



ADVANCE
GSTEEL

TUTORIAL

GGRAITEC

www.graitec.com

Advance Steel

Tutorial

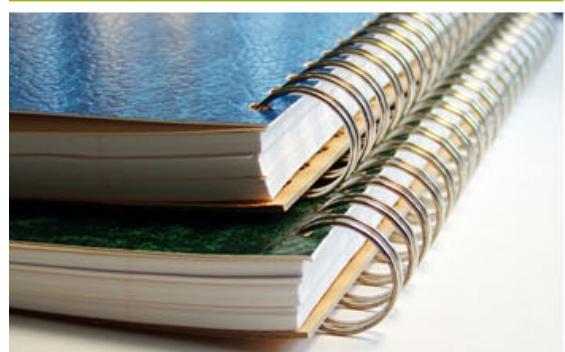


Table of contents

About this tutorial	7
How to use this guide	9
Lesson 1: Creating a building grid.....	10
Step 1: Creating an axis group in the X direction.....	10
Step 2: Creating an axis group in the Y direction.....	12
Lesson 2: Creating columns.....	15
Step 1: Creating a column with an I section.....	15
Step 2: Copying a column	17
Lesson 3: Creating beams	20
Step 1: Creating beams with I sections.....	21
Step 2: Creating beams with I sections between columns flange middle	23
Step 3: Modify properties for several beams.....	24
Step 4: Splitting beams at a given point.....	26
Step 5: Splitting beams at a given distance	28
Step 6: Copying beams at a distance below	29
Step 7: Creating a curved beam.....	31
Step 8: Creating purlins	35
Lesson 4: Creating structural elements	39
Step 1: Creating cross bracings	39
Step 2: Creating a joist	44
Step 3: Moving the joist	47
Step 4: Other adjustments.....	48
Lesson 5: Defining model views	50
Defining a view with 1 point in the UCS	51
Defining a view with 2 points in UCS and front and rear depth.....	54
Defining a view at a grid line	57
Lesson 6: Working with intelligent connections	60
Step 1: Creating a base plate joint	61
Step 2: Creating a group of base plate connections	62
Step 3: Adding a connection to an existing group.....	64
Step 4: Removing a joint from a joint group	65
Step 5: Modifying the properties of a simple joint	65
Step 6: Modifying joint properties within a group	68
Step 7: Creating independent joints of the same type	74

Lesson 7: Creating intelligent connections	78
Step 1: Creating a moment end plate	78
Step 2: Creating double side end plates with safety bolts	81
Step 3: Creating gable wall end plates.....	85
Step 4: Creating front plate splices	88
Step 5: Creating clip angle joints.....	91
Step 6: Creating a clip angle skewed joint	98
Step 7: Creating copes.....	102
Step 8: Creating purlin connections	104
Step 9: Creating a flat bracing with tension bolt for a base plate.....	106
Step 10: Creating a flat bracing with tension bolt for a beam	109
Step 11: Creating middle gusset plates	112
Step 12: Creating a tube connection with sandwich plates for the base plate.....	114
Step 13: Creating a tube connection with sandwich plates for a beam as an additional object	120
Step 14: Creating a tube connection middle with sandwich plates	122
Lesson 8: Collision check.....	126
Step 1: Checking for collisions	126
Step 2: Displaying a collision.....	127
Step 3: Repairing the collisions	128
Lesson 9: Displaying connected objects	130
Step 1: Displaying objects connected in shop.....	130
Step 2: Displaying connected objects	131
Step 3: Displaying connection means	133
Lesson 10: Creating cameras	135
Step 1: Inserting a camera according to the UCS.....	135
Step 2: Moving the camera	138
Step 3: Changing the camera color.....	142
Lesson 11: Numbering	143
Step 1: Setting prefixes	144
Step 2: Numbering	145
Step 3: Displaying the results	146
Lesson 12: Drawings	147
Step 1: Automatic detailing of all the single parts in the model.....	148
Step 2: Modifying the properties of a detail.....	149
Step 3: Automatic detailing of all the main parts in the model	151
Step 4: Inserting a structured list in a drawing	154

Lesson 13: Generating details using cameras	155
Step 1: Inserting a level symbol in the model.....	155
Step 2: Creating details using cameras	157
Step 3: Inserting a level symbol in the drawing.....	159
Lesson 14: Exploding details	160
Step 1: Adding to explode details.....	160
Lesson 15: Creating external lists.....	163
Step 1: Creating a model extract.....	163
Step 2: Creating a list	164
Step 3: Exporting a list	165

About this tutorial

GRAITEC provides a multimedia tutorial to help you learn how to use various Advance tools.

The tutorial contains step-by-step instructions for creating Advance elements, connections, drawings, lists and understanding Advance functions.

During the first 7 lessons, you will learn how to use the most common Advance construction elements (i.e., beams, plates, connection elements, structural elements) and joints and you will create a steel structure.

In the next lesson, the model is checked for errors using Advance tools for checking collisions in the model.

The tools for displaying connected objects are described in Lesson 9.

In the following lessons, the model is numbered and then used to create drawings.

In this tutorial:

Lesson 1: Creating a building grid

Lesson 2: Creating columns

Lesson 3: Creating beams

Lesson 4: Creating structural elements

Lesson 5: Defining model views

Lesson 6: Working with intelligent connections

Lesson 7: Creating intelligent connections

Lesson 8: Collision check

Lesson 9: Displaying connected objects

Lesson 10: Creating cameras

Lesson 11: Numbering

Lesson 12: Drawings

Lesson 13: Generating details using cameras

Lesson 14: Exploding details

Lesson 15: Creating external lists



How to use this guide

Simply go through the tutorial and create the model from scratch following the procedures described in each lesson!

The first page of each lesson presents the lesson content (what you will learn). If some additional concepts or techniques are useful, these are also presented.

Additionally, recommended practices and tips and tricks are included during the lesson.

This tutorial does not explain all Advance features and commands. For more details, see the online help.

Lesson 1: Creating a building grid

Placing a building grid is the first step of 3D modeling in Advance. In this lesson, you will create a building grid that is used in the following lessons of this tutorial. Grids facilitate placing of the construction elements and for orientation in the 3D view.

You will learn how to:

- Create an axis group by distances
- Define the axis labels

The following examples describe how to create a building grid with axes in the X and Y directions, with the distances given in Figure 1.

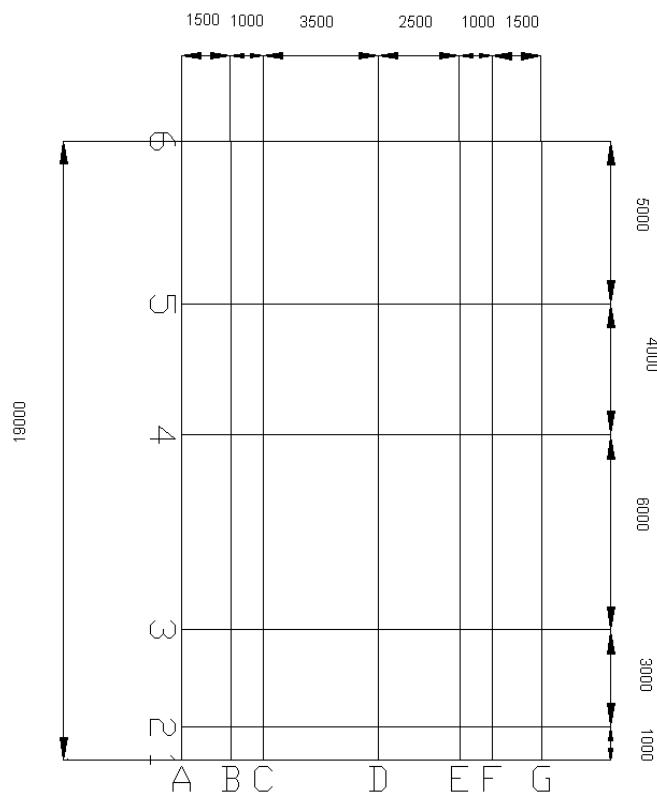
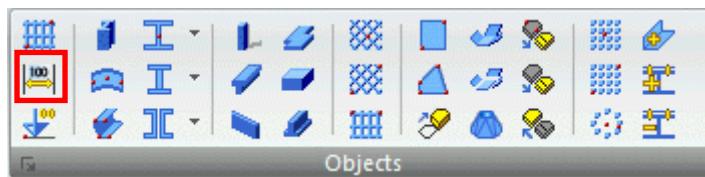


Figure 1: Building grid – Top view

Step 1: Creating an axis group in the X direction

In this step, create an axis group in the X direction and label it with capital letters.

1. On the **Home** tab, **Objects** panel, click  to create an axis group at a given distance.



2. On the command line, enter the starting point of the first axis of the group **0,0,0** and press **Enter**.

3. Move the mouse pointer in the Y direction (the **Ortho** setting causes an exact orientation entry) and enter **19000** to define the end point of the grid line.

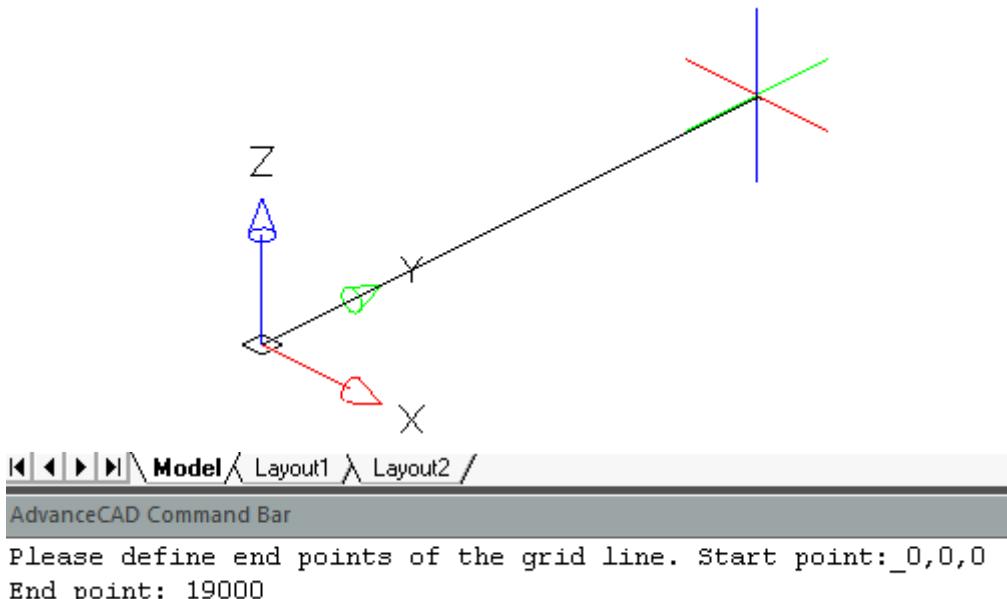


Figure 2: Defining the end point of the grid line

4. Select a point in the X direction on the screen to define the direction of the axis group.

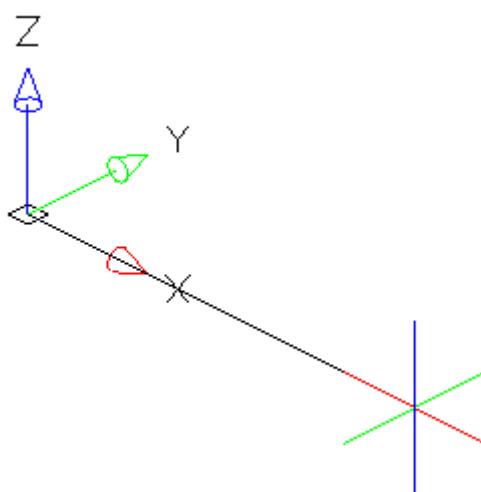


Figure 3: Defining the direction of the axis group

5. Define the distances between grid lines by entering each distance between axis: **1500, 1000, 3500, 2500, 1000, 1500**.

The command line displays a total value of **11000**.

6. Press **Enter** twice to confirm the axis group creation.

The axis group in the X direction is created and the properties dialog box appears.

7. On the **Total** tab, from the **Label Type** drop-down list, select **Capital Letters**.

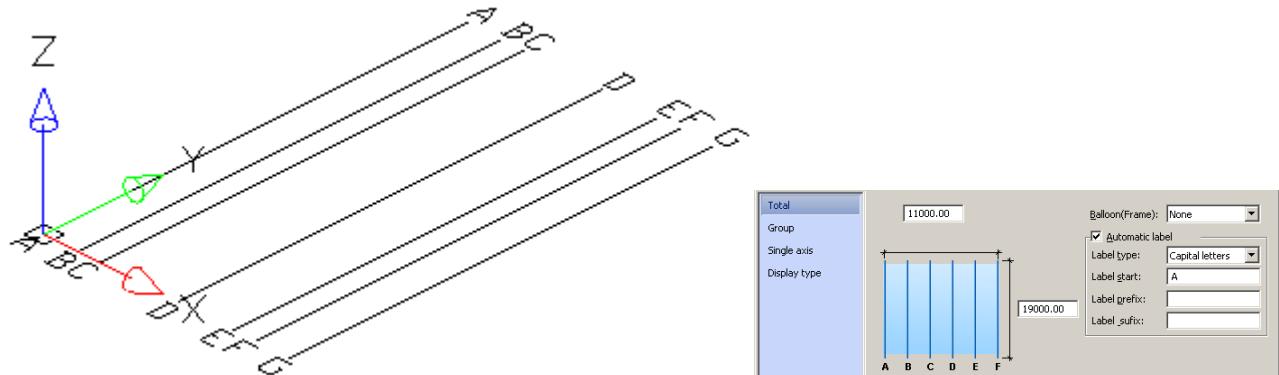
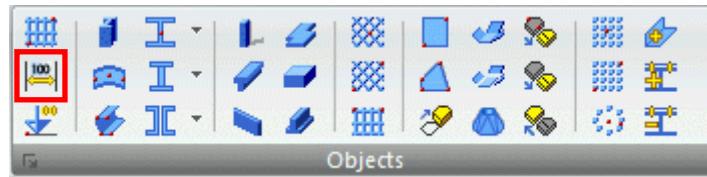


Figure 4: Axis group in the X direction – labeling options

Step 2: Creating an axis group in the Y direction

Next, create an axis group in the Y direction and label it with numbers.

1. On the **Home** tab, **Objects** panel, click



2. Define the start point of the first axis of the group: click on the **G** axis corresponding end point as in the Figure 5:

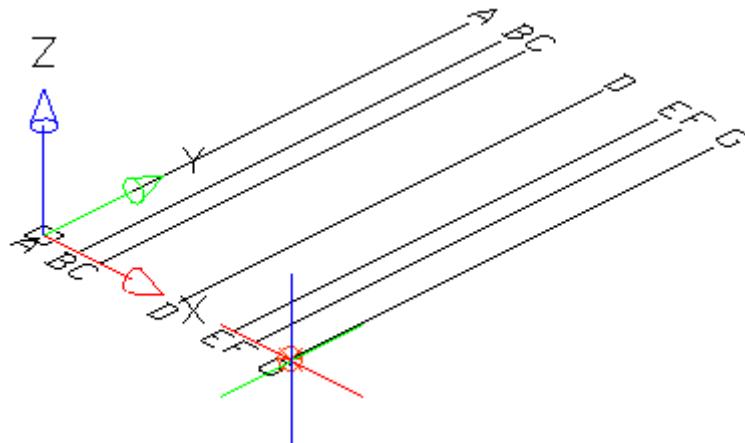


Figure 5: Defining the starting point of the axis group

3. Define the end point of the grid line by dragging the mouse pointer (with the **Ortho** mode on, F8) to the **A** axis corresponding end point as in the Figure 6:

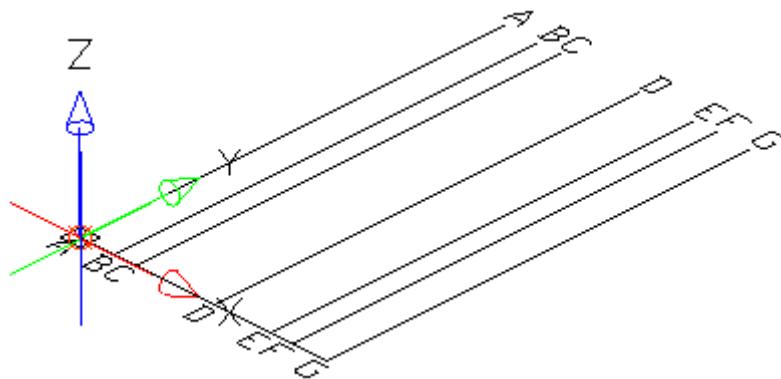


Figure 6: Defining the end point of the axis group

4. Select a point in the Y direction on the screen to define the direction of the axis group.

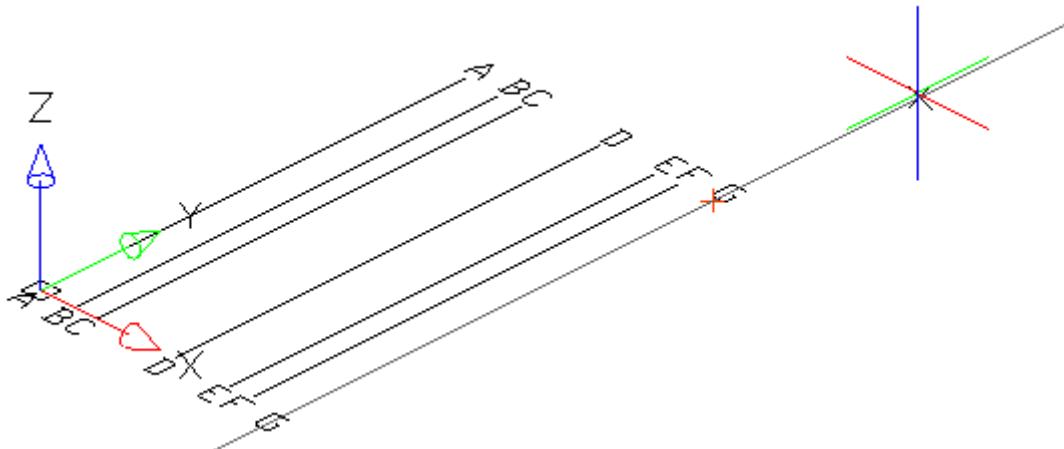


Figure 7: Defining the direction of the axis group

5. Define the distances between grid lines by entering each distance between axes: **1000, 3000, 6000, 4000, 5000**.

The command line displays a total value of **19000**.

6. Press **Enter** twice to confirm the axis group creation.

The axis group in the Y direction is created and the properties dialog box appears.

7. On the **Total** tab, from the **Label Type** drop-down list, select **Numbers**.

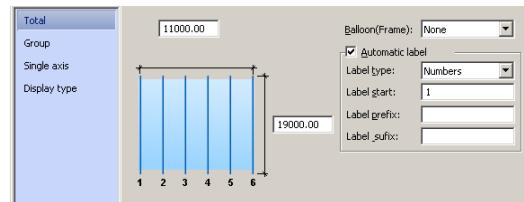


Figure 8: Properties dialog box

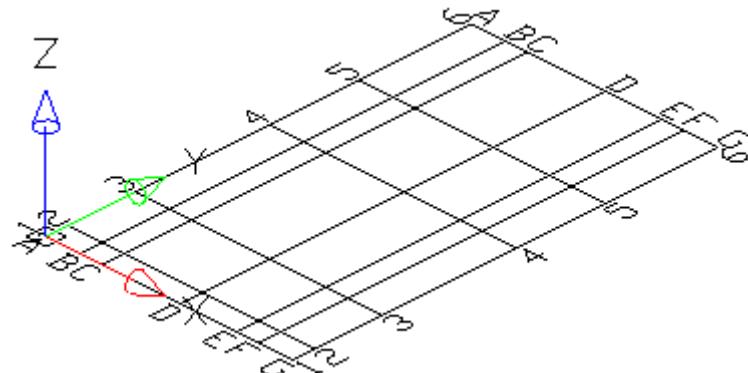


Figure 9: Final Grid Axis

Lesson 2: Creating columns

In this lesson, you will create columns using the grid axis from Lesson 1 – “Creating a building grid”.

You will learn how to:

- Create a column with an I Section
- Create a column with a Rectangular Hollow Section
- Modify column properties
- Copy a column

Concept

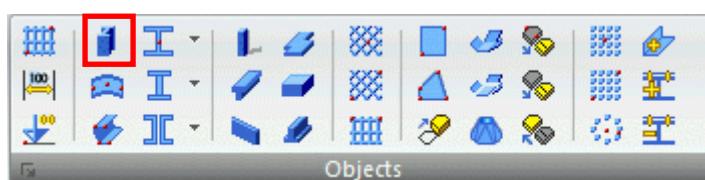
Columns of a default length are created in the Advance 3D model vertically, regardless of the current user coordinate system (UCS), by entering one base point.

The column length can be changed by using the grips. The default length of the column is defined in the GRAITEC Advance Manager.

Step 1: Creating a column with an I section

In this step, create a column with an I section at the **C1** axis intersection and modify its properties.

1. On the **Home** tab, **Objects** panel, click 



3. Select the base point of the column at the **C1** axis intersection.

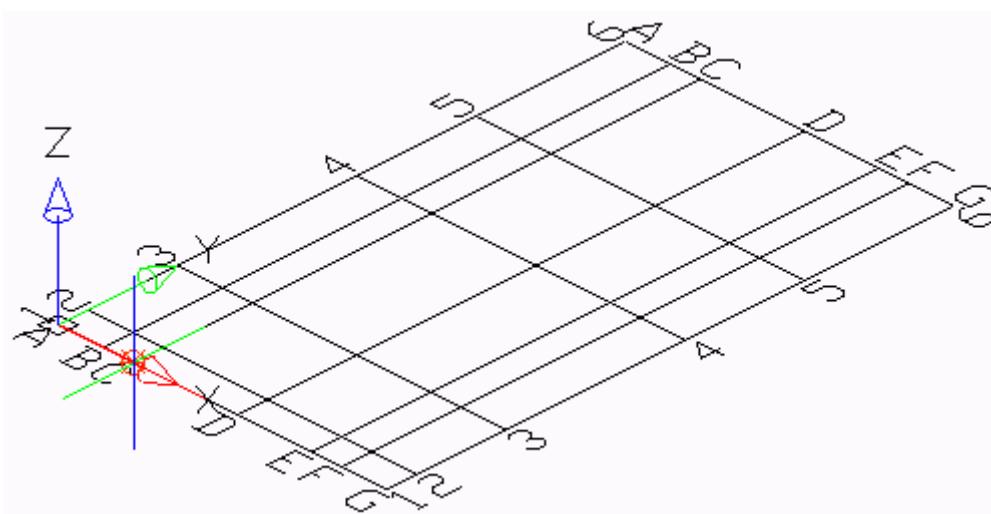


Figure 10: Creating the first column with I section

The properties dialog box appears.

Modifying column properties

Once the column is created, in this example, change the section, the material and rotate the column.

In the properties dialog box, define the following settings:

- On the **Section & Material** tab:
 - Use the  buttons and make the following settings for the column section:

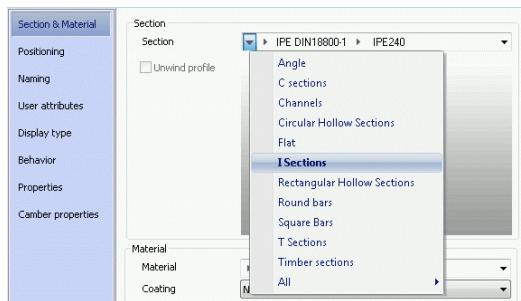


Figure 11: “Beam” dialog box – **Section & Material** tab

- On the **Positioning** tab:
 - Select the **90.00°** angle.

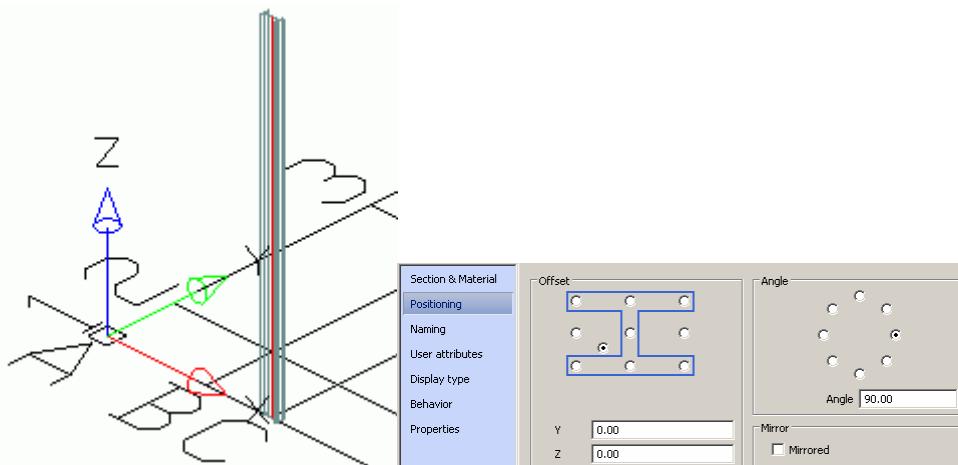


Figure 12: Column with I section

Note: Note that on the **Naming** tab, the model role is set by default to **Column**.

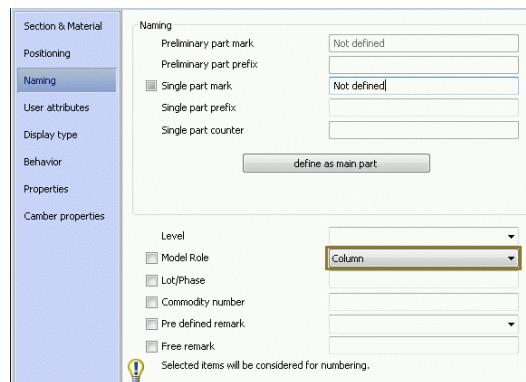


Figure 13: **Naming** tab – the model role

Using a similar process, create a column with a rectangular hollow section (**RHS140x8**) at the **G6** axis intersection.

In the properties dialog box, on the **Section & Material** tab, use the buttons and make the following settings for the column section:

- Select the section class: **RHS square cold**.
- Select the section size: **RHS140x8**.

