest to use the mouse wheel to:

- Zoom in** – scroll your mouse wheel **up**
- Zoom out** – scroll mouse wheel **down**
- Pan** (move) – **hold down** the middle mouse wheel and **drag**
- Rotate** – Right click & drag



The following keyboard shortcut keys are useful :

- Zoom All** – **Home** key → Zoom so that all content is visible in the viewport
- Origin of Interest** – **F9** key → Select an origin of interest by clicking on any node or any object (such as bolt or plate). The model will then rotate around the selected point.

5. Selection Methods

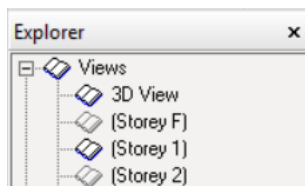
- **Left-click** on the entity to select the entity.
- To select multiple entities, hold down **CTRL** key while you pick them

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

- Drag from **left to right** to create a rectangular box. When you release the mouse button, all entities **completely contained** within the box will be selected
- Drag from **right to left** similarly and all the entities that are **contained within** the box and which **cross its boundaries** will be selected
- To **deselect** all entities, left-click on any empty space anywhere in the viewport.

6. Views & View Settings

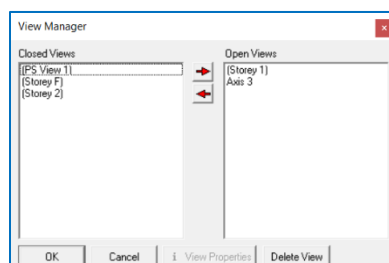
By default, the 3D view general is created & shown when you open the project. The plan of each storey is also created.



The views are shown under the **View** folder in the left **Explorer** pane.

- **Double-click** on the desired view in the left-hand Explorer pane to open it.

You can open as many views as you like.

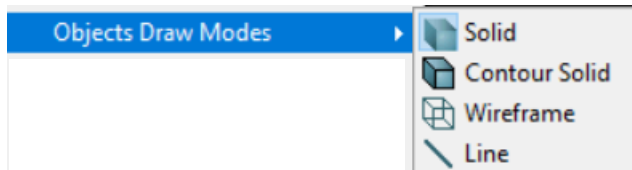



- **Double-click** on **Views** will open the **View Manager** where you can quickly :

- Open a view
- Close a view
- Delete a View
- Display the View Properties

If you have multiple views opened, you can manage them, e.g, fit them into the screen by going to **Windows** (top menu) → **Tile Horizontal / Tile Vertical / Cascade**.

You can change the **Object Draw Mode** by going to **Views** (top menu) → **Object Draw Modes**

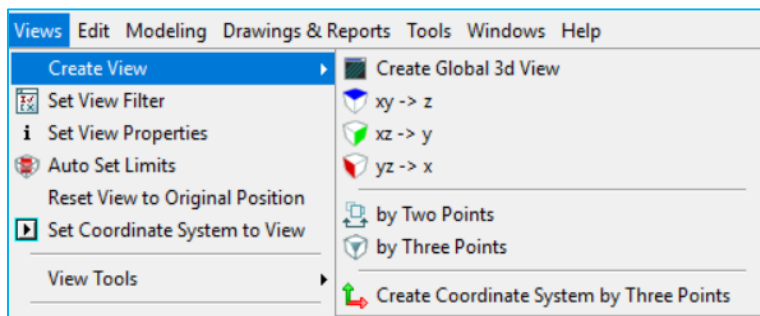


The shortcut icon can also be found on the right-hand most vertical toolbars 

- Press **Alt + D** continuously will cycle through the draw modes.

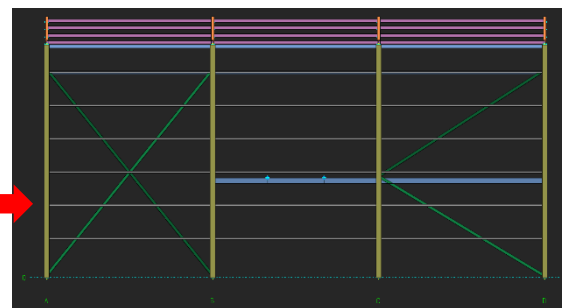
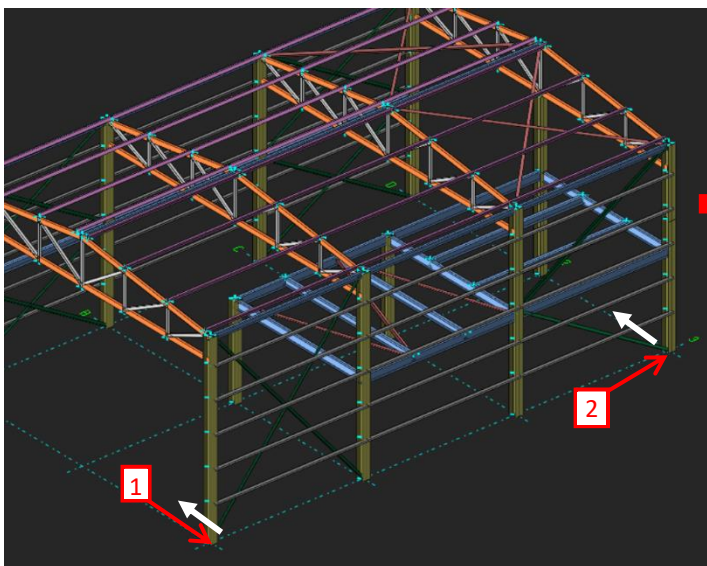
7. Creating new Views

New views can be created by going to **Views** → **Create View**



Let us create a new sectional view along grid 3 :

- Go to **Views** → **Create Views** → **By Two Points**
- **Click two points** to define a plane → the view will be created as shown below

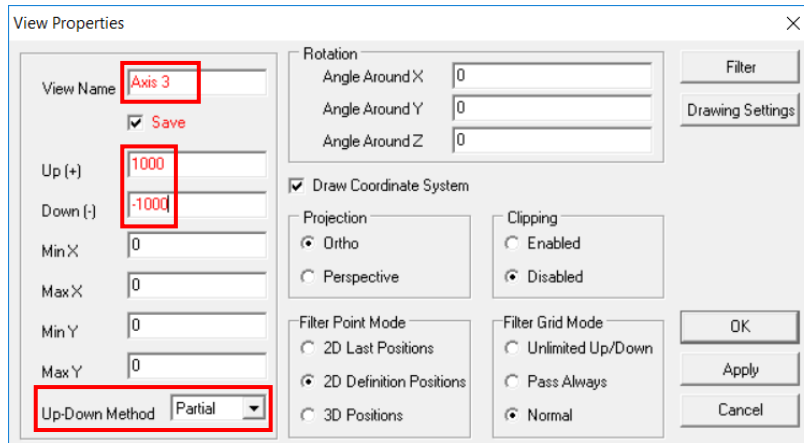


Tip: A white line arrow will appear once the mouse cursor hovers on 2nd point. This white line indicates the direction of viewing. In this case, the above view is looking into the building.

Note : The sectional view is actually created from the same 3D model but viewed from a different camera angle or orientation. Changes made to the active view are automatically reflected in all other views. The extent, boundary and objects to display can be set via **View Properties**.

To access the **View Properties**:

- **Double-click** anywhere on the new view → The **View Properties** will appear



- Give the view a new name **“Axis 3”**
- Change the **Up(+)** & **Down (-ve)** to **1000mm** & **-1000mm** respectively
This means that the view will only show elements 1m into & out of the plane. This will make the view less cluttered as elements along other grids will be hidden. If zero values are entered, then it will be unlimited.
- **Up-Down Method :**
 - **Partial** : Any element that exists partially within the set extents will be shown
 - **Full** : Only elements that exist fully within the set extents will be shown
- Click **OK** to close the dialog & view will be updated.

Any sectional view is inherently a 3D view; hence you can still rotate the view.

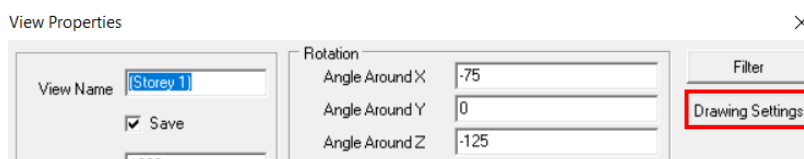
- **Right-click & drag** to **rotate** the Axis 3 view
Notice that only members within the 1 m extent is shown.
- Go to **View Properties** again
- Change the **Up-Down method** in View properties to **FULL** → **Apply**
Notice all the beams out of the plane of Axis 3 will hidden.
- Press **F4** to **restore lookup** (i.e. restore to original viewing direction)

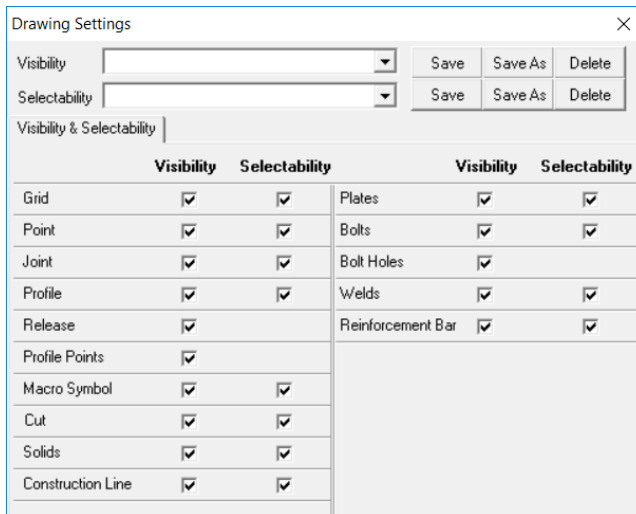
Important : You must give the view a new name and check / tick “Save” to ensure the view is saved. Only saved views can be used to generate drawings in the drawing module.

8. Visibility & Selectability

You can control what objects are visible and selectable by assessing the **Drawing Settings** dialog :

- Pick **Drawing Settings** in **View Properties** dialog





➤ Alternatively, click on the **Visibility/Selectability** icon “...” in the middle toolbar

➤ Turn off the **Grids & Points**

➤ **Apply** & see effect

➤ Go back & turn them back on





You can **Save** these different settings as different name to quickly re-use them.

The shortcuts to visibility / selectability objects are in the middle toolbar.

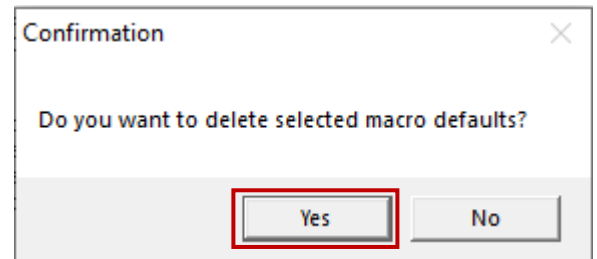
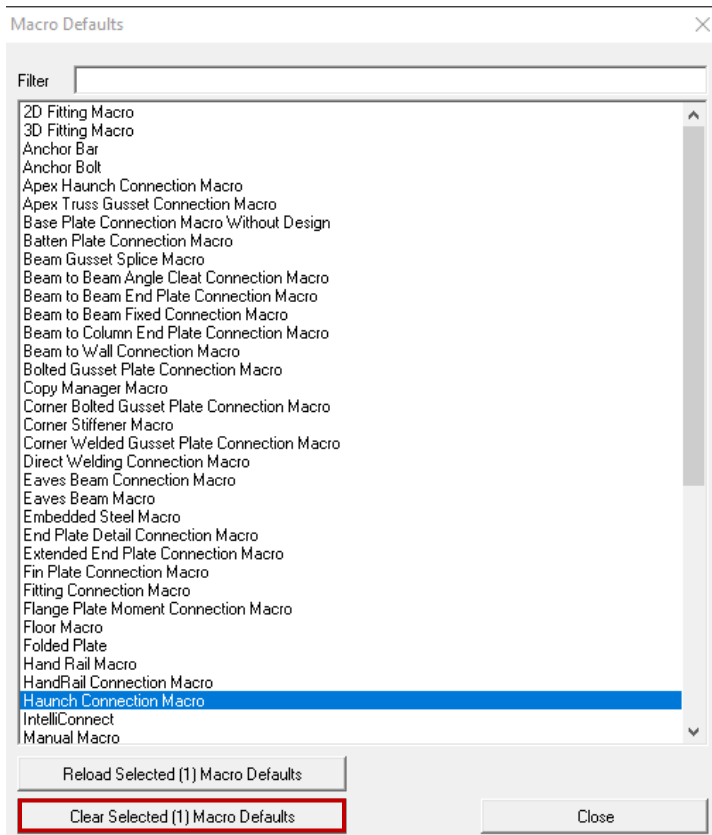


9. Macros

Most of the icons in ProtaSteel such as purlins, sag rods, girts & connections are called “macros”. A macro is generally defined as a shortcut tool that specifies how a certain input sequence will result in a desired output according to a defined procedure. The instruction on how to use each macro is shown in the tooltip if you hover the mouse cursor over the icon.

- Before creating the element using the macro, you can set the macro settings, e.g. member profile, plate offsets, bolts, etc. by **right-clicking** on the macro icon which will bring up the **Macro dialog**.
- Once a macro is clicked, the mouse cursor will change to the hand icon  meaning that it is waiting for object(s) selection or crosshair  meaning it is waiting for point selection. Perform the required action(s) to get the desired result, as stated in the tooltip.
- The macro will remain activated; you can continue use the macro without having to click on the macro icon again.
- Once you have finished, you must press **ESC** to cancel & stop the macro. The mouse cursor will change back to default arrow  (selection mode).
- After creating the elements in a group such as the purlins, each element will have its individual properties; which can be changed by **double-clicking** on it → this will bring up the **Element dialog**.
- However, all the elements are still associated with the macro & can be changed in a group. This can be done by selecting the **macro sign**  & then press **ENTER** will bring up the macro dialog. After making the changes, clicking **Apply** will apply to all the elements associated with the macro. Clicking **Get** will set it default for this macro for this and future projects.
- If you have problems selecting the macro sign, you can select any element or component of the macro (e.g. a plate of a connection) then press “**M**” → this will select the macro → followed by **ENTER** which will expose the Macro dialog.
- You can select all the elements associated with the macro by selecting the macro sign or an element of the macro → press “**CTRL + M**”.
- You can also delete all the elements & components of a macro by selecting the macro sign & then press **DELETE**.
- It is highly recommended you review settings of the macro before executing it.


- You can reload/clear the changes in macro default setting by going to **Tools** (top menu) → **Macro Defaults** → Select macro → **Reload/Clear Selected Macro Defaults**

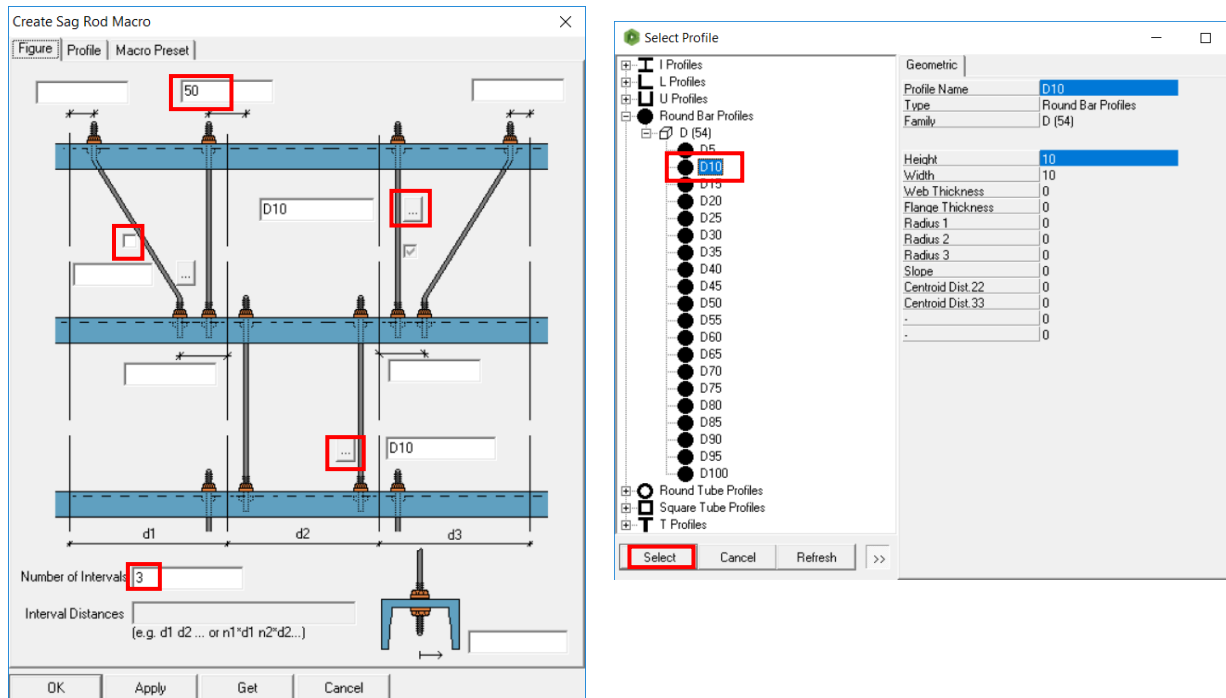


10. Insert Sag rods

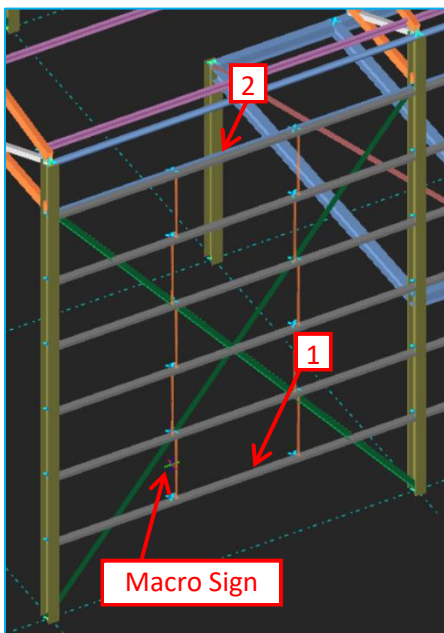
In ProtaStructure, the main steel members are modelled & designed; usually omitting the minor or non-structural members to keep the model size manageable & efficient. After the model is transferred to ProtaSteel, you might then want to add ancillary steel members such as sag rods.


We will insert some sag rods between the girts along axis 3 :


- Double-click on **3D View** in the Explorer pane to go to the default 3D view
- Right-click on **Sag Rod** icon  → This will expose the Sag Rod Macro Dialog.
- Key in the **offsets 50mm** & **Number of Intervals = 3** (as shown below)




- **Untick** the diagonal so that it will not be included
- Choose the **section profile** by clicking on “...” (as shown above) → Pick Round Bar **D10** → **Select**
- Pick **Get** to make these input the default from now on → **OK**



- **Left-click** on the **Sag Rod** icon 


Notice the cursor turn to a hand icon  meaning that it is waiting for required element selection action(s).

- **1** – **Left-click** on bottommost girt
- **2** – **Left-click** on topmost girt
- **Right-click** → Sag Rods will be inserted
- Insert sag rods for another panel between grid **B, C & D** using the same steps above, i.e. start by picking the bottommost girt (no need to pick the sag rod icon again)
- Once complete, press **ESC** to cancel the Sag Rod macro action.

Notice the cursor changes back to default arrow .

Each sag rod has its individual properties & can be selected, deleted or changed.

- **Double-clicking** it will reveal its properties.

All the sag rods are represented by the macro sign  They can be changed in a group by selecting this macro sign → press “**Enter**” as stated previously.

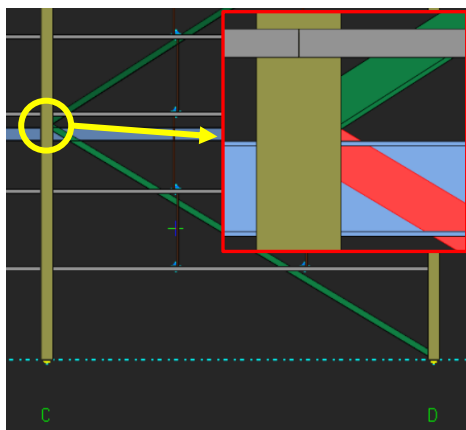
You can use **Sag Rod Connection** or **Sag Rod Holes** macros to create connections to sag rods and girts/purlins.

11. Adjustment of members by creating points

In ProtaSteel, you might also want to adjust the position of members that are not exactly at the correct position, i.e. fine-tuning of final member positions for detailing purposes.

In ProtaStructure, all the braces inserted will join at the top of beams instead at the middle (centroid) of the beam. This is because the analytical position of any beam in elevation is at the top of the beam.

- Double-click on previously created Axis 3 view *in the Explorer pane*
- **Zoom in** to the location of the diagonal braces between grid **C & D**

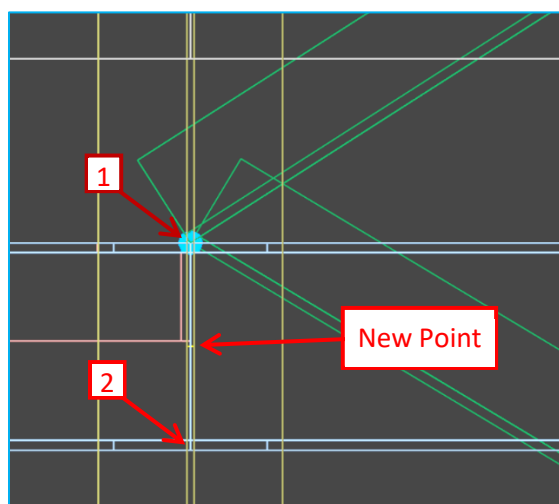



As you can see, the diagonal bracings are connected to the top of the beam. It is more accurate for the bracings to join at the middle of the beam.

To do that, we must create a point (or node) in the middle of the beam.

With the point created, we can then move the ends of the bracing to this point.

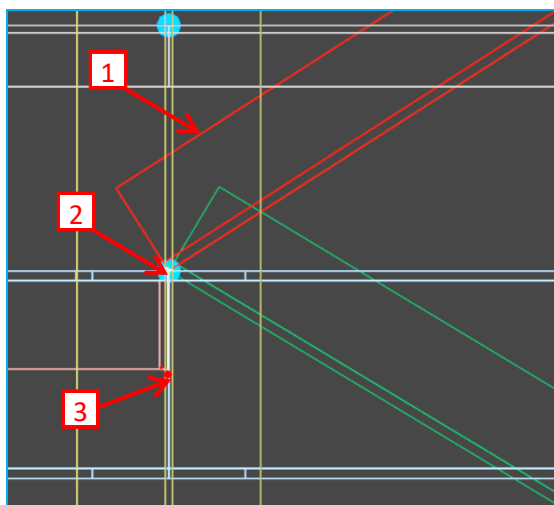
TIP : Press **Alt + D** continuously will cycle through the draw modes.




- Press **Alt + D** until you see the outline of members only (wireframe mode)
- Click "**Create Dividing Points**" icon  at the right-hand tool bar
- **1** – Click top point of the beam
- **2** – Click bottom point of the beam

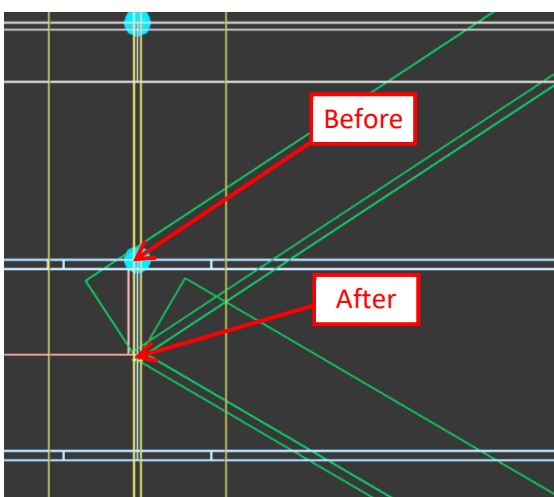
A new point "+" is created as shown in the left figure.

- Press **ESC** to stop the function



- Click **Adjust Profile Manually** icon  in the top menu toolbars
- **1** – Select the diagonal top bracing
- **2** - Click on existing end point
- **3** - Click on the new point

The brace end will be shifted to the new point.

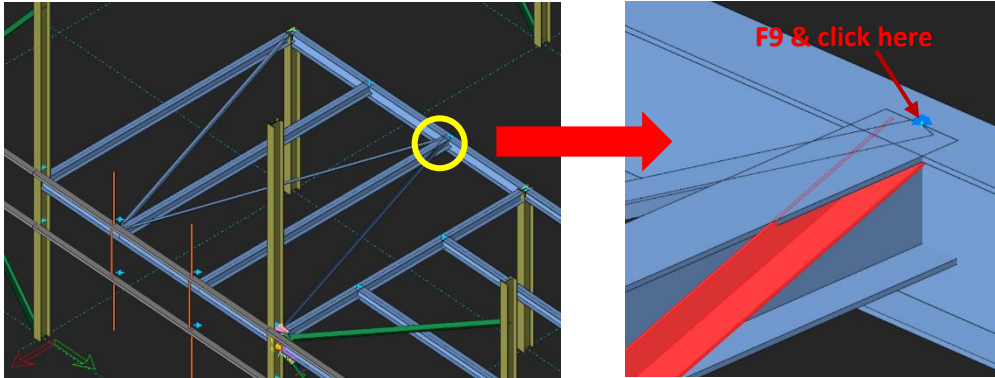


- **Repeat** the above **Step 1 to 3** for the other bottom diagonal brace.
- The final positions of the diagonal braces are as shown in the left figure.
- **Double-click** on **PS View 1** in the Explorer pane to go to the 3D view.
 - **Zoom** in the same location & check the diagonal braces final positions to ensure they are correct.