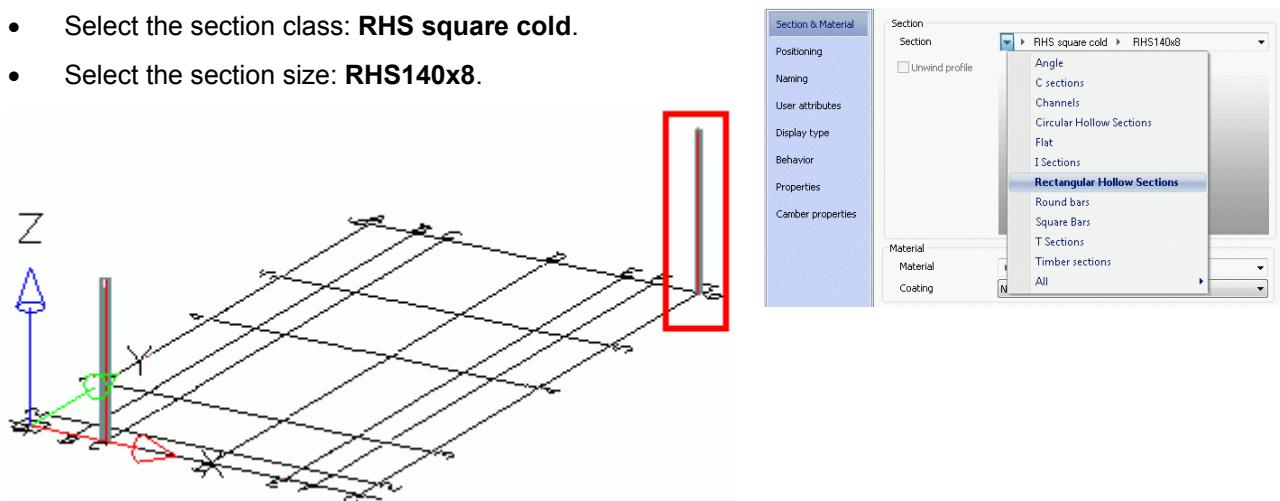


Figure 14: Creating first column with Rectangular hollow section

Step 2: Copying a column

In this step, the column situated at the **C1** axis intersection is copied to the **A4, A5, B2, B3, B4, E1, F2, F3, G3, G4, and G5** axis intersections.

To create the columns at the axis intersection, activate the **Grid intersection** OSNAP mode.

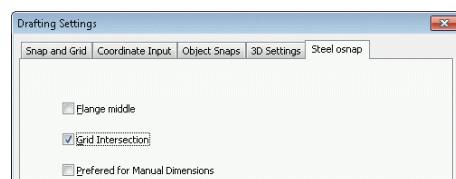


Figure 15: Drafting settings – **Steel Osnap** tab

1. Select the column.
2. On the **Tools** tool palette, click .
3. Select the base point of the copying procedure: the base point of the **C1** column.

Use the **Real-Time Sphere** tool to rotate the drawing in 3D and get a suitable view.



4. Define the end points for the newly created columns: axis intersections **E1, B2, F2, B3, F3, G3, A4, B4, G4, A5, G5**.

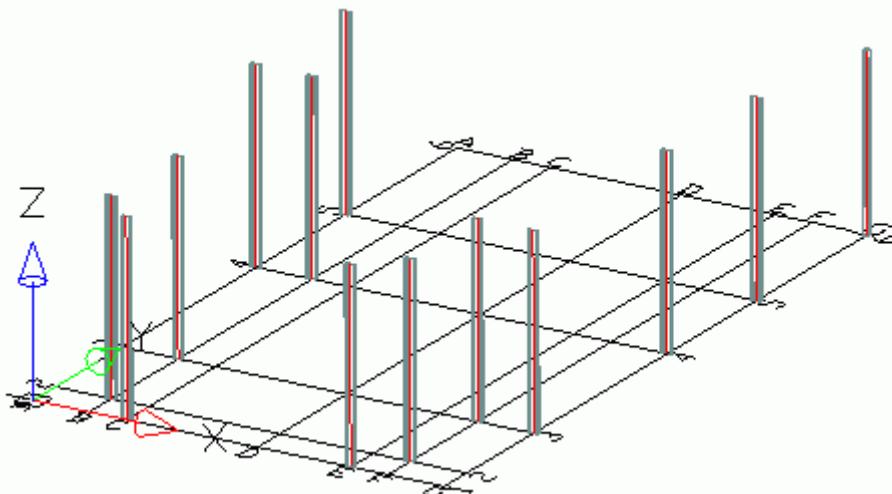


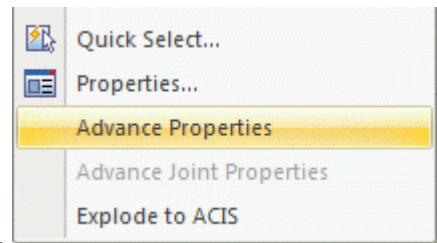
Figure 16: The 12 columns with I sections

5. Press **Esc** to finish.

Modifying column positions

The position of columns can be changed using Advance Steel properties:

1. Select the **B2** and **F2** columns.



2. Right click and select **Advance Properties** from the context menu.

3. In the properties dialog box, on the **Positioning** tab, select the **-90.00°** angle.

The column position is modified.

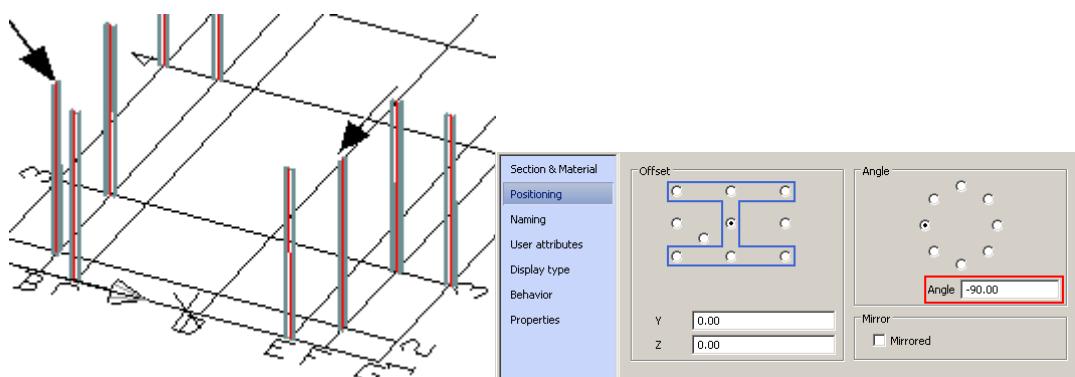


Figure 17: Changing orientation of the two columns

Using the same process from step 2, copy the column with Rectangular hollow section to the **D6**, **C6**, and **A6** axis intersections.

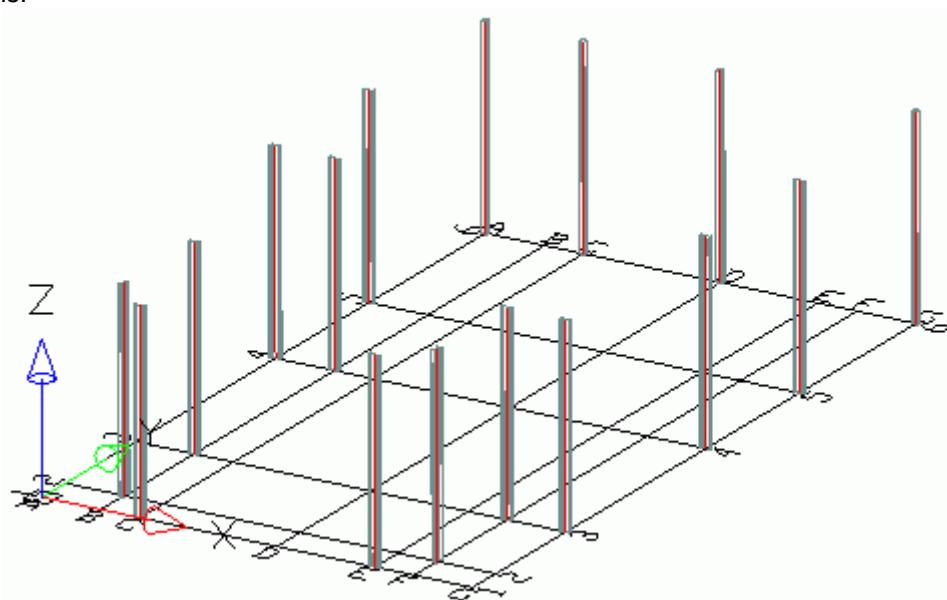


Figure 18: 12 columns with I sections and 4 with rectangular hollow sections

Lesson 3: Creating beams

In this lesson, you will create beams using the columns created in Lesson 2.

You will learn how to:

- Create beams with I Sections
- Split beams
- Copy beams
- Select objects according to different criteria
- Create a curved beam
- Create a channel section beam

Concept

Beams are created in the Advance 3D model relative to the current user coordinate system (UCS) by entering one start point and one end point.

The current user coordinate system (UCS) determines the position of the sections' main axes: the web of a beam runs in the Y direction of the UCS (i.e., the 'top' of the section is in the Z direction).

Before starting



*Make sure that the **Node** OSNAP mode is active. It is useful for start and end point selection.*

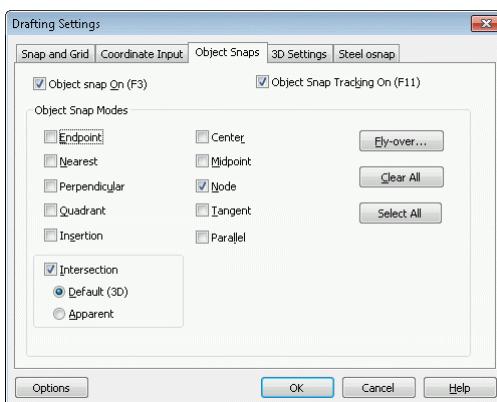


Figure 19: Properties dialog box – Activating **Node** OSNAP mode

Step 1: Creating beams with I sections

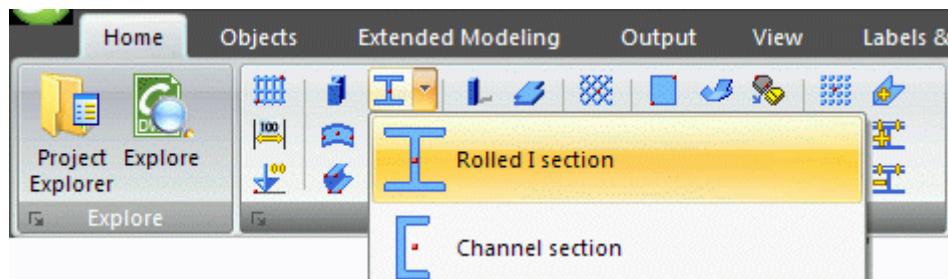
In this step, create a beam with an I section between the **C1** and **E1** columns and adjust its position.



Make sure the current UCS is in a suitable position (see Figure 20), otherwise use the three buttons on the UCS tool palette to place it correctly



1. On the **Home** tab, from the **Objects** panel, select the I section.



2. Use the upper system line end point of the **C1** column as the start point.

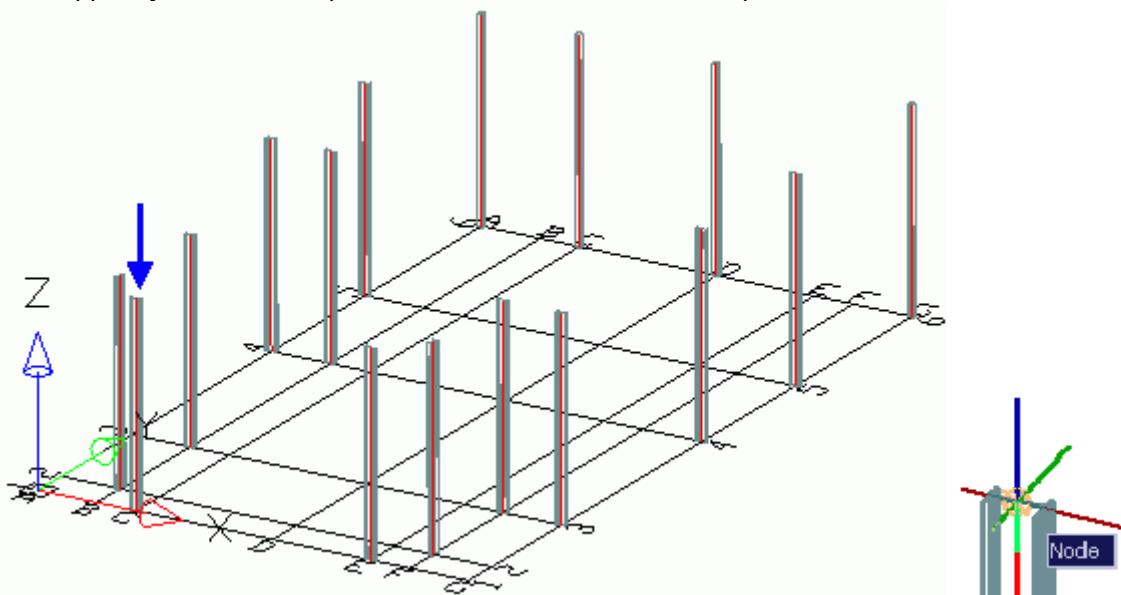


Figure 20: Selecting the start point of the beam: the upper system line end point of the **C1** column

3. Use the upper system line end point of the **E1** column as the end point.

The properties dialog box appears.

Modifying beam properties

In the beam properties dialog box, define the following settings:

- On the **Section & Material** tab:
 - Select the section class: **IPN**.
 - Select the section size: **IPN280**.

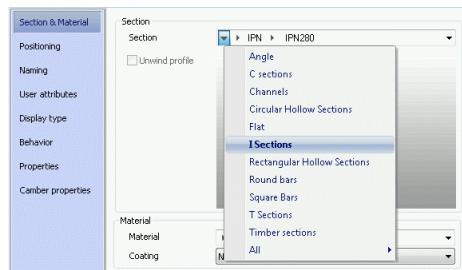


Figure 21: **Section & Material** tab – Selecting the section type

Note: All the following I section beams will have the same section class and section size.

- On the **Positioning** tab:
 - Define the offset in Z direction: **-140**.

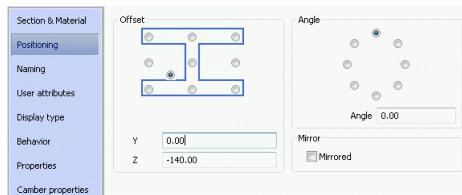


Figure 22: **Offset** tab – Setting the offset

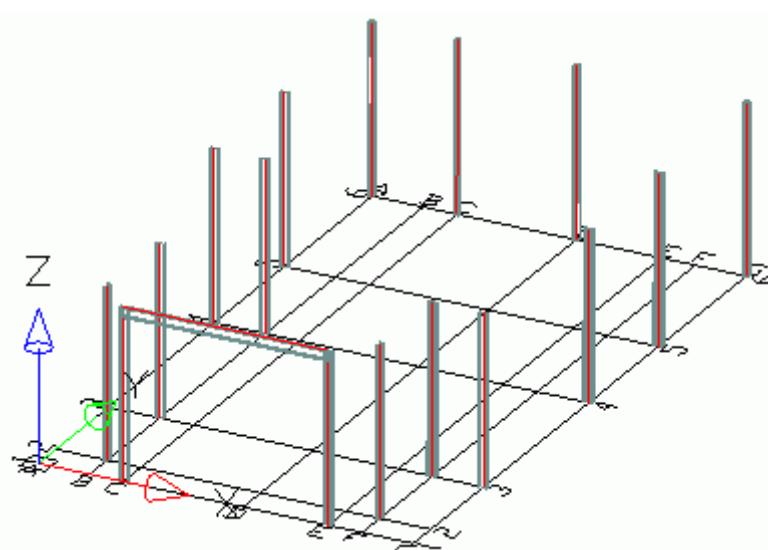


Figure 23: The beam with I section

Using the same process, draw other beams with the same sections between the **F2-F3**, **F3-G3**, **G3-G6**, **G6-A6**, **A6-A4**, **A4-B4** and **B4-B2** columns. Next, draw another beam between the **B4** and **G4** columns.

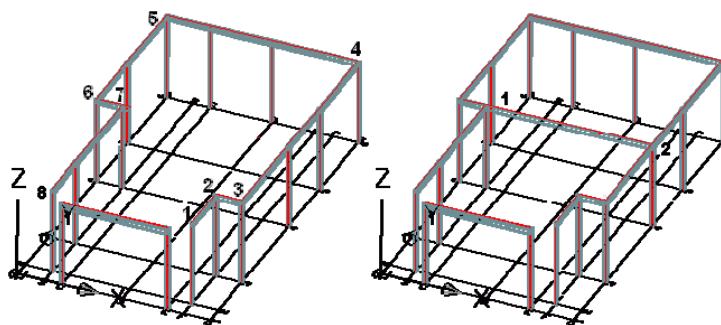
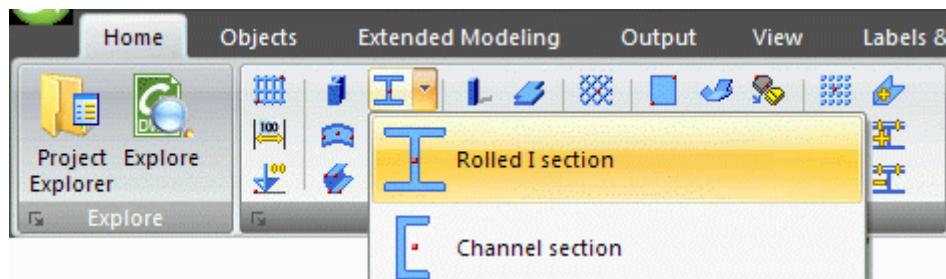


Figure 24: Creating the beams

Step 2: Creating beams with I sections between columns flange middle

Next, create 2 beams with I sections between the **E1-F2**, **C1-B2** columns flange middle and modify their section, position and model role.

1. On the **Home** tab, from the **Objects** panel, select the I section.



2. Use the lower flange middle point of the **E1** column as the start point.
3. Using the same process, use the upper flange middle point of the **F2** column as the end point (see Figure 25).

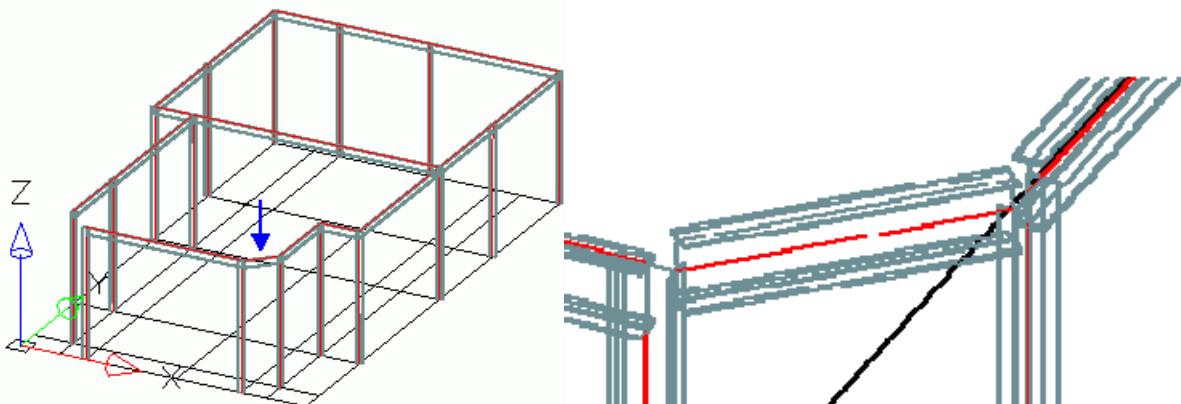


Figure 25: The beam with an I section between columns flange middle: **E1-F2**

Using the same process, draw another beam with the same I section between the **B2** and **C1** columns flange middle.

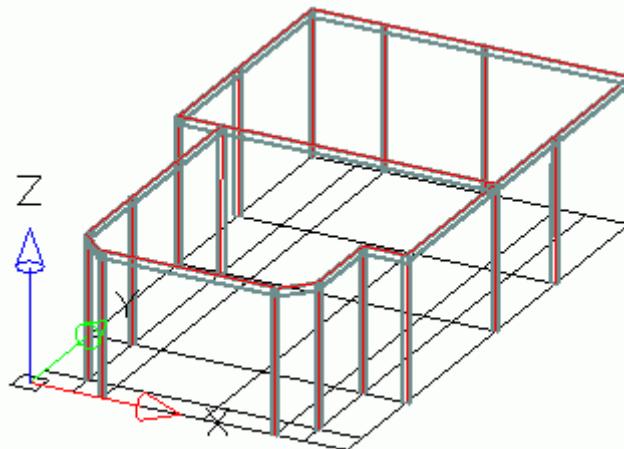


Figure 26: The final result: all created beams

Step 3: Modify properties for several beams

In this step, adjust the position of all beams of the model, in the same time. To do this, it is necessary to put the objects having the same properties in a selection set.

As all beams have the same section, it is possible to identify them defining a search criteria based on the section type and size in the “Search filter”. All objects matching the search criteria will be marked in red.

Additionally, define the model role. You can assign a role to several objects at once as long as they are of the same type (e.g. beam, plate, etc.).

Best practice: Always use model roles, as this defines the prefix in numbering and also has influence with the drawing output. The drawing styles link to the Model roles to set the style within the process.

Marking objects

Mark all the profiles with an **IPN280** section.



- On the **Selection** tool palette, click .
- In the “Search and mark objects” dialog box, define the following settings:
 - On the **Objects** tab, select the type of objects to select: **Steel beam**.

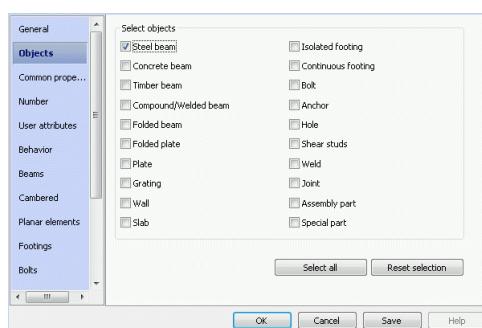


Figure 27: Search filter dialog box – **Objects** tab

- On the **Beams** tab:
 - Select the section class: **IPN**.
 - Select the section size: **IPN280**.

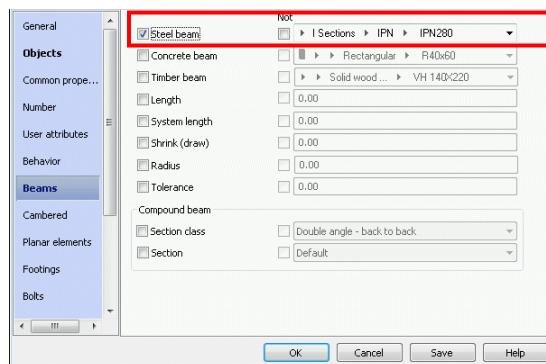


Figure 28: Search filter dialog box – **Beams** tab

Note: If the section class to be selected is not in the list, unwind **All** in the drop-down list.

- Click **OK**.

The beams are marked in red.

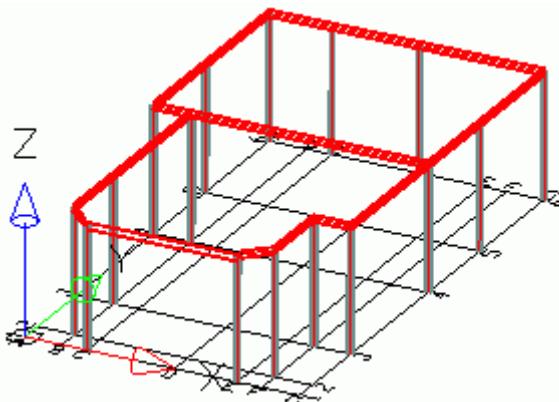


Figure 29: The profiles with an **IPN280** section

- On the **Selection** tool palette, click to select the marked objects.

Modifying beam properties

- Right click and select **Advance Properties** from the context menu.
- In the properties dialog box, define the following settings:
 - On the **Positioning** tab, select the orientation of the beam.

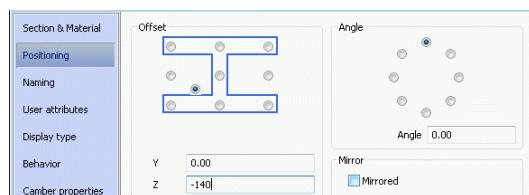


Figure 30: **Positioning** tab – Modifying the orientation

- On the **Naming** tab:
 - Set the model role to **Rafter**.

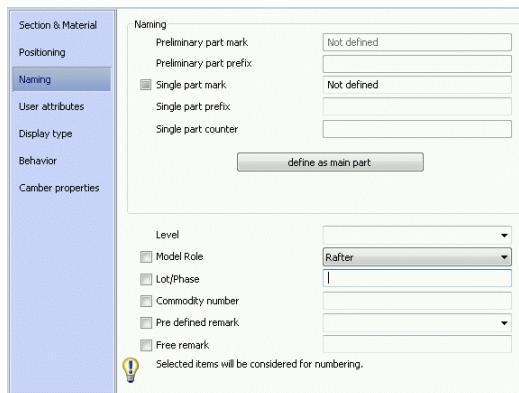


Figure 31: **Naming** tab – Defining the model role

- On the **Selection** tool palette, click  to cancel the markings.

Step 4: Splitting beams at a given point

Next, split the **G3-G6**, **A4-A6**, **B4-B2** beams at the upper system line end points of the **G4**, **G5**, **A5**, and **B3** columns.

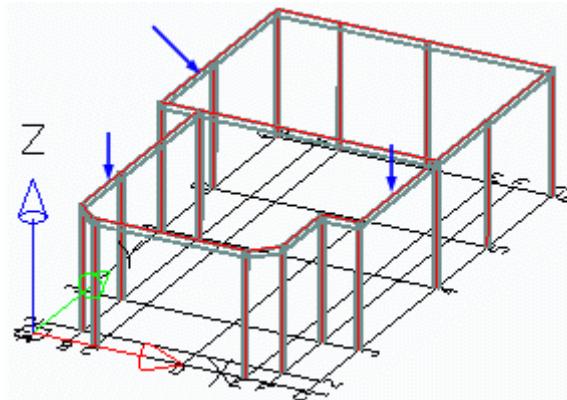


Figure 32: The beams to split

- On the **Beam features** tool palette, click .
- Select the **G3-G6** beam and press **Enter**.

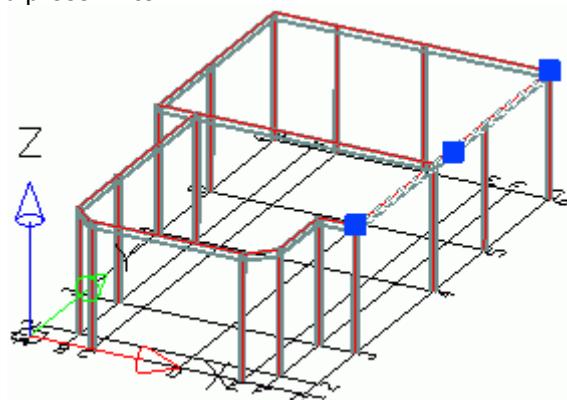


Figure 33: Selecting the **G3-G6** beam

3. Select the splitting points: the upper system line end points of the **G4** and **G5** columns and press **Enter**.

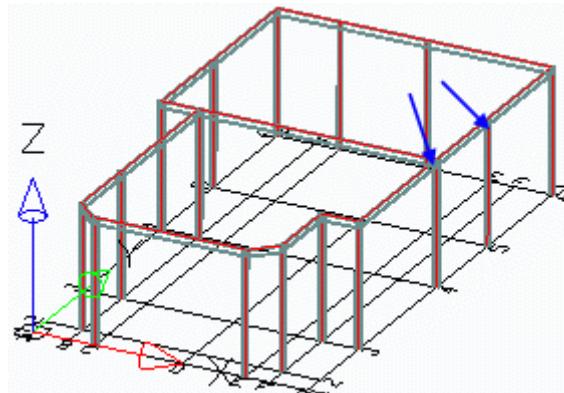


Figure 34: The splitting points for the **G3 – G6** beam

The beam is split.

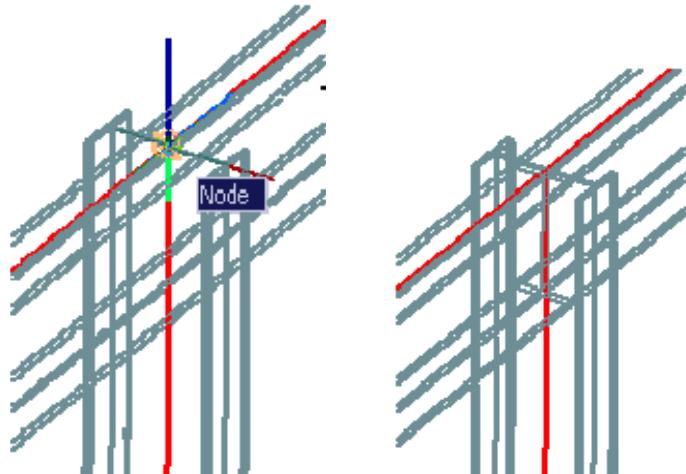


Figure 35: Splitting a beam

Using the same process, split the **A4-A6**, **B4-B2** beams at the upper system line end points of the **A5** and **B3** columns, respectively.

Note: Use the **Repeat** command to repeat the last executed command – in this case, **Split beams** command.

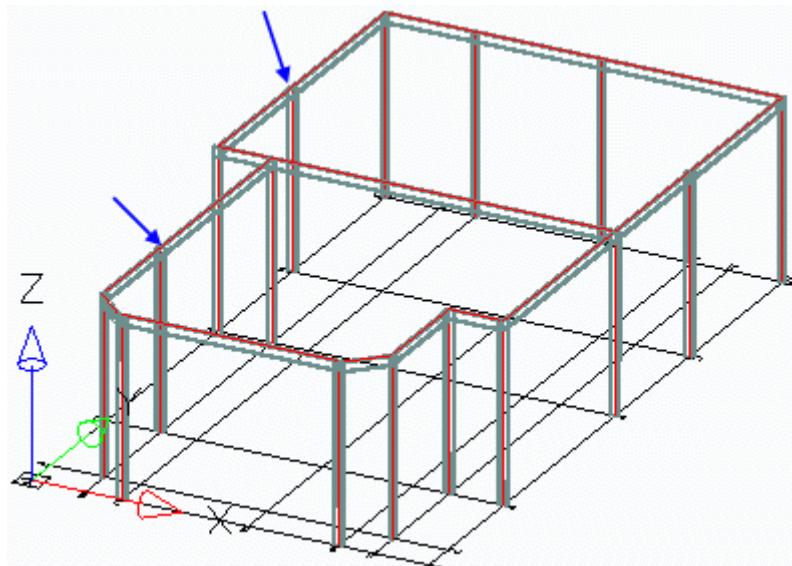


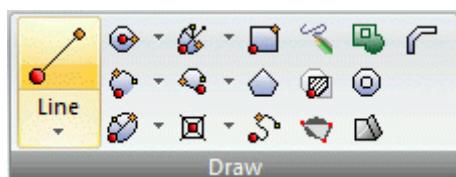
Figure 36: Splitting points for the **A4-A6** and **B2-B4** beams

Step 5: Splitting beams at a given distance

First create two auxiliary lines along the **A6-G6** beam using the **Line** tool. The beam will be split at the end points of the created lines.

Drawing the auxiliary lines

1. On the **Utilities** tab, **Draw** panel, click .



2. Use the upper system line end point of the **C6** column as the start point.
3. Move the mouse pointer in the X direction (the setting **Ortho** causes an exact orientation entry) and enter **700** to define the line end point.

Following the same steps, create another line with the **D6** system line end point as the start point, oriented in the **-X** direction, with a value of **400** to define the line end point.

 Make sure the **Dynamic Input** is off.

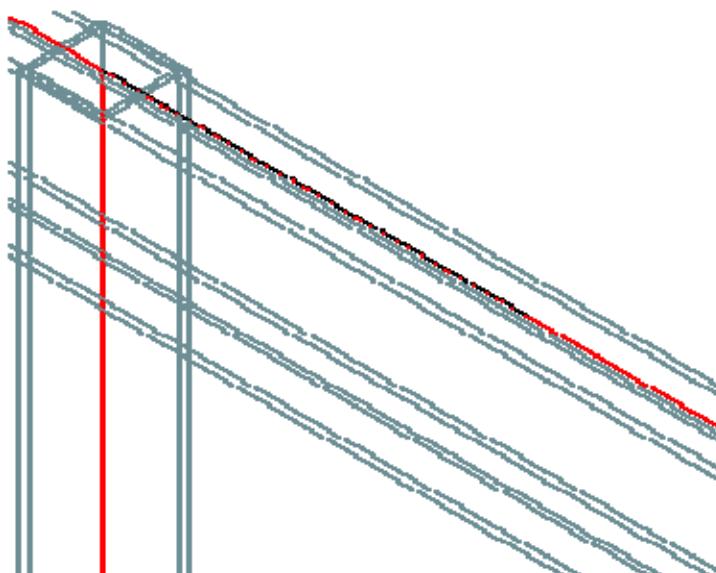


Figure 37: The auxiliary line

4. Press **ESC**.

Splitting the beam

1. On the **Beam features** tool palette, click 
2. Select the **A6-G6** beam and press **Enter**.
3. Select the first splitting point: the end point of the line from the **C6** column.
4. Select the second splitting point: the end point of the line from the **D6** column.
5. Press **Enter**.

The beam is split.

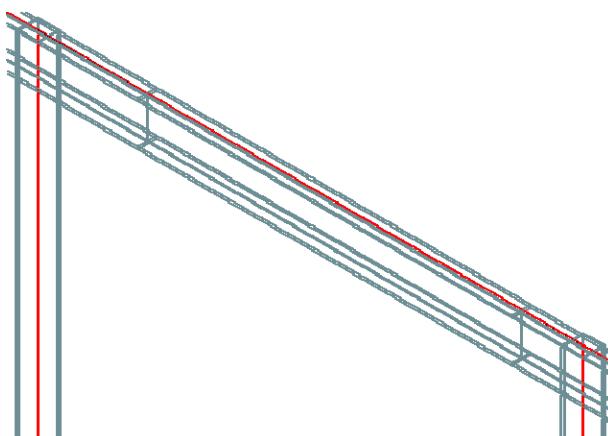


Figure 38: The split beam

Step 6: Copying beams at a distance below

In this step, copy the **G5-G6**, **G3-G4**, **G3-F3**, **A4-B4**, **A4-A5**, and **A5-A6** beams **2500** distance below.

1. Select the following beams: **G5-G6**, **G3-G4**, **G3-F3**, **A4-B4**, **A4-A5**, **A5-A6**.

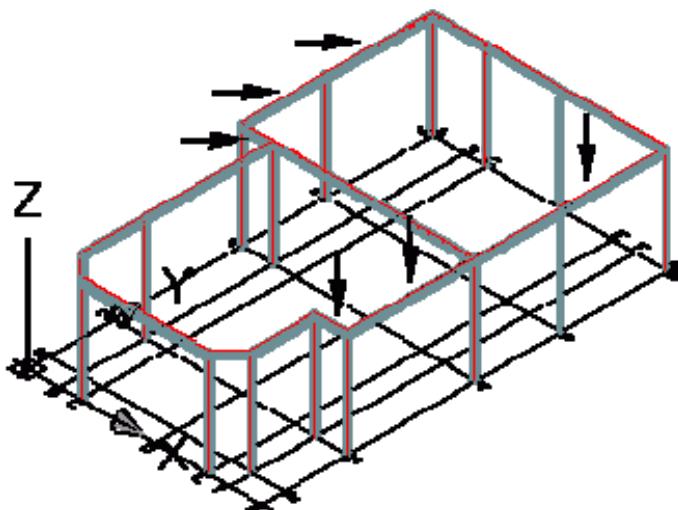


Figure 39: The beams to be copied

2. On the **Tools** tool palette, click  to copy the selected beams.
3. Select the base point (with **Polar** mode on) anywhere in the drawing.
4. On the command line enter **2500** and press **Enter**.

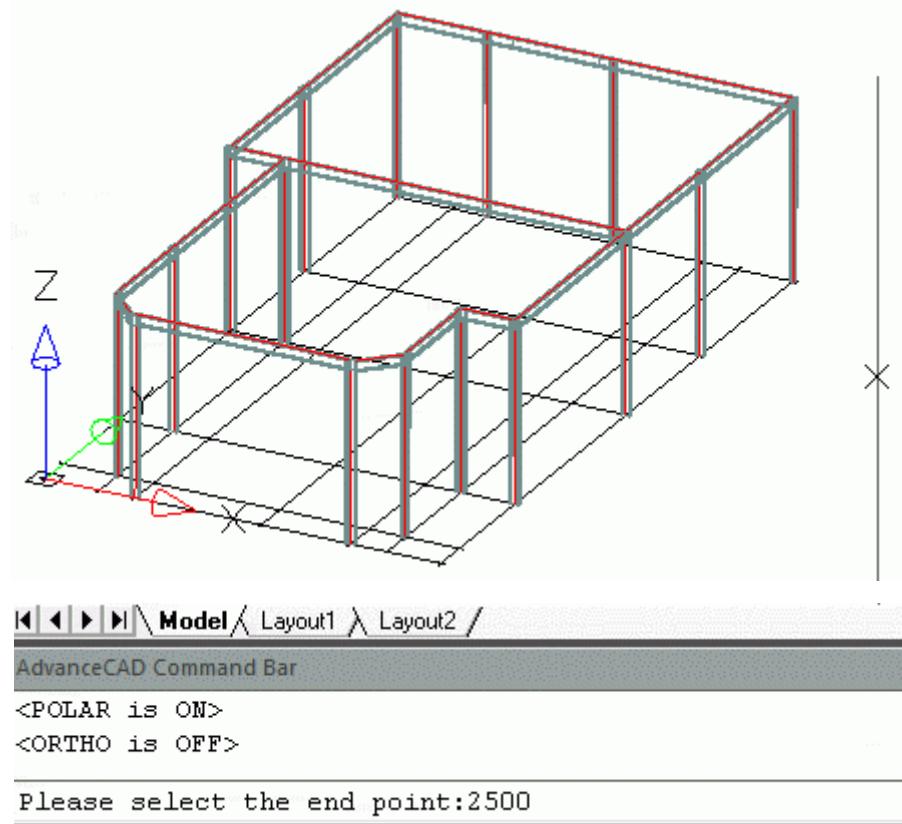


Figure 40: Selecting the end point of the newly copied beams

The beams are copied **2500** distance below.

*Use the options on the **Viewpoints** panel of the **Home** tab to change the view angle.*

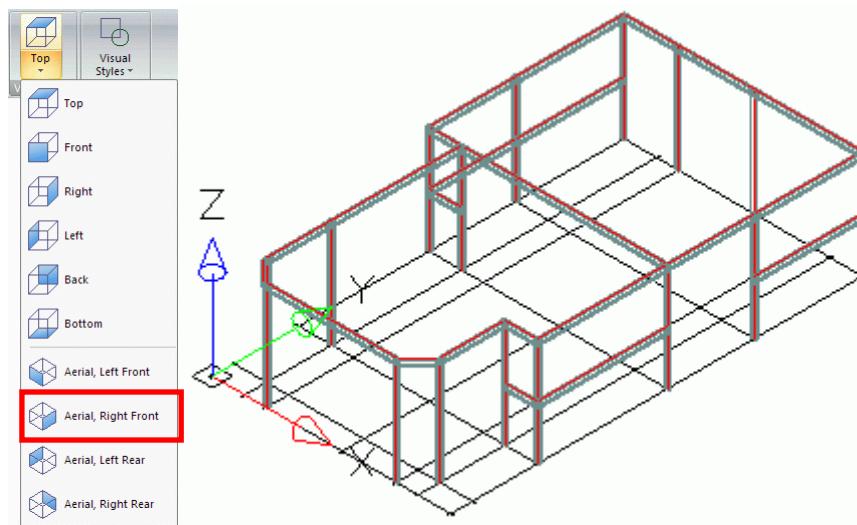


Figure 41: The final result: 6 new beams.

Step 7: Creating a curved beam

Create a curved beam between the **C1** and **E1** columns. The beam has an **UAP150** section.

Use the **Orbit** tool to rotate the drawing in 3D and get a suitable position.



1. On the **UCS** tool palette, click  to change the placement of the UCS.
2. Specify the new origin point in the upper system line end point of the **C1** column.

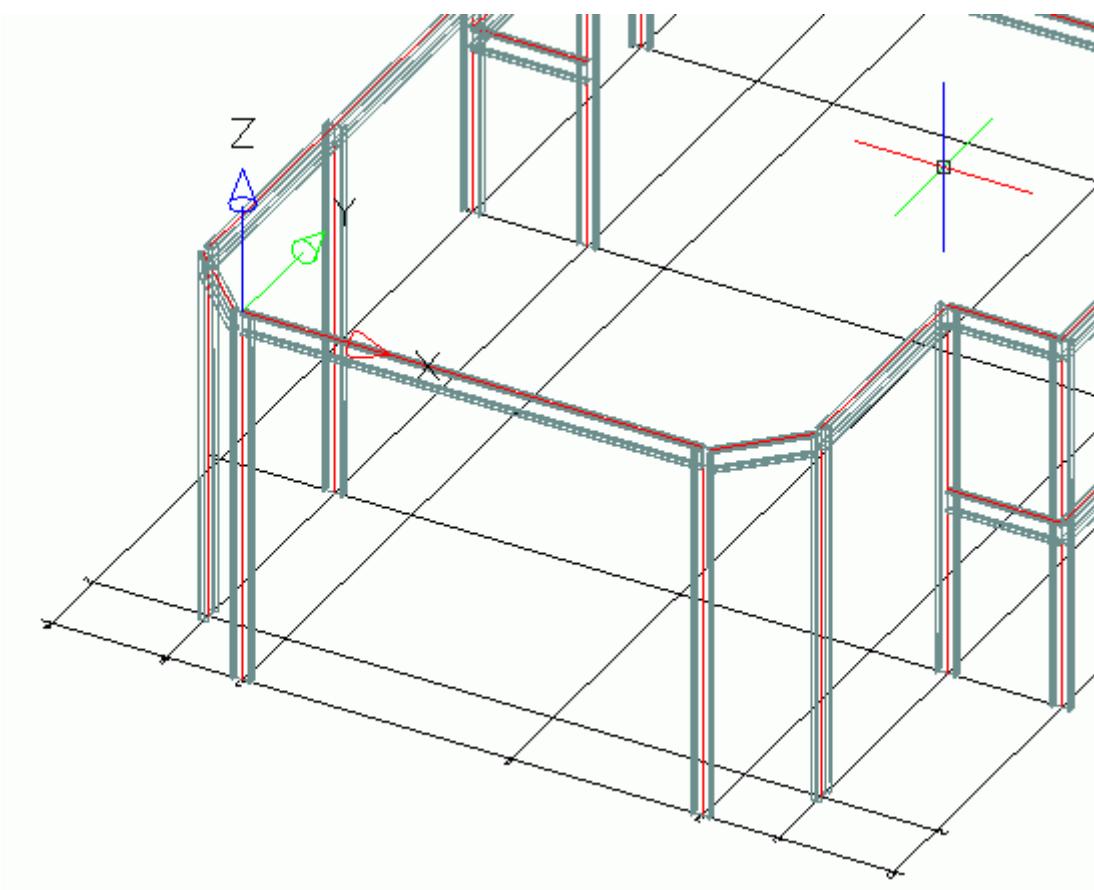
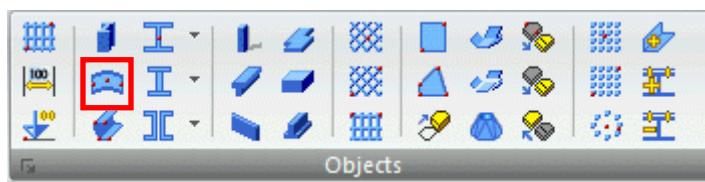


Figure 42: UCS in a suitable position

3. On the **Home** tab, **Objects** panel, click .



4. Use the upper system line end point of the **E1** column as the start point.
5. Use the upper system line end point of the **C1** column as the end point of the curved beam (with **Polar** mode on).

6. Click in the -Y direction to define the circle point.

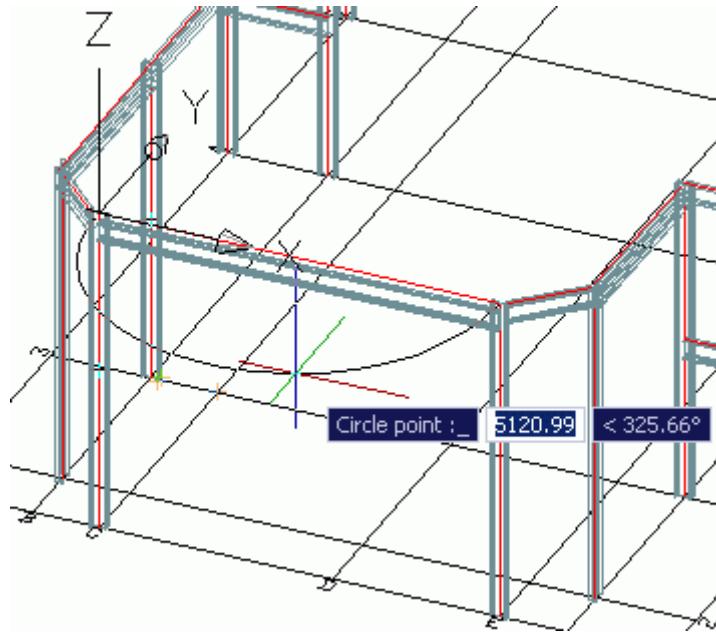


Figure 43: Creating a curved beam

7. In the properties dialog box define the following settings:

On the **Section & Material** tab:

- Use the  buttons and make the following settings for the curved beam:

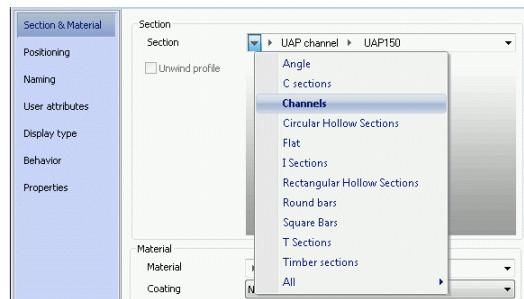


Figure 44: **Section & Material** tab – Selecting the section class

On the **Positioning** tab, select the position of the beam.

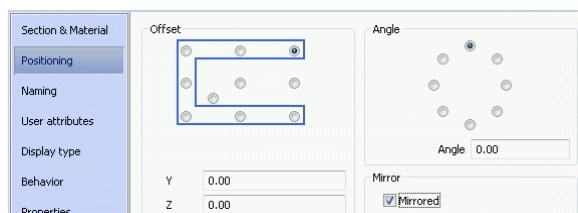


Figure 45: Properties dialog box – **Positioning** tab

On the **Naming** tab:

- Set the model role to **Curved beam**.

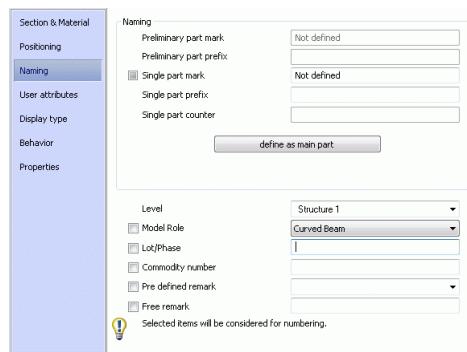


Figure 46: Naming tab – Selecting the model role

Adjusting the curved beam end point position

Offset the curved beam end points using the grip points.

1. Select the curved beam end point from the **E1** column.
2. Move the grip point in the **-X** direction.
3. On the command line, enter **50** for the displacement distance.

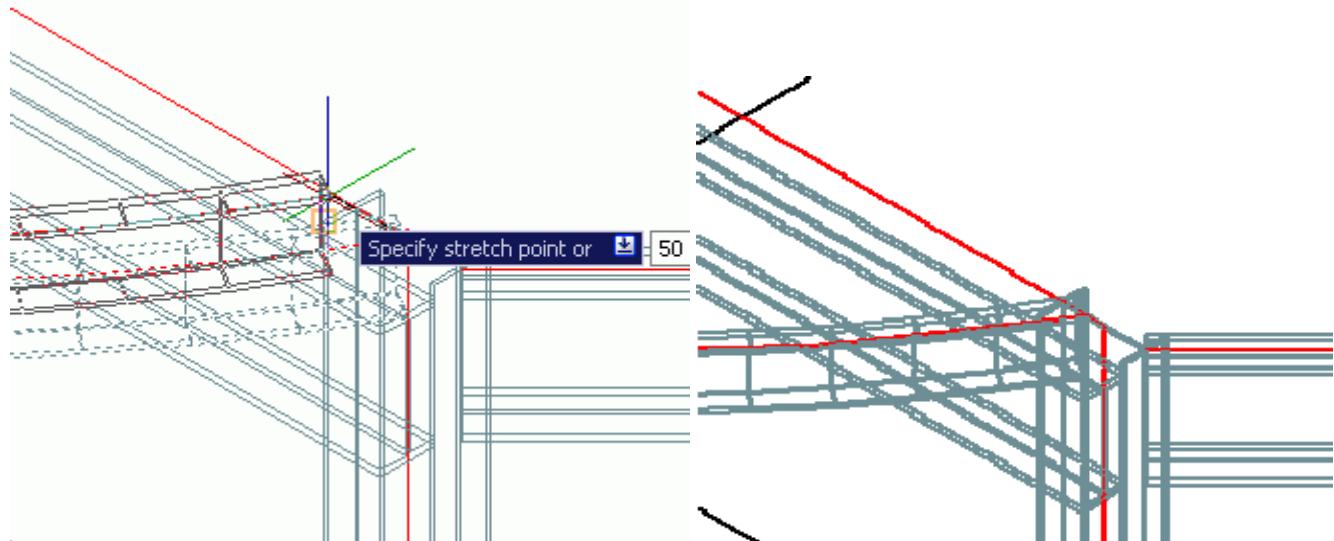


Figure 47: Adjusting the end point position

Using the same process, adjust the curved beam position from the **C1** system line end point by dragging the mouse pointer in the X direction.

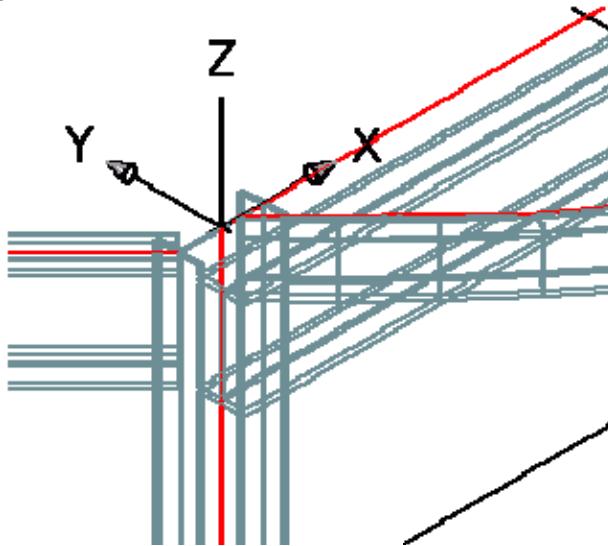


Figure 48: Adjusting the curved beam end point position from the **C1** column

Modifying the curved beam radius

1. Select the curved beam.
2. Right click and select **Advance Properties** from the context menu.
3. In the beam properties dialog box, on the **Positioning** tab enter **4000** for the radius:

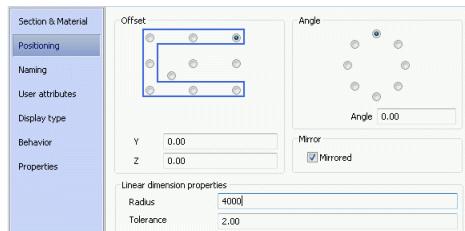


Figure 49: **Positioning** tab – Setting the curved beam radius

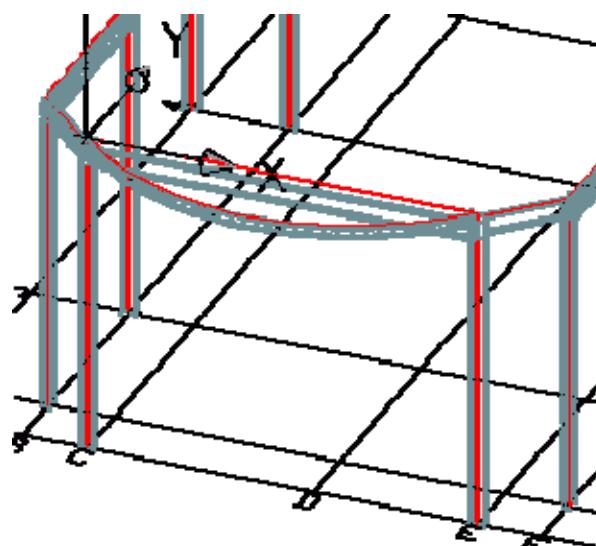
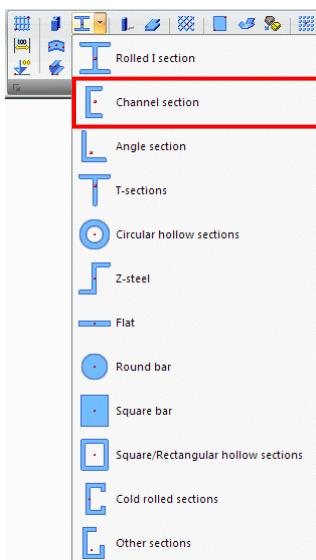


Figure 50: The **E1-C1** curved beam

Step 8: Creating purlins

In this step, create 4 purlins with a **UPN140** section between the **E1-C1** straight beam and the **E1-C1** curved beam.