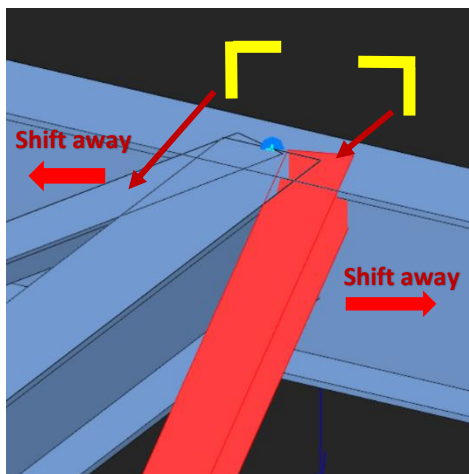
12. Adjustment of members orientation & coordinates

All member position & orientation can also be changed without creating new points. We will make such minor adjustments to some of the floor braces:

- Double-click on **Storey 1** view to open that view
- **Right-click & drag** to rotate it a 3D view
- Zoom to the joint **GL 2/B-C** where the secondary, main beam & 2 horizontal “L” brace meet
- Press **F9** & **click on the joint** to set it as point of interest so that rotation is about this point



Notice L braces orientation is not symmetrical with the other. We will rotate the brace so that flange of the brace is aligned with the flange of the beam.

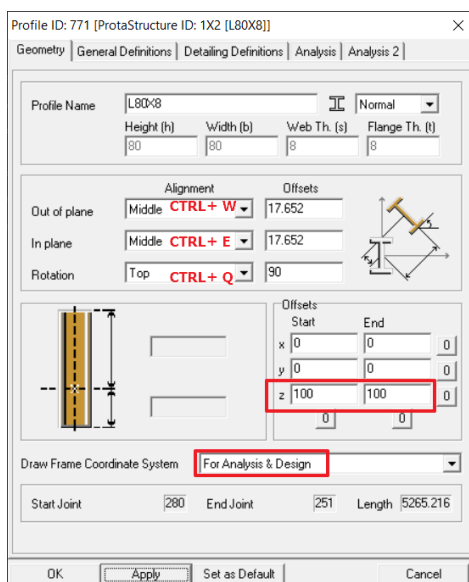


- **Select** the brace
- Press **Ctrl + Q** to rotate it until the correct orientation is achieved as shown in the left figure
- Repeat for the other brace

The braces are coming in too close to the secondary beam. We can shift it slightly away.

- Press **Ctrl + W** to offset the brace horizontally until it is farthest away from the secondary beam
- Repeat for the other brace

There is still a problem in the vertical position of the brace as it is too close to the top of the beams. Usually, braces are connected to the middle of the beam so transfer of force is to the centroid of the members and there will less clashes in the connection.



- **Double-click** on the right L brace

Profile Properties dialog allows you to change :

- Profile /Section type by picking "..."
- Alignment (Out of Plane, In-plane, Rotation)
- Offsets Start & End
- Draw Frame Coordinate System

- Change Frame **Coordinate System** to **For Analysis & design** → **Apply**

Notice 3 arrows showing local axis appear at the frame end. **Red** arrow = x-axis; **Green** arrow is y-axis; **Blue** arrow is z-axis.

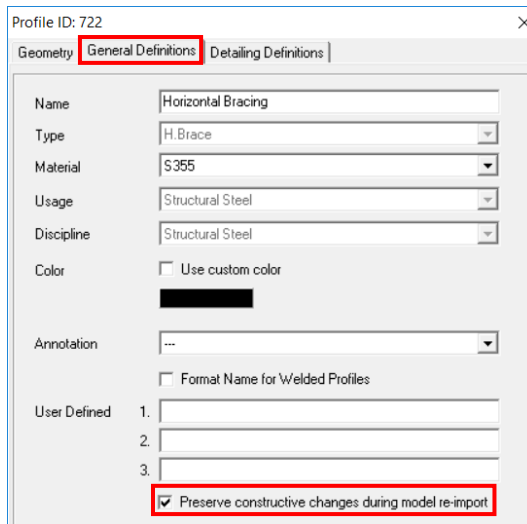
If the axes is not visible, please **ALT+D** to change to **Wireframe**

- Enter **z Start & End Offset** to **100 mm** → **Apply**
- Check the 3D view to ensure brace is moved down
- **Repeat** for the other brace (**-100mm**) → **OK** to exit

Minor shift in the vertical alignment can be done by pressing **Ctrl + W**. However, if you try this you will find that the offsets are too small. Further detail adjustment can be done by exposing the **Profile properties**.

Preserve constructive changes during model re-import

In the **General Definitions** tab, the option “**Preserve constructive changes during model re-import**” is by default checked.



This will preserve the member offset adjustments (e.g. member position & orientation) made in ProtaSteel during subsequent re-import from ProtaStructure. During the next import, the import log file will show a warning :

*Imported Frame's ID in *.cdf file[xxx]: Frame is locked. Therefore, in plane, out of plane and start/end offsets were not applied.*

Ignore the above warning if you want the member offsets to be retained in ProtaSteel.

If this option is unticked (disabled), offsets done in ProtaSteel will be discarded during the next re-import from ProtaStructure.

13. Steel Connections

Once we have finalized position of the steel members, we can start “connecting” the steel members, i.e. create connections. Connections can be created in two ways:

1. Automatically using the connection macros
2. Manually by using the Plates, Bolts, Welds & Cuts commands

The automatic connection macros will automatically create the appropriate connection. If there are internal forces transferred from ProtaStructure, they will be automatically taken in the connection design.

Detail design check is performed for the following connections in accordance to EuroCode 3, AISC (LRFD), AISC (ASD), BS5950 :

- Beam to Beam Connections: End Plate, Stiffened End Plate, Fin Plate
- Beam to Column Connections: End Plate, Stiffened End Plate, Fin Plate
- Beam to Beam End Plate Connection
- Haunch Connection
- Direct Welding Connection
- Simple Base Plate Connection
- Flange Plate Moment Connection
- Extended End Plate Connection
- Bolted Gusset Plate Connection
- Corner Bolted Gusset Plate Connection
- Welded Gusset Plate Connection

- Corner Welded Gusset Plate Connection
- Truss Seating Connection

For the above connections, detail design report can be produced which includes both geometric checks and capacity checks. In addition, intermediate calculations, references to the appropriate clauses of the design code, and corresponding equation are shown for easy checking.

Basic material capacity checks are performed for the following for the standard connections:

- All Bolted & welded gusset plate
- Purlin & Girt
- Splice connections
- Flange Plate Moment connections

For the above connections, the following basic material strength checks are performed:

- Plate bearing check
- Bolt shear & Bolt tension check
- Weld stress check
- Plate tension & plate bending check

The above are basic checks and not clause by clause code specific checks. No design report can be produced. You are advised to perform your own additional calculation checks if required.

The force used in the automatic connections are obtained from the higher of the two values:

1. The critical design forces of the connected members imported from ProtaStructure
2. User-defined **Capacity Ratio** in the Profile Properties > **Detailing Definitions**.

Capacity Ratio = Design Force / Maximum Member Capacity (equal less than 1)

This is to prevent unrealistic design when the design forces are very small or negligible.

For example, in determination of the number of bolts for a simple beam to beam fin plate:

- The critical design shear imported from ProtaStructure is 30 kN. This requires 1 number of bolts.
- The Shear Capacity Ratio of the beam is set to 0.3. This works out to be 100 kN which requires 2 number of bolts.
- The final design shear force used is 100kN and hence 2 number of bolts is chosen.

There is no check or design at all for the following connections: Apex Haunch Connection, Direct Welding Connection, Base Plate (Except Simple Base Plate), Eaves Beam Connection, Profile Fit Connection, Sag Rod Connection & Sag Rod Holes, Welded Pipe Connection, Beam to Wall Connection, Apex Truss Gusset Connection & Truss End Plate With Gusset Plate Connection.

Before performing connection design, we should customize the default design parameter:

- Go to **File** → **Settings** → **Project Preferences**
- Ensure that **Design Standards for Connections** is set to **EC3**
- **General Overdesign Factor for Connections = 1**

This means that the connections will generally design for 100% of the member capacity. This setting works in combination with **capacity ratios** of the connecting members (changed via Profile dialog →

Detailing Definitions). The Capacity Ratio is automatically calculated = Design force divided by the actual capacity of the member. Hence, **Capacity Ratio x Overdesign Factor = Final Factor.**

➤ For example, under **Steel Detailing** ensure the following

- **Plates** : Preferred Plate Material = **S355**
- **Bolts** : Preferred Bolt Material = **Class 8.8**
- **General Weld** : Preferred Weld Material = **Class 42**

➤ **Apply** → **OK**

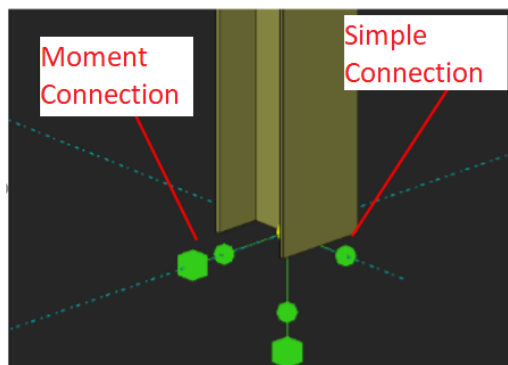
14. Base Plate Connection



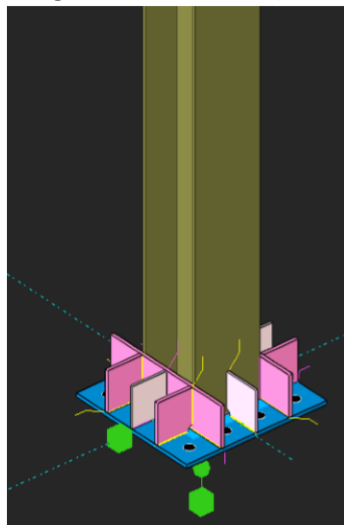
In order to create a base plate connection to columns, we can use the base plate macros. There are two base plate macros in ProtaSteel for creating the base plate connection.

- Base Plate Connection (without design)
- Simple Base Plate Connection (with design)

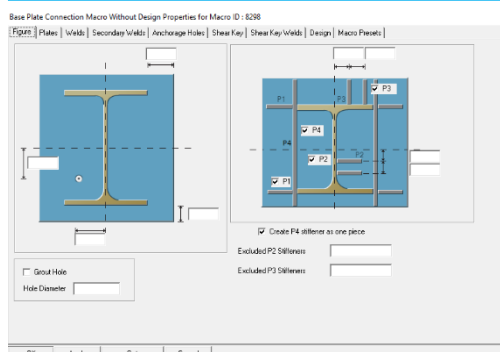
In ProtaSteel, restraints are indicated with the ball and cubic objects in green. If the connection is defined as moment connection in ProtaStructure, it is shown with ball+cubic object. If the connection is designed as a pinned connection, it is shown with only a ball object.



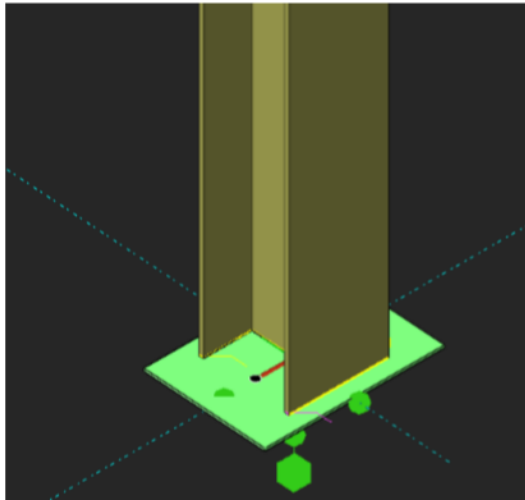
Using with “Base Plate (without design) Macro”, all types are created.



- Select the **“Base Plate Connection Macro”**
- Select the **column.**
- The connection is created under of column.



Pinned connections can be created by the “Simple Base Plate Macro”. This macro can design and create calculation report. All parameters can be changed in macro properties.



- Select the “**Simple Base Plate Connection Macro**”
- Select the **column**.
- The connection is created under of column.

15. Fin Plate Connection

We will create a **fin plate connection** between the adjacent main & primary beam joint.

- **Right-click** of the **Fin Connection macro** 

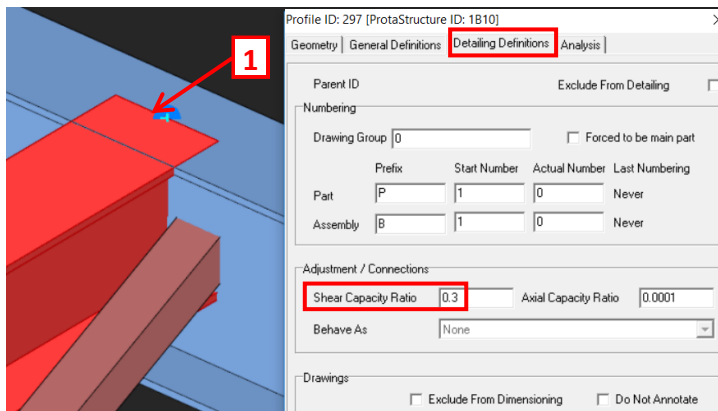
The Fin Plate Macro dialog will appear which allows you to customize the connections.

- Go to the **Design Tab** & ensure the following:

- Design Standard = **EC3**

Note : Slip Coefficient is blank & hence will follow the Project Preferences → Bolts settings

- Pick **Get** to make it **default** settings for this connection type for all projects from now on.
- **Zoom GL 2/B-C** to the previous joint where the secondary, main beam & 2 “L” brace meet.



- **Double-click** on the secondary beam
- In **Profile Properties**, go to **Detailing Definitions tab**

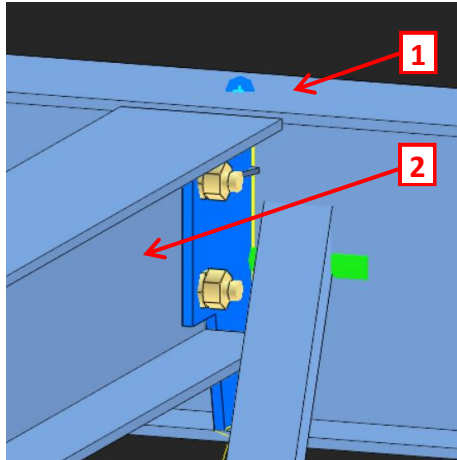
The **Capacity Ratio** defaults to 0.3 for this beam.

This will be used to calculate the minimum design shear used for the connection.


The **Analysis tab** shows the geometric properties of the member.

- **Apply → OK**

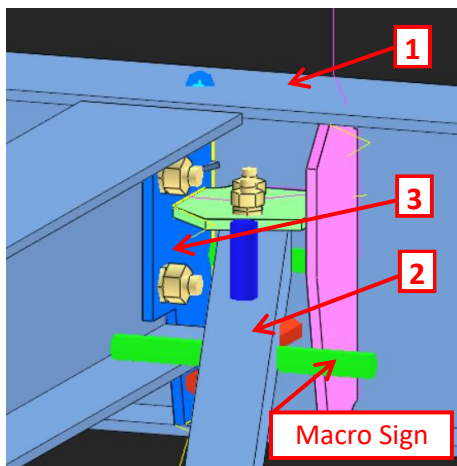
Important Note : The capacity ratio works in combination with **General Overdesign factor for Connections** (in Project Preference). Hence, **Capacity Ratio x Overdesign Factor = Final Factor**.




We will now insert a fin plate connection at the primary & secondary beam joint.

- Pick **Fin Plate Connection** 
- **1** – Pick the main beam
- **2** – Pick the secondary beam
- **Right-click 3 times**

A fin plate connection will be created.




We will now insert corner bolted gusset plate for the L brace that uses the beam fin plate as boundary.

- Pick **Corner Bolted Gusset Plate Connection** 
- **1** – Pick the main beam
- **2** – Pick the brace
- **3** – Pick the blue fin plate

The connection is created. Repeat the same for the other brace.

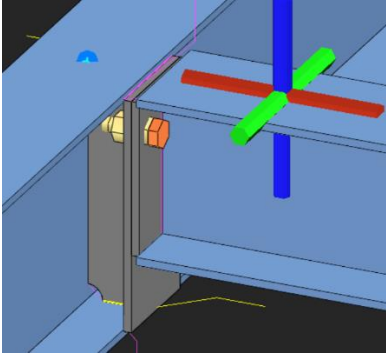
Tip :

- ❖ The connection consists of plates, bolts & welds. Each element has its individual properties & can be selected, deleted or changed. **Double-clicking** it will reveal the element properties.
- ❖ The entire connection is represented by the macro sign . **Double-clicking** on it will bring out the Macro dialog box where the entire connection can be changed.
- ❖ It is highly recommended you review settings of the connection macro before executing it.

16. Beam to Beam End Plate Connection

We will create **Beam to Beam End Plate Connection** between the main & secondary beam.

- Zoom to the adjacent joint **GL 2/B-C** where the secondary & main beam meet



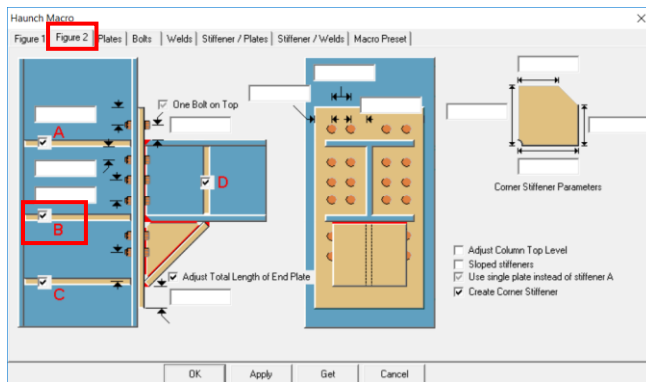
- Pick **Stiffened End Plate Connection** 


- **1** – Pick the main beam
- **2** – Pick the secondary beam

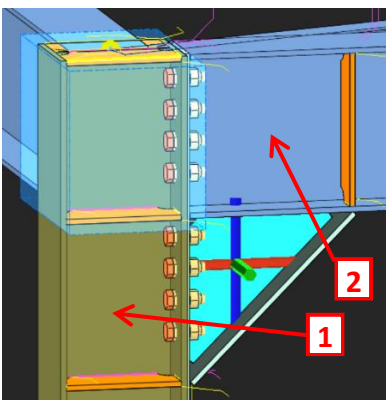
The connection is created. This maybe a more preferable in terms of load transfer & ease of assembly & construction.

17. Haunch Connection


We will create a haunch connection between the column and beam at **GL B/2**.



- Right click on the **Haunch Connection** icon 
- In Haunch Macro, go to **Figure 2** tab
- Ensure that all the **end plates A,B,C & D** is checked
- **Apply** → **OK** to exit



We will now insert the haunch connection.

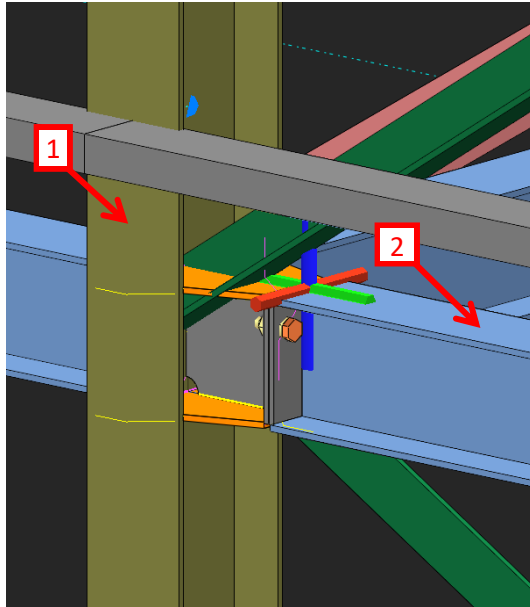
- Pick **Haunch Connection** 
- **1** – Pick the column
- **2** – Pick the beam


The connection is created. The blue transparent box is the cut object. This can be turn on/off in Visibility dialog.

- Repeat the same for the other end of beam to column

18. Stiffened End Plate Connection

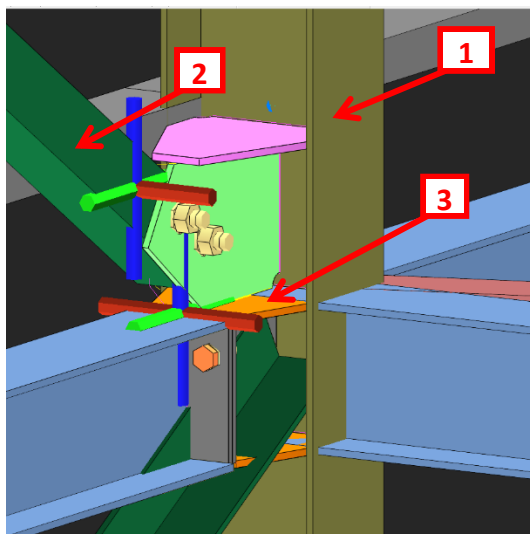
We will create a stiffened end plate connection at the column beam joint at **grid C/3**.




- Double-click on **Axis 3** view to open view
- Zoom in the joint where the column, beam & 2 diagonal braces meet at grid **C/3**
- Press **F9** on this joint to set the point of interest
- Rotate the view slightly about this joint
- Pick **Stiffened end plate connection** icon 
- **1** – Pick the column
- **2** – Pick the beam
- **Right-click** → Connection is created

19. Corner Bolted Gusset Plate Connection

We will connect the diagonal brace at the same joint. There is problem because the existing connection is getting in the way, i.e. we cannot connect the diagonal brace directly to the column due to the top stiffener plate. This can be solved by using the **Corner Bolted Gusset Plate Connection**.

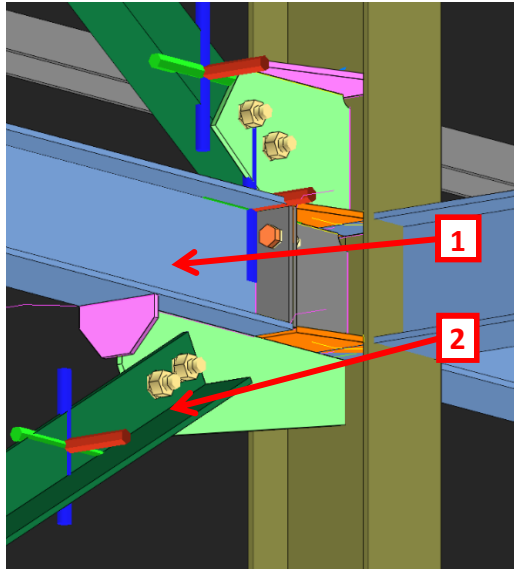


- **Rotate** the view around so the connection is not obstructed by the girt
- Pick **Corner Bolted Gusset Plate Connection** 
- **1** – Pick the column
- **2** – Pick the top brace
- **3** – Pick top orange plate

A logical & constructible gusset plate connection will be created joining the top plate of the previous connection.

20. Bolted Gusset Plate Connection

For the bottom brace, it is a slightly different situation as the brace should connect to the bottom of the beam flange directly. Hence, we should use the **Bolted Gusset Plate Connection**.



➤ **Rotate** the view around the connection so you are viewing the bottom of the brace

➤ Pick **Bolted Gusset Plate Connection** 

➤ **1** – Pick the beam


➤ **2** – Pick the bottom brace

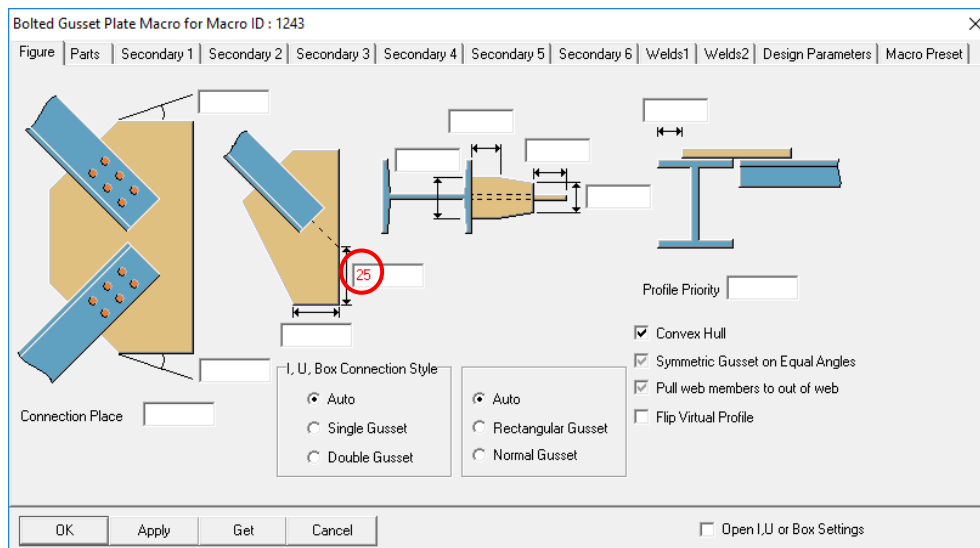
➤ **Right-click**

A logical & constructible gusset plate connection will be created joining the bottom flange of the beam.