1. On the **Home** tab, **Objects** panel, select 



2. Use as start point the upper system line end point of the **E1** column.

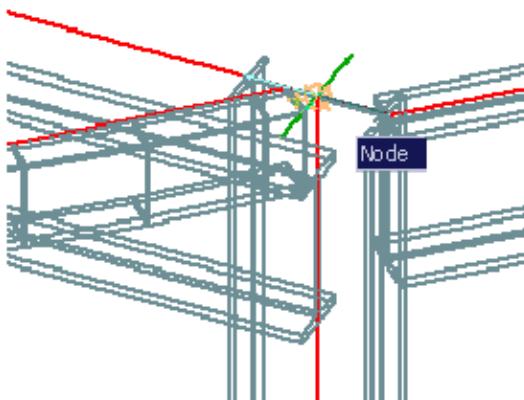


Figure 51: Selecting the start point of the channel section beam: the system line end point of the **C1-E1** straight beam.

3. Move the mouse pointer in the **-Y** direction (the setting **Ortho** causes an exact orientation entry) and enter **1400** to define the channel section beam end point.

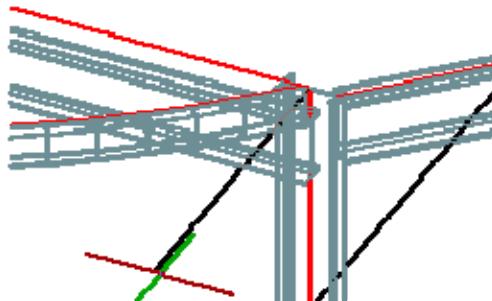


Figure 52: Defining the direction of the channel section beam

4. In the beam properties dialog box, on the **Section & Material** tab, use the  buttons and make the following settings:

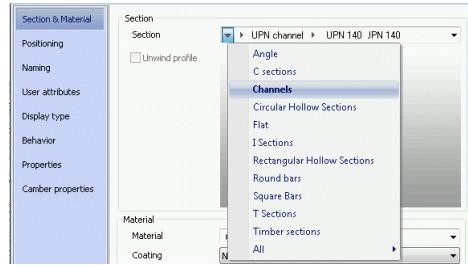


Figure 53: **Section & Material** tab – Selecting the section

Copying the channel section beam

Next, copy the purlin to the **1200, 2400, 3600, 4800** distances from the first purlin in the –X direction.

1. Select the channel section beam.



2. On the **Tools** tool palette, click  to copy the selected beam.
3. Define the reference point anywhere in the drawing.
4. Move the mouse pointer in the –X direction (the **Ortho** setting causes an exact orientation entry). On the command line, enter the distances: **1200, 2400, 3600, 4800**. Press **Enter** after each one.

Delete the first created channel section beam.

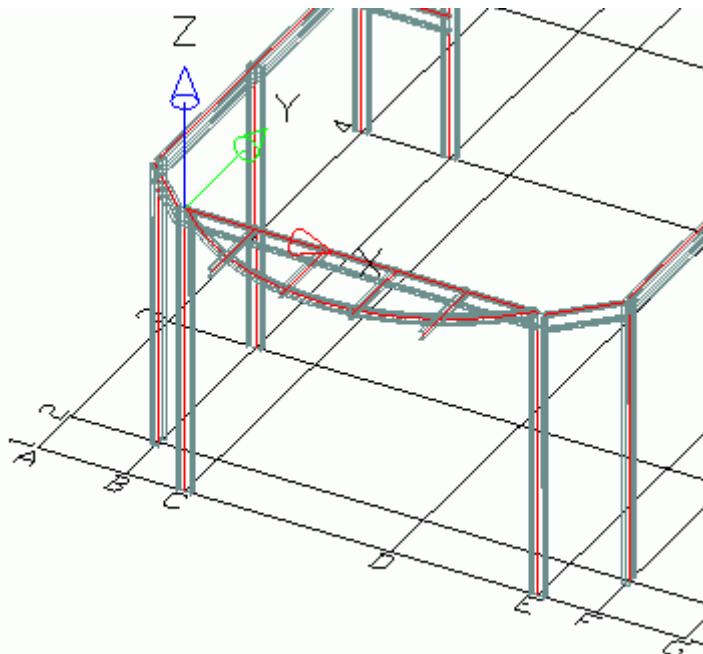


Figure 54: 4 channel section beams

Modify the position and the model role of the channel section beams

1. Select all the channel section beams.

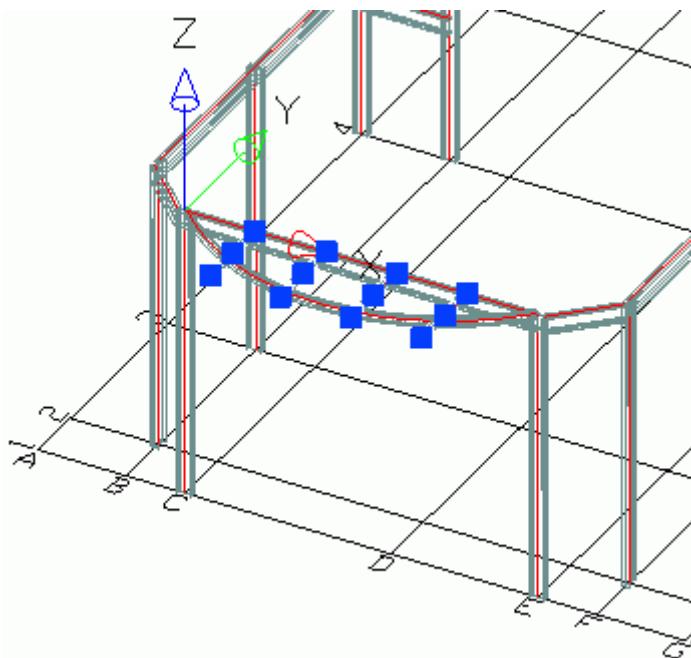


Figure 55: Selecting the channel section beams

2. Right click and select **Advance Properties** from the context menu.

3. In the beam properties dialog box, define the following settings:

On the **Positioning** tab, select the position of the channel section beam.

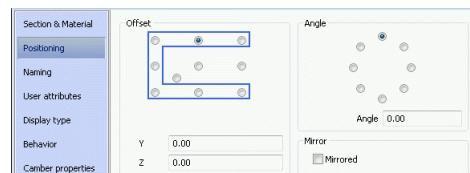


Figure 56: **Positioning** tab – Selecting the position

On the **Naming** tab:

- Set the model role to **Purlin**.

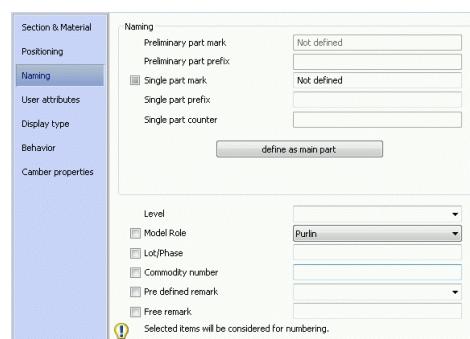


Figure 57: **Naming** tab – Selecting the model role

Adjusting the purlin position

1. Select all the channel section beams.
2. Right click and select **Move** from the context menu.
3. Specify the base point as in Figure 59 and move the purlins in the Y direction until they reach the edge of the **C1-E1** straight beam.



*Make sure that the **Node** and **Perpendicular** OSNAP modes are active.*

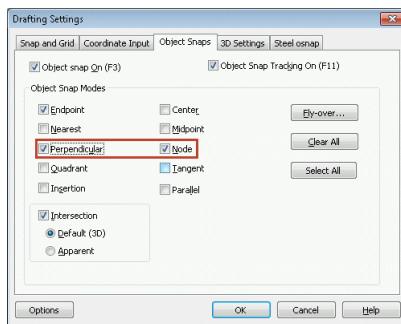


Figure 58: Activating the **Perpendicular** OSNAP mode

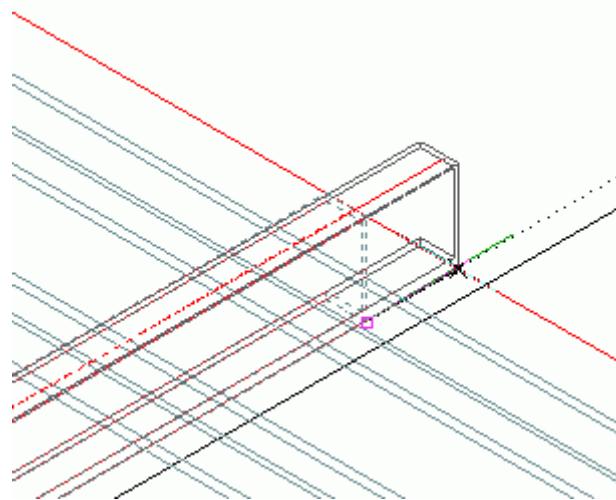


Figure 59: Adjusting the purlin position

Lesson 4: Creating structural elements

In this lesson, you will create structural elements.

You will learn to:

- Create cross bracings
- Create a joist
- Move a structural element

Step 1: Creating cross bracings

In this step, create cross bracings between the **G4** and **G6** columns and the **B3** and **B4** columns.

First, move and rotate the UCS in a suitable position:

1. On the **UCS** tool palette, click .
2. Specify the new origin point in the lower system line end point of the **G4** column.
3. Next, rotate the UCS by X and by Y so that the Y-axis points upwards and the X-axis is in the **G** axis direction as in Figure 60.

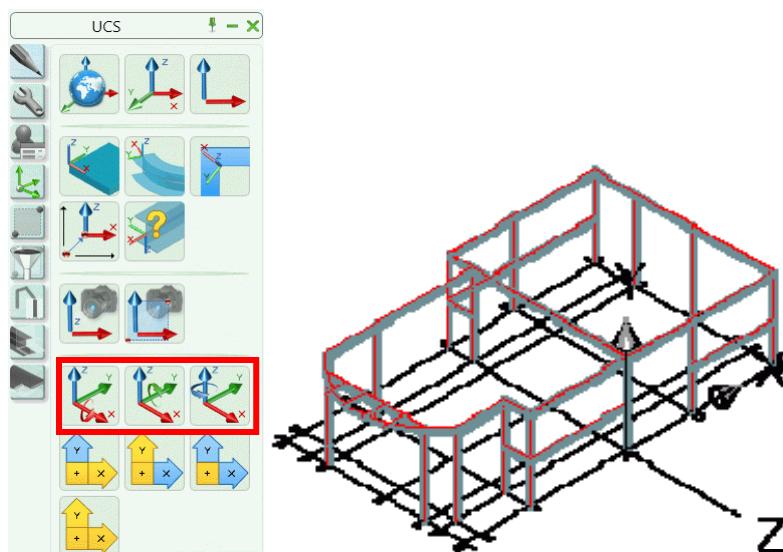
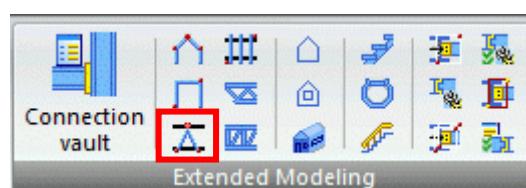


Figure 60: Placing the UCS in a suitable position

4. On the **Home** tab, **Extended modeling** panel, click .



5. Use the lower system line end point of the **G4** column as the base point of the area to stiffen.

6. Select the diagonal point: the upper system line end point of **G5** column.

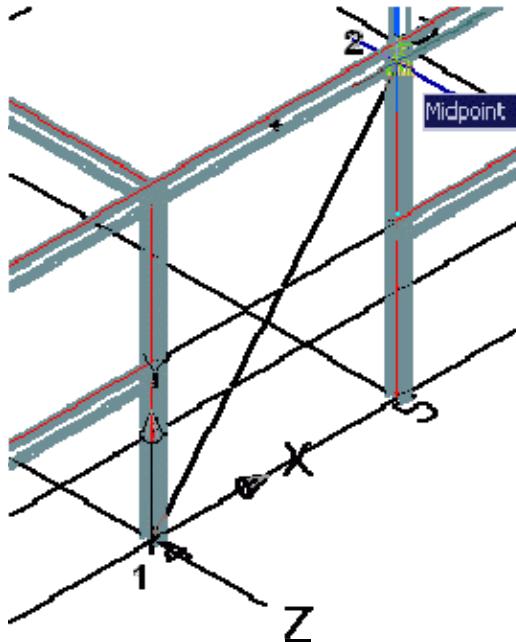


Figure 61: Selecting two diagonal points defining the cross bracings

The properties dialog box appears.

Modifying cross bracing properties

In the structural element properties dialog box, define the following settings:

On the **Type & Section** tab:

- Select the bracing type: **Crossed**.
- Use the  buttons and make the following settings:
 - Select the section class: **Flat**.
 - Select the section size: **FL100X10**.
- From the **Member split** drop-down list, select **Diagonal 1**, to split the bracing.

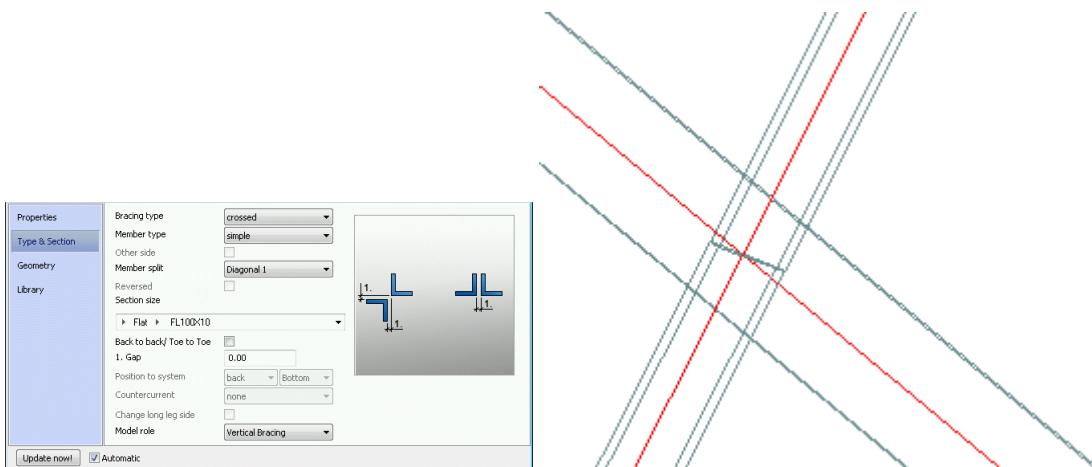


Figure 62: **Type & Section** tab - Splitting the bracing

On the **Geometry** tab:

- Define the offset from top (upper distance): **120**.
- Define the offset from bottom (lower distance): **0**.

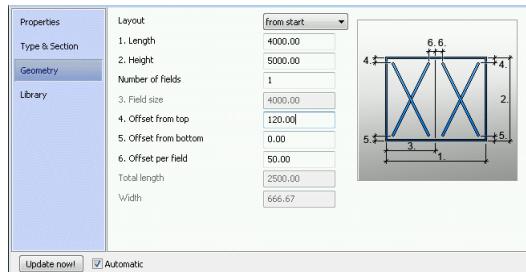


Figure 63: Properties dialog box – **Geometry** tab

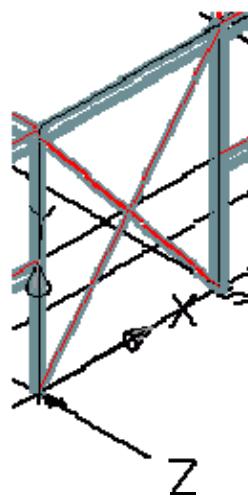
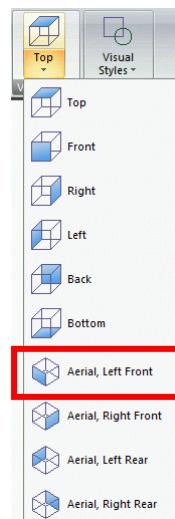


Figure 64: The flat cross bracings

For the second cross bracing, use the tools on the **UCS** tool palette to move and rotate the UCS in a suitable position as in Figure 65. The new origin point is the lower system line end point of the **B4** column.

*Use the buttons on the **Viewpoints** panel to change the view angle.*



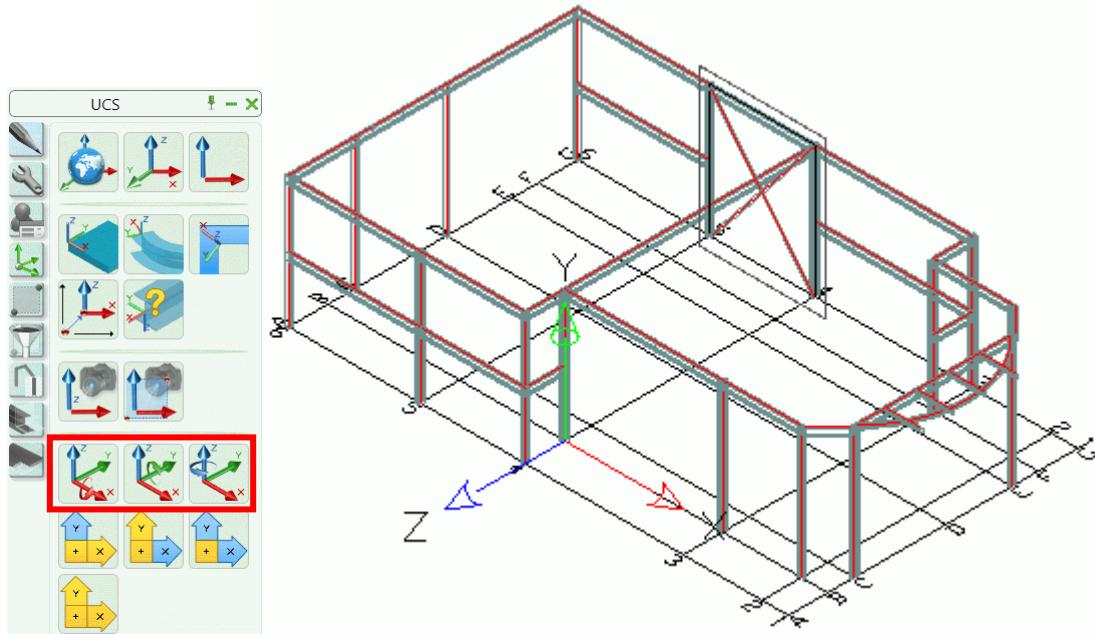
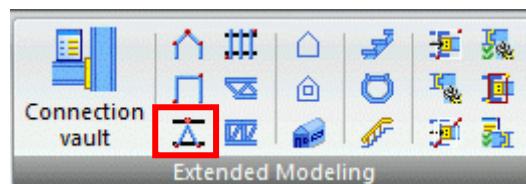


Figure 65: Setting the UCS in a suitable position

4. On the **Home** tab, **Extended Modeling** panel, click .



5. Use the lower system line end point of the **B4** column as the base point of the cross bracing.
 6. Select the diagonal point: the upper system line end point of the **B3** column.

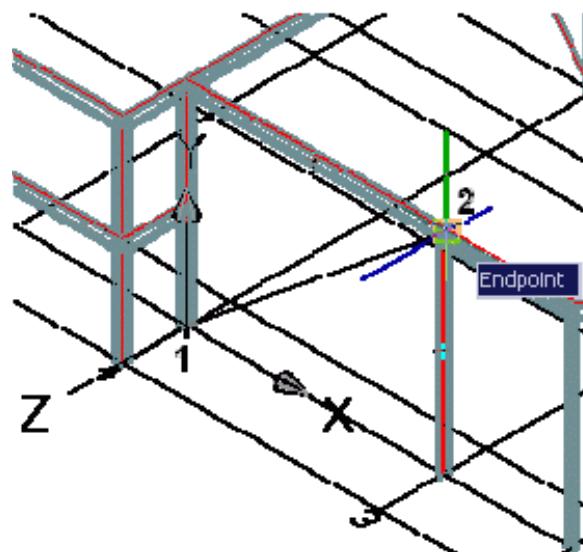


Figure 66: Selecting two diagonal points defining the cross bracings

The properties dialog box appears.

Modifying cross bracing properties

In the structural element properties dialog box, define the following settings:

- On the **Type & Section** tab:
 - Select the bracing type: **Crossed**.
 - Select the section class: **RHS rectangle cold**.
 - Select the section size: **RHS60X40X4**.
 - From the **Member split** drop-down list, select **Diagonal 2**, to split the bracing.

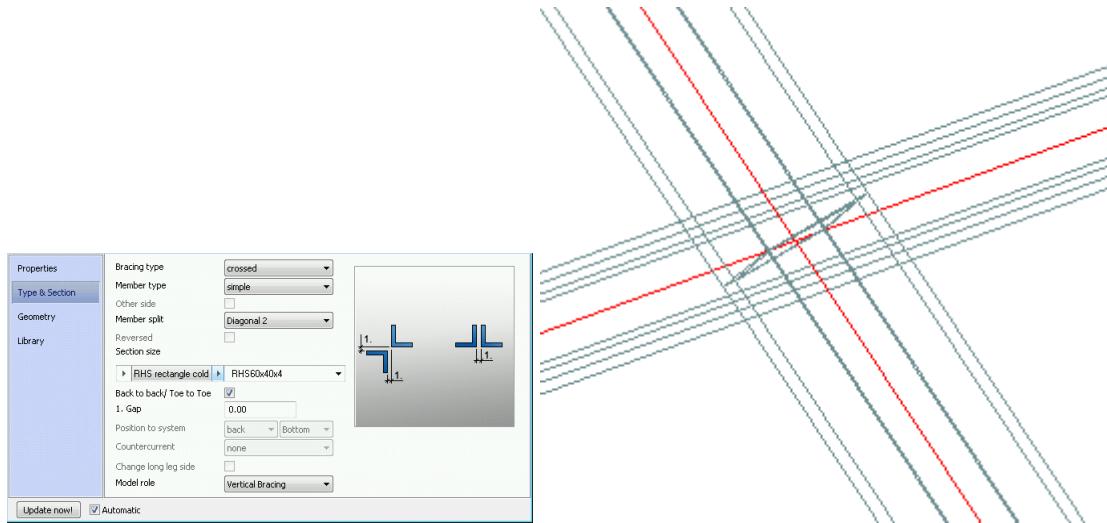


Figure 67: Type & Section tab - Splitting the bracing

- On the **Geometry** tab:
 - Define the offset from top (upper distance): **120**.
 - Define the offset from bottom (lower distance): **0**.

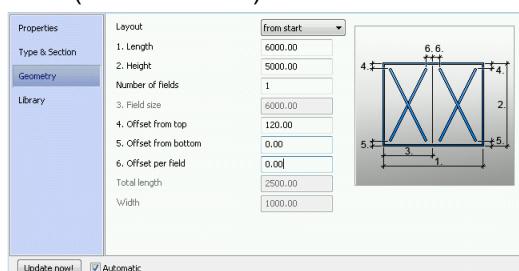


Figure 68: Properties dialog box – Geometry tab

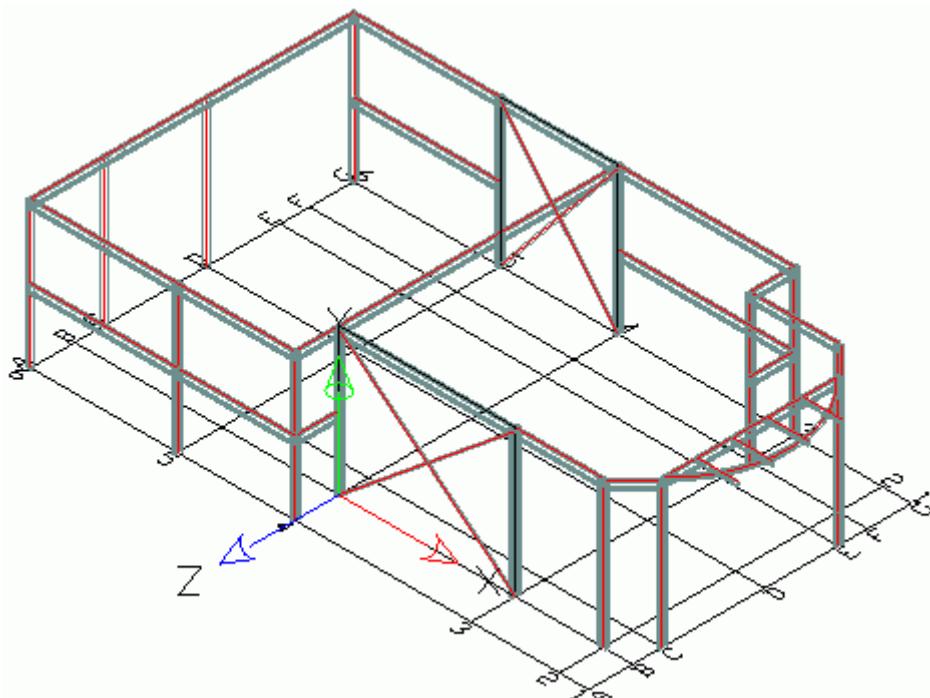


Figure 69: The model with the 2 cross bracings

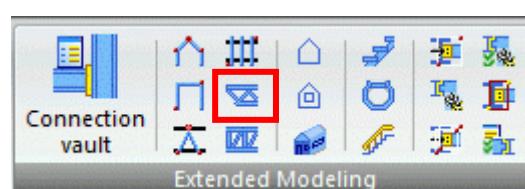
Step 2: Creating a joist

Next, create a joist between the **B3** and **F3** columns.

First, use the tools on the UCS tool palette to move and rotate the UCS in a suitable position as in Figure 70. The new origin point at the upper system line end point of the **C1** column.



1. On the **Home** tab, **Extended Modeling** panel, click .



2. Use the upper system line end point of the **B3** column as the start point of the joist.

3. Use the upper system line end point of the **F3** column as the end point of the joist.
4. Move the mouse pointer in the **-Z** direction to define the orientation and enter **700** for the joist height.

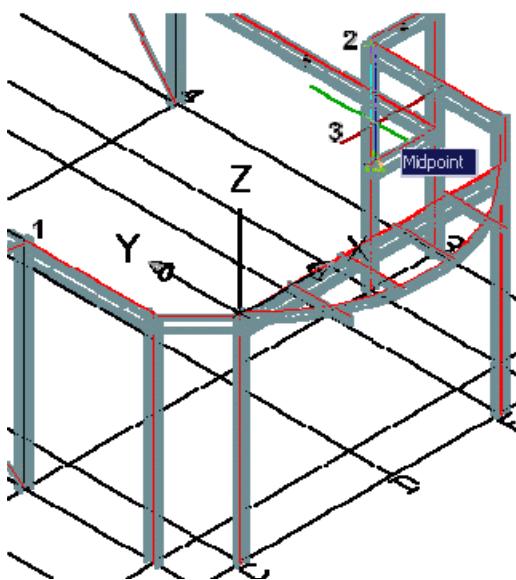


Figure 70: Defining the height and orientation for the joist

The properties dialog box appears.

Modifying joist properties

In the joist properties dialog box, define the following settings:

- On the **Sections** tab:
 - Select the section class: **Angle not identical**.
 - Select the section size: **L80X40X6**.

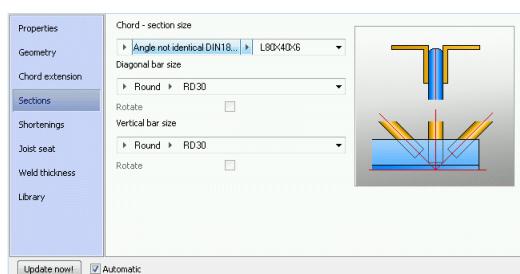


Figure 71: Properties dialog box – **Sections** tab

- On the **Joist seat** tab, modify the thickness of plate 2: **10**.

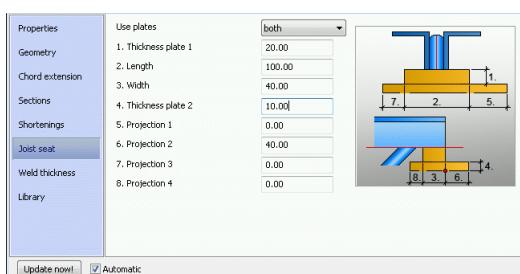


Figure 72: Properties dialog box – **Joist seat** tab

- On the **Geometry** tab, modify if necessary the joist height and width.

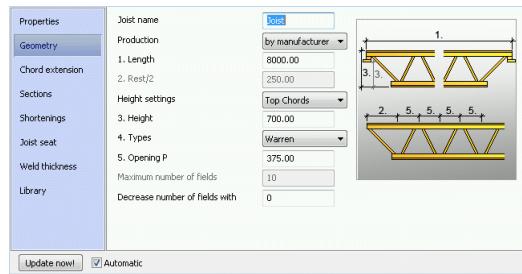


Figure 73: Properties dialog box – **Geometry** tab

- On the **Chord extension** tab, adjust the length of the chords.

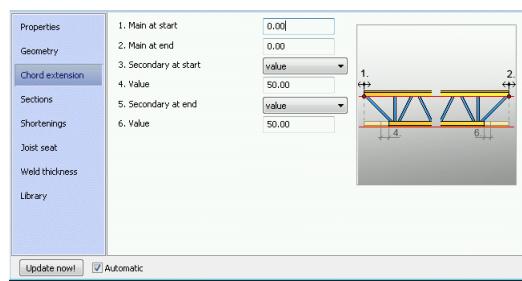


Figure 74: Properties dialog box – **Chord extension** tab

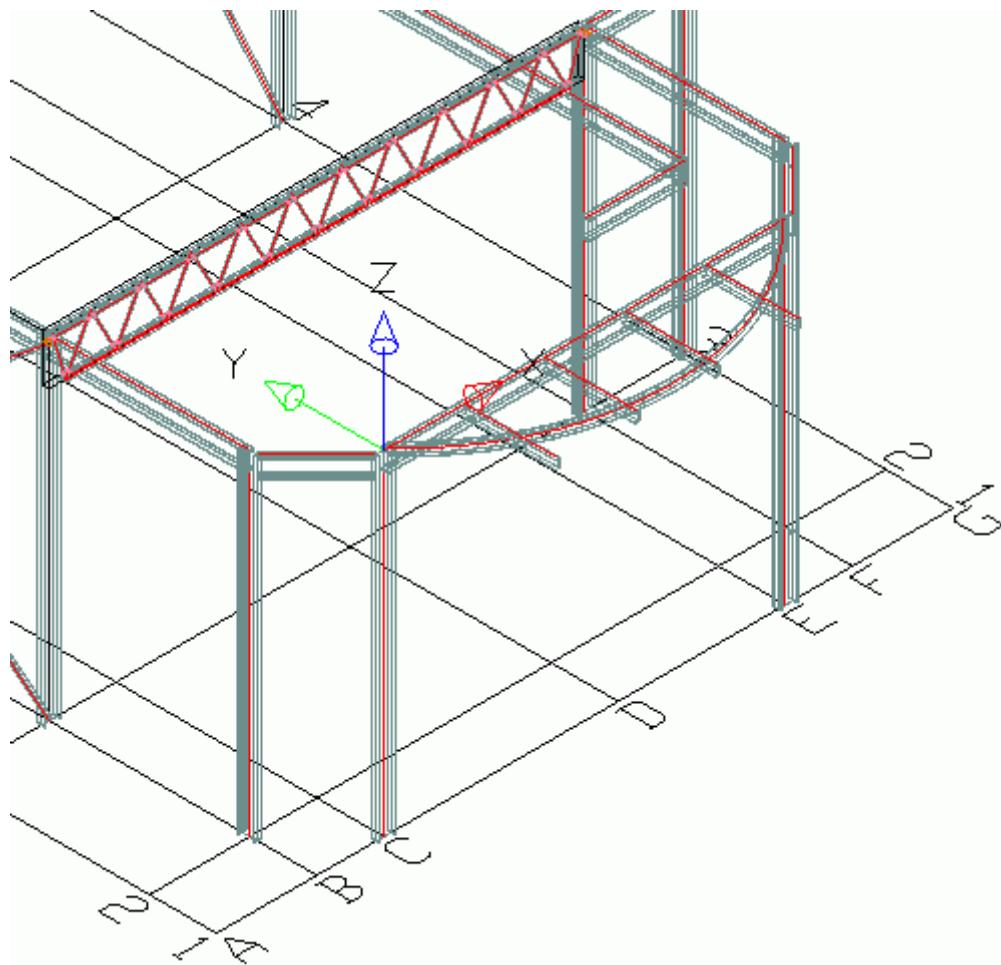


Figure 75: The joist

Step 3: Moving the joist

You must move the joist at an **800** distance in the **-Y** direction to adjust to the context.

1. Select the structural element of the joist.

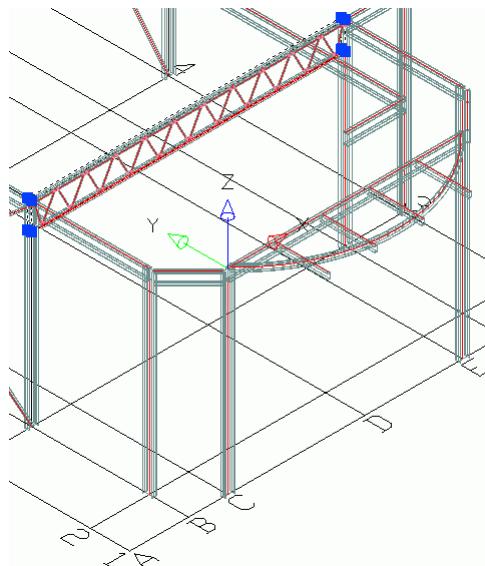


Figure 76: Selecting the joist structural element

2. Right click and select **Move** from the context menu.
3. Define the base point anywhere in the drawing.
4. Move the mouse pointer in the **-Y** direction (the **Ortho** setting causes an exact orientation entry) and enter **800** for the displacement.

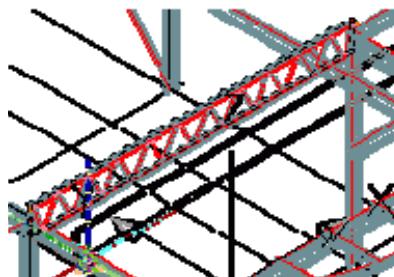
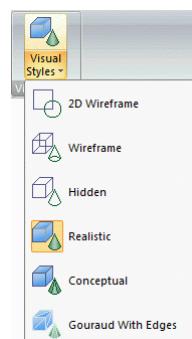


Figure 77: Moving the joist

Note: For a more realistic presentation of the model, use the **Realistic** visual style.



Step 4: Other adjustments

Notice that the model is not right: the joist plate is inside the beam. To correct this situation, adjust the beam positions.

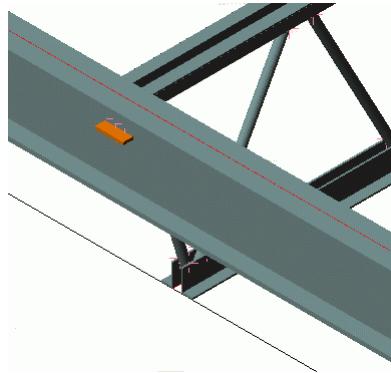


Figure 78: Joist plate incorrectly placed

1. Select the **B2-B3** and the **F2-F3** beams.

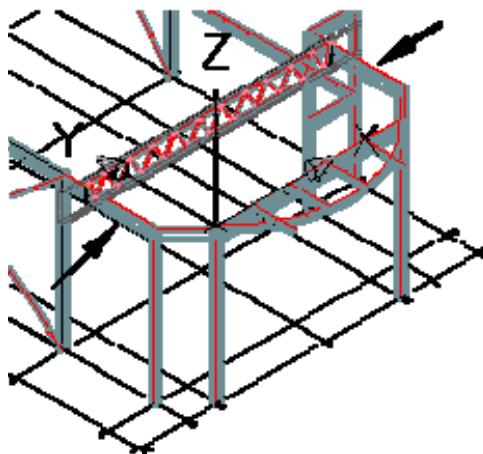


Figure 79: Selecting the **B2-B3** and **F2-F3** beams

2. Right click and select **Advance Properties** from the context menu.
3. In the properties dialog box, on the **Positioning** tab, define the offset value in the Z direction: **-110**.

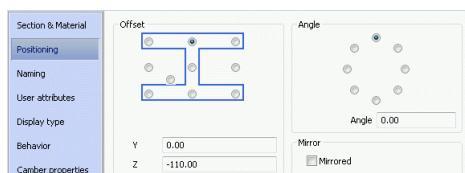


Figure 80: **Positioning** tab – Setting the offset

For a more realistic presentation of the model, use the **Visual styles** panel.

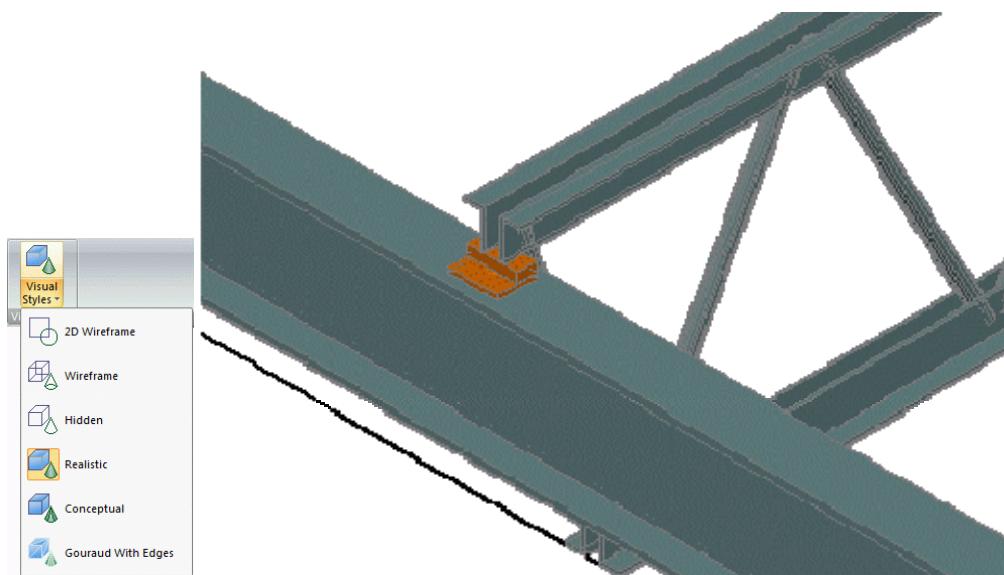


Figure 81: The corrected model

Lesson 5: Defining model views

When working with large models with many elements an isometric view can be confusing. A model view creates local view areas that show only the elements that lie within specific boundaries. Slices are cut from the model for better visualization.

Model views are created using different definition methods. In this lesson, you will define model views:

- With 1 point in the UCS
- With 2 points in the UCS, front and rear depth
- At the selected grid line

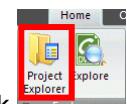
Note: All the views defined in this lesson are further used in Lesson 7 – “Creating intelligent connections”.

All created views are listed in the Model views section of the Project Explorer.

The display of a model view can be turned on or off using the light bulb in front of the name. If you double click on a model view name the model view is turned on and the model rotates to the selected Default View direction.



Figure 82: Project Explorer – Model views category



For quicker access, display the Project Explorer: On the **Home** tab, **Explore** panel, click

Defining a view with 1 point in the UCS

Define a view on the **E1** column and the **C1-E1** straight beam, which will be connected by a clip angle in Lesson 7 – “Creating intelligent connections”.

*Use the buttons on the **Viewpoints** panel to change the view angle.*

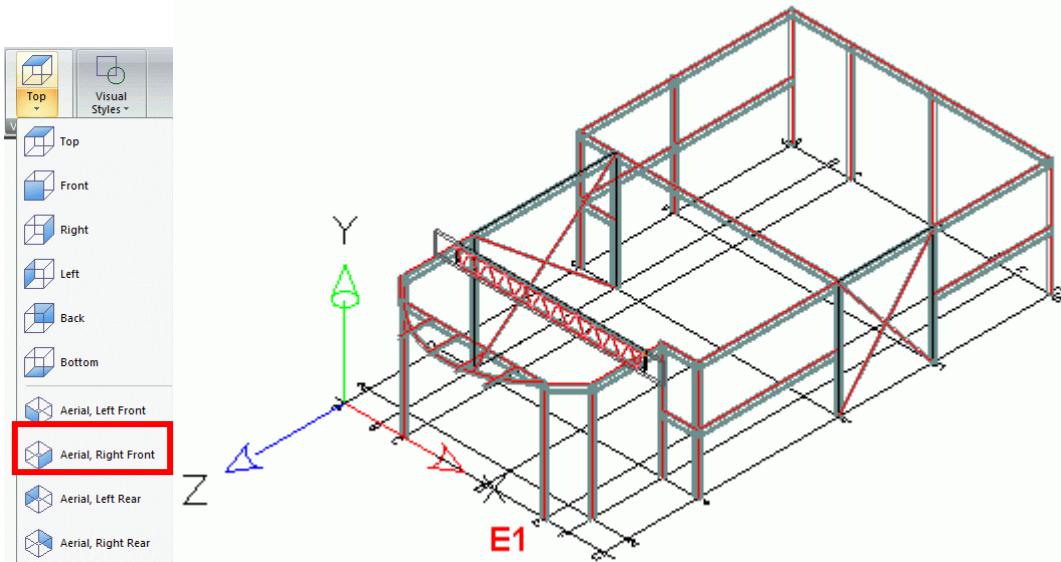


Figure 83: The **E1** column

1. Place the UCS in a suitable position as in Figure 83. The XY plane is in the elevation plane.
2. On the **Quick views** tool palette, click .
3. In the “Choose the definition method” dialog box, select the **One point in UCS** method.

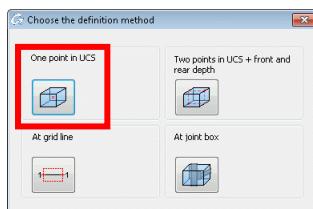


Figure 84: Selecting the view definition method

4. Select the axis intersection of the **E1** column and the **C1-E1** straight beam.

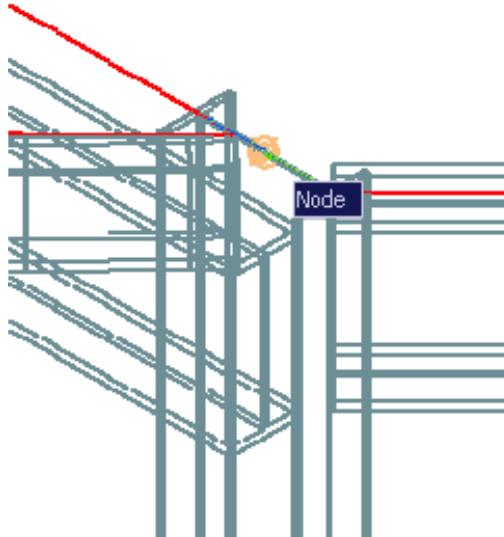


Figure 85: The axis intersection of the **E1** column and the **C1-E1** straight beam

5. On the command line, enter **ClipAngle** for the name of the current view and press **Enter**.
 6. Define the view direction (in this case, from the top) by selecting the top arrow and press **Enter**.
 7. You can modify the view box size using the grips as in the following figure.

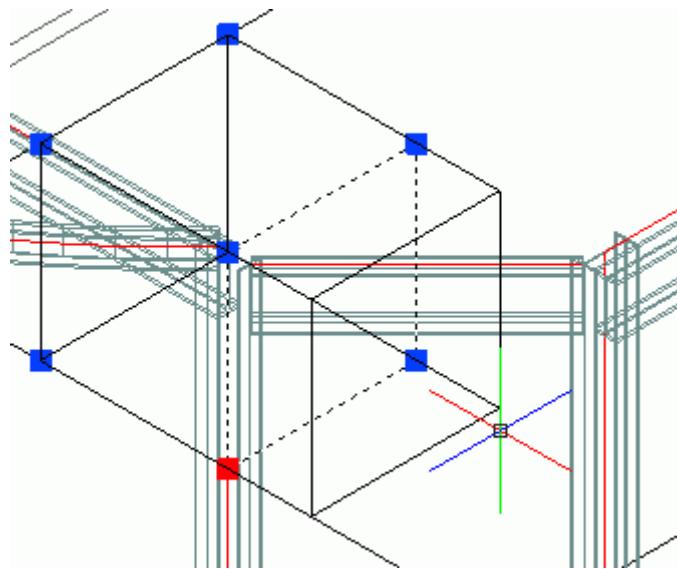


Figure 86: Modifying the view box size

The view is created and appears in the Project Explorer. Clicking the bulb displays only the ClipAngle model view.

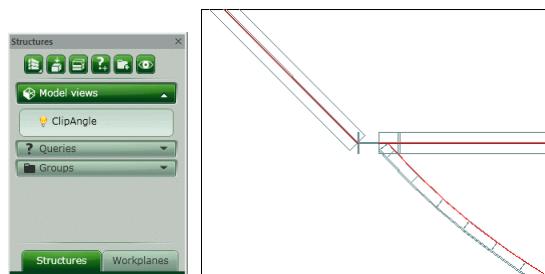


Figure 87: Top view on the **E1** column

Use the buttons on the **Viewpoints** panel to change the view angle.

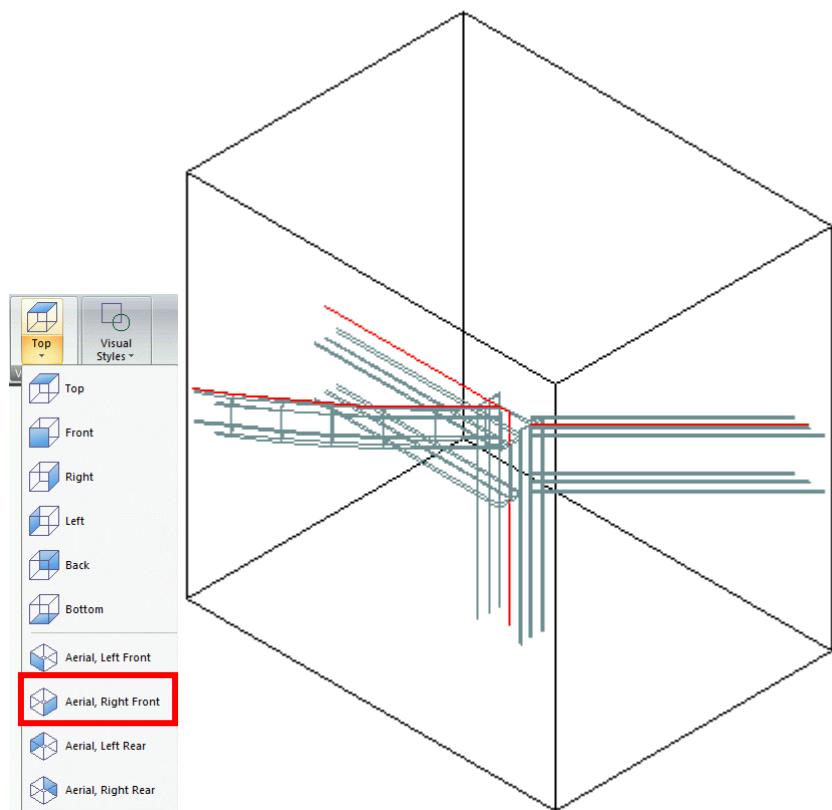


Figure 88: The “Clip angle” view on the E1 column

On the **Quick views** tool palette, click to redisplay all the elements of the model or double click the view box.



Defining a view with 2 points in UCS and front and rear depth

In this step, define a view on the **E1**, **F2** columns and the **E1-F2** beam which will be used for creating the clip angle skewed connection in Lesson 7 – “Creating intelligent connections”.

*Use the buttons on the **Viewpoints** panel to change the view angle.*

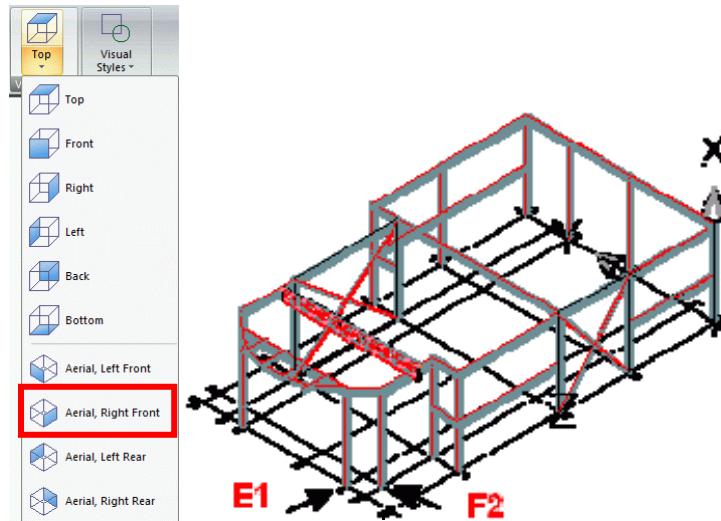


Figure 89: The **E1** and **F2** columns

1. Place the UCS in a suitable position as in the Figure 91 using UCS 3 points  tool on the UCS tool palette. The XY plane is in the elevation plane.



2. On the **Quick views** tool palette, click .

3. In the “Choose the definition method” dialog box, select the **Two points in UCS + front and rear depth** method.

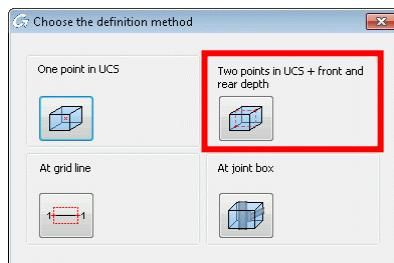


Figure 90: Selecting the view definition method

4. Select the first point of the view: the exterior lower end point of the **E1** column.
5. Select the second point of the view: the exterior upper end point of the **F2** column.

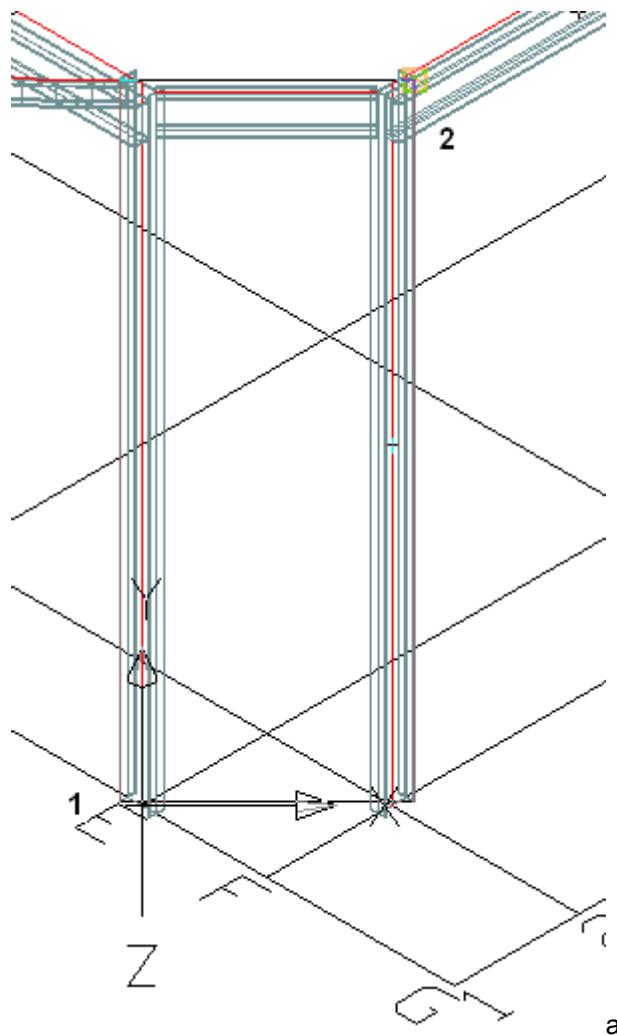


Figure 91: Selecting the two points defining the view

6. On the command line, define the view depth:
– Front depth: **500** and press **Enter**
– Rear depth: **500** and press **Enter**.

7. On the command line, enter **ClipAngleSkewed** for the name of the current view and press **Enter**.
6. Define the view direction (in this case, from the top) by selecting the top arrow and press **Enter**.

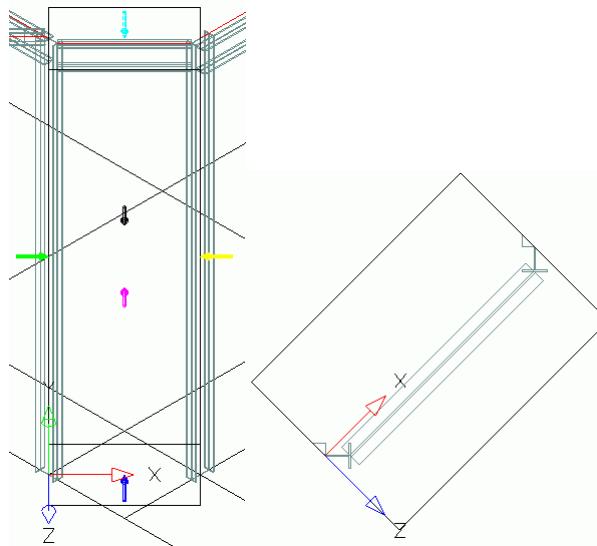


Figure 92: **ClipAngleSkewed** view in top view

The view is created and appears in the Project Explorer. Clicking the light bulb displays only the ClipAngleSkewed model view.



Figure 93: Project Explorer – **ClipAngleSkewed** view

Tip: The view box can be enlarged using the grip points. Right click and select **Activate** from the context menu to view the result.