➤ Press **ESC** to cancel the macro

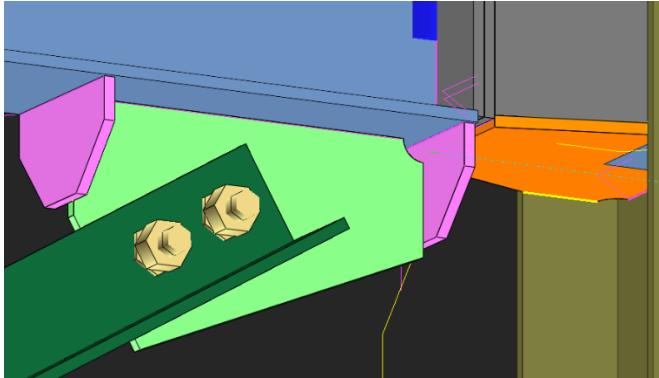
Let's push back & shorten the green gusset plate so that it ends exactly at the beam end for constructability.

- **Select** connection macro by selecting the macro sign  → Press **Enter**
- Alternatively select the gusset plate > press "**M**" > Press **Enter** (wait for the macro dialog)



- Enter **25mm** in the above plate **set-back** value → **Apply**

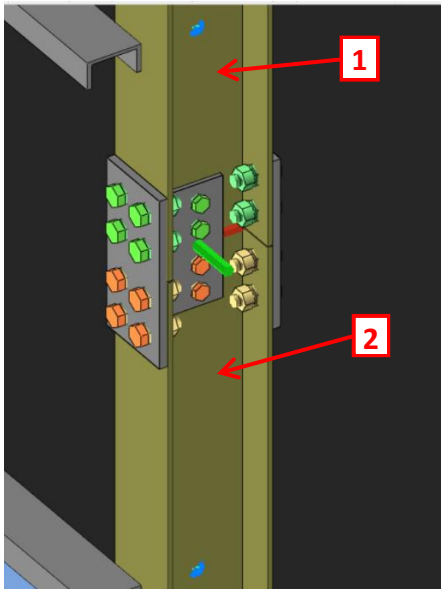
You can see the plate is shortened and the stiffener plate is added.



- This is more constructible because the gusset plate & stiffener can be assembled off-site as part of the beam assembly.
- The previous stiffened end plates will be part of the column assembly.
- Hence, when the column & beam are brought to site, it's easy & quick to install & "connect" them together.

21. Splice Connection

Splice connection must first be inserted in a column or beam in ProtaStructure. For this model a splice connection has been inserted at the corner column at **GL D/3** at 800mm above the 1st Storey.



- **Zoom** the corner column at **GL D/3**, slightly above Storey 1

Notice the column is broken into 2 separate elements as a column splice was created in ProtaStructure.

- Pick **Splice Connection** 

- **1** – Pick the top column

- **2** – Pick the bottom column

The column splice connection is added.

- Press **ESC** to cancel the macro

- The **green** color bolts are called **Workshop Bolt**. These bolts are pre-installed in workshop before being brought to site.
- The **orange** bolts represent the bolts to be assemble on site.

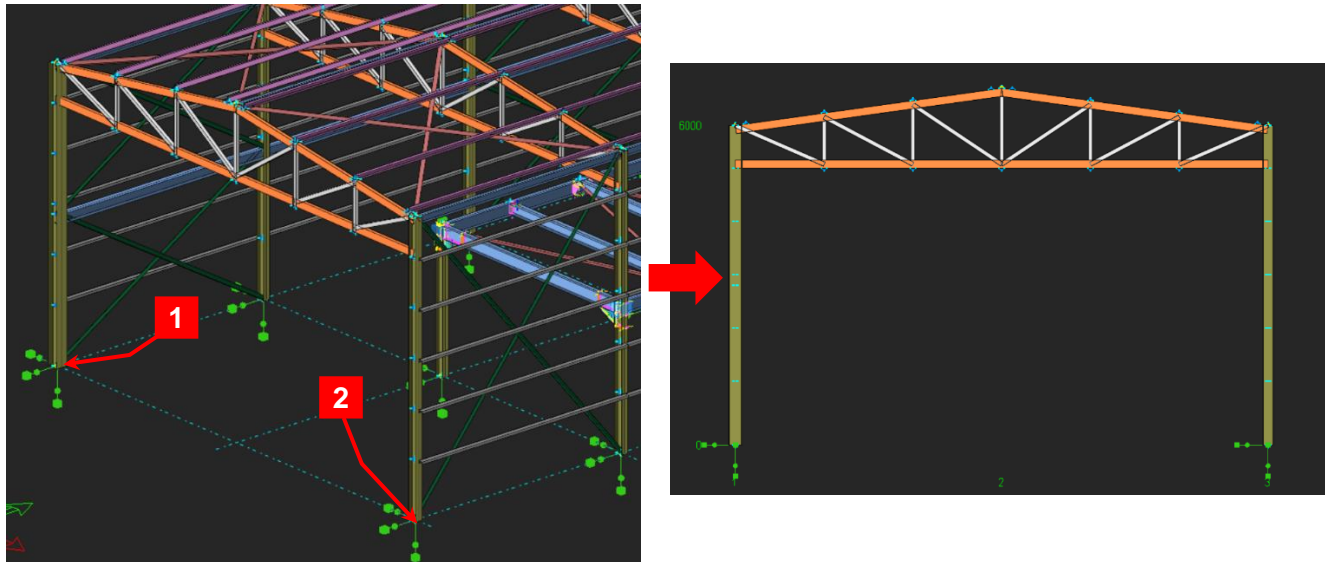
The above setting & other bolt parameters can be viewed & changed in the **Bolt Group property** :

- **Double-click** on any of the bolt → Bolt Group property will appear
- Go to **Settings 2** tab → **Assembly Location** → two choices are in the dropdown menu :
 - **Workshop**
 - **Site**

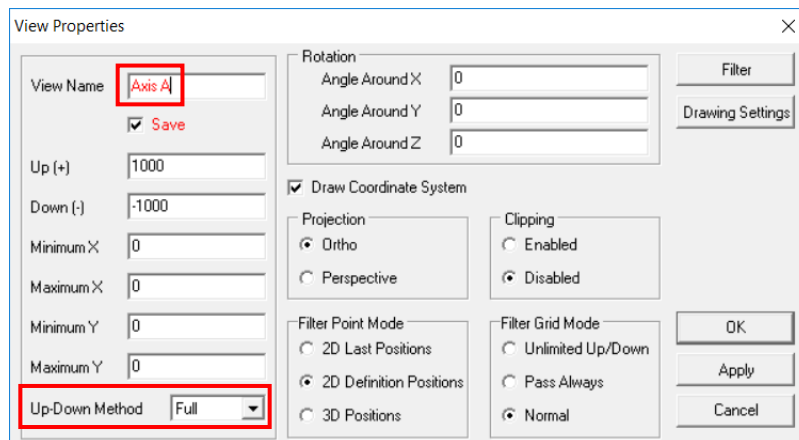
22. Truss End Plate With Gusset Plate

Truss End Plate with Gusset Plate connection macro creates a connection between a column and secondary trusses with gusset & end plates. Let us focus on the truss by creating a new view along axis A.

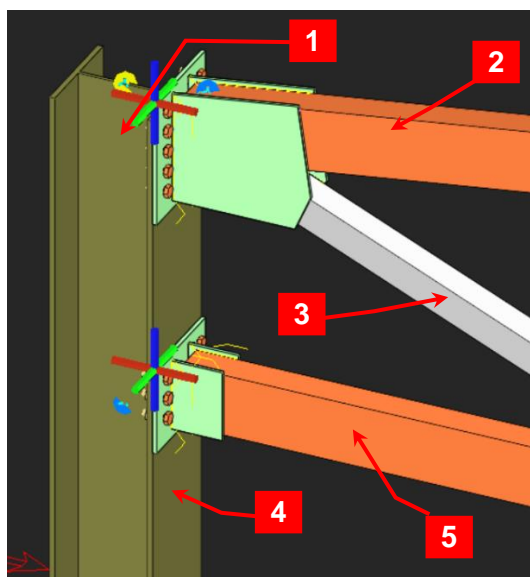
- Go to **Views** → **Create Views** → **By Two Points**
- **Click two points** along Axis A → the view will be created as shown below




- **Double-click** anywhere on the new view → The **View Properties** will appear



- Give the view a new name **"Axis A"**
- Change the **Up-Down method** in View properties to **FULL** → **Apply**
- Zoom to the left side of the truss (Axis A/1)
- Press **F9** & click on the **top of the column node** to set it as point of interest & rotation

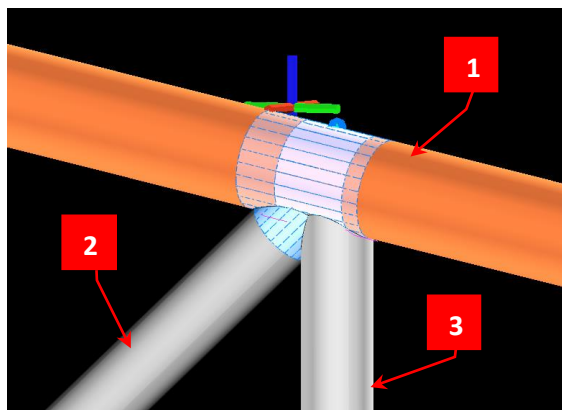



- Pick **Truss End Plate with Gusset Plate** 
- **1** – Pick the main column
- **2** – Pick the top chord
- **3** – Pick the diagonal
- **Right-click** to end & create the connection
- **4** – Pick the main column
- **5** – Pick the bottom chord
- **Right-click** to end & create the connection

The two connections are created as shown.

23. Welded Pipe Connection (info only)

Welded pipe connection macro can be used to connect tube profiles.

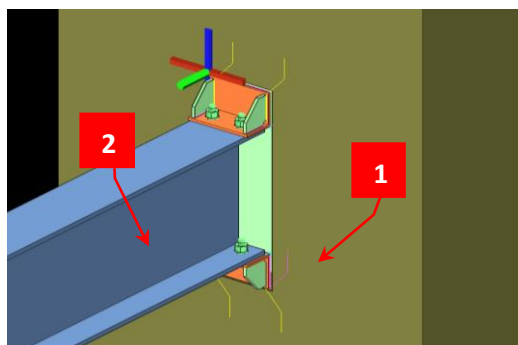



- Click on **Welded Pipe Profile** macro 
- **1** – Select the main top member
- **2 + 3** – Select the secondary profile, e.g. diagonal and vertical element.
- **Right-click** to end & create the connection

Profiles are automatically cut & trimmed around the joint.

24. Beam to (RC) Wall/Column Connection

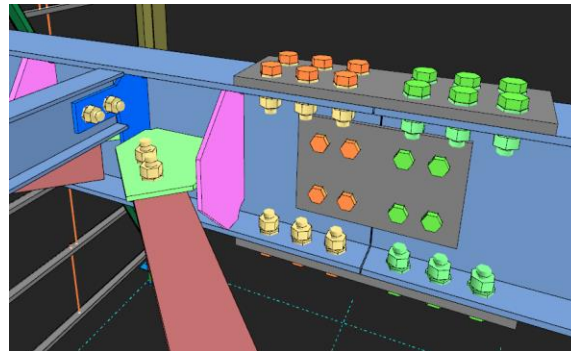
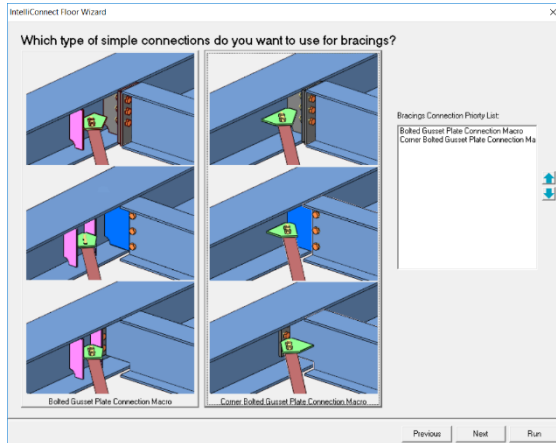
This connection macro creates & designs an embedded steel connection between a steel beam and a concrete wall or column. The steel member must be perpendicular to the reinforced concrete surface. It is important to have enough space on the concrete surface, else the macro will not work.



- Click on **Beam to Wall Connection** macro 
- **1** – Select the concrete wall or column
- **2** Select the steel beam
- **Right-click** to end & create the connection

25. 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.

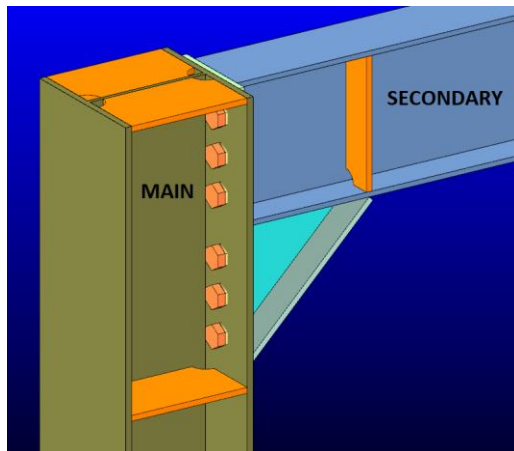


Figure 1

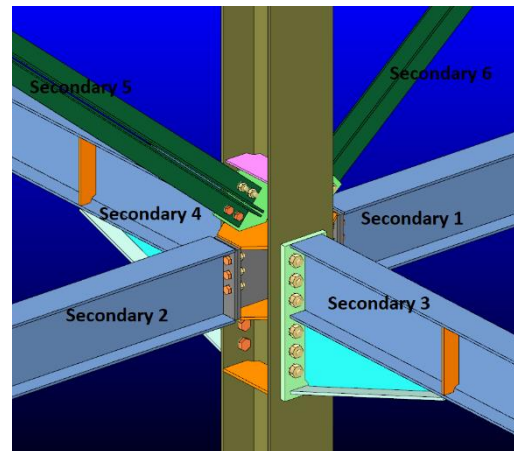
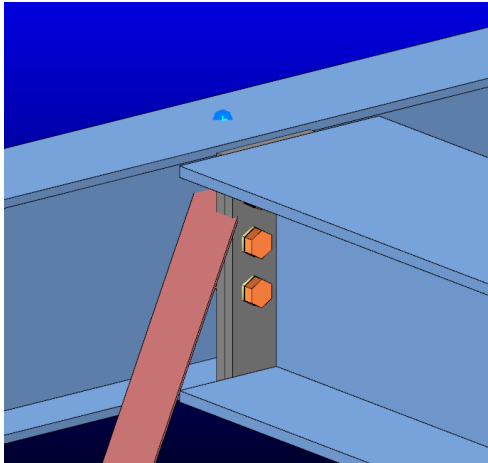


Figure 2

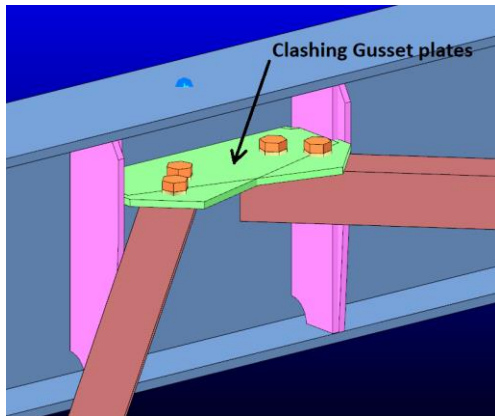
While this feature of “Rule based joints” is beneficial in simple cases where the main and the secondary (incoming profile) are the only two profiles present at a joint (see Figure 1 above), it fails to deliver for complex joints when more than two elements meet at the same joint (see Figure 2) :

- ❖ 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.

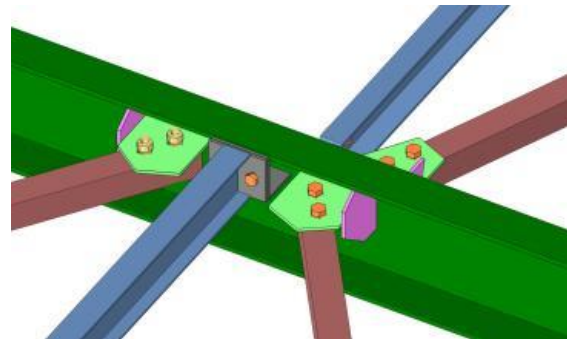
❖ For example (see Figure 4), two horizontal braces will connect to the same point separately, creating a clash between the gusset plates, rather than using a single common gusset plate.

IntelliConnect is an advanced technology which attempts to address these issues which are common in building detailing. IntelliConnect automatically determines which connection type is more appropriate for the selected profiles by checking their type, location and orientation. It automatically alters connection parameters if necessary, and creates a feasible & practicle connection.

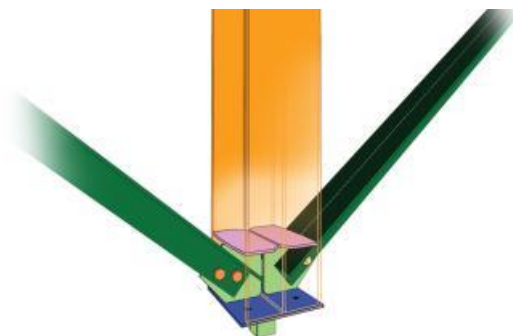
IntelliConnect Scope

IntelliConnect can automatically create connections for various types of elements and profiles. IntelliConnect can provide solution for 36 different cases grouped in 9 categories. These categories can be listed as follows:

- Splice connections
- Batten plate connections
- Sag rod connections
- Vertical bracing connections



- Truss connections
- Purlin and girt connections
- Floor beam connections (with horizontal bracing connections if there are any)
- Beam – Column connections (with horizontal and vertical bracings if there are any)
- Base plate connections (with vertical bracings if there are any)



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 Stiffened 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 maybe I or channel profiles. The type of beam to beam connection can be any one of the following 1) The Stiffened 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 use **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 use 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.
- ❖ A detail report will automatically be displayed listing nodes that are successfully connected or failed to connect.
- ❖ **Intelliconnect Node Control** tool is an interactive navigator to browse & examine points that partially connected or not connected; so the user can connect / correct them manually.

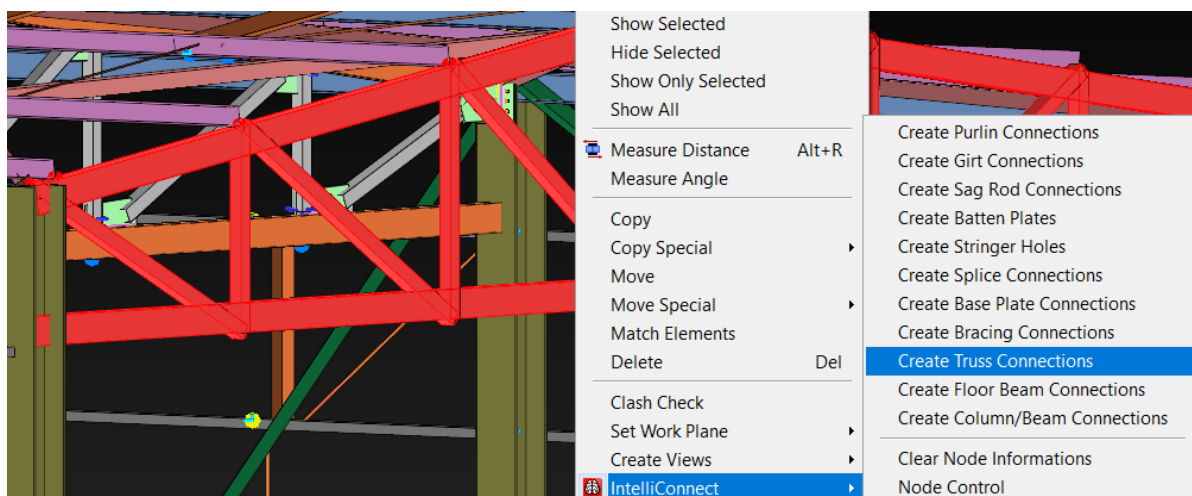
Truss Example

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

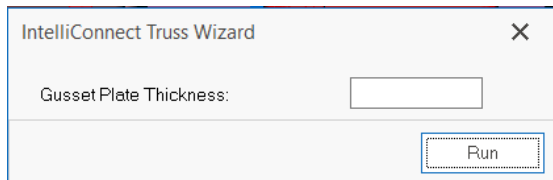
Joints are shown as blue dots at the end of the member. For simplicity, you can click and drag a box to select all members which will then also select the nodes.

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

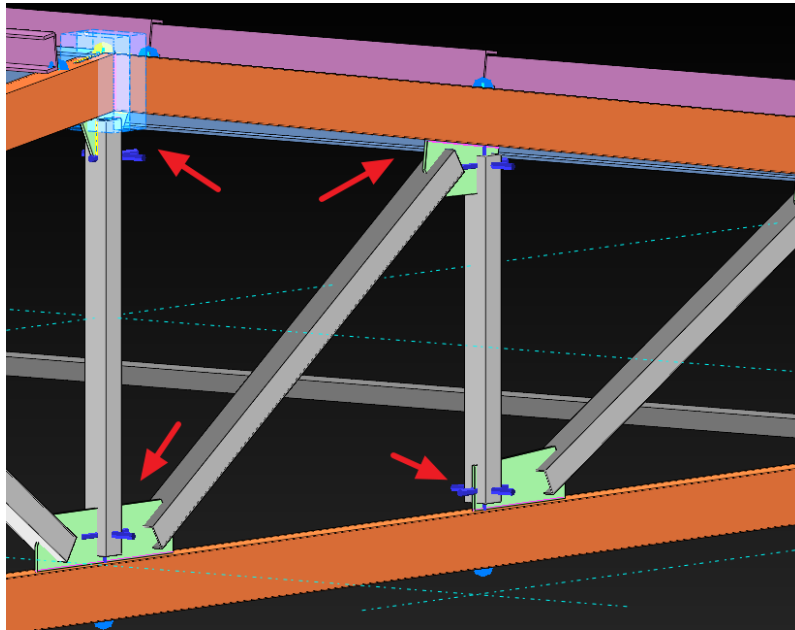
- *Go to the **3D View***
- *Select all the truss members by left-click a drag a box around them*



- Right-click → **IntelliConnect** → Pick **Create Truss Connections**



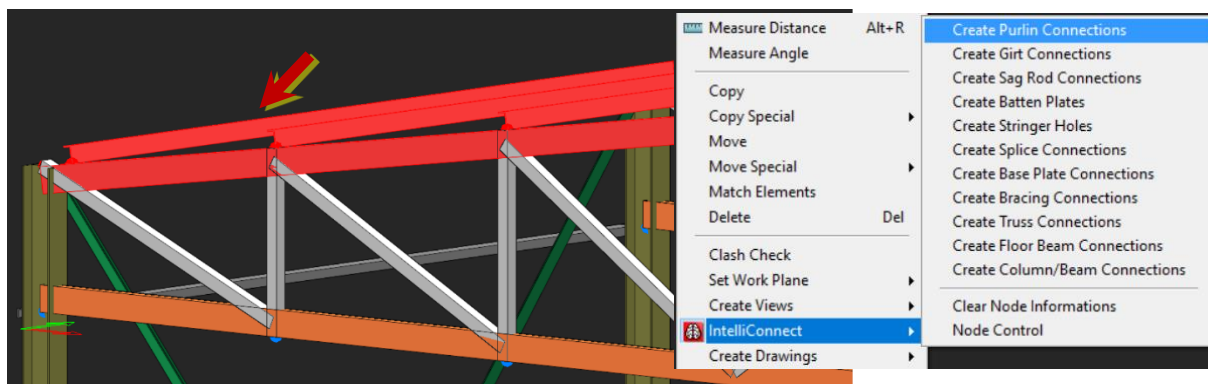
- In the Truss Wizard, input the preferred Gusset Plate Thickness or left it blank to automatically select thickness for your truss → **Run**