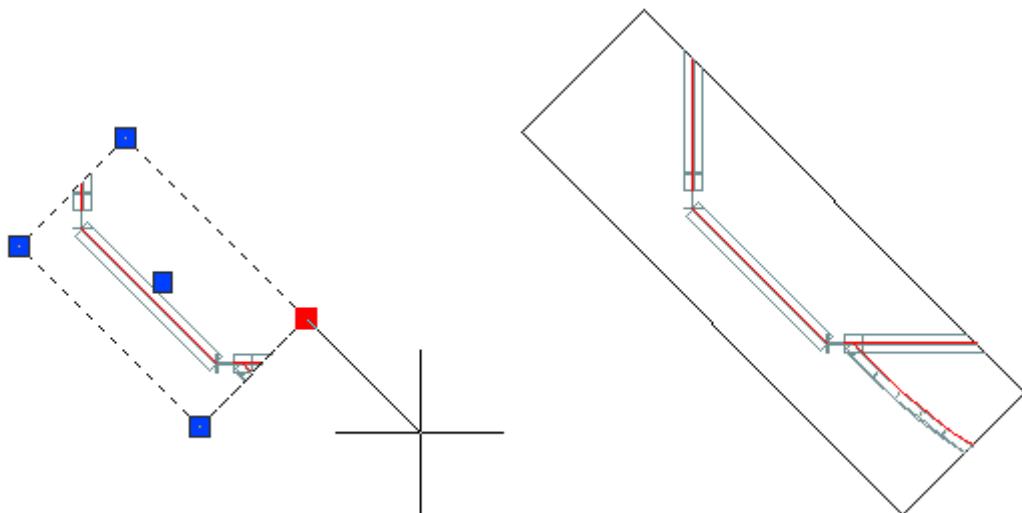


Figure 94: Resizing the view

Use the buttons on the **Viewpoints** panel to change the view angle.

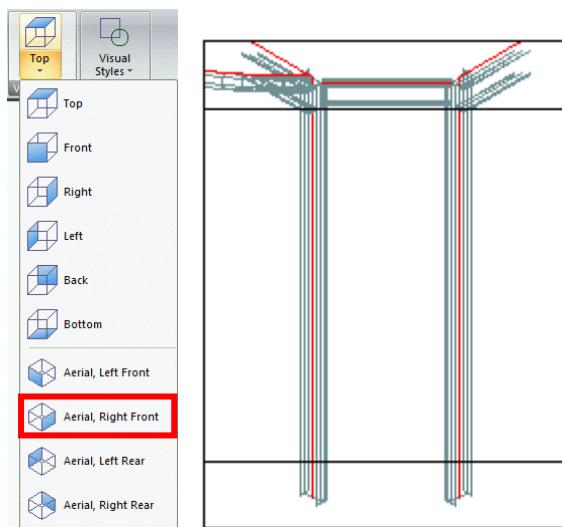


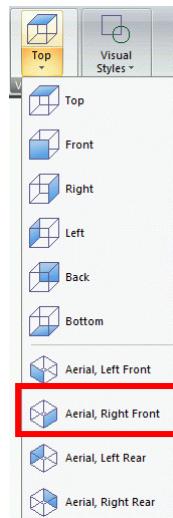
Figure 95: The “ClipAngleSkewed” view on the E1 and F2 columns

On the **Quick views** tool palette, click to redisplay all the elements of the model or double click the view box.

Defining a view at a grid line

Define a view on the **C6, D6** rectangular columns and the **A6-G6** beam, which will be used for creating gable wall and front plate splice connections in Lesson 7 – “Creating intelligent connections”.

Use the buttons on the **Viewpoints** panel to change the view angle.



1. On the **Quick views** tool palette, click .
2. In the “Choose the definition method” dialog box, select **At a grid line** method.

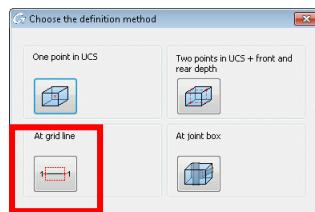


Figure 96: Selecting the view definition method

3. Select the grid line labeled with **6** and press **Enter**.
4. On the command line, enter **GableWall** for the name of the current view and press **Enter**.

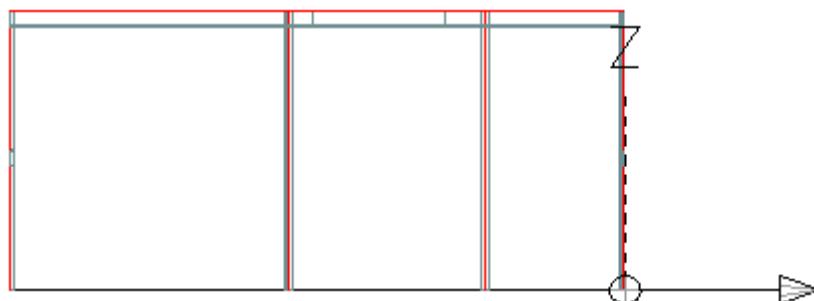


Figure 97: **GableWall** view

5. Define the view direction by selecting the left arrow and press **Enter**.

The view is created and appears in the Project Explorer. Clicking the light bulb displays only the GableWall model view.

*Use the buttons on the **Viewpoints** panel to change the view angle.*

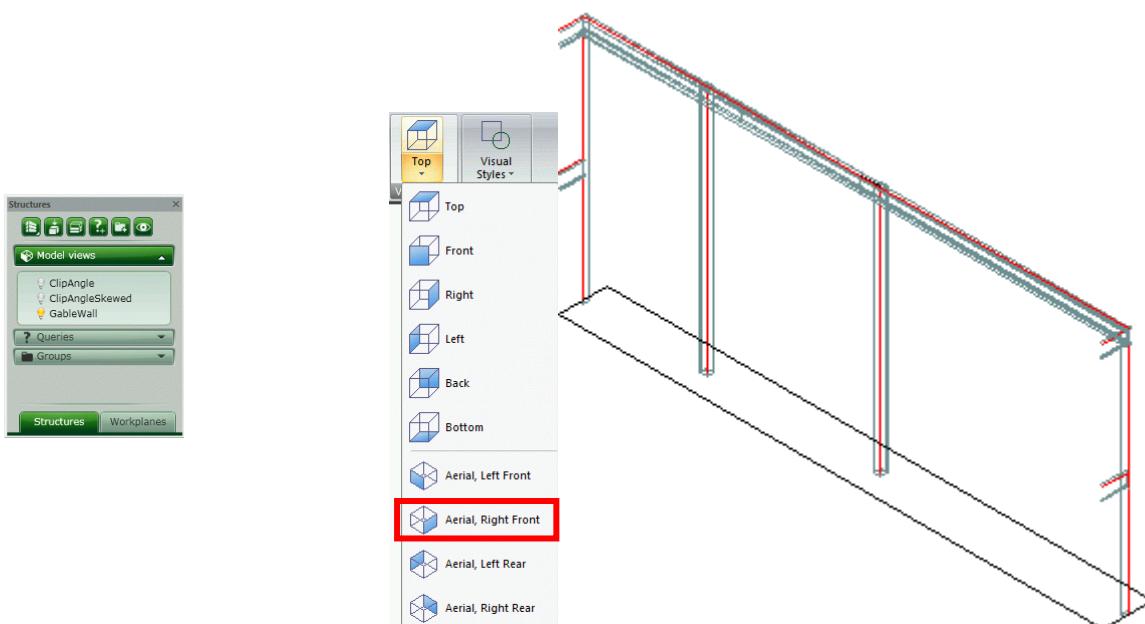
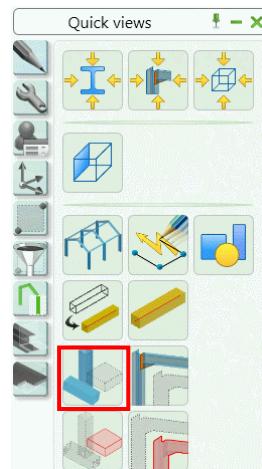


Figure 98: The “GableWall” view on the **C6** and **D6** columns



On the **Quick views** tool palette, click to redisplay all the elements of the model or double click the view frame.

If necessary, the view boxes can be hidden using the “Selected objects off” tool.



Lesson 6: Working with intelligent connections

In this lesson you will create base plates for all columns of the model and end plate connections between beams and columns.

You will learn how to:

- Create simple connections – a base plate connection and a single side endplate connection
- Modify the connection properties
- Handle several similar connections using groups of connections
- Defining the master connections
- Creating slave connections
- Adding connections to an existing group
- Removing a connection from a group of connections
- Create other connections by template

Concepts

Joints

Joints are intelligent connections that consist of basic elements such as plates, stiffeners, beams, bolts and welds and dependent elements (shortenings, copes) controlled by construction rules. An Advance joint is created with all its parts and features using a single operation.

A joint is represented in the model by a gray box connection object that contains all the connection parts.

Joints are available in the **Connection vault** which is accessible from the **Extended modeling** panel of the **Home** tab.

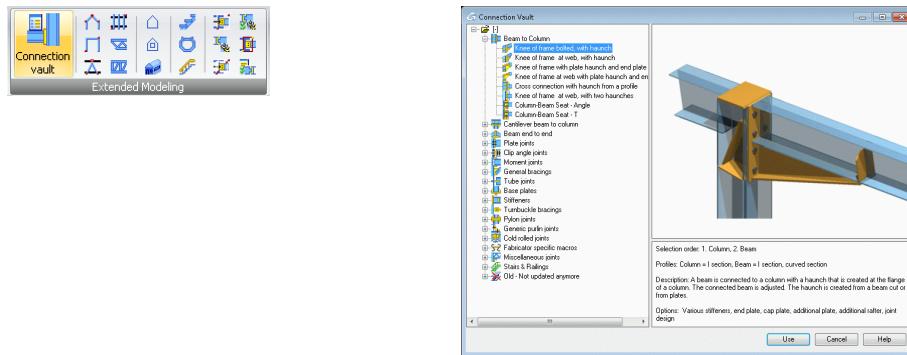


Figure 99: Accessing Connection vault

Joint group

Sometimes it is useful to link several joints together so that if one changes all other joints change and continue to match.

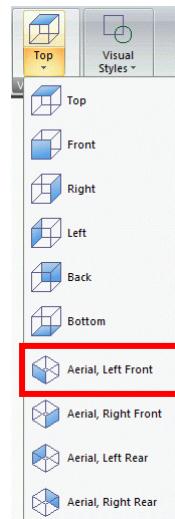
In a **joint group** one joint is the *master* and the other joints in the group are *slaves* and must always match the master. Any changes to the master joint are immediately applied to all slave joints.

In the joint properties dialog box of a slave joint all controls are disabled to prevent editing.

Step 1: Creating a base plate joint

In this step, create a base plate with 4 bolts for the C1 column.

*Use the buttons on the **Viewpoints** panel to change the view angle.*



1. On the **UCS** tool palette, click
2. Specify the new origin point in the **A1 axis** intersection.
3. Next, rotate the UCS by X and by Y so that the XY plane is in the grid plane as in Figure 100.

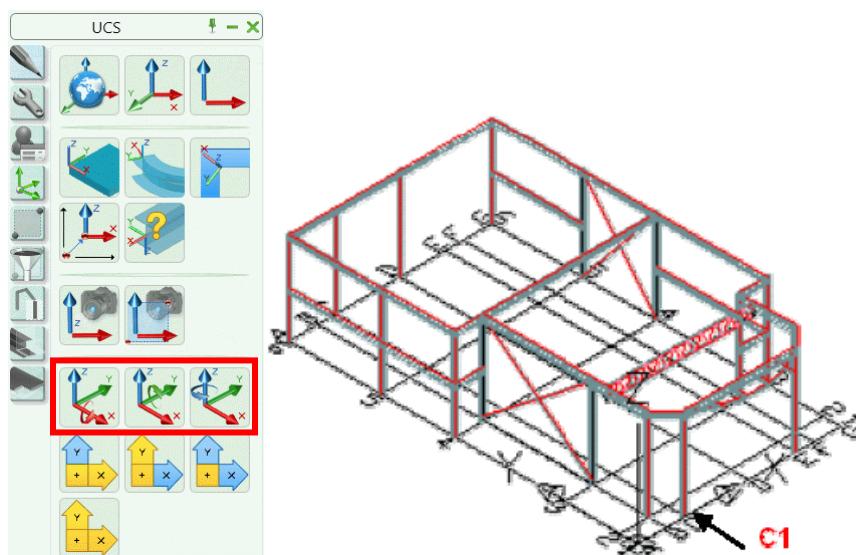


Figure 100: Placing the UCS in a suitable position

4. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click

5. From the **Base plates** category, click



6. Select the C1 column and press **Enter**.

The base plate with anchor bolts is created at the end of the column. The plate is welded to the column. The properties dialog box appears.

Step 2: Creating a group of base plate connections

In this step, the base plate created for the **C1** column is defined as the master connection for other base plates in new locations.

Defining the master connection

The connection between the master connections and the slave joints is made through the name of the master connection.

Note: *The name of the master connection must be unique. The slave connections receive the same name automatically.*

1. Select any part of the **C1** base plate joint, for example the plate.
2. Right click and select **Advance joint properties** from the context menu.
3. In the properties dialog box, in the Name field, enter the name of the master connection.

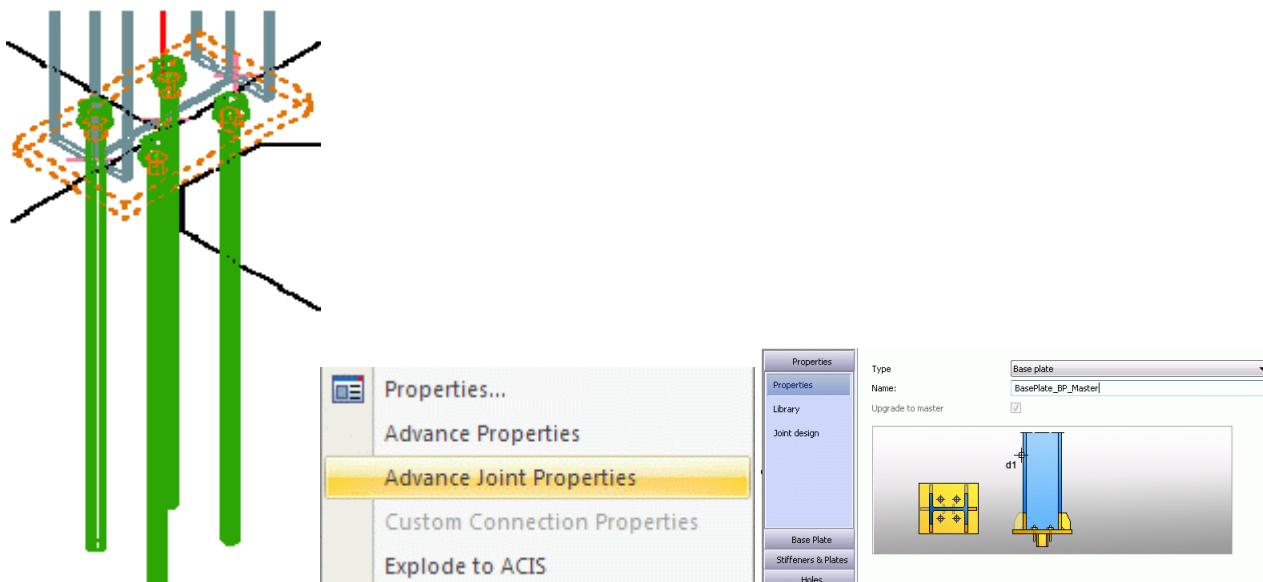
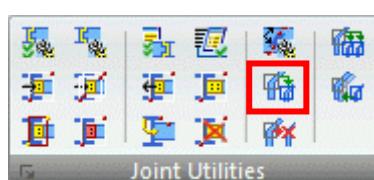


Figure 101: Defining the master connection

Creating slave connections

In this step, the base plate created for the **C1** column is used as template for other base plates in new locations.

1. On the **Extended modeling** tab, **Joint Utilities** panel, click .



2. Select any part of the template joint (e.g. the plate of the **C1** base plate joint) and press **Enter**.

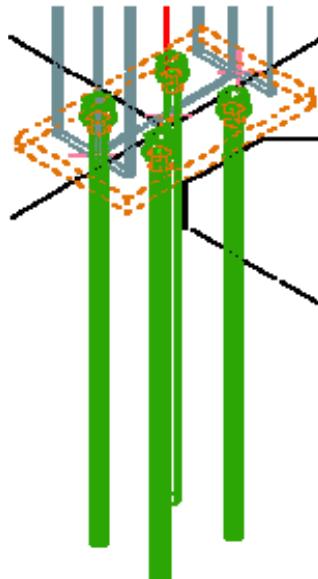


Figure 102: Selecting the base plate

3. One by one, select the columns to which the base plate will be copied and right click after each one (all columns of the model).

The base plates are created on the selected columns.

Note: Notice that the columns along the 6 axes differ in geometry from the other columns from the model (have a square section) and the plate size is according to the column section (Figure 104). In this case, the slave joints are not the same as the master connection and cannot be part of the group.

4. Click **Yes** in the warning message box.

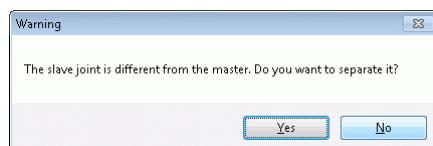


Figure 103: Warning message box

The slave joints receive the same name automatically.

The base plates of the square columns are separated from the joint group. As the columns have the same section, the connections can be part of another group.

For a more realistic presentation of the model, use the **Realistic** visual style.

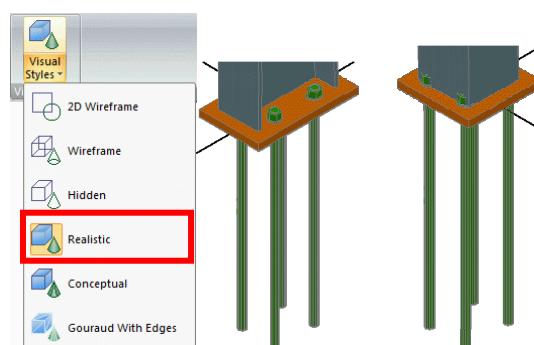


Figure 104: Base plate for the square section columns

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.

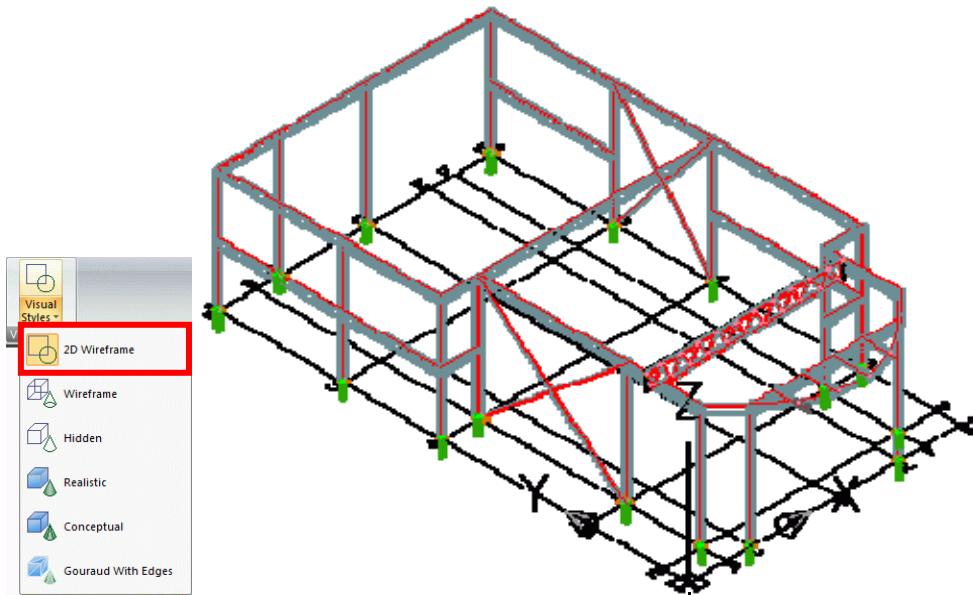


Figure 105: The model with all base plates

Step 3: Adding a connection to an existing group

In this step, the base plates created for the column along the **6** axes will be grouped.

Define the **A6** base plate as the master connection by setting a name in the properties dialog box.

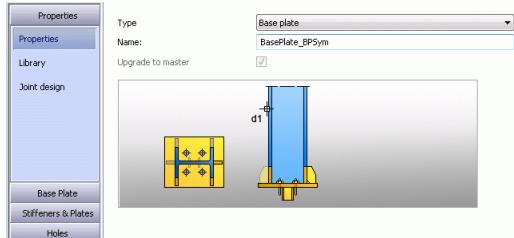
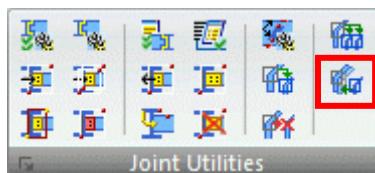


Figure 106: Defining the master connection

1. On the **Extended modeling** tab, **Joint Utilities** panel, click .



2. Select the joint to add to the new group – the base plate of the **C6** column – and press **Enter**.
3. Select the master joint – the base plate of the **A6** column – and press **Enter**.

Using the same process, add the base plates of the **D6** and **G6** columns to the group.

The four connections are linked. Any modification in the master connection is transferred immediately to all the connections in the group.

Step 4: Removing a joint from a joint group

In this step, the **B3** and **B4** base plate joints will be removed from the group. This will allow editing them independently from other joints of the group.

1. On the **Extended modeling** tab, **Joint Utilities** panel, click .



2. Select any part of the joint to be removed from the group, in this case, the **B4** base plate and press **Enter**.

The **B4** base plate joint is no longer part of the group. The properties can be changed independently from the group.

Using the same process, remove the **B3** base plate connection from the group.

Step 5: Modifying the properties of a simple joint

In this step, create asymmetrical base plates of the **B3** and **B4** columns by modifying the base plate size, the bolt properties, the number of bolt lines and columns.

Creating an asymmetrical base plate

Modify the size of the base plate.

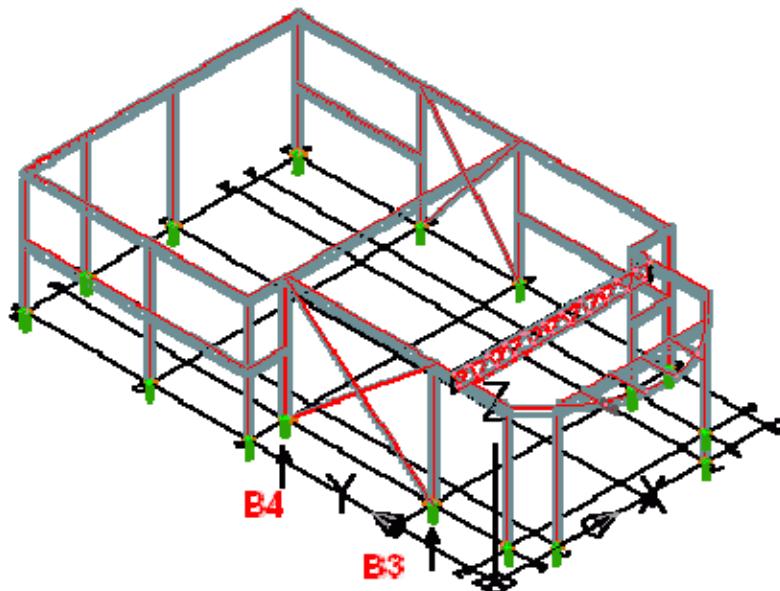


Figure 107: The **B3** and **B4** columns with asymmetrical base plates

1. Select any part of the **B4** base plate.
2. Right click and select **Advance Joint Properties** from the context menu.

3. In the properties dialog box, select the **Base plate** category and define the following settings:

On the **Base plate layout** tab:

- Define the plate thickness: **20**.

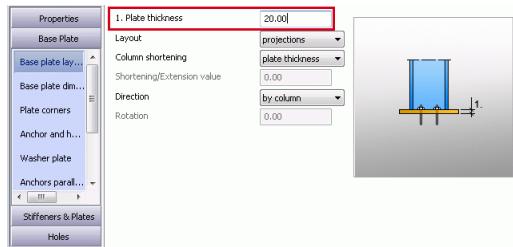


Figure 108: Properties dialog box – **Base plate layout** tab

On the **Base plate dimensions** tab:

- As the base plate is not symmetric, unselect **All projections equal**.
- Set the Projection 1, 2 and 3 to **30**.
- Set the Projection 3 value to **200**.

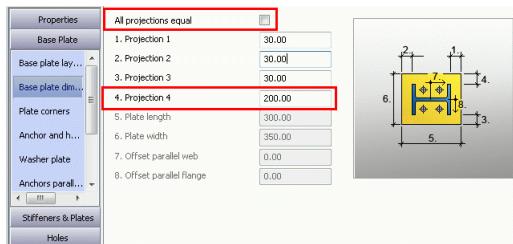


Figure 109: Properties dialog box – **Base plate dimensions** tab

On the **Anchor and holes** tab:

- Select the anchor diameter: **20.00 mm**.

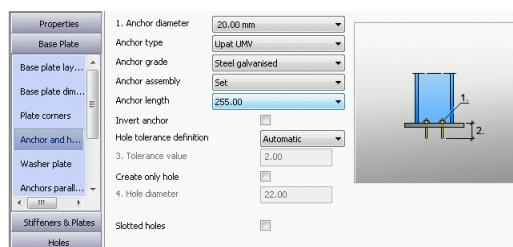


Figure 110: Properties dialog box – **Anchor and holes** tab

On the **Anchors parallel web** tab:

- Define the number of bolt lines in the first group (parallel to the column's web): **2**.
- Define the distance between the bolt lines: **100**.
- Define the offset from the center of the beam section: **0**.

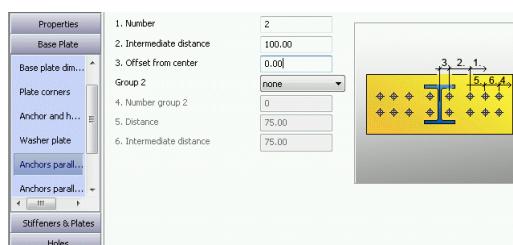


Figure 111: Properties dialog box – **Anchors parallel web** tab

On the **Anchors parallel flange** tab:

- Define the number of bolt lines (parallel to the beam's flange): **2**.
- Define the intermediate distance between bolt lines: **120**.
- Define the offset from the center of the beam section: **0**.

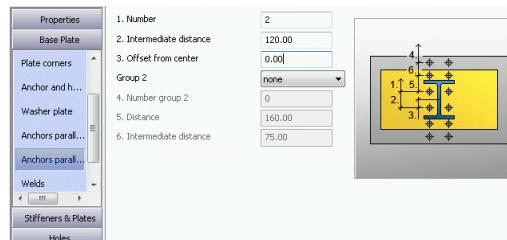


Figure 112: Properties dialog box – **Anchors parallel flange** tab

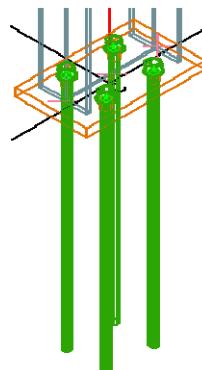


Figure 113: The asymmetrical base plate

*For a more realistic presentation of the model, use the **Realistic** visual style.*

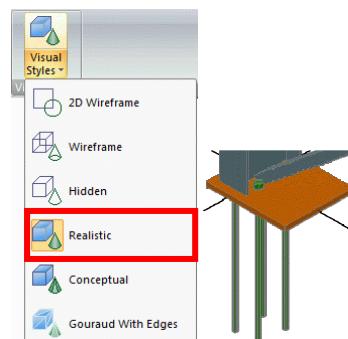


Figure 114: Asymmetrical base plate of the **B4** column

Using a similar process, define the same properties for the **B3** base plate joint:

- On the **Base plate dimensions** tab:
 - Set the Projection 4 value to **200**.
 - Set the Projection 3 value to **30**.

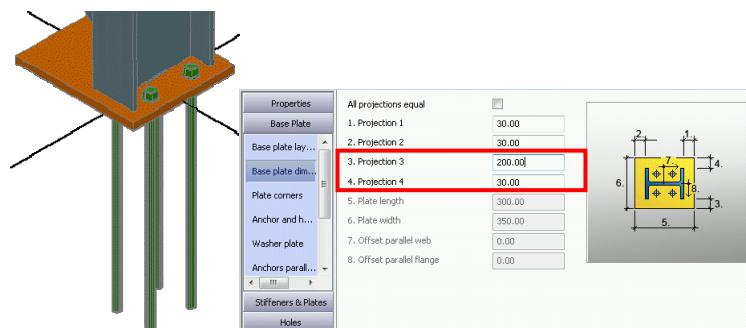
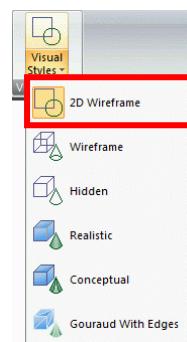


Figure 115: Asymmetrical base plate of the **B3** column – Base plate dimensions

*To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.*



Step 6: Modifying joint properties within a group

Next, modify the base plate size, the bolt properties, the number of bolt lines and columns. The modifications will be transferred automatically to the other joints of the group.

To change the properties of joints of a group the master joint must be modified as all editing of slave joints is disabled. However you may not know which joint is the master.

There are two methods to modify the properties of grouped joints:

- Use the **Search filter** to search the master joint and modify the properties.

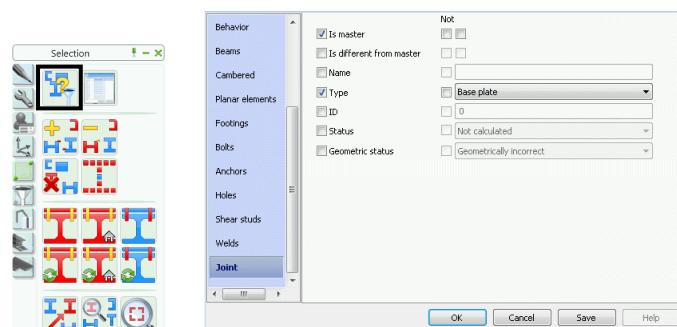


Figure 116: Searching the master joint

- Access the properties of any joint of the group and enable **Upgrade to master** option on the **Properties** tab. The current joint becomes the master of the group and any of its properties can be modified in the current dialog box.

1. Select any part of a base plate joint, for example the plate of the **E1** base plate joint.
2. Right click and select **Advance Joint Properties** from the context menu.

Note: Notice that in the properties dialog box all controls are grayed as the **E1** base plate joint is not the master joint in the group.

3. In the properties dialog box, on the **Properties** tab, enable **Upgrade to master**.

The **E1** base plate becomes the master joint of the group and its properties can be modified.

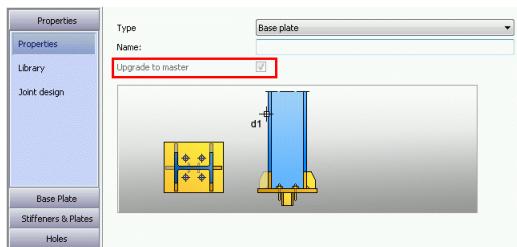


Figure 117: Upgrading the joint to master

4. Select the **Base plate** category and define the following settings:

On the **Base plate layout** tab:

- Define the plate thickness: **20**.
- From the **Layout** drop-down list, select **projections** to calculate the base plate length according to the set projections.
- From the **Column shortening** drop-down list, select **plate thickness** to shorten the column by the plate thickness.

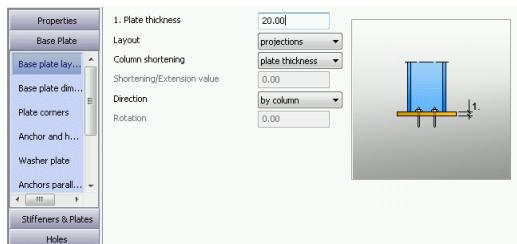


Figure 118: Properties dialog box – **Base plate layout** tab

On the **Base plate dimensions** tab:

- Select **All projections equal** to define the plate size by equal projections.
- Define the projection value: **30**.

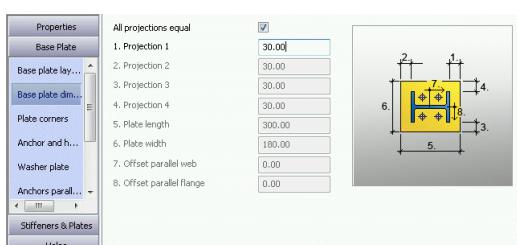


Figure 119: Properties dialog box – **Base plate dimensions** tab

On the **Anchor and holes** tab:

- Select the anchor diameter: **20.00 mm**.

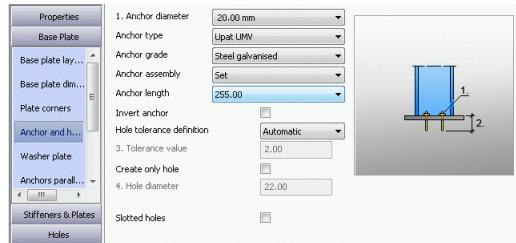


Figure 120: Properties dialog box – **Anchor and holes** tab

On the **Anchors parallel web** tab:

- Define the number of bolt lines in the first group (parallel to the column's web): **2**.
- Define the distance between the bolt lines: **100**.
- Define the offset from the center of the beam section: **0**.

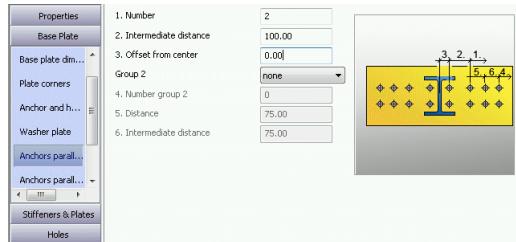


Figure 121: Properties dialog box – **Anchors parallel web** tab

On the **Anchors parallel flange** tab:

- Define the number of bolt lines (parallel to the beam's flange): **2**.
- Define the intermediate distance between bolt lines: **120**.
- Define the offset from the center of the beam section: **0**.

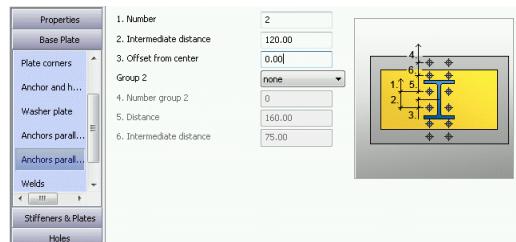


Figure 122: Properties dialog box – **Anchors parallel flange** tab

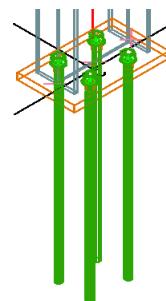


Figure 123: The base plate

The modifications are transferred automatically to the other base plate joints from the group.

View the result

For a more realistic presentation of the model, use the **Realistic** visual style.

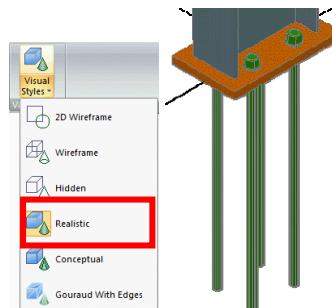


Figure 124: Base plate joint in shaded mode

Note: If the bolts have the Standard representation, for a clear view, display the bolts in the solid representation:

1. Select the anchor bolts.
2. Right click and select **Advance Properties** from the context menu.
3. In the properties dialog box, on the **Display type** tab, select the **Solids** option.

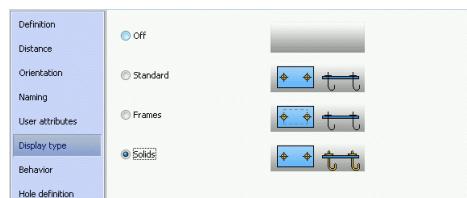


Figure 125: Displaying bolts in the solid representation

Using a similar process, modify the base plate size and the distances between the bolts of the **A6** base plate joint to match the square section column. The modifications will be transferred automatically to the other three joints of the group.

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.

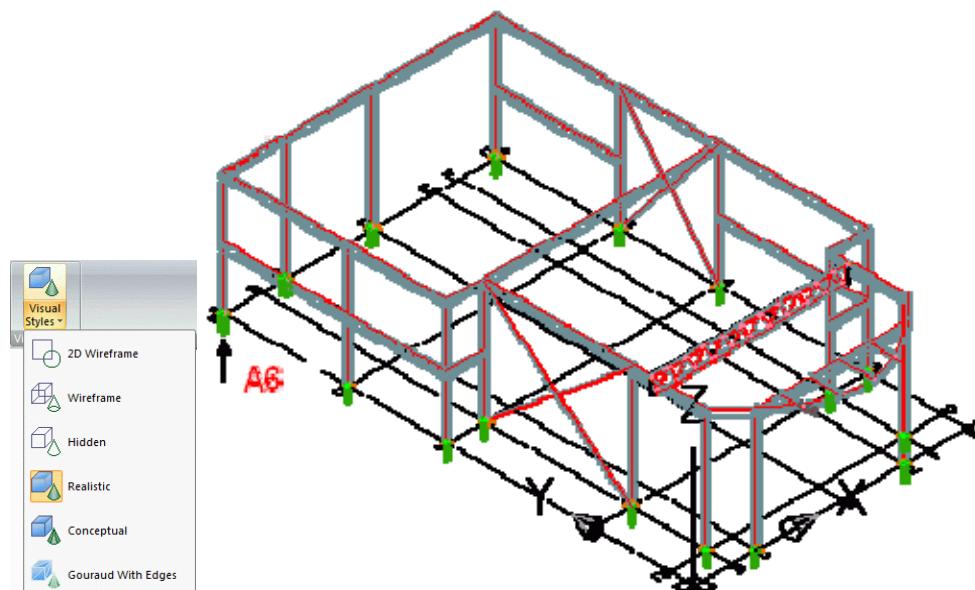


Figure 126: The **A6** column

In the properties dialog box, select the **Base plate** category and define the following settings:

- On the **Base plate layout** tab:
 - Define the plate thickness: **20**.
 - From the **Layout** drop-down list, select **projections** to calculate the base plate length according to the set projections.
 - From the **Column shortening** drop-down list, select **plate thickness** to shorten the column by the plate thickness.



Figure 127: Base plate thickness

- On the **Base plate dimensions** tab:
 - Modify the Projection 1 value to **80**.

Note: As the base plate is symmetrically placed, **All projections equal** option is selected.

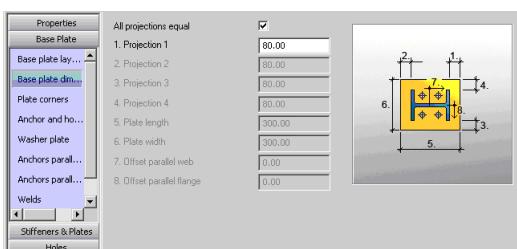


Figure 128: Properties dialog box – Base plate dimension tab

- On the **Anchors parallel web** tab:
 - Define the distance between bolt columns: **200**.

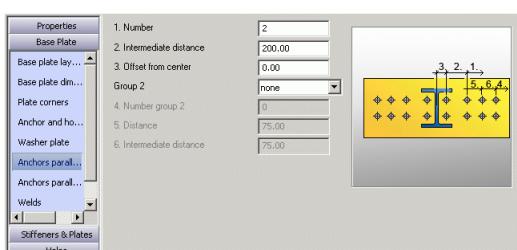


Figure 129: Properties dialog box – Anchors parallel web tab

- On the **Anchors parallel flange** tab:
 - Define the distance between bolt lines: **150**.

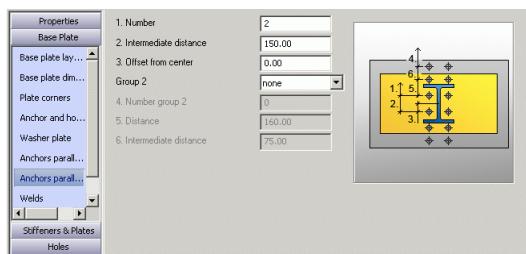


Figure 130: Properties dialog box – **Anchors parallel flange** tab

For a more realistic presentation of the model, use the **Realistic** visual style.

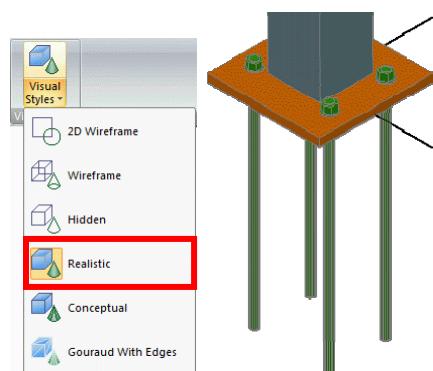
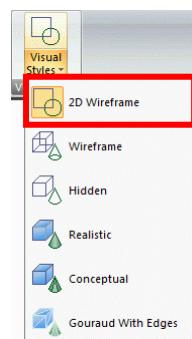


Figure 131: The base plate modified for the square section column

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 7: Creating independent joints of the same type

This step describes how to connect a column to a beam using single side end plate connections. Create several single side end plate joints in the model starting from the **A4** column.

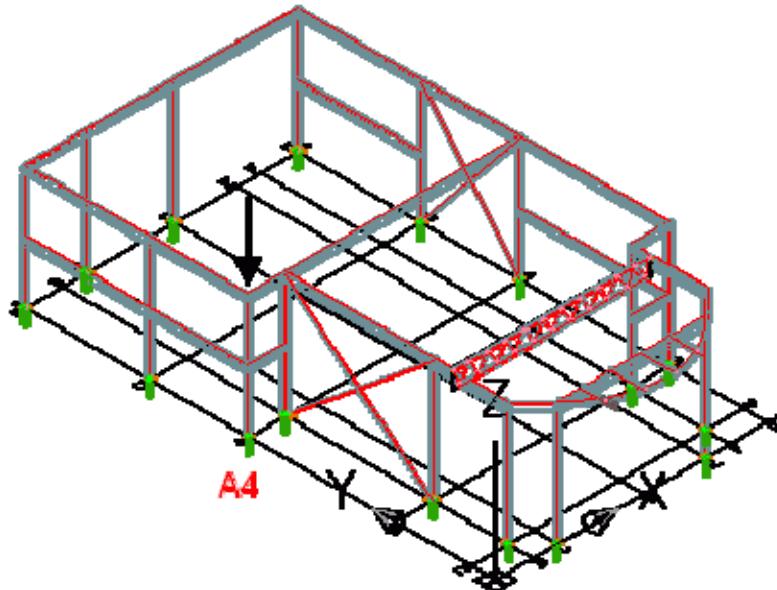


Figure 132: The **A4** column

Creating single side end plates

1. Open the Connection vault: On the **Home** tab, **Extended modeling panel**, click .
 2. From the **Plate joints** category, click .
- Plate joints
 - Single side end plate
 - Moment end plate
 - Double side end plate with safety bolt
 - Platform plate
 - Platform T
 - End plate of flange, reference beam
 - Two end plates, reference beam
 - Shear plate
 - Shear splice plate
 - Through plate
2. Select the **A4** column and press **Enter**.
 3. Select the upper **A4-B4** beam and press **Enter**.

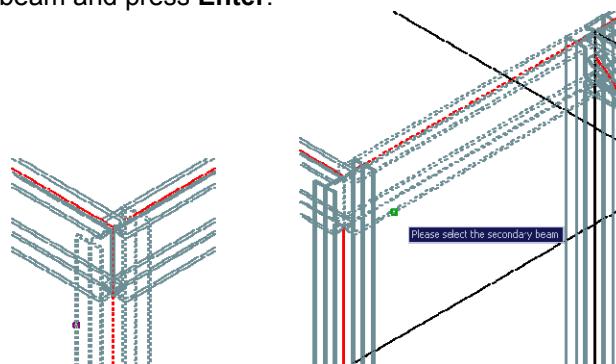


Figure 133: Selecting the elements to connect

4. Click **OK** in the “Attention” dialog box.

The joint is created and the beam is connected by a plate to the flange of the column.

The properties dialog box appears.

Modifying single side end plate properties

In the properties dialog box, select the **Plate & bolts** category and define the following settings:

- On the **Horizontal bolts** tab modify the distance between the bolt columns:
 - Unselect the **Bolts on gauge lines** option.
 - In the **Center distance** field enter **75**.

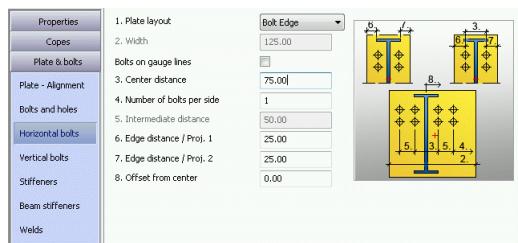


Figure 134: Properties dialog box – **Horizontal - bolts** tab

- On the **Vertical bolts** tab modify the plate height according to the number of bolts and the distance between them:
 - From the **1. Layout** drop-down list select **Plate 1** as reference for the bolt distances.
 - Define the **Plate height** by **Bolt distances from the top edge** of the plate.
 - Define the plate layout distance: **0**.
 - Define the number of bolt lines in the first group: **3**.
 - Define the first bolt distance to the plate edge: **60**.
 - Define the distance between the bolt lines within the group: **80**.
 - Define the end distance: **60**.

The plate height is automatically calculated (**3. Plate height** field).

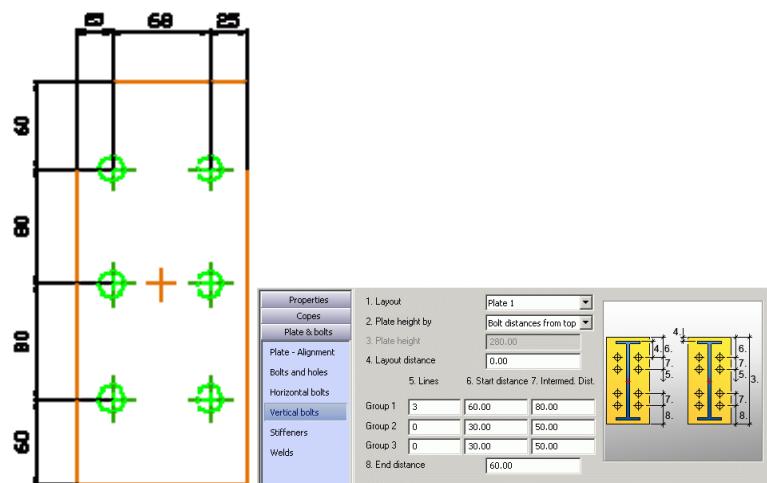
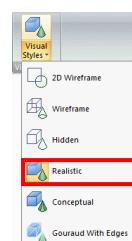


Figure 135: Properties dialog box – **Vertical bolts distances**

For a more realistic presentation of the model, use the **Realistic** visual style.



Note: For a clear view, display the bolts in the solid representation:

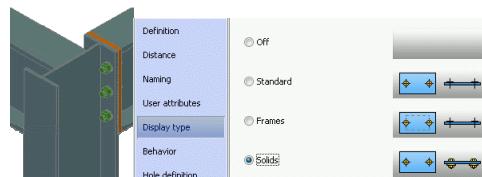
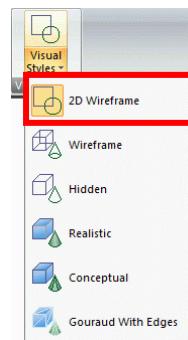


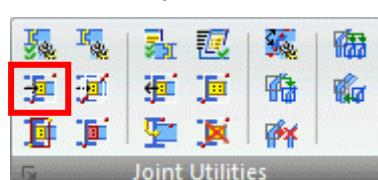
Figure 136: Single side end plate in shade mode, with bolts in solid view

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Creating by template single side end plates

1. On the Extended modeling tab, **Joint Utilities** panel, click .



2. Select any part of the template joint (e.g. the plate of the upper **A4-B4** joint) and press **Enter**.
3. Select the column to which the beam is connected through a single side end plate: the **A4** column.
4. Select the beam to connect to the column: the **A4-A5** upper beam.

For a more realistic presentation of the model, use the **Realistic** visual style.



Figure 137: Single side end plate connections

Note: All the joints remain independent of each other.

Using the same process, create single side end plates between:

- The **A4** column and:
 - The **A4-A5** middle beam
 - The **A4-B4** middle beam
- The **B4** column and:
 - The **B4- A4** middle beam
 - The **B4- G4** beam
 - The **B4- A4** upper beam
 - The **B4 - B3** upper beam
- The **B2** column and the **B2-B3** beam

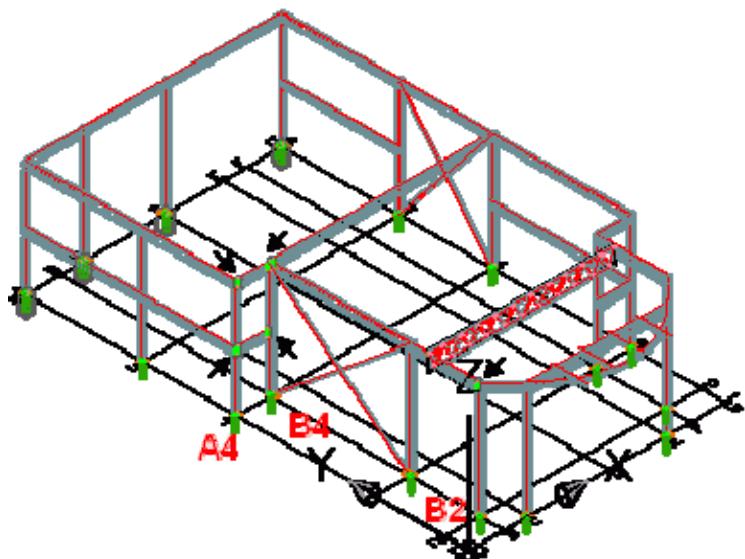


Figure 138: Joint locations in the model

Use the buttons on the **Viewpoints** panel to change the view angle.

Using the same process, create the single side end plates between:

- The **F2** column and:
 - The **F2-F3** beam
- The **F3** column and:
 - The **F3-F2** upper beam
 - The **F3-G3** upper beam
 - The **F3-G3** middle beam
- The **G3** column and:
 - The **G3-F3** upper beam
 - The **G3-G4** upper beam
 - The **G3-F3** middle beam
 - The **G3-G4** middle beam
- The **G4** column and:
 - The **G4-B4** upper beam
 - The **G4-G3** middle beam
- The **G5** column and:
 - The **G5-G6** middle beam

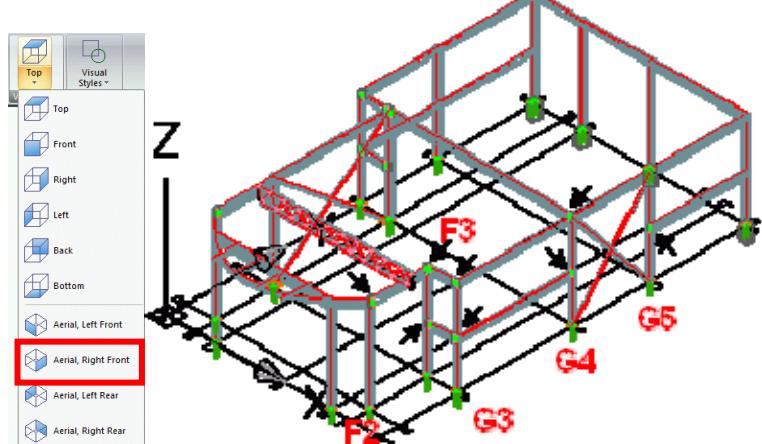


Figure 139: Joint locations in the model

Lesson 7: Creating intelligent connections

In this lesson, you will create various connections:

- Double side end plates
- Gable wall end plates
- Front plate splices
- Clip angles
- Clip angles skewed
- Copes
- Purlin connection elements
- Flat bracings with tension bolt and base plate
- Middle gusset plate
- Tube connection with sandwich plates
- Tube connection middle with sandwich plate

Step 1: Creating a moment end plate

Next, connect the **C1-E1** curved beam to the **C1** and **E1** columns using moment end plates connections.

Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.

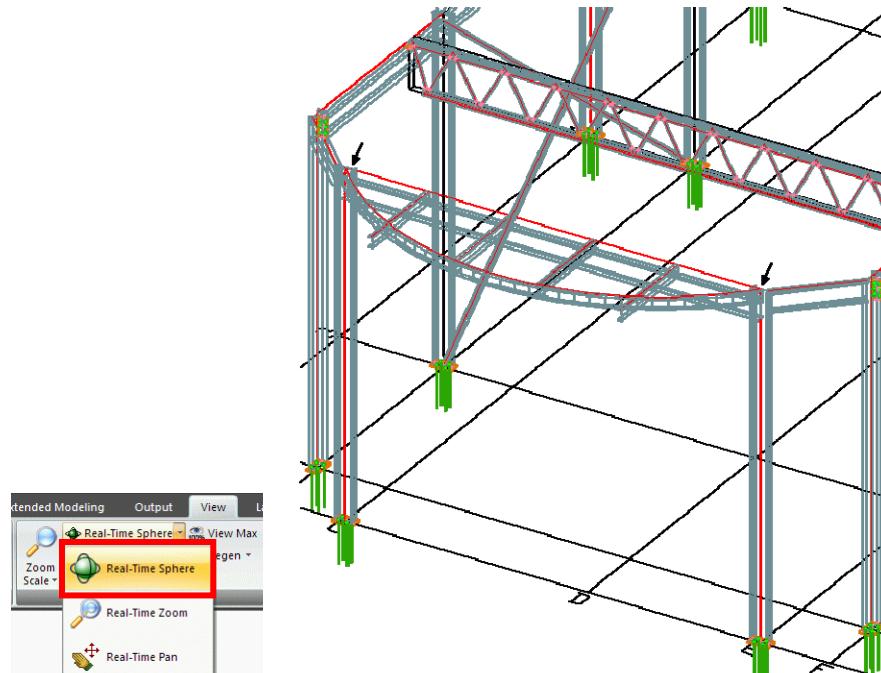
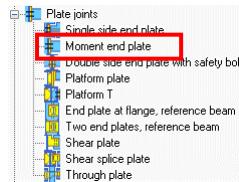


Figure 140: Moment end plate – joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click 
2. From the **Plate joints** category, click 



3. Select the **C1** column and press **Enter**.
4. Select the **E1-C1** curved beam and press **Enter**.
5. Click **OK** in the “Attention” dialog box.