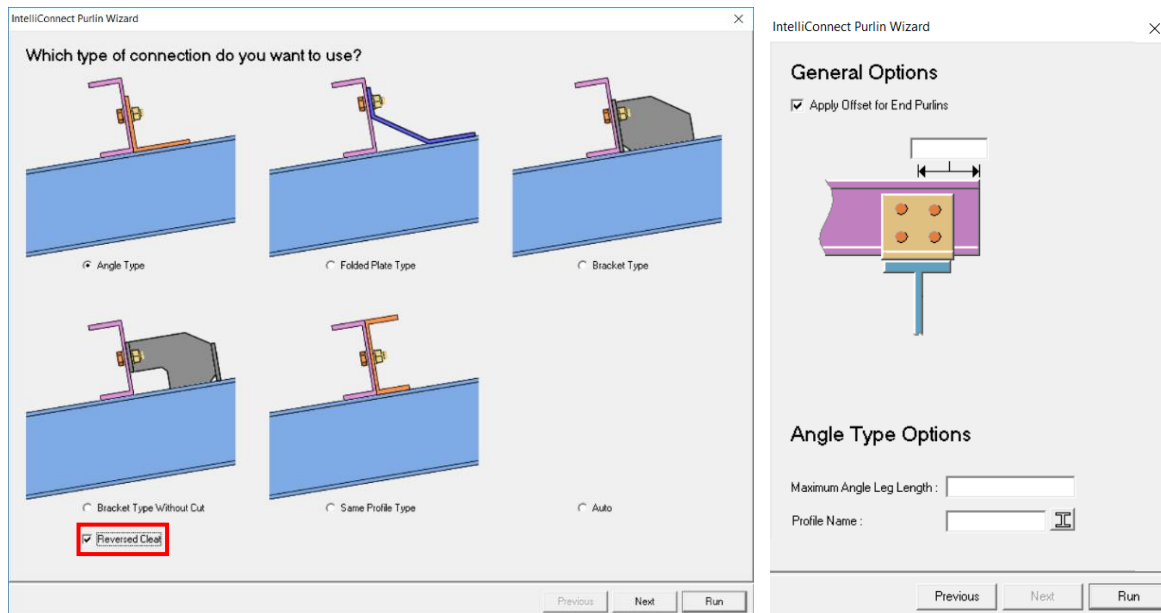
Purlin Example

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

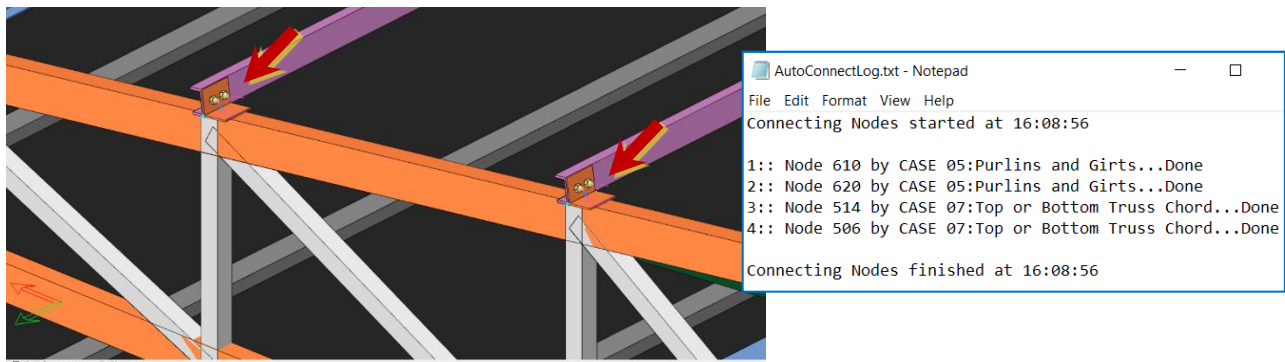
- Go to the **3D View**
- Select all the truss top chords members and purlins by left-click a drag a box around them



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



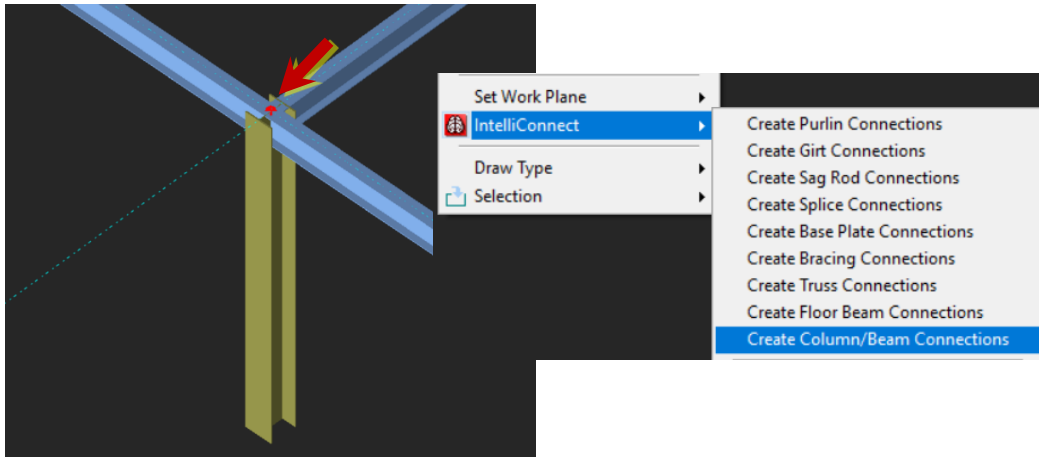
- In the Purlin Wizard, choose the preferred type of connection
- For this example, choose **“Angle Type”** & **“Reversed Cleat”**
- Input further parameters in the next wizard dialog → **Run**
 - Max Angle Leg Length or profile can be specified
- AutoConnectLog text file will open → Review its contents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



Beam – Column – Brace Example

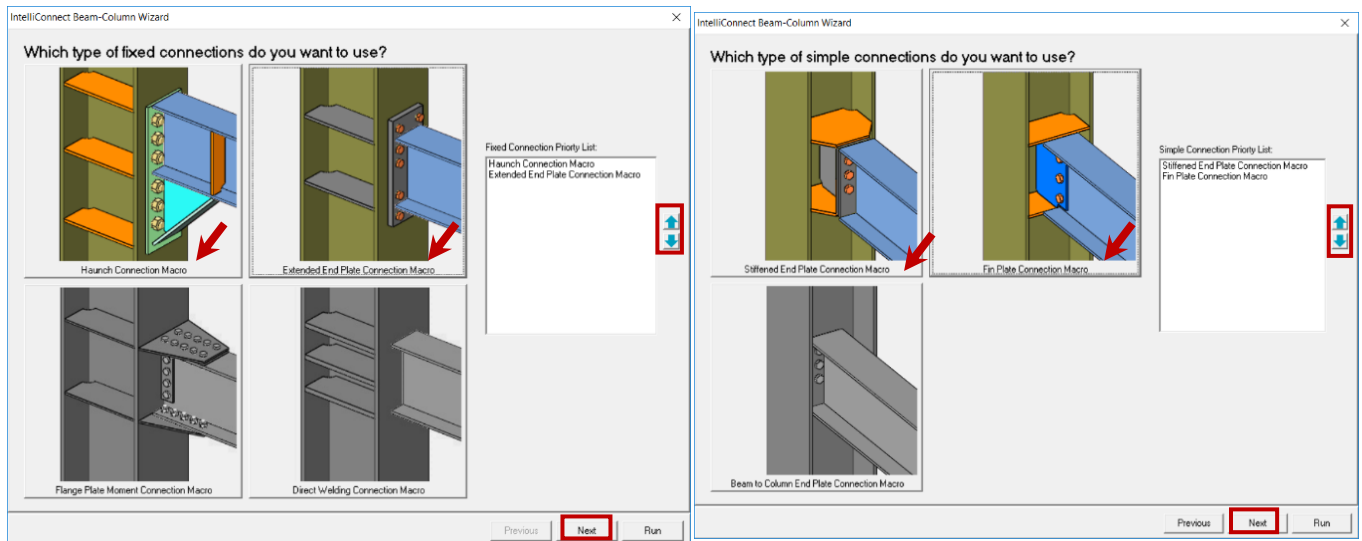
For illustration purpose, we will auto create a beam – column – brace connection based on the Quick Start Guide Steel model.

- Go to **Storey 1 view** → Select the all the column and beam joints along axis 2



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

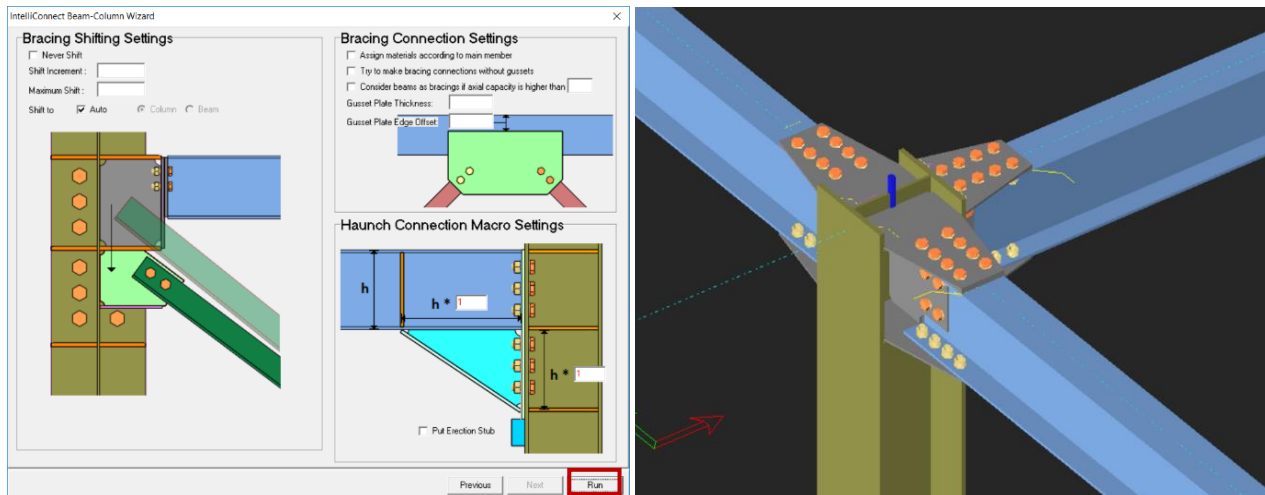
Where there is multiple possible connection configuration, the subsequent dialog allows you to choose which connection type to consider and its priority.



- 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** → Then connections will be created



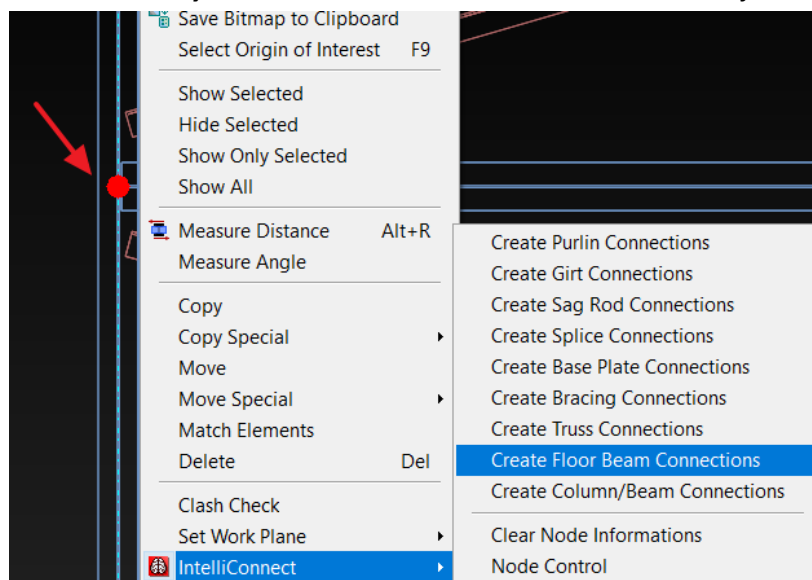
Note : Beam - Column Connections (IntelliConnect) does not work for connection between beam and middle of 2 separate columns (upper and lower). To perform this connection, you can either:

- Create as single column with LEN in ProtaStructure, then import to ProtaSteel,
- or Create only using connection macro

Beam – Beam – Brace Example

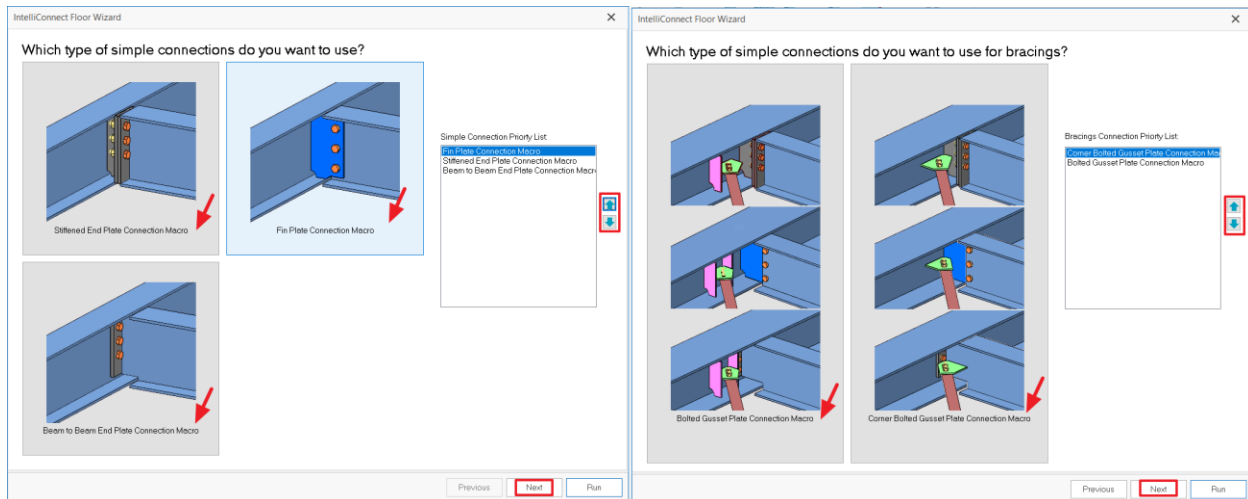
For illustration purpose, we will auto create a beam – beam – brace connection based on the Quick Start Guide Steel model.

- Go to **Storey 1 view** → Select the all the beam and beam joints along axis 2



- Right-click → **IntelliConnect** → Pick **Create Floor Beam Connection**

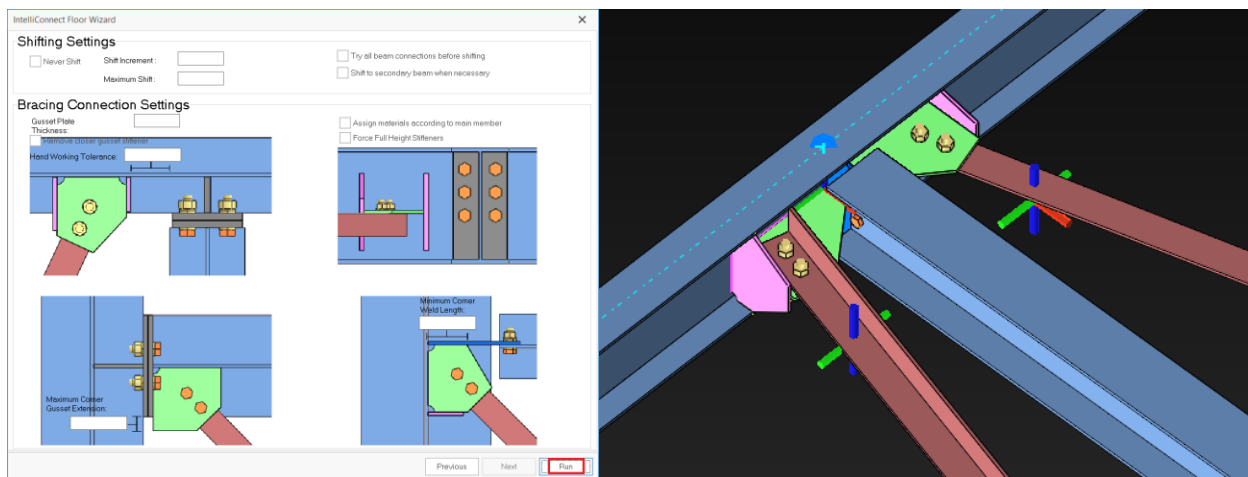
Where there is multiple possible connection configuration, the subsequent dialog allows you to choose which connection type to consider and its priority.



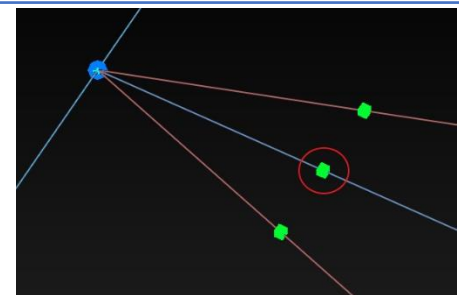
➤ Pick the connection types to consider → Adjust the priority using up/down arrow → **Next**

The relevant bracing shifting, bracing connection settings will appear next

➤ Make desired adjustment → **Run** → Then connections will be created



Note : Floor Beam Connections (IntelliConnect) works when beam to beam connection is hinged in ProtaStructure.
Change to “Line Mode”, **Green Dot** defined hinged connection.



IntelliConnect and Connection Ordering

Although detailing order often depends on the structure itself, it is usually better to start from large connections to small ones. The connection order depends on the size and shape of the structure. For example, for the high-rise buildings, starting the connection process from the bottom is more useful. Most structures will have similar, symmetrical, or identical axes. Whether you use IntelliConnect or not, it is a good practice to finish the frame connections first.

In a typical warehouse structure with truss, the recommended connection order is given below :

- ❖ Splice connections (Columns, Beams and Truss members)
- ❖ Truss connections
- ❖ Apex connections
- ❖ Truss top and bottom chord connections to columns
- ❖ Base plate connections
- ❖ Manual (User defined) connections (User defined connections can be saved for later use and repeatedly used for similar joints)
- ❖ Connections of the compression members located between the column or truss system.
- ❖ Vertical and roof bracing connections
- ❖ Purlin or girt connections
- ❖ Sag Rod Connections
- ❖ Batten Plate Connections

For building type multi-story structures, the recommended order to use IntelliConnect is given below;

- ❖ Splice Connections
- ❖ Beam-to-column connections (Moment connections first)
- ❖ Beam-to-Beam connections
- ❖ Horizontal bracing connections
- ❖ Base Plate Connections
- ❖ Vertical bracing connections
- ❖ Purlin, sag rods and batten plate connections

When working with IntelliConnect, it may be necessary to change the order for two reasons:

- ❖ The IntelliConnect operates on a joint. All connections on this joint are simultaneously taken into account. For instance, in the case of column-beam connection (beams connecting to both sides of the column) with vertical bracing, Intelliconnect will try to create all connections at the same time to prevent clashing by making use of common parts. That will change the detailing order.
- ❖ Since IntelliConnect operates on a joint, it is unclear whether the other joints are too close or too far away. If the members or joints are close enough to collide with each other, IntelliConnect may face conflict problems. In this case, the Clash Check command can be used to detect collisions. IntelliConnect can run with the user's requested order so that it does not use the connection information of other joints.

In summary, connection design should be initiated from the main connections. IntelliConnect can be triggered on grid and storey basis. The girt, purlin and sag rod connections can follow. Batten plate connections should be created finally.


26. Manual Connection

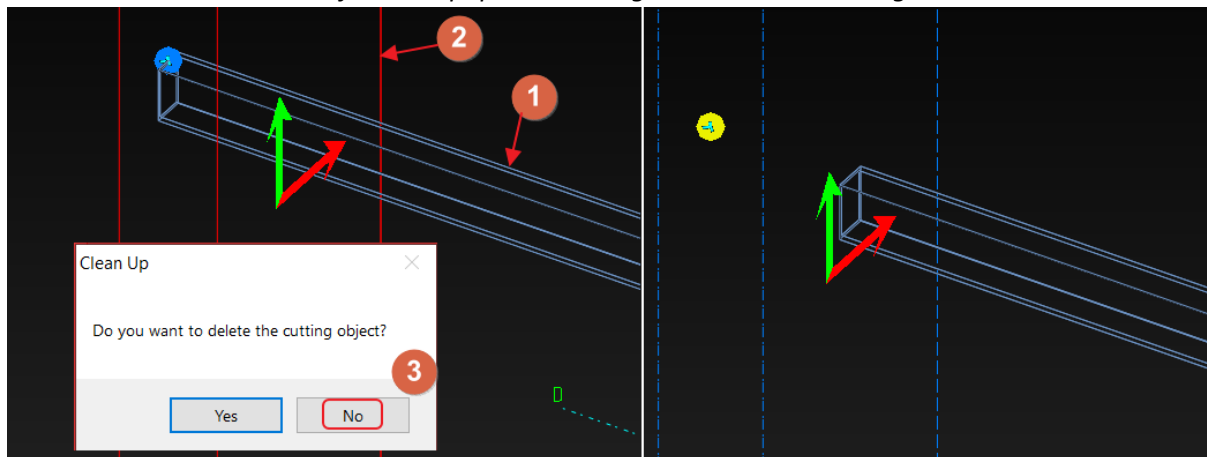
In ProtaSteel, user can also create connection manually and save them into macro library to re-use them at similar nodes. Example of steel beam to RC column manual connection will be demonstrated below.

Steel Beam to RC Column Manual Connection


Steel beam to reinforced concrete column connection can be done using manual embedded steel plate connection in ProtaSteel.

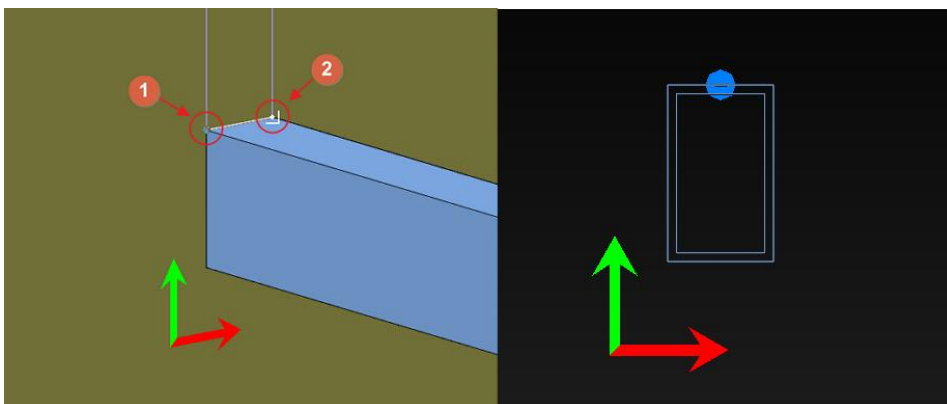
The following example illustrates the steps of creating embedded steel plate connection.

1. Firstly, we can cut of excessive steel beam using the **Cut This With That** command .
 - I. First, select Member to Cut.
 - II. Second, select the Cutting Member.
 - III. Select “No” from the pop out warning to remain the Cutting Member.

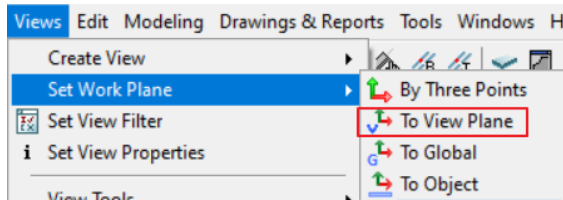



Note : Remember to turn off the “Cut” in Visibility Filter when necessary.

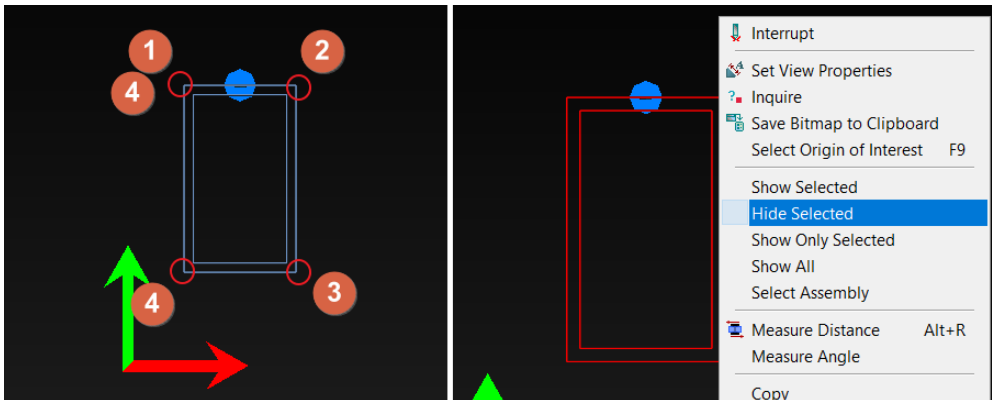
2. Create the section view plane with the command **By Two Points**  by accessed from **View** tab → **Create View** → **By Two Points**
 - Select point between end of the steel beam and the surface of concrete.
 - Double click on the background to open the **View Properties** → save View Name: **Cut Section A** or any preferred name. (Refer to **chapter 7 – Creating new Views**)



3. In the **Cut Section A View**, go to **Views → Set Work Plane → To View Plane**. This sets active work plane based on current view.




4. Select **plate** command  and select the 4 corners of the steel beam clockwise → A plate will be created. Then right click on the beam and select **Hide Selected**.

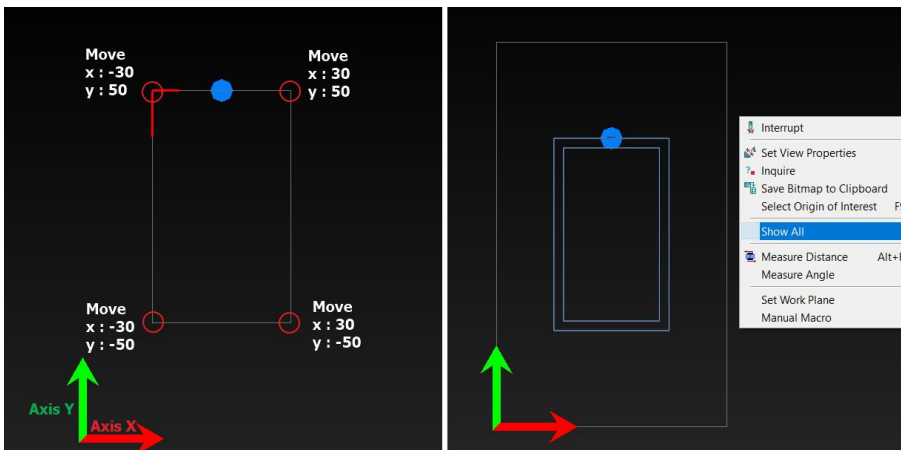


5. Use **ALT** to select each corner and right click to use **Move Special → Linear** to move each corner accordingly.

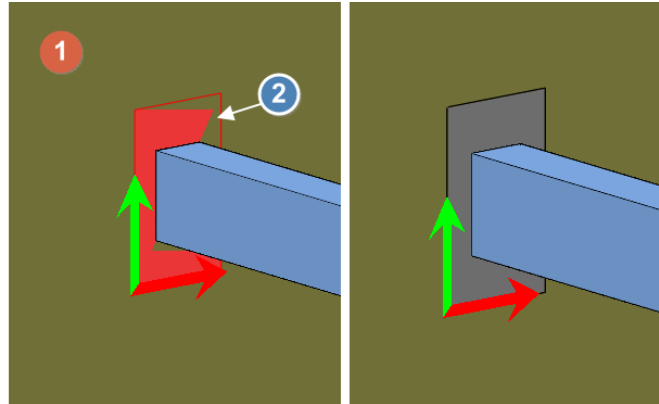
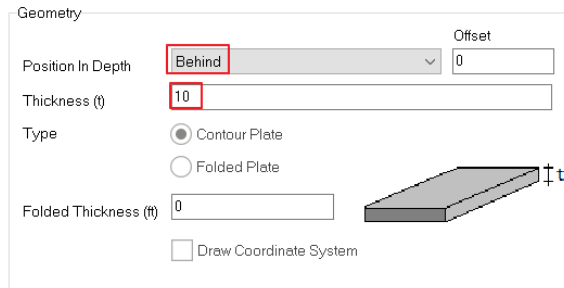
- Left Top Corner – X: -30, Y: 50
- Right Top Corner – X: 30, Y: 50
- Right Bottom Corner – X: 30, Y: -50
- Left Bottom Corner – X: -30, Y: -50


Note : Direction x, y, z must always check based on the workplane (axis direction icon ) showed in the modelling view. The axis can be based on colour: Red = x. Green = y. Blue = z.

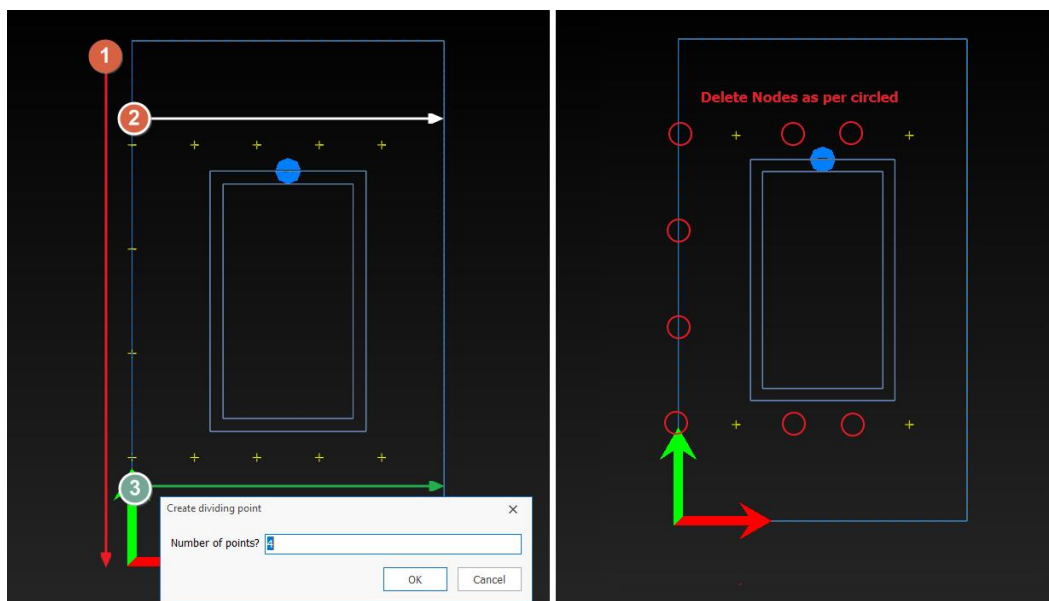
Then, right click at the background to **Show All**.



6. Right click on the plate, set:
 - *Position In Depth: "Behind"*
(Alternatively, can be adjust via hotkey **CTRL+W** in modelling view)
 - *Thickness (t): 10*
7. Go to **3D views**, remove the overlapping concrete using the Object Cut command.
 - *First, select concrete member.*
 - *Second, select the steel plate.*
 - *Select "No" from the pop out warning to remain the Cutting Member.*



8. Go to the **Section Elevation View**, right click on command "Create Dividing Points"  to set to "4" in **Number of Points?** and press **OK**. Left click on the command to start creating reference points.
 - **Step 1** - start at left top corner of the steel plate and end at left bottom corner of plate.
 - **Step 2** - start at the first point created from the top corner of plate, drag horizontally and end at the right side of the steel plate.
 - **Step 3** - repeat **step 2** but start at the Fourth node which created at the **step 1** instead.
 - Delete all points created in **Step 1**. Delete 2nd and 3rd node created in **Step 1** and **Step 2**

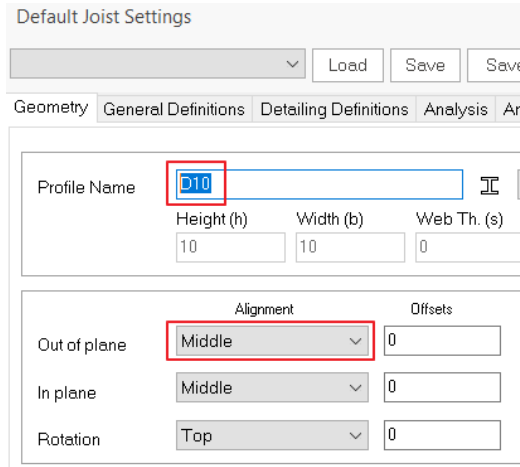
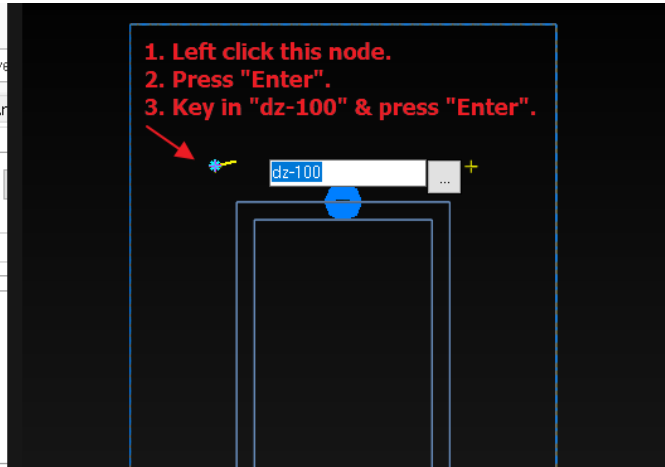


9. After finished creating the reference points, now we can create steel stud. Right click on **Joist**



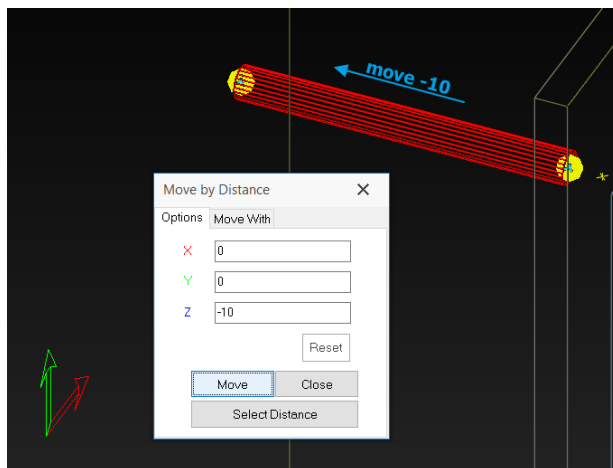
to access the **Default Joist Setting** and set:

- **Profile name:** **D10**
- **Out of plane:** **Middle**

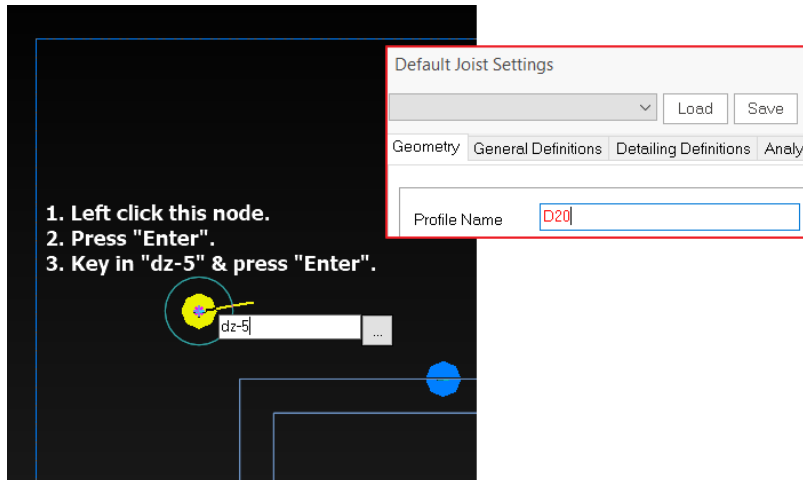



10. Left click on the reference point at left top corner to start creating the rebar → press **Enter** → key in “dz-100” and press **Enter** again.

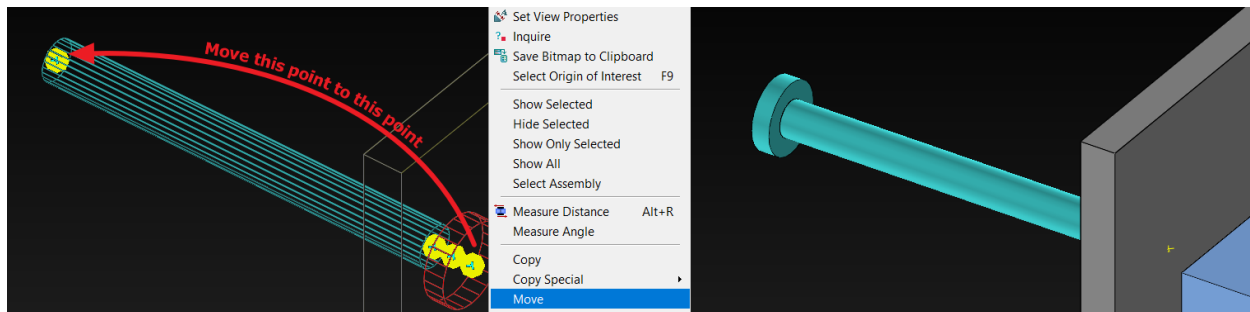
Then, go to **3D view** with **wireframe view mode**, move the stud by -10 on axis Z to avoid overlapping with steel plate
(Important to always check your workplane to move linear).




11. Go back to **Section Elevation View**, right click on **Joist** to access the **Default Joist Setting** and set **Profile name:** **D20**. Then, left click on the reference point at left top corner to start creating the rebar → press **Enter** → key in “dz-5” and press **Enter** again.



12. Go to **3D view** with **wireframe view mode**, select and right click the created circle to **move** it to end of the stud.

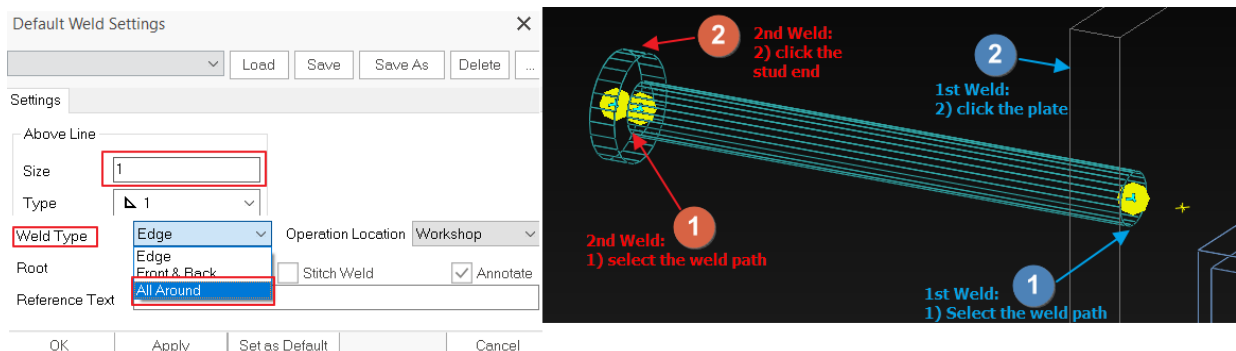


13. Right click **Weld** command  to access the **Default Weld Settings** and set:

- **Weld Type: All Around,**
- **Size: 1**

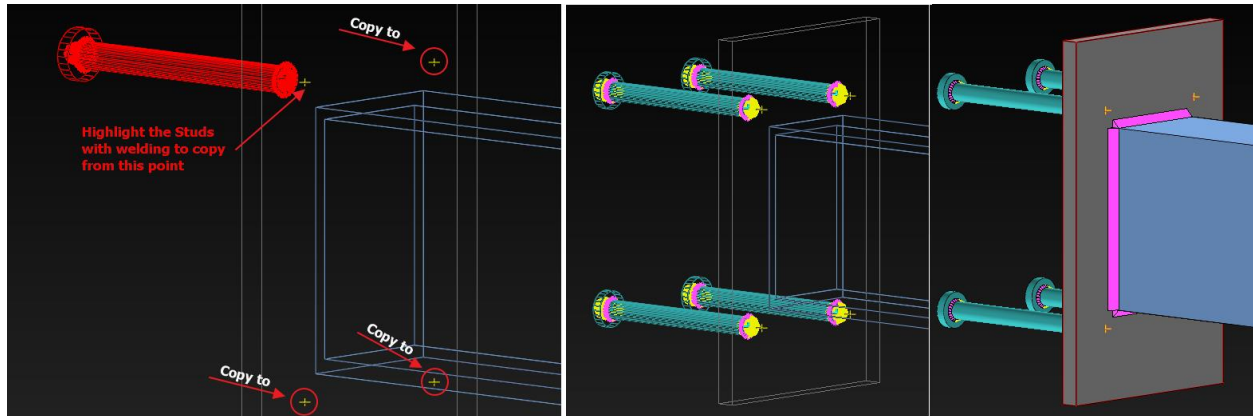
Click **OK** after setting. Start creating weld by Left clicking on the command:

- **Create Weld between Steel Plate and the stud:**
 - Click on the weld path that should be welded.
 - Click on the steel plate (which is the larger object)
- Repeat the same steps to create weld between stud and the stud end.




14. Once the welding is finished, highlight and copy the stud including with the welding and paste to the rest of the reference points.
15. Repeat the welding steps for the beam to the steel plate.

Note : You may right click the RC column → **Hide Selected** if necessary.
Right click on the background → **Show All** to reveal all the hidden item(s).



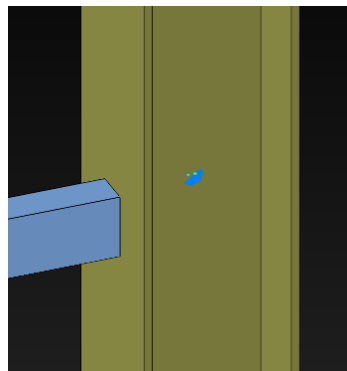
Steel Beam to Steel Column Manual Connection

Steel beam to steel column connection can be done using manual steel plate connection in ProtaSteel too. The steps is rather similar to **Steel Beam to Reinforced Column Manual Connection**.

1. We can cut of excessive steel beam using the **Cut This With That Boundary** command .

Note : (**Cut this With That** command is not suitable due to the cutting member shape is only able to cut the overlapping volume.)

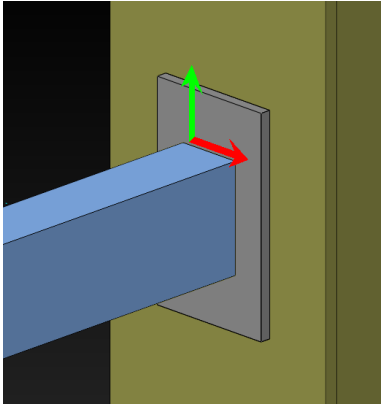
- I. First, select Member to Cut.
- II. Second, select the Cutting Member.
- III. Select "No" from the pop out warning to remain the Cutting Member.



2. Repeat steps 2 to 5 in **Steel Beam to Reinforced Column Manual Connection**.
3. Right click on the plate, set:
 - I. Position In Depth: "Front"
(Alternatively, can be adjust via hotkey **CTRL+W** in modelling view)

II. Thickness (t): 10

4. Use **Cut This With That** command to cut the beam and the plate. Select beam → Select the plate → select “No” in the warning.



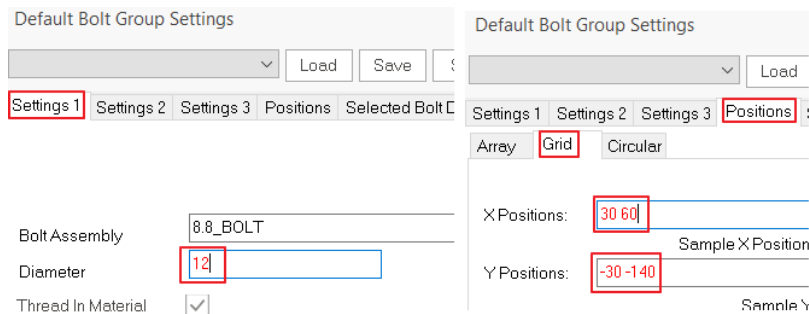
5. Now, we can create bolt using **Bolt Group** command , right click to preset the default setting:

I. In Settings 1 tab → Diameter: **12**

II. In Positions → Grid:

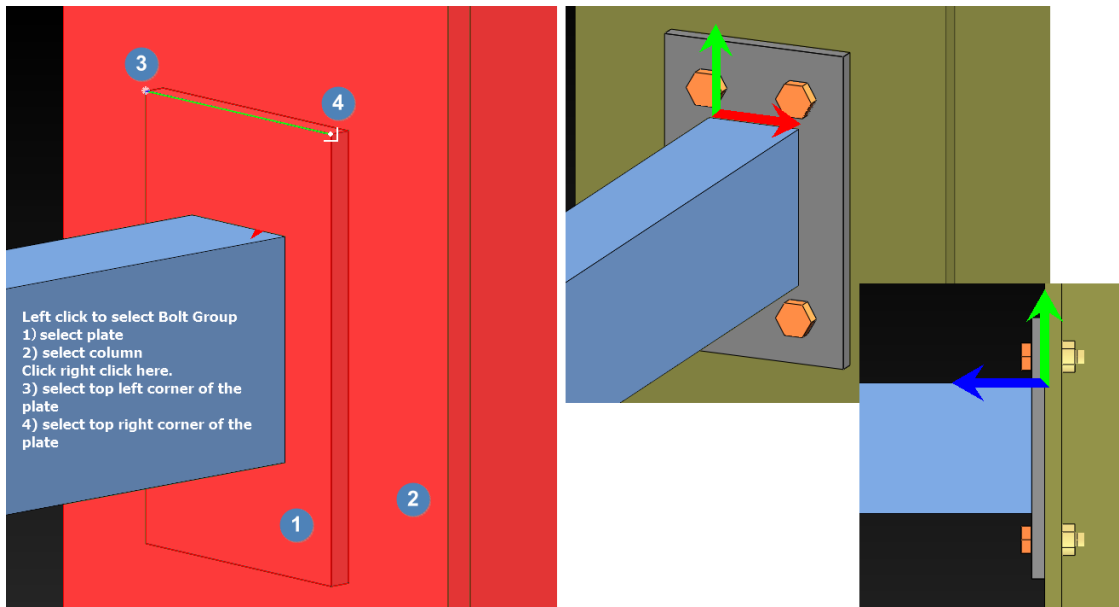
➤ X Positions = **30 60**

➤ Y Positions = **-30 -140**

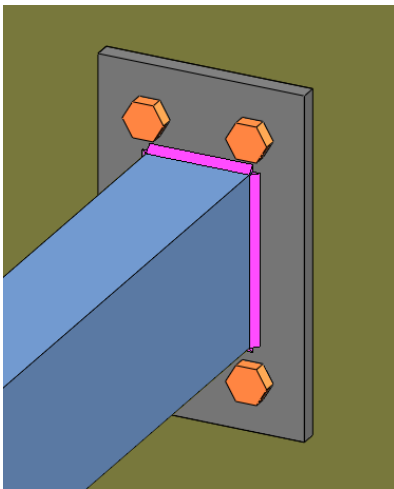


Click **OK** once finished.

6. Left click to use the **Bolt Group**.
 - I. Select plate and column. Then, right click once to enter next stage.
 - II. Select top left corner of steel plate and then select top right corner of the steel plate.



7. Lastly, weld the beam and the plate using **Weld** command.



27. Connection Design Reports

ProtaSteel can create design check reports for the following connections in accordance to EuroCode3, AISC (LRFD), AISC (ASD), BS5950:

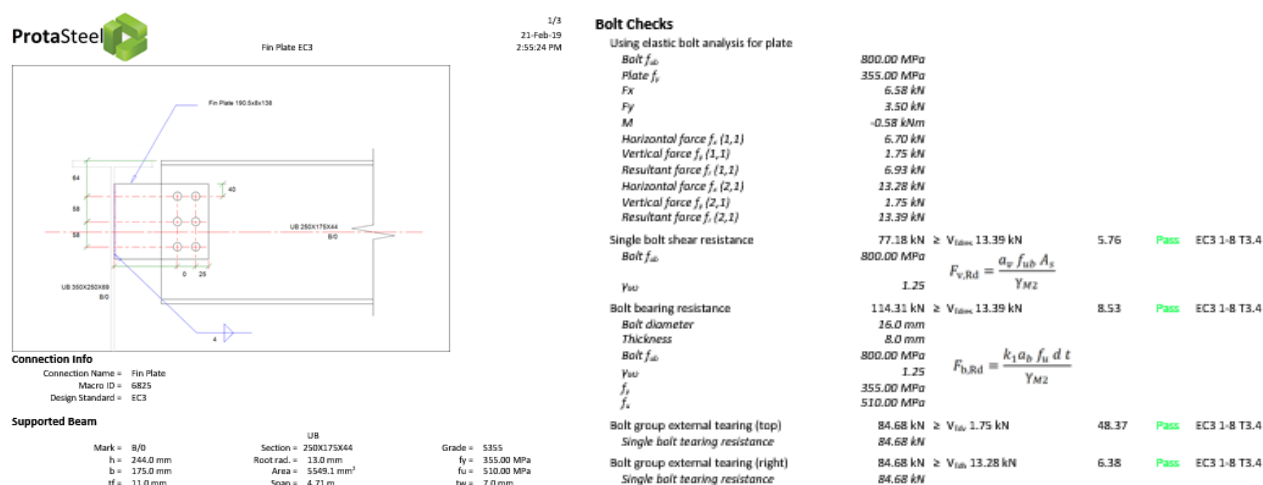
- Beam to Beam Connections: End Plate, Stiffened End Plate, Fin Plate
- Beam to Column Connections: End Plate, Stiffened End Plate, Fin Plate
- Beam to Beam End Plate Connection
- Haunch Connection
- Direct Welding Connection
- Simple Base Plate Connection
- Flange Plate Moment Connection
- Extended End Plate Connection
- Bolted Gusset Plate Connection
- Corner Bolted Gusset Plate Connection

- Welded Gusset Plate Connection
- Corner Welded Gusset Plate Connection

ProtaSteel can't create any calculation check for the below connection types.

- Base Plate Connection
- Eaves Beam Connection
- Fitting Connection
- Sag Rod Connection
- Purlin and Girt Connection
- Splice Connection
- Welded Pipe Connection
- Apex Haunch Connection
- Apex Truss Gusset Connection
- Truss End Plate with Gusset Plate
- Batten Plate Connection
- Stiffened Web Opening
- Multiple Stiffened Web Opening
- Beam to Wall Connection
- Embedded Plate Connection
- Manual Connections

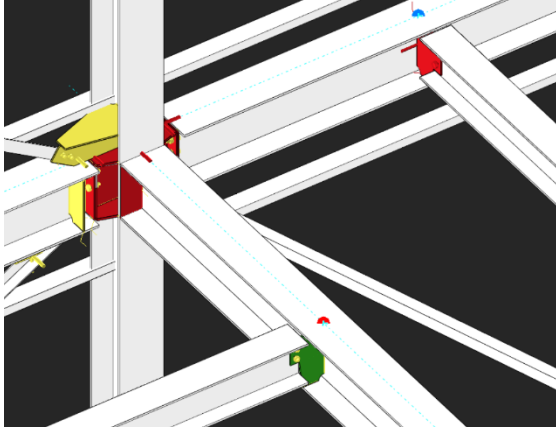
Reports include both geometric checks and capacity checks. In addition, intermediate calculations, references to the appropriate clauses of the design code, and corresponding equation are displayed for easy checking.




The reports can be created on demand in rich text format (RTF). Design Status colors can be seen on model view before report creation. 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

28. Connection Design Summary Table


Connection design summary table presents details about selected connection macros. The summary table shows the “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.

- **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**

Connection Design Summary Table



Frame IDs	Profiles	Macro IDs	Connection Type	Capacity Ratio	Design Status	Design Report
2C4 - 2X8 [L100X10]	HE200A - L100X10	10909		0.041 < 1	Pass	<input checked="" type="checkbox"/>
1B9 - 1X4 [L80X8]	IPE160 - L80X8	8894		0.011 < 1	Pass	<input checked="" type="checkbox"/>
1C3 - 1B1	HE200A - IPE240	10647	Haunch Connection Macro	1 >= 1	Fail	<input checked="" type="checkbox"/>
1B9 - 1B10S	IPE160 - IPE100	8837	Fin Plate Connection	1 >= 1	Fail	<input checked="" type="checkbox"/>
1B7 - 1X1 [L80X8]	IPE240 - L80X8	9308		0.012 < 1	Pass	<input checked="" type="checkbox"/>
1B6 - 1X1 [L80X8]	IPE240 - L80X8	8544		0.012 < 1	Pass	<input checked="" type="checkbox"/>
1B7 - 1X1 [L80X8]	IPE240 - L80X8	9286		0.012 < 1	Pass	<input checked="" type="checkbox"/>
1B2 - 1X1 [L80X8]	IPE240 - L80X8	8647		0.012 < 1	Pass	<input checked="" type="checkbox"/>
1B9 - 1X3 [L80X8]	IPE160 - L80X8	8975		0.009 < 1	Pass	<input checked="" type="checkbox"/>
1B6 - 1X3 [L80X8]	IPE240 - L80X8	8514		0.009 < 1	Pass	<input checked="" type="checkbox"/>
1B9 - 1X3 [L80X8]	IPE160 - L80X8	8941		0.018 < 1	Pass	<input checked="" type="checkbox"/>
1B2 - 1X3 [L80X8]	IPE240 - L80X8	8617		0.018 < 1	Pass	<input checked="" type="checkbox"/>

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

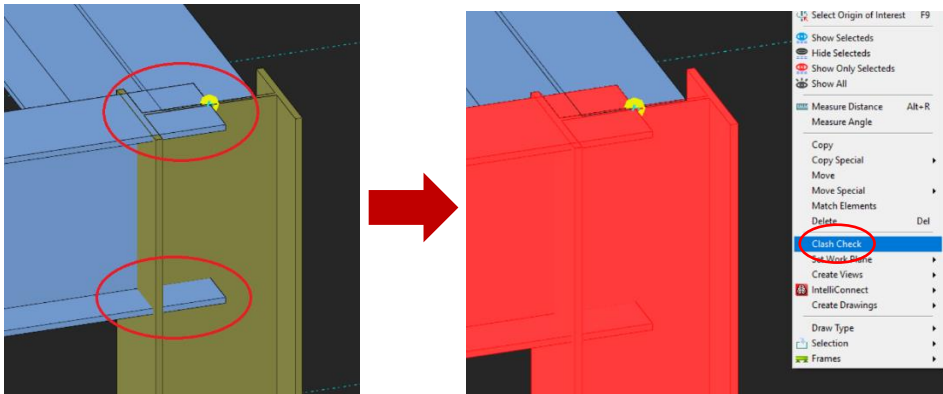
If the connection fails, change the macro properties. When the connection parameters are enough, the connection check again and color will be in green.

29. Collision / Clash Checks

Prior to creating detail general arrangement drawings, it is recommended that a collision check be performed for all elements. The collision check shows elements which are clashing or overlapping :

- **Select** all or some elements of the model
- Go to **Tools** (top menu) → **Diagnostic Tools** → **Clash Check**
- Alternatively, right click on the model screen and pick **Clash Check**

As a result, the clashing elements will be left selected (in red). Generally, collision occurs when two or more elements overlap each other. It usually occurs when connections are not made yet. For example, before the haunch connection was made, the beam end flange & web plate is overlapping into the column (as shown below).

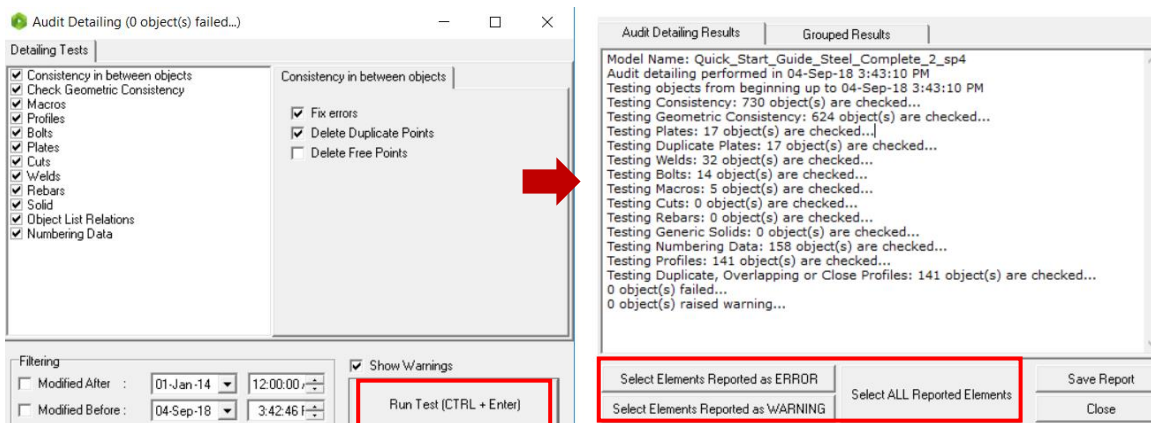


If a Clash Check is performed by selecting both the elements, they will remain selected after the check.

30. Audit Model

Before creating detail drawings, it is important to determine errors and inconsistencies that may exist in the model. This function will perform several model tests, such as duplicate points :

- Go to **Tools** (top menu) → **Diagnostic Tools** → **Audit Model**
- **Select the checks desired** → **Run Test**

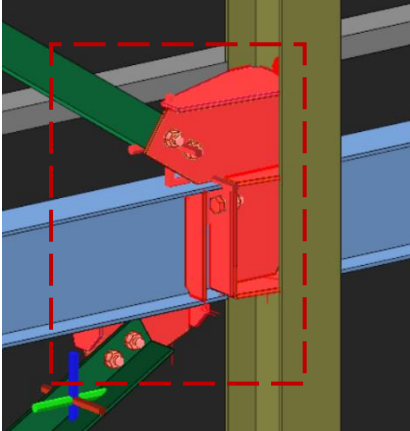



You can select the object(s) specified in the result report and automatically navigate to it in the model.

31. Detail Item Manager

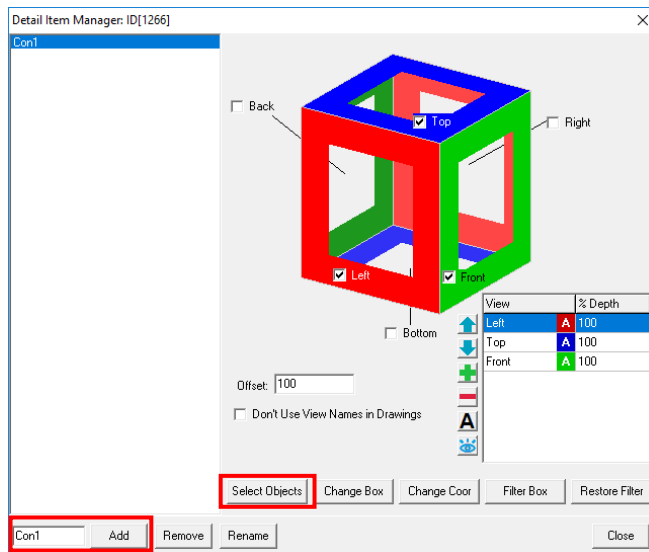


The Detail Item Manager creates and manages detail items such as the connections. We can use this function to create exploded drawings of the connection with different views & detail information such as dimensions, location of bolts, welds, etc.



- Zoom into the connection that we have created at **Grid C**
- **Click & drag a window** to select all components of the connections, including all the plate, bolts & welds
- Pick **Detail Item Manager** 

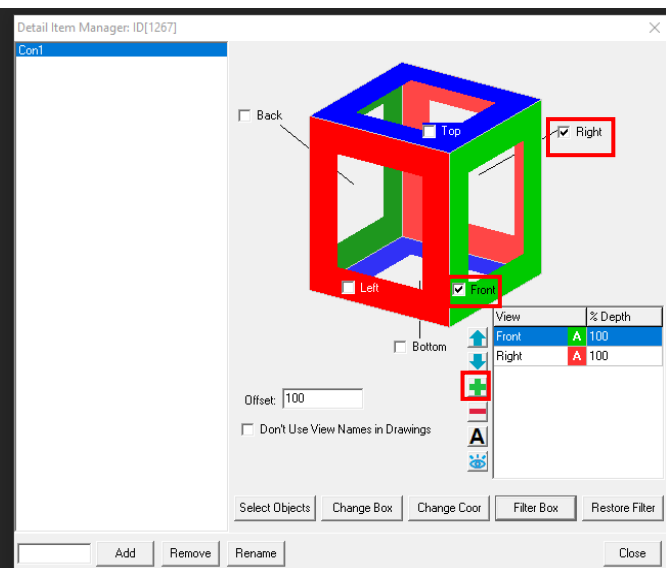
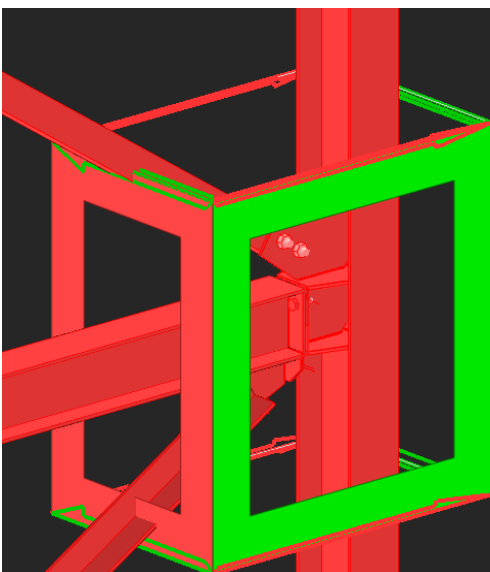
The Detail Item Manager dialog will appear.



In the Detail Item Manager :

- Type the name **"Con1"** at the bottom
- Pick **Add** → Con 1 will be added
- Pick **Select Objects**
- Select the **beam**, the **column** & the **2 braces**
- **Right-click**

The moment we do so, a "3D detail box" appears around the connection in the 3D view (as shown below)




This box controls the various directional views that we want to show in the drawings. The view is shown in 3 colors, red, blue and green. Each color has a front & back view. Hence there is a total of 6 possible

views. The view names can be changed – so if the top of the connection is not the actual top, change the name manually.

- For this connection, pick **Front & Right** only
- Pick **Filter Box** to show only the connection elements (**Restore Filter** will restore all elements)
- Click on “+” icon to add a 3D view of the connection (if desired)
- **Close** to save & exit the dialog
- Repeat the same steps for the **Splice Connection** at corner column at **GL D/3**

For practice purposes, we can similarly insert connection box on the connections we have previously created in Story 1. The summary of steps are :

- Open the view that is most suitable to insert the connection, try **Storey 1 view**
- **Zoom** into connection
- Click & drag a window to select all components of the connections, including all the plates & bolts
- Pick **Detail Item Manager** 
- **Add** the connection name
- Click **Select Objects** & **pick all the members** of the connection
- **Tick** the desired view direction box
- **Close** the dialog

We have completed the connections and are now ready to generate the drawings.

32. Preparing Engineering Drawings

ProtaSteel has a powerful tool to prepare engineering drawings – it's simply called the "Drawing module". The drawings are based on the views prepared in the modelling module. The steps are summarized as below :

- Create the desired **Views** in main modelling view.
- Ensure the orientation of the view you want to use in the drawing is correct as the drawing module will automatically use the latest views.
 - Double-click on **Axis 3** view → Press **F4** to reset the view to default
 - Double-click on **Storey 1** view → Press **F4** to reset the view to default
- Create any **detail views** such as connection details using **Detail Item Manager**
- Start the Drawing module by double-clicking **Drawings** in the Explorer pane (**Ctrl + D**)

33. Drawing Module

In the Drawing Module, the same views that was shown in the main model will be duplicated under the Explorer → **Modelling Views** :

- **3D View** – 3D view of the model
- **Storey F** – foundation plan view (ST00 in ProtaStructure)
- **Storey 1** – Storey 1 plan view as defined in ProtaStructure
- **Storey 2** – Storey 2 plan view as defined in ProtaStructure
- **Axis 3** – New view along Axis 3
- **Axis A** – New view along Axis A

If you do not have the same views, please close the drawing module to return to the main model & create the view as outlined in previous sections.

34. Creating Drawings via Drawing Manager

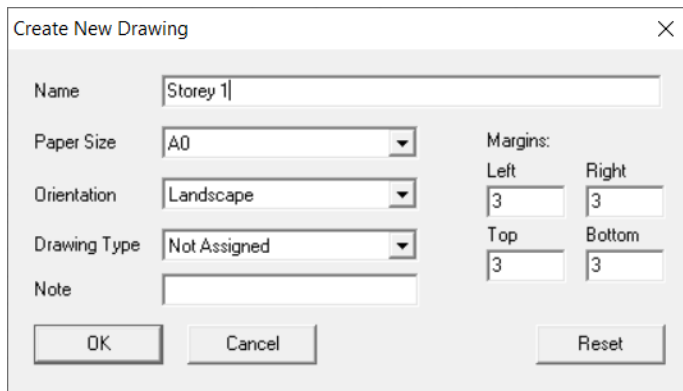
- Double-click on **Drawing Manager** (or **F6**)

The Drawing Manager is where you can start a new drawing.

The View Manager allows you to manage the drawings:

- Create new drawing using **Create Drawing** → Create Drawing Dialog will appear.

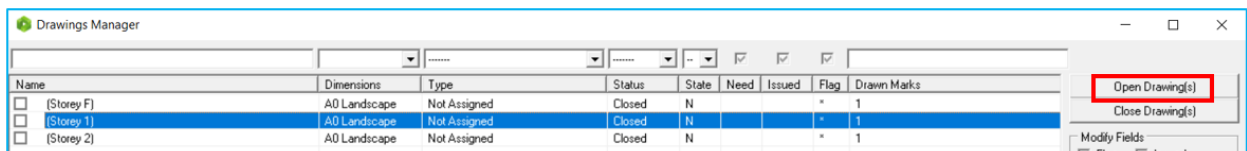
- Name the drawing **Storey 1**, Set **Paper Size** to A1 → click **OK**.



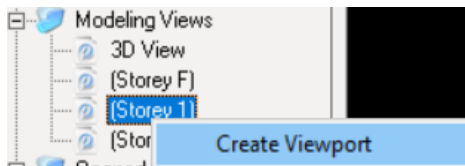
Create New Drawing dialog box showing the following settings:

- Name: Storey 1
- Paper Size: A0
- Orientation: Landscape
- Drawing Type: Not Assigned
- Note: (empty)
- Margins: Left: 3, Right: 3, Top: 3, Bottom: 3
- Buttons: OK, Cancel, Reset

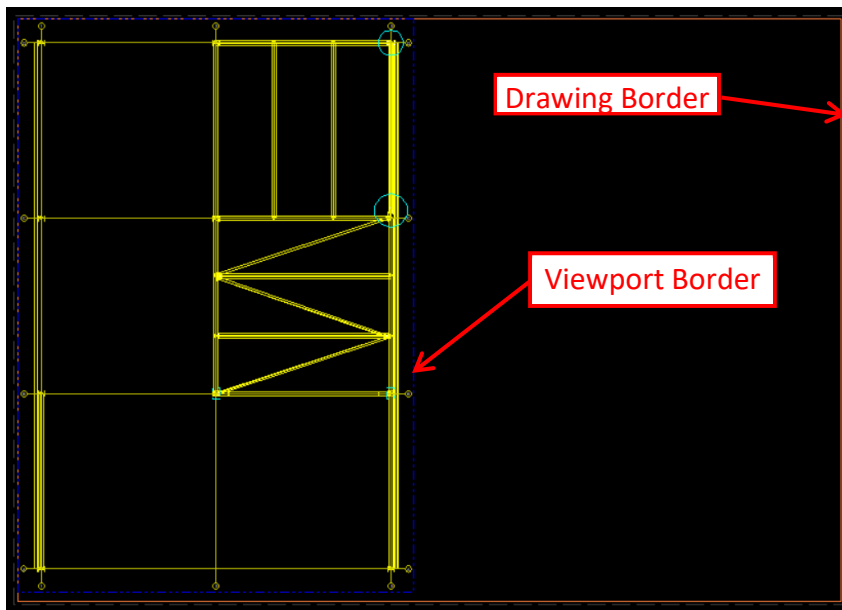
- Select **Storey 1** → **Open Drawing**



- Alternatively, simply double-click on **Storey 1** to open it.
- Right click on **Storey 1** located at the explorer panel → **Create Viewport**

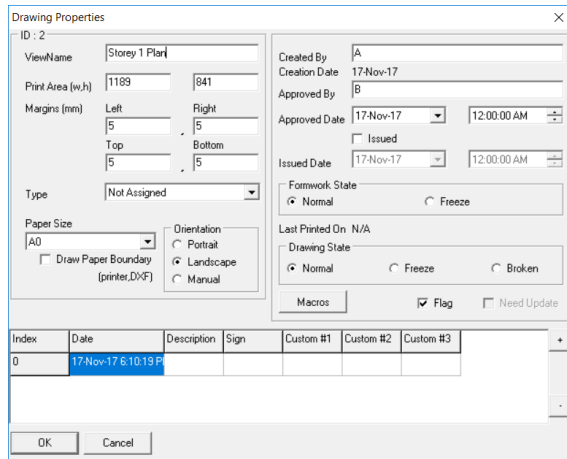


- Close the **Drawing Manager** & review the drawing.



The outermost brown line is the Drawing Border (A0 size). The size & other settings of the drawing can be changed by :

- **Double-click** on any empty space within the drawing border to access the **Drawing Properties**



Index	Date	Description	Sign	Custom #1	Custom #2	Custom #3
0	17 Nov-17 6:10:19 P					

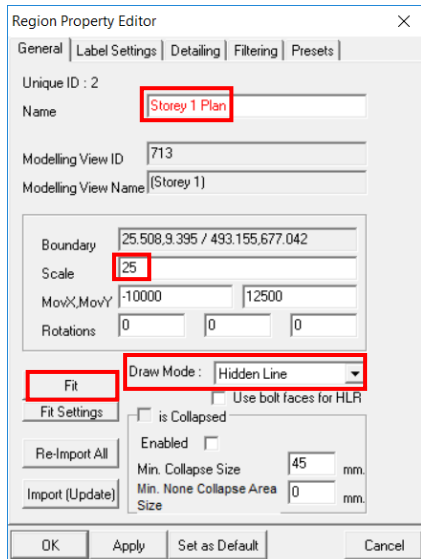
Documents Properties will appear where the following can be changed:

- Drawing Name
- Print Area & Margin
- Paper Size & Orientation
- Created, Approve by, Approved Date
- Document State : Normal / Freeze / Broken

The inner **dotted blue line** is the Viewport border which is directly linked to the view :

- Place the mouse cursor at the **lower-right corner** → cursor will change to resize ↘
- Click & drag to resize & ensure all the elements is visible
- Place the mouse cursor at **the boundary line** → click & drag to move the position of the region
- Double-click any place within the **Viewport** to access the **Viewport Properties**

Note : Region has been renamed **Viewport** in current release



Region Property Editor

General | Label Settings | Detailing | Filtering | Presets

Unique ID : 2

Name: **Storey 1 Plan**

Modelling View ID: 713

Modelling View Name: (Storey 1)

Boundary: 25,508.9,395 / 493,155,677,042

Scale: **25**

MovX, MovY: -10000 12500

Rotations: 0 0 0

Fit: ☒ **Draw Mode:** Hidden Line

Fit Settings: ☐ is Collapsed Use bolt faces for HLR

Re-Import All: ☐ Enabled: ☐

Min. Collapse Size: 45 mm.

Min. None Collapse Area Size: 0 mm.

Import (Update):

Viewport Proper Editor dialog will appear. In the General tab, the following can be changed :

- Name of the region
- Scale
- MovX, Mov Y : Coordinates of view
- Rotation : States the rotation angle
- Drawing Mode : Line/All Visible/Hidden Line/Full Dashed

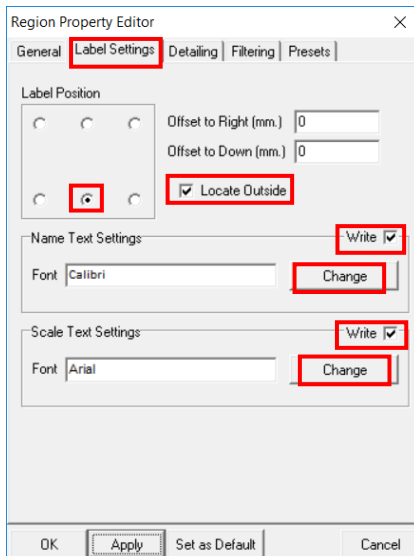
➤ Change the Name to **Storey 1 Plan**

➤ Change **Scale** to **25**

➤ Change **Draw Mode** to **Hidden Line** → **Apply**

The drawing will be updated to scale & hidden line.

➤ Pick **Fit** to auto fit all the elements & the new title



Region Property Editor

General | **Label Settings** | Detailing | Filtering | Presets

Label Position

Offset to Right (mm.): 0

Offset to Down (mm.): 0

☒ **Locate Outside**

Name Text Settings: ☒ **Write**

Font: Calibri **Change**

Scale Text Settings: ☒ **Write**

Font: Arial **Change**

➤ Go to **Label Settings** tab

The Label Settings dialog allow you to change :

- Label Position
- Offsets
- Name & Scale Text Settings

➤ Pick **Locate Outside** so the label will be place outside the region border

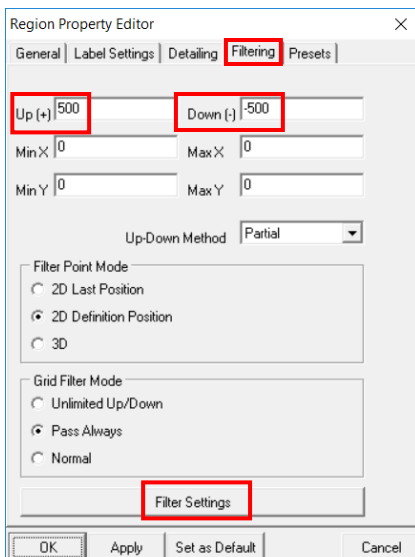
➤ Tick **Write** next to **Name & Scale Text Settings**

➤ Pick **Change** under **Name Text Settings**

➤ **Change Name** Text Height to **10mm** → **OK**

➤ **Change Scale** Text Height to **10mm** → **OK** → **Apply**

Notice that the name & scale appears larger in the drawing.



Region Property Editor

General | Label Settings | Detailing | **Filtering** | Presets

Up (+) **500** Down (-) **-500**

Min X: 0 Max X: 0

Min Y: 0 Max Y: 0

Up-Down Method: Partial

Filter Point Mode

☐ 2D Last Position

☒ **2D Definition Position**

☐ 3D

Grid Filter Mode

☐ Unlimited Up/Down

☒ **Pass Always**

☐ Normal

Filter Settings

➤ Go to **Filtering** tab

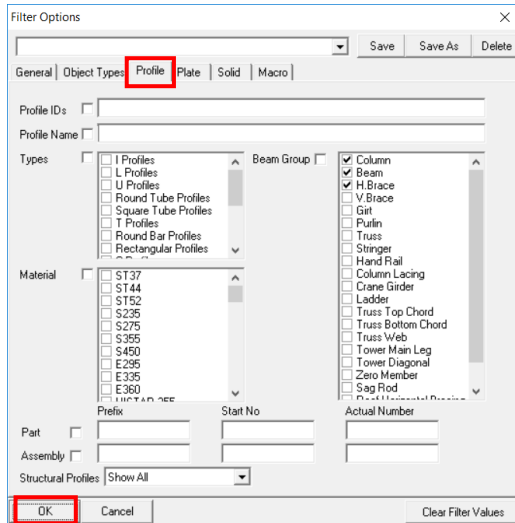
The options here controls what is shown in the drawing :

- Up (+) & Down (-) filters objects in & out of the paper
- Min X & Max X filters objects in the X plan direction
- Min Y & Max Y filters objects in the Y plan direction
- **Up-Down Method** :
 - **Partial** : Any element that exists partially within the above extents will be shown
 - **Full** : Only elements that exist fully within the above extents will be shown

- **Filtering Settings** – expose more filter options

➤ Change **Up(+)** to **+500** & **Down (-)** to **-500** → **Apply** → **Girts disappears as they are placed slightly above the floor level**

➤ Pick **Filter Settings**



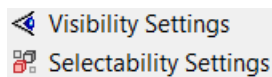
Filter Options enable advance filtering :

- General : Steel, Concrete, Unique ID, etc
- Object types : Grid, Profile, Joint, Weld, etc
- Profile : Columns, Beams, Brace, etc
- Plate : Plate IDs, Types & Usage
- Solid & Macro
 - Go to Profile tab
 - Tick **Column, Beam, H. Brace**
 - Pick **OK** to exit

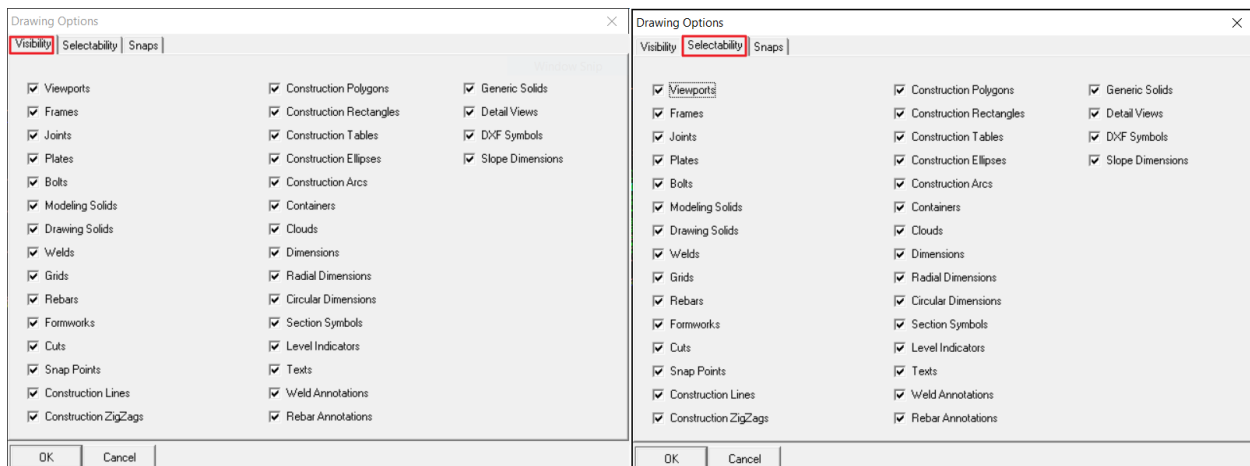
This ensures that only column, beam and horizontal elements are shown.

35. Visibility and Selectability in Drawing Module

Visibility settings allows user to show and hide elements that wish to hide/show in the drawings, while **Selectability settings** allows user to lock the elements and prevent from mis-click or accidentally deleting the elements.



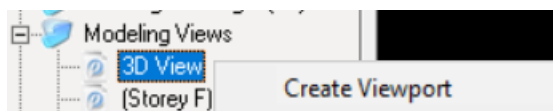
By default, all elements will be ticked. If selected element is unticked and press “OK”, the effect will immediately apply to the selected element in the drawing.



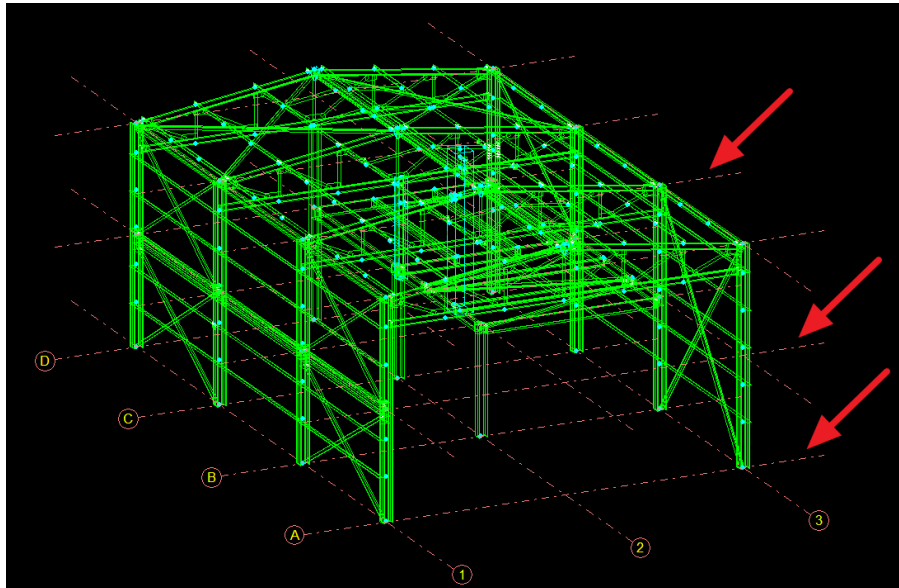
Grid Level Visibility

Now, we will insert the 3D view into the drawing.

- Right-click on **3D View** → **Create Viewport**



In the 3D view, the grids are appeared in each storey.

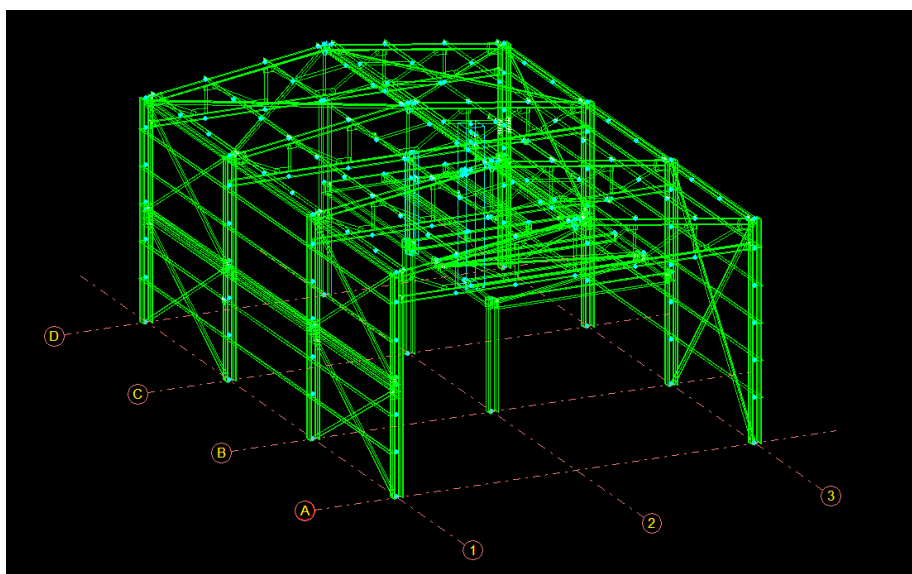


To show the grids at the storey 0 only,

- Double click on the grids → **Grid Properties Editor** will appear.
- **Untick 1. Level and 2. Level** under **Grid Level Visibility** → Click OK.




- The 3D view will now looks clean.

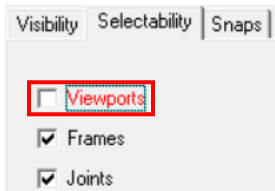


36. Profile Annotation

We now add steel profile annotation, i.e. show the member sizes. Firstly, we should select the members we want to annotate. To select the members easily, we can control what elements is selectable:

- Click on **Selectability** icon  at the top menu (or **View** → **Selectability Settings**) to access the **Drawing Options** dialog

Drawing Options

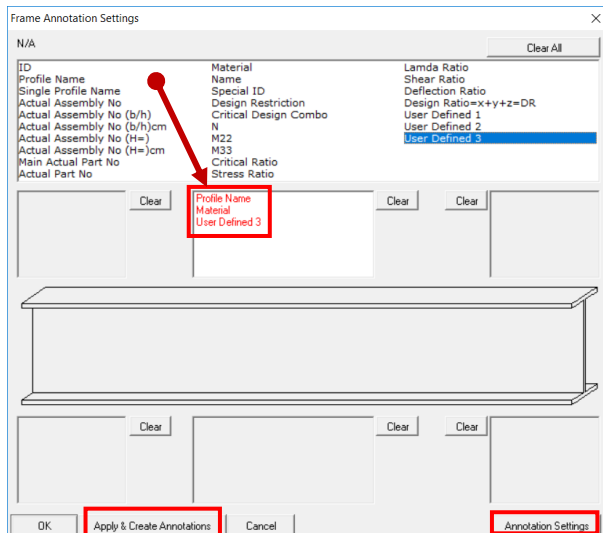


- **Untick Viewport** → **OK**

This prevents you from selecting the region border.

The same options are available in **Visibility** tab which controls what is shown in the drawing.

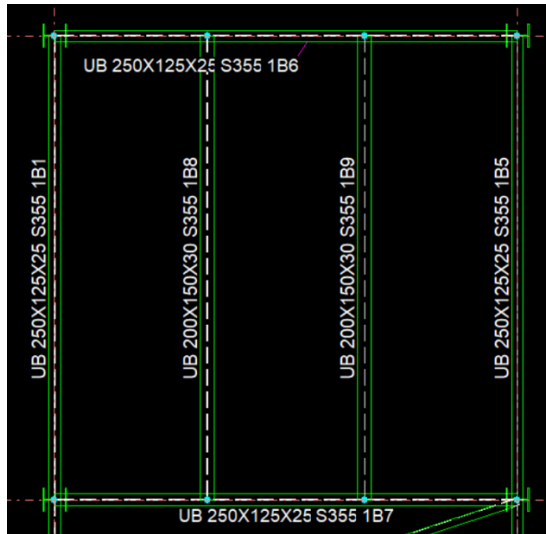
- In the drawing, **left click & drag a box** around all the steel members to select them
- Right-click on **Profile Annotation** icon  to access the **Annotation Settings**




The settings here controls how steel members are annotated (labelled) :

- The various type of annotations are listed in the table
- Click & drag any annotation name into the box which also defines the position of the annotation
- Click and drag **Profiles, Material & ProtaStructure Label** in the box
- Click **Annotation Settings** to view or change the text settings :
 - Font type & size
 - Format or style of the annotations
- Change the **Text Height** to **7 mm**
- Click **Apply & Create Annotations**


The annotations are added next to all steel members as shown below.




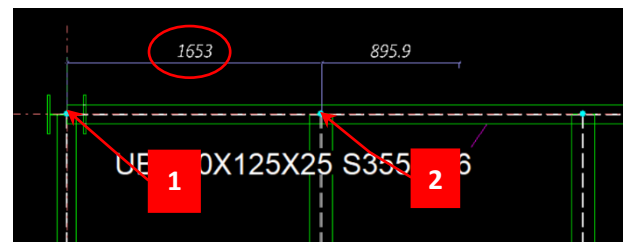
Notes :

- ❖ Alternatively, click **OK** to exit the Frame Annotation dialog → Select elements → **Left-click** on the **Profile Annotation** icon 
- ❖ **ProtaStructure Labels** are member labels such as 1B1, 1C1
- ❖ The annotation position can be moved by **left click & drag**
- ❖ Annotations can be deleted using **Delete** key
- ❖ Annotation text size, font height, etc can be changed by selecting them → **ENTER** → **Annotation Text Property Editor**

37. Dimension

Dimension tools  can be used to insert dimensions in the drawing.

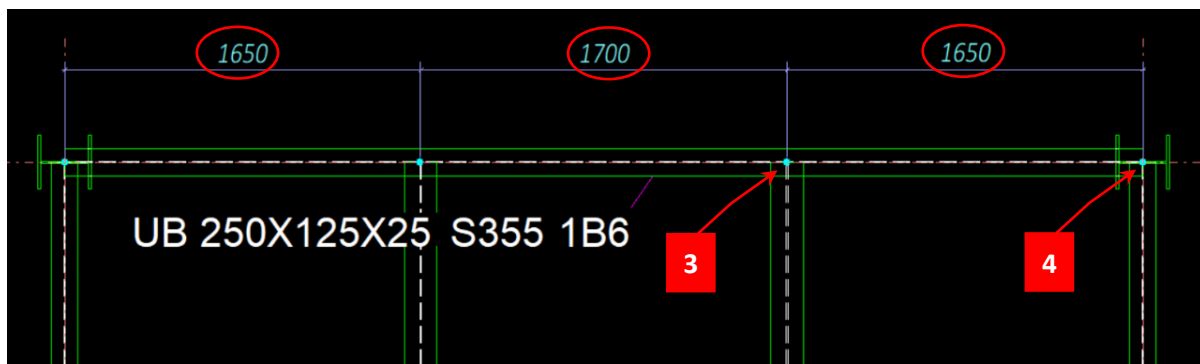
- Zoom in the top around primary beam along axis D
- Click **Horizontal Dimension** 
- Left click on the **1st** point & then the **2nd** point between the points to be dimensioned.
- Move the mouser cursor away to the right



Notice the dimension is shown dynamically.

- Left click on the **3rd** point & then the **4th** point
- **Move** the mouse cursor to the **exact position** where you want the text to be placed
- **Right-click** to confirm & create the dimensions

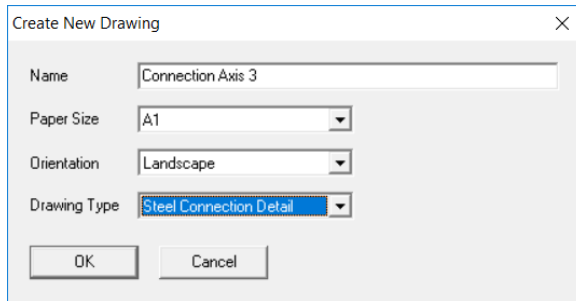
The dimensions will be created as shown below.



38. Creating Connection Detail

Let us now create a new connection drawing for the connection we created along **Grid 3**.

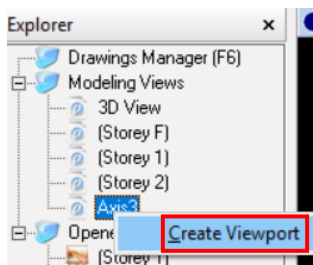
- Double-click on **Drawings Manager** (or **F6**)
- Pick **Create Drawing** → Create Drawing Dialog will appear



- Name it "**Connection Axis 3**"
- Change the **Paper Size** to **A1** → Orientation = **Landscape**
- Pick Drawing Type = **Steel Connection Detail**
- Pick **OK** → The drawing will be created & automatically opened
- **Close** the Drawing Manager dialog (Pick OK if prompted to save)

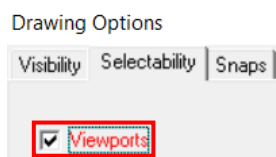
The drawing "Connection Axis 3" will be opened and brought to focus. The drawing is empty & we will insert the required view into the drawing.

- **Right-click** on **Axis 3 View** → **Create Viewport**

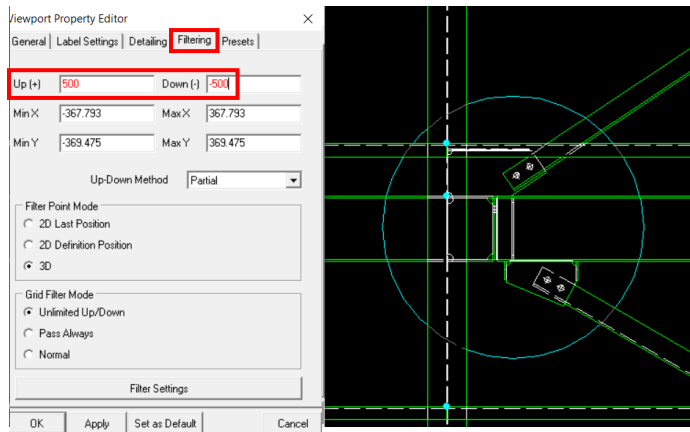


The view region will be inserted but region may not be positioned correctly. Let us move it within the drawing border. In order do this, we need to be able to select the blue region border.

- Click on **Selectability** icon  at the top menu to access the **Drawing Options** dialog



- **Tick Viewport** → **OK**
- **Click & drag** the viewport into the Drawing Sheet

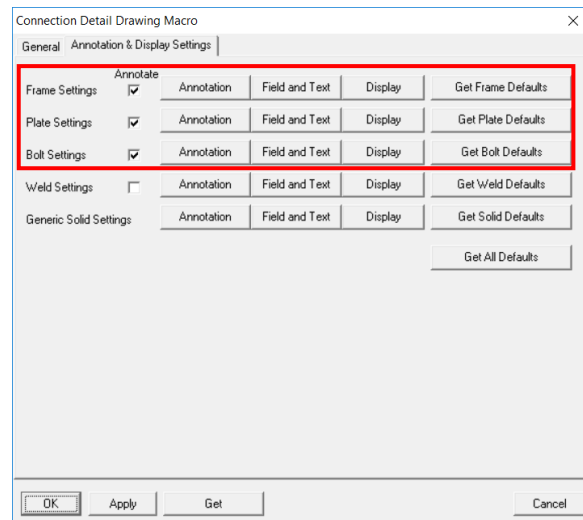
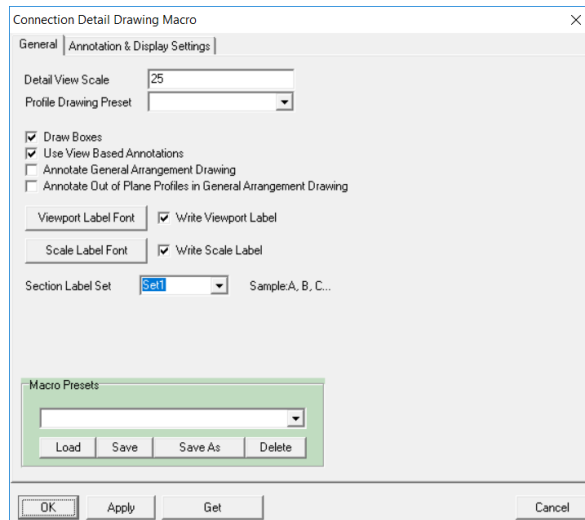



- **Double-click** on the region to access the **Viewport Property Editor**
- Scale can be change but let's leave it as default
- Go **Filtering** tab → change **Up = 500** & **Down = -500**
- **OK** to close the dialog

Notice there is a circle around the connection
– this is directly linked to the connection box created in the main model.

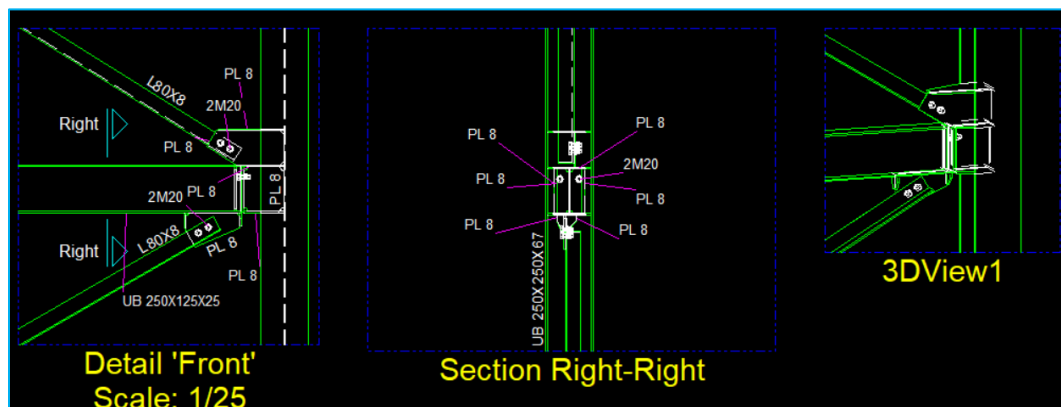
- **Right-click** on Connection Detail Drawing macro icon 

In this dialog you can set the details of the connection drawing such as the scale and Annotation

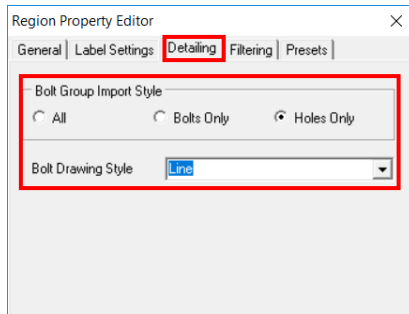


- Under **Annotation & Display Settings**, review **Annotation**, **Field and Text** & **Display**
- Click on **Connection Detail Drawing** Macro icon 

The Connection Detail will be inserted into the drawing as shown below.



Further customization of the connection detail can be done.

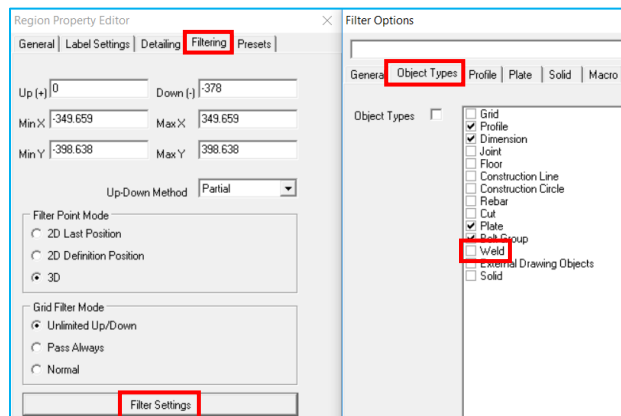


- **Double-click** in the detail region of the connection to access the **Viewport Property Editor**

Under **Detailing** tab, you can change :

- **Bolt Import Style**
- **Bolt Drawing Style**

- Try the various options & click **Apply**



- Go to **Filtering tab** → Pick **Filter Settings**

- Go to **Object Types**

You can choose to turn on/off objects.

- **Untick Weld** → **OK** → **Apply**

The welds shown in thicker lines will be hidden.

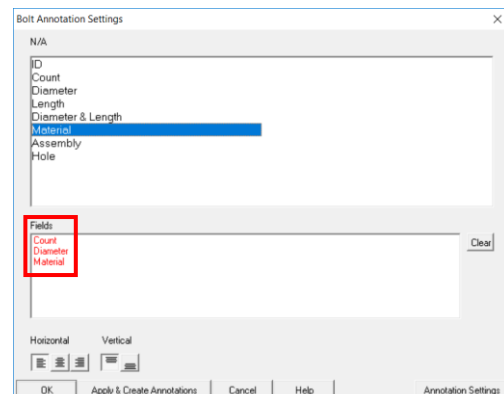
Plate, Bolts & Weld Annotation Tools




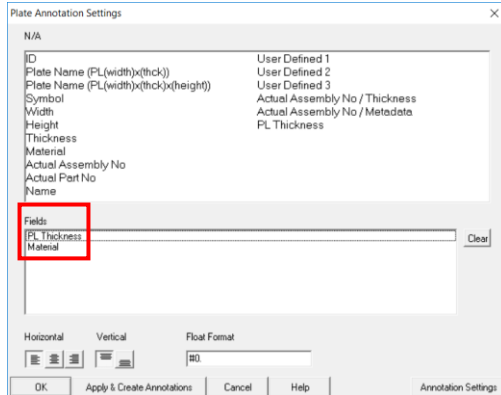
The bolts, plates and welds are automatically annotated in the above example. If you wish, you can manually do so by using the plate, bolts & weld annotation tools




- Hold down **CTRL** key → select all the annotations of the “Front” detail → delete them
- **Click & drag a window** (right to left) to select bolts & plates to be annotated (does not matter if other objects are selected)

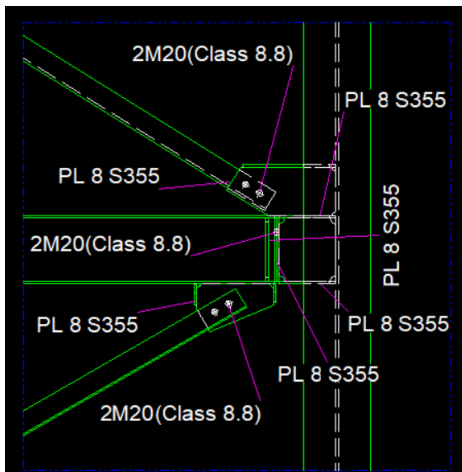


- **Right-click** on **Bolt Annotation**  access the **Settings**
- Pick **Clear** to remove existing settings
- Click & Drag **Count**, **Diameter** & **Material** to the bottom Fields
- Pick **Annotation Settings** to view/change font type, color, etc → **OK**
- Pick **“Apply & Create Annotations”**



- Right-click on **Plate Annotation**  access the **Settings**
- Pick **Clear** to remove existing settings
- Click & Drag **PL Thickness**, & **Material** to the bottom **Fields**
- Pick **Annotation Settings** to view/change font type, color, etc → **OK**
- Pick **“Apply & Create Annotations”**

You should get details which are similar to figure shown below (depending on your preference).



The drawing can be further improved by :

- Go to **Viewport Properties Editor** and change the **Scale**
- Change **Offset (X,Y)** to reposition the connection
- **Delete** any annotations that you think is not required
- **Click & drag** the annotation to reposition it

39. Numbering Objects

Before parts and assembly views & drawings can be created, the objects must be numbered. By default, all objects are included in the same numbering group. To assign objects to different groups:

- Double-click on object to open the **Profile** dialog
- In the **Detailing Definitions** tab, input **Drawing Group** number

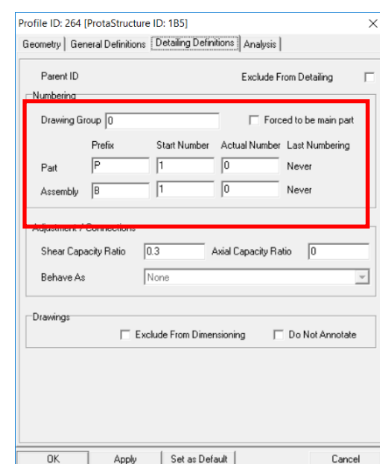
This feature was developed to number the different parts of the structure separately.

For each group of objects, the Prefix & Start Number can be specified.

After numbering process, the actual part and assembly number will be shown under **“Actual Number”**.

- Go to **Drawings & Reports** (top menu) → **Numbering** → **Perform Numbering**

The Numbering Options dialog will appear.



Numbering Options						
Group	Do Numbering	Unique Number	Renumber All	Reuse Old Number	Modified Part	New Part
0	YES	Not Unique	Modified	-	Keep Number If Possible	Compare to Old
1	YES	Not Unique	Modified	-	Keep Number If Possible	Compare to Old

As the model is continuously being revised, the numbering process need to be repeated. **ProtaSteel** has extensive options for re-numbering. Numbering Options can also be configured according to the preferences of the manufacturer. For example, there is option to force geometrically identical part to have the same number.

Do Numbering

- ❖ Specifies whether to number the selected group

Unique Number

- ❖ **Unique:** Can be selected for reinforced concrete elements. Objects are given a different number regardless of its geometric properties (beyond the scope of **ProtaSteel**)
- ❖ **Not Unique:** Should be used for steel model. Makes numbering by looking at geometry and material properties.
- ❖ **Spatial Sort:** Can be selected for reinforced concrete model. Performs numbering according to the object's position. The Object's reference point is sorted first by X and then by Y. Different groups should be made for different levels. Consecutive numbered column and beam applications can be an example.

Renumber All

- ❖ **Modified:** Number only the modified objects. Ignores objects that have not changed.
- ❖ **Renumber:** Number all objects.

Reuse Old Numbers

- ❖ **Reuse:** Uses old numbers.
- ❖ **-** : Don't use old numbers. After the changes, there may be no elements belonging to a number. Specifies whether to use this number for a new element.

Modified Part

As an example, we have 3 nos of P5, 2 nos of P7; then 3 nos of P5 have the same change and have become the same as the P7.

- ❖ **Keep Number If Possible:** Retains the numbers if possible after the change. If setting is used, the result is still 3 nos of P5, 2 nos of P7.
- ❖ **Compare to Old :** Compares and assigns the part-marks from previous numbering if possible. If this option is used, the result is 5 nos of P7.
- ❖ **Take New Number:** Assigns new part-marks. This option will lead to 2 nos of P7 and 3 nos of P8.

New Part

Similar to “Modified Settings”. Only applies to newly created objects. Hence, there is no option to “Keep the Number If Possible”.

- ❖ **Compare to Old** : Compares and assigns the part-marks from previous numbering if possible.
- ❖ **Take New Number**: Assigns new part-marks.

40. Creation of Parts and Assembly Drawings

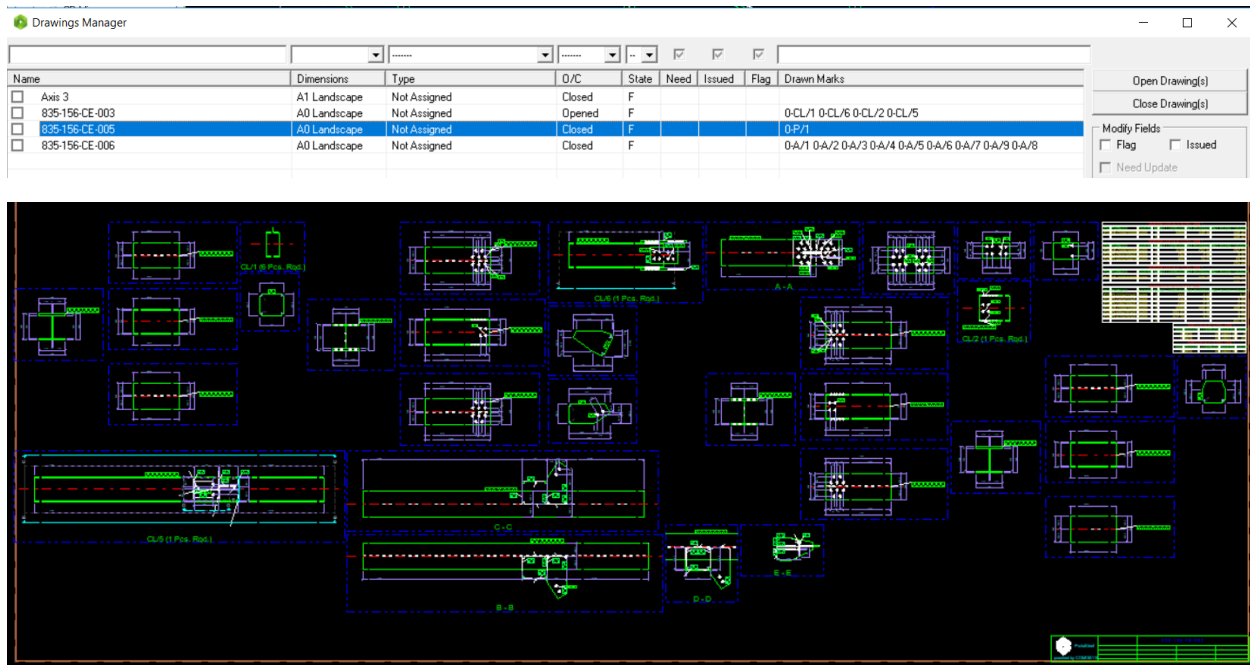
If there are no problems after Audit Model, Clash check and Numbering, the parts and assembly drawing can be produced.

You can create the part & assembly drawing for the entire model, or only for some objects.

- *Select the entire model or objects*
- Go to **Drawings & Reports** → **Drawings** → **Create Assembly Drawing / Create Part Drawing**

The Drawing Module will be launched and the drawings will be created.

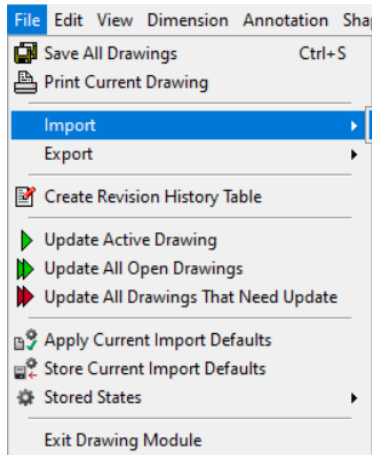
- In the **Drawing Manager** → Double-click to open & review the drawing



41. Import Titleblock using DXF Import

In ProtaSteel, user can import titleblock and print drawings directly.

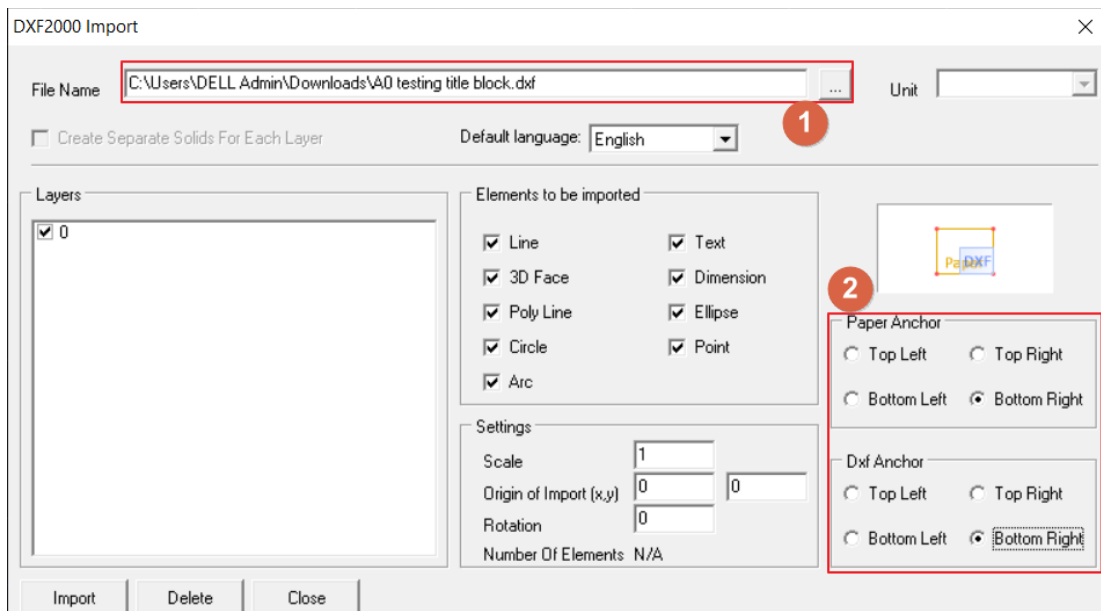
- User can simply import company standard titleblock as DXF in **Drawing Module** by going to **File** → **Import** → **Import DXF**



- Select the company standard titleblock DXF directory.

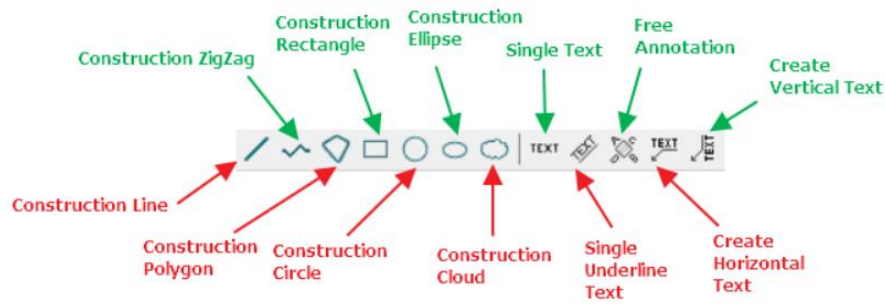
Note : Ensure the titleblock drawing should contain only titleblock, not in block format and drawing dimension unit is correctly set.

- Paper Anchor and DXF Anchor set to **Bottom Right**.



- Titleblock annotation can be insert/modify using "Text" Function.

Points, Construction Objects and Text



Points, Construction Objects and Text allow user to add details to the drawings such as the opening dash line, "to be continued" line, revision cloud, description and etc.

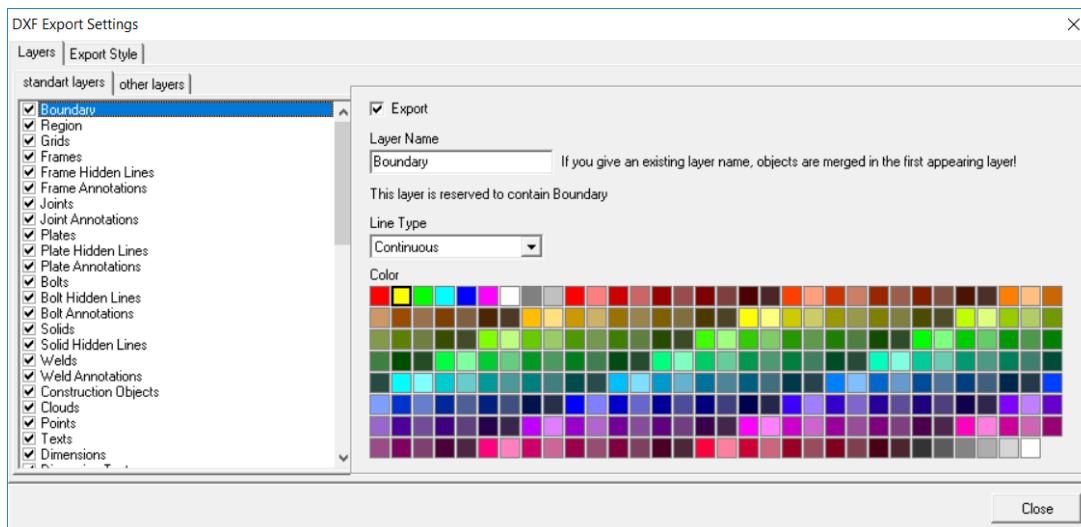
- If you wish to select a point where has no object snap, you can hold **left ALT** to select a free point.
- If you wish to draw it horizontally, please hold **left CTRL** before selecting the start point.
- If you wish to draw it vertically, please hold **left SHIFT** before selecting the start point.

42. DXF Export

All the drawings created in ProtaSteel can be exported as DXF format:

- Double-click on **Drawing Manager** (or **F6**)
- Under **DXF Export** → Pick **Settings**

In the DXF Export Settings, you can choose the layers to be exported, change the Layer Name & Line Type.



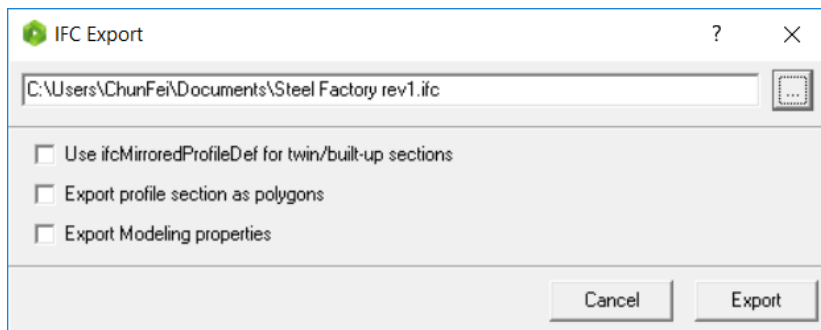
- **Close** the DXF Export Settings
- Pick **Export** → Choose the destination folder → **OK**

The dxf drawing will be created in the folder selected.

43. IFC File Export

ProtaSteel model can be exported as Industry Foundation Classes (IFC) file. IFC is a Building Information Modelling (BIM) data structure created by buildingSmart International and is the ISO 16739: 2013 standard. 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. This enables sharing of building information models between various BIM platforms.

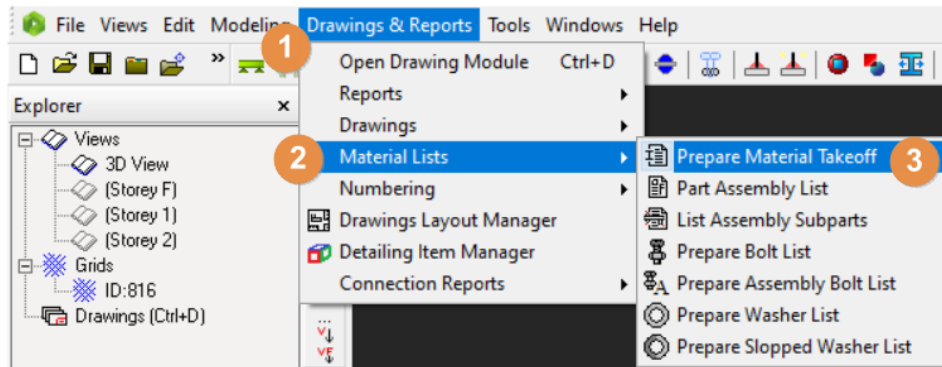
- **Select objects to be exported as IFC file format.**
If no object is selected, then all objects will be exported.
- **The File → Export → IFC Export**



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

44. Material Take off

ProtaSteel automatically creates material lists of the whole of the steel structure. **Material Lists** is located under the **Drawing and Reports** tab.

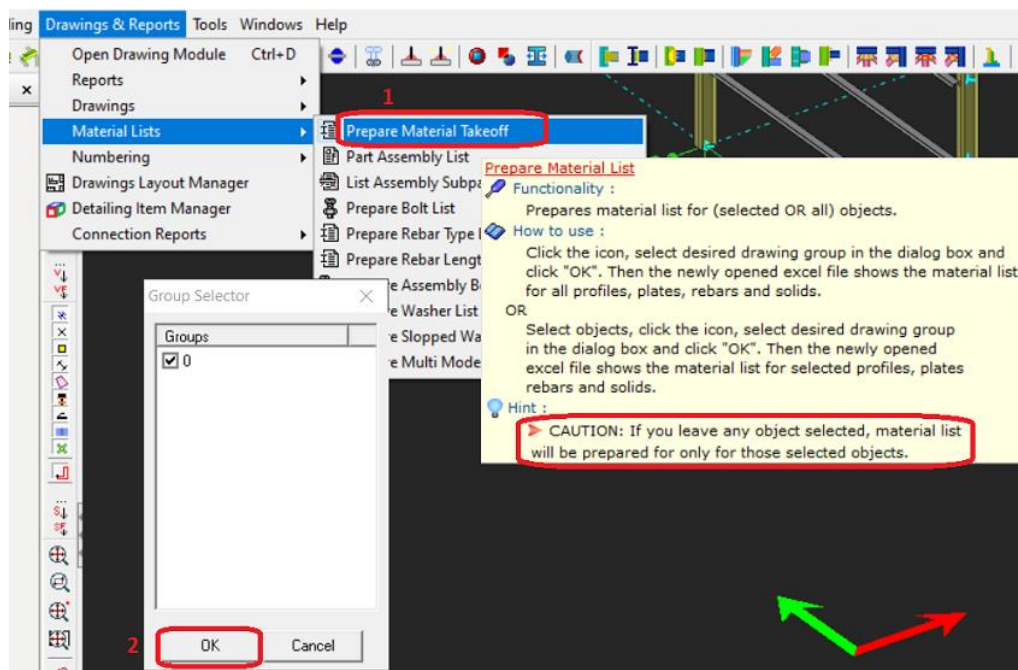


There are two methods to prepare for your Material Takeoff and will be presented below:

Method 1

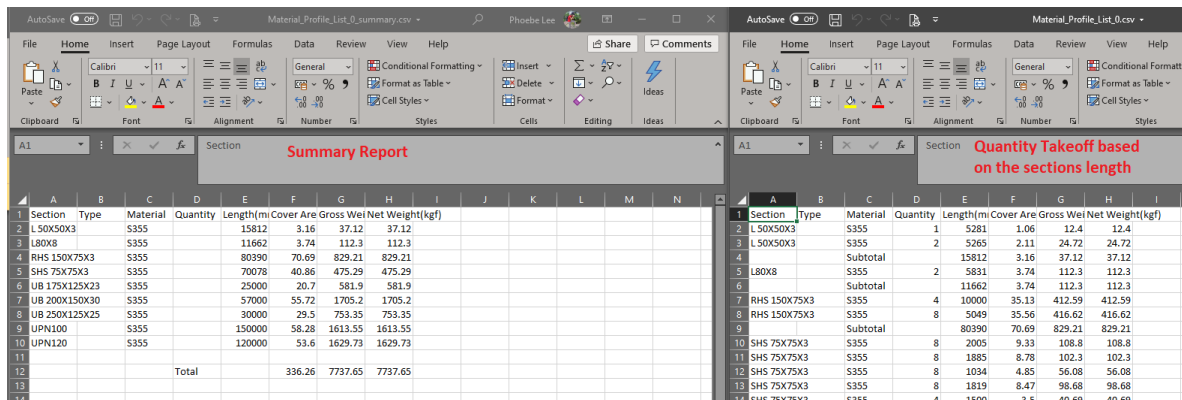
To generate quantity takeoff, for example **steel members** which will be sorted by sections used in project, please refer to following steps: -

1. Go to **Drawings & Report** → **Material List** → **Prepare Material Takeoff**
2. Click **OK** on the **Group Selector** Dialog.



3. Two Quantity Takeoff Tables (with different format) will automatically export as CSV file.
 - Summary Report and Quantity Takeoff based on the section length

Note : Repeat step 1 to 3 for the other Material Lists option in order to export to CSV file.

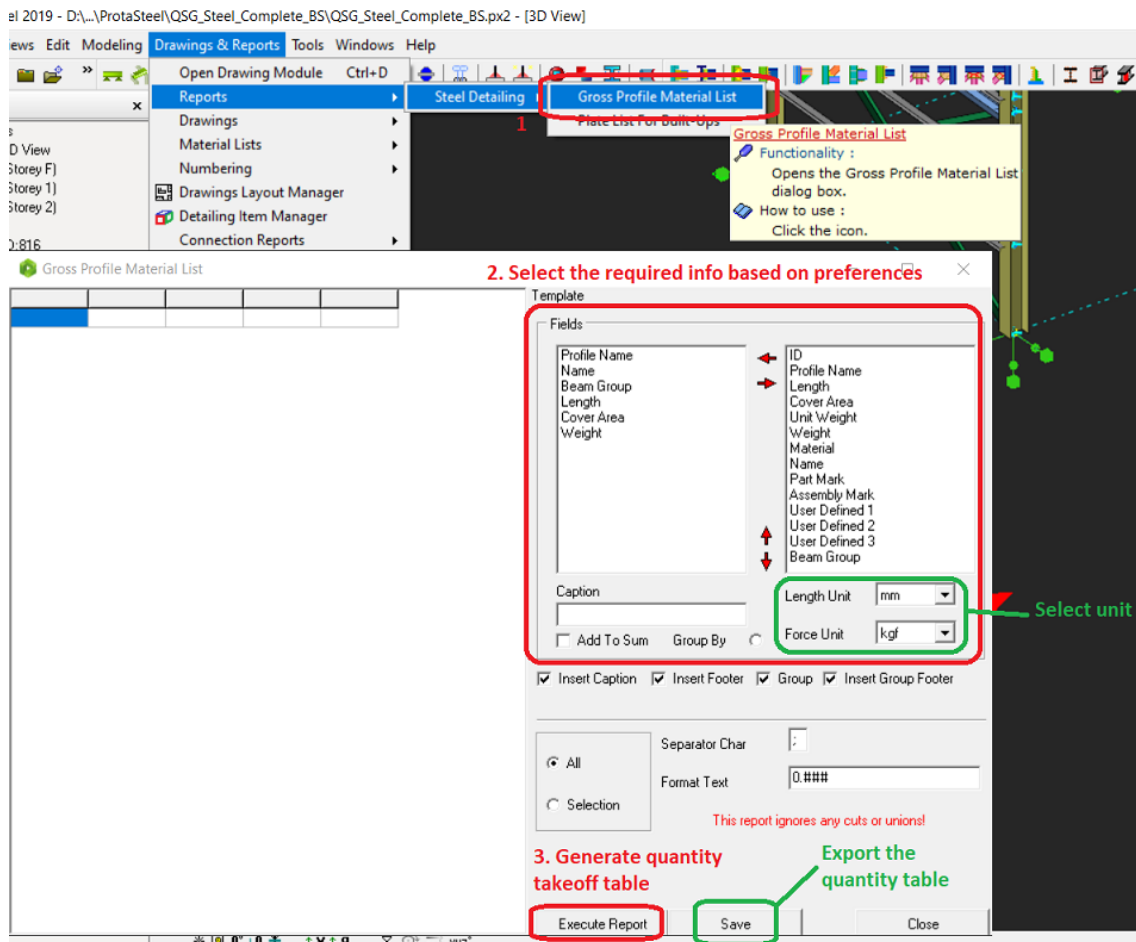


Section	Type	Material	Quantity	Length(m)	Cover Area	Gross Weight(kg)	Net Weight(kg)
1	L50X50X3	S355	1	15812	3.16	37.12	37.12
2	L80X8	S355	1	11662	3.74	112.3	112.3
3	RHS 150X75X3	S355	80390	70.69	829.21	829.21	
4	SHS 75X75X3	S355	70078	40.86	475.29	475.29	
5	UB 175X125X23	S355	25000	20.7	581.9	581.9	
6	UB 200X150X30	S355	57000	55.72	1705.2	1705.2	
7	UB 250X125X25	S355	30000	29.5	753.35	753.35	
8	UPN100	S355	150000	58.28	1613.55	1613.55	
9	UPN120	S355	120000	53.6	1629.73	1629.73	
Total				336.26	7737.65	7737.65	

Method 2

Method 2 generates more detail quantity takeoff of steel members with choices of categories to be included in the table. Please refer to following steps: -

- Go to **Drawings & Reports** → **Reports** → **Steel Detailing** → **Gross Profile Material List**
- Select corresponding format and units and click **Execute Report**.
- Click **Save** to export generated report as CSV file.



el 2019 - D:\ProtaSteel\QSG_Steel_Complete_BS\QSG_Steel_Complete_BS.p2 - [3D View]

ews Edit Modeling Drawings & Reports Tools Windows Help

Open Drawing Module Ctrl+D

Reports

Drawings

Material Lists

Numbering

Drawings Layout Manager

Detailing Item Manager

Connection Reports

1

Gross Profile Material List

Plate List For Built-ups

Gross Profile Material List

Functionality :

Opens the Gross Profile Material List dialog box.

How to use :

Click the icon.

2. Select the required info based on preferences

Template

Fields

Profile Name

Name

Beam Group

Length

Cover Area

Unit Weight

Weight

Material

Name

Part Mark

Assembly Mark

User Defined 1

User Defined 2

User Defined 3

Beam Group

Length Unit

mm

Force Unit

kgf

Select unit

3. Generate quantity takeoff table

Execute Report

Save

Close

Export the quantity table

This report ignores any cuts or unions!

45. Closing Summary

Congratulations! You have completed your steel model & generated the engineering drawings in **ProtaSteel**.

In this Quick Start Guide, we have imported the ProtaStructure model in ProtaSteel. We then inserted ancillary steel members such as sag rods, adjusted members to their final positions, created some steel connections & generated the engineering drawings. This will give you the necessary knowledge to proceed to a real project.

For more help and guidance, please refer to **ProtaSteel** help reference accessible from Help menu.

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