The joint is created and the beam is connected by a plate to the flange of the column.

The properties dialog box appears.

Modifying moment end plate properties

In the properties dialog box, select the **Plate & bolts** category and define the following settings:

- On the **Plate layout** tab define the plate height by bolts:
 - Define the plate thickness: **15**.
 - Select the plate height layout **by bolts**.
 - Enter **50** for the top and bottom edge distance.

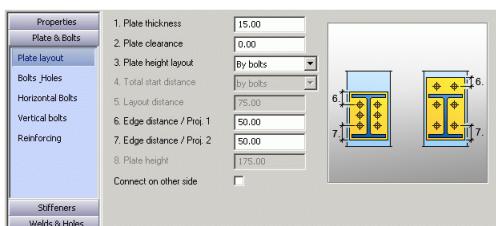


Figure 141: Properties dialog box – **Plate layout** tab

- On the **Horizontal bolts** tab:
 - Define the center distance: **100**.
 - Define the edge distance / Proj. 1: **30**.
 - Define the edge distance / Proj. 2: **30**.
 - Define the offset from center: **20**.

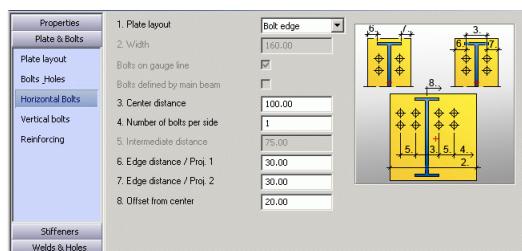


Figure 142: Properties dialog box – **Horizontal bolts** tab

- On the **Vertical bolts** tab define the number of bolt lines and the distances between bolts:
 - Define the layout distance: **0**.
 - Define the number of bolt lines in the first group: **2**.
 - Define the distance to the plate edge for the first bolt: **50**.
 - Define the distance between bolt lines within the group: **80**.

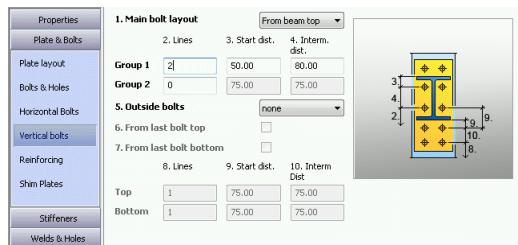


Figure 143: Properties dialog box – **Vertical bolts** tab

Note: For a clear view, display the bolts in the solid representation.

For a more realistic presentation of the model, use the **Realistic** visual style.

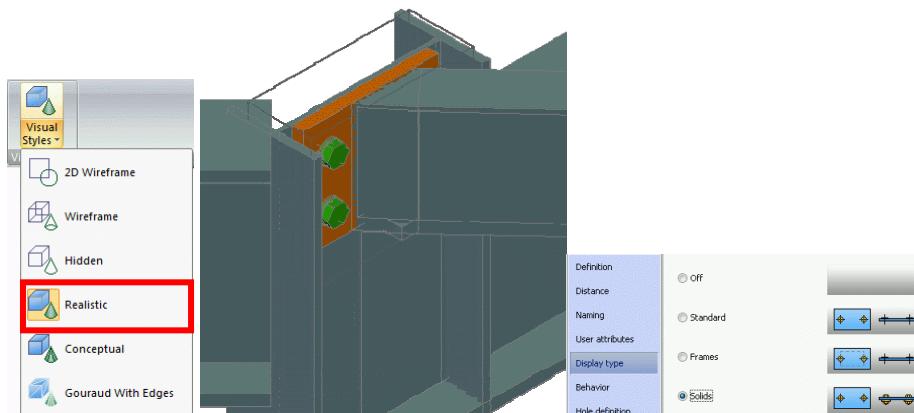
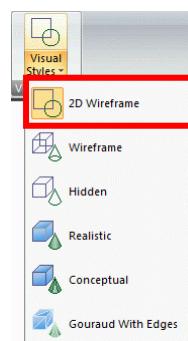


Figure 144: Displaying the connection in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Using the same process, create a moment end plate between the **E1** column and the **E1-C1** curved beam.

Note: In the moment end plate properties dialog box, on the **Horizontal bolts** tab, change the offset from center to **-20**.

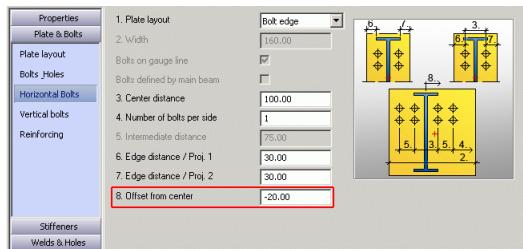


Figure 145: Properties dialog box – **Horizontal bolts** tab

Step 2: Creating double side end plates with safety bolts

Connect the **B3** column with the two rafters by a double side end plate with safety bolts.

Use the buttons on the **Viewpoints** panel to change the view angle.

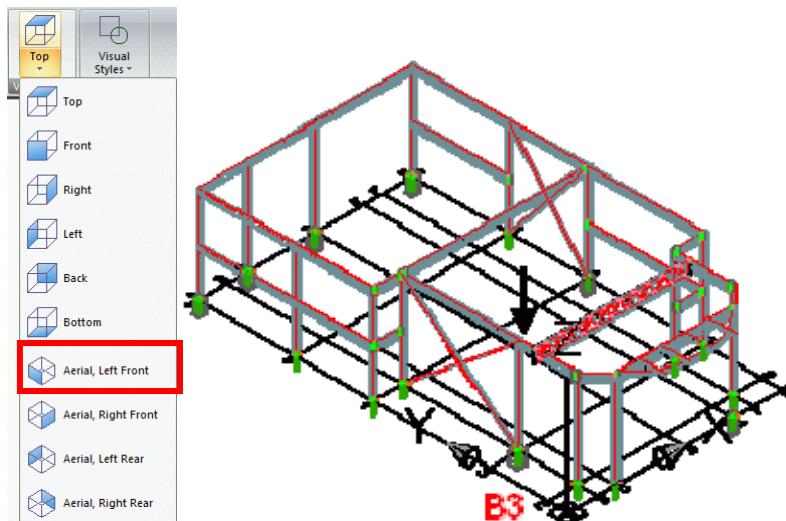
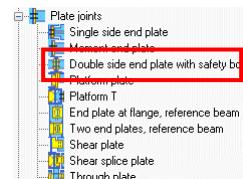


Figure 146: Joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click
2. From the **Plate joints** category, click



3. Select the **B3** column and press **Enter**.
4. Select the **B3-B4** beam and press **Enter**.
5. Select the **B3-B2** beam and press **Enter**.

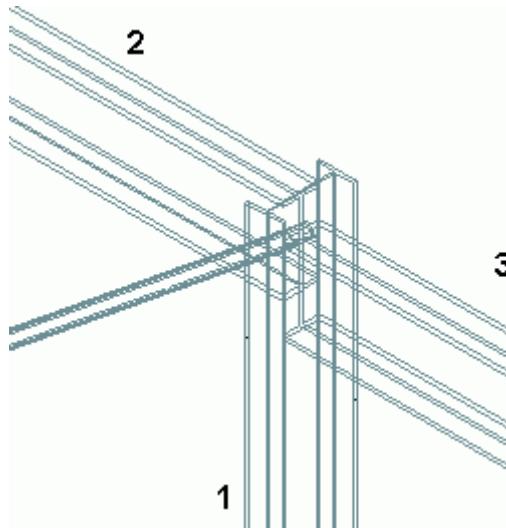


Figure 147: Creating a double side end plate connection

6. Click **OK** in the “Attention” dialog box.

The joint is created and the beams are connected. The properties dialog box appears.

Modifying double side end plate properties

In the properties dialog box, select the **Plate & bolts** category and define the following settings:

- On the **Plate-Alignment** tab, enter **15** for the thickness of each plate.

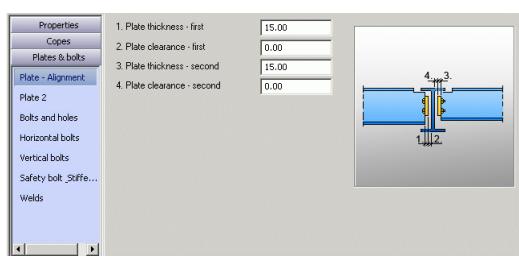


Figure 148: Properties dialog box – **Plate - Alignment** tab

- On the **Vertical bolts** tab define the plate height by bolt distances, from the top edge of the first plate:
 - From the **Layout** drop-down list, select **Plate 1**.
 - From the **Plate height by** drop-down list, select **Bolt Distances from top**.
 - In the **Layout distance** field, enter **0** to align the plate to the top edge.
- Then define the bolt distances:
- Define the number of bolt lines in each group: **1**, for the first group, **2** for the second.
 - Define the distance to the plate edge for the first group: **60**.
 - Define the distance of the upper bolt line of the second group to the last bolt line of the previous bolt group: **110**.
 - Define the distance between bolt lines within the groups: **160** for the first group, **160** for the second.
 - Define the last bolt distance to the bottom edge of the plate: **60**.

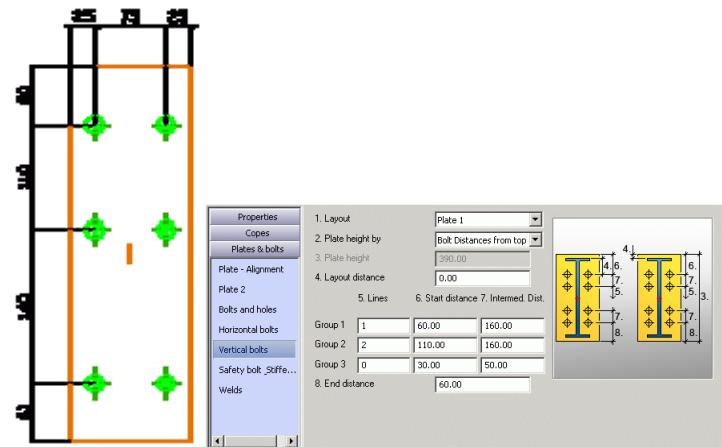


Figure 149: Vertical distances between bolts on the **Vertical bolts** tab

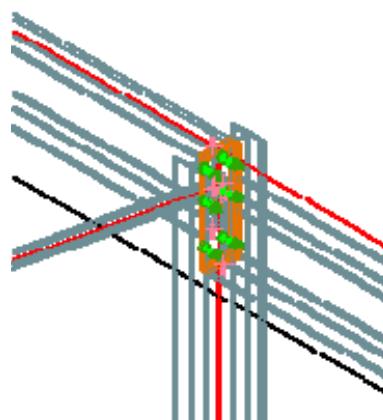


Figure 150: Double side end plate connection at the **B3** column

Using a similar process, create another double side end plate connection between the **A5** column and the **A5-A4** upper beams. The vertical distances between bolts must be modified to adjust the plate height.

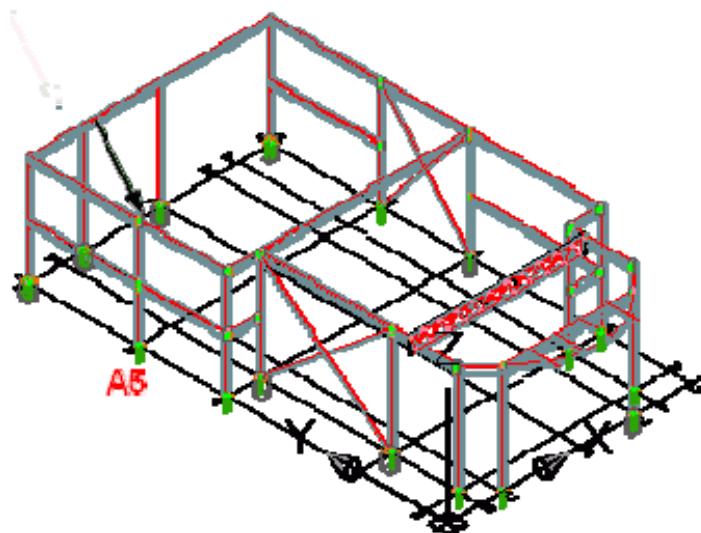


Figure 151: Joint location in the model

In the properties dialog box, define the following settings:

- On the **Vertical bolts** tab:
 - Define the number of bolt lines in the first group: **3**.
 - Define the distance to the plate edge for the first group: **60**.
 - Define the distance between bolt lines within the group: **80**.
 - Define the end distance: **60**.

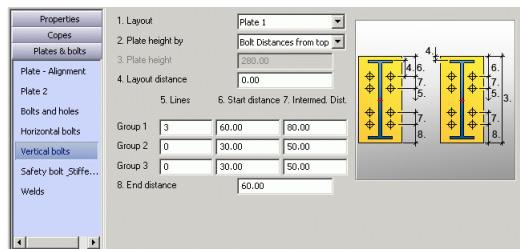


Figure 152: Properties dialog box – **Vertical bolts** tab

Creating by template double side end plates

Next, create by template other double side end plate connections in the locations indicated in Figure 153.

- The **A5** column and:
 - A4-A5** and **A5-A6** middle beams
- The **G4** column and:
 - G4-G3** and **G4-G5** upper beams
- The **G5** column and:
 - G5-G4** and **G5-G6** upper beams

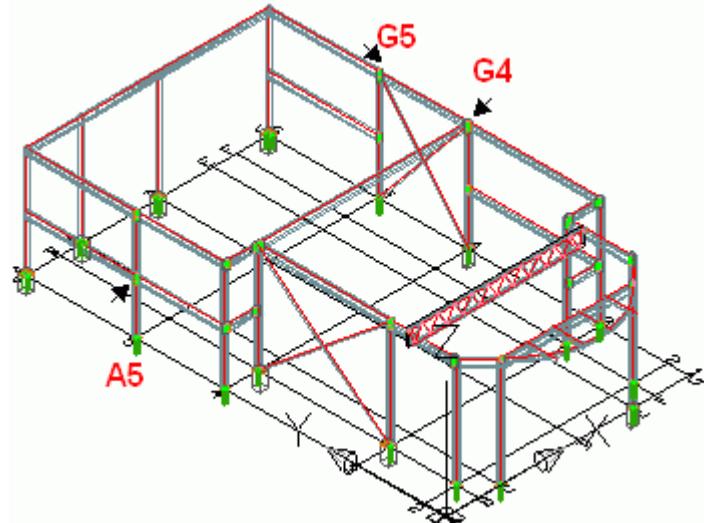


Figure 153: Joint location in the model

Step 3: Creating gable wall end plates

In this step, create gable wall end plates at the **C6** and **D6** columns.

Before starting

For a clear view, use the **GableWall** view defined in Lesson 5 – “Defining model views”.

From the Project Explorer enable the **GableWall** view.

*Use the buttons on the **Viewpoints** panel to change the view angle.*

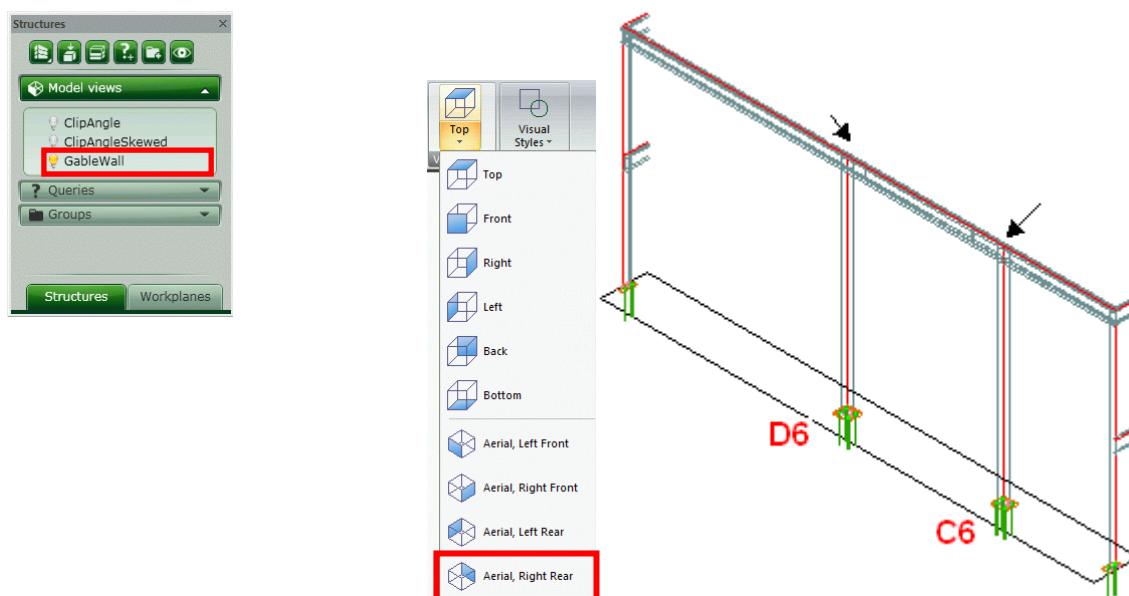
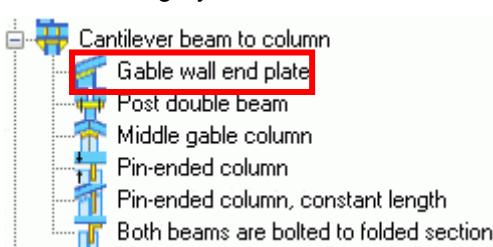


Figure 154: “Gable wall” view – Joint location

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .
 2. From the **Cantilever beam to column** category, click .
- 
 - Cantilever beam to column
 - Gable wall end plate
 - Post double beam
 - Middle gable column
 - Pin-ended column
 - Pin-ended column, constant length
 - Both beams are bolted to folded section
3. Select the **G6-D6** rafter and press **Enter**.
 4. Select the **D6** column and press **Enter**.

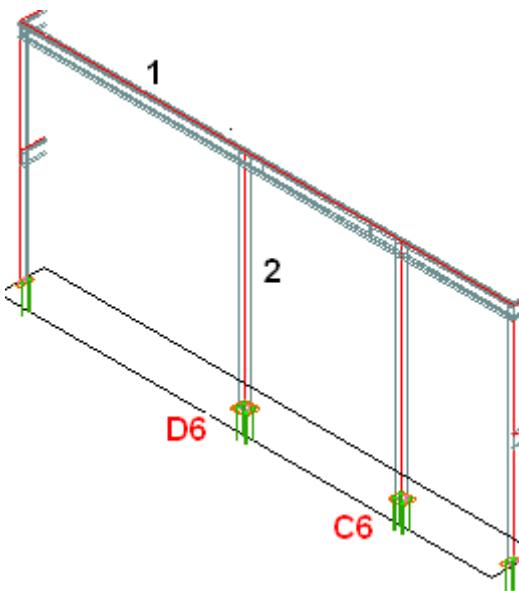


Figure 155: Selecting the elements to connect

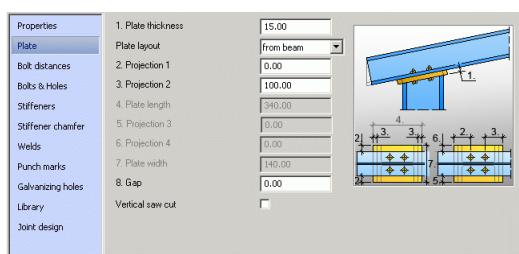
5. Click **OK** in the “Attention” dialog box.

The gable wall end plate connection is created and the properties dialog box appears.

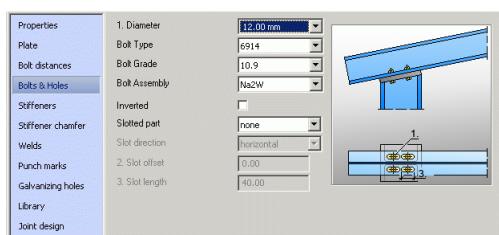
Modifying gable wall end plates properties

In the properties dialog box, define the following settings:

- On the **Plate** tab:
 - Define the cover plate thickness: **15**.
 - Define the Projection 2 value: **100**.

Figure 156: Properties dialog box – **Plate** tab

- On the **Bolts & Holes** tab:
 - Select the diameter of the bolts: **12 mm**.

Figure 157: Properties dialog box – **Bolts & Holes** tab

- On the **Bolt distances** tab:

Note: *The On gauge line option must be unselected!*

- Define the intermediate distance in the X direction: **220**.
- Define the intermediate distance in the Y direction: **80**.

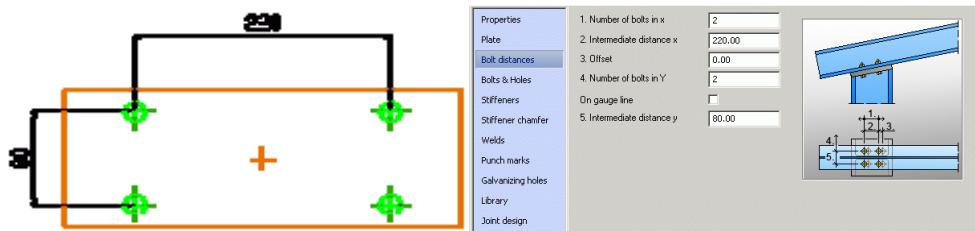


Figure 158: The distances between bolts in the **Bolt distances** tab

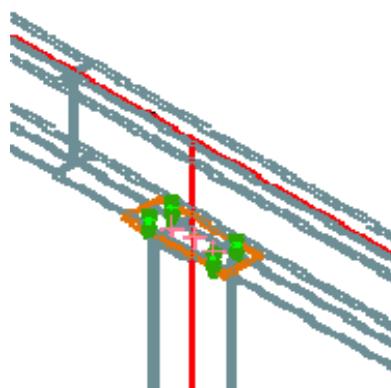
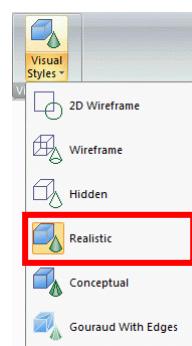


Figure 159: The gable wall end plate connection at the **D6** column

Using the same process, create another gable wall end plate connection for the **C6** column and the corresponding rafter.

*For a more realistic presentation of the model, use the **Realistic** visual style.*



Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.



Note: For a clear view, display the bolts in the solid representation.

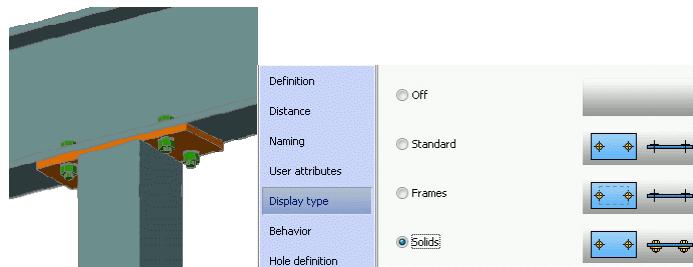
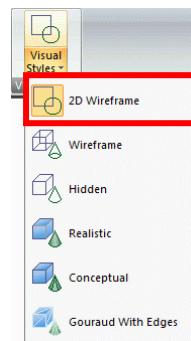


Figure 160: Displaying the joint in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 4: Creating front plate splices

In this step, create front plate splice connections at the two splitting points of the **A6-G6** rafter.

Note: Use the **GableWall** view from Lesson 5 – “Defining model views” - for a clearer display of the working area.

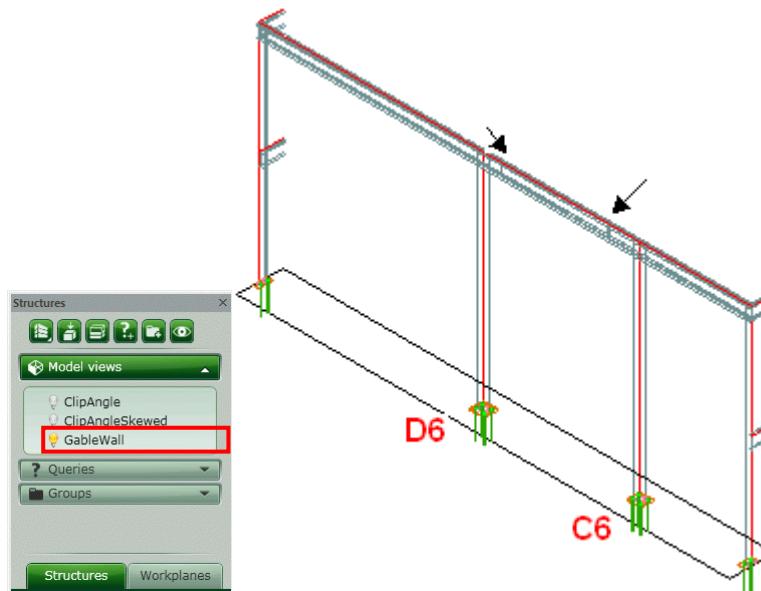
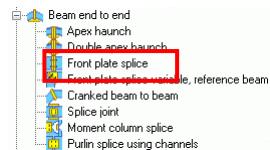


Figure 161: Joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .

2. From the **Beam end to end** category, click .



3. Select the **D6-G6** rafter and press **Enter**.

4. Select the **D6-C6** rafter and press **Enter**.

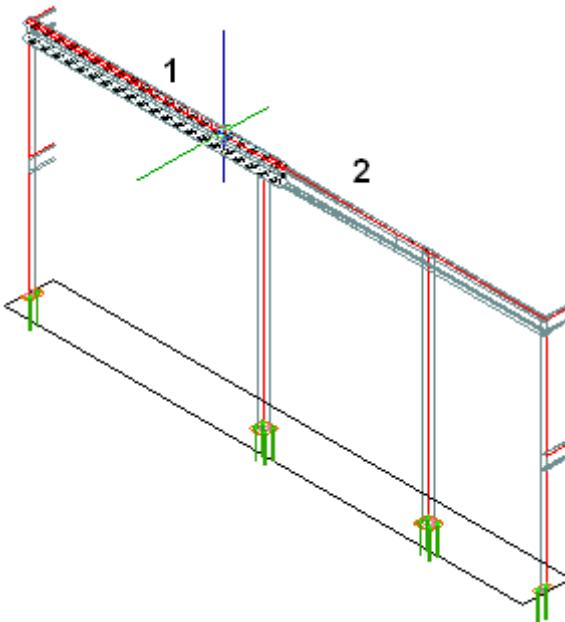


Figure 162: Selecting the elements to connect

5. Click **OK** in the “Attention” dialog box.

The joint is created and the rafters are connected. The properties dialog box appears.

Modifying front plate splice properties

Next, modify the plate size, the thickness and the bolt distances.

In the properties dialog box, define the following settings:

- On the **Plate 1** tab, define the plate size:
 - Thickness: **15**
 - Width: **150**
 - Height: **310**



Make sure the Same as other option is enabled in the Plate 2 tab.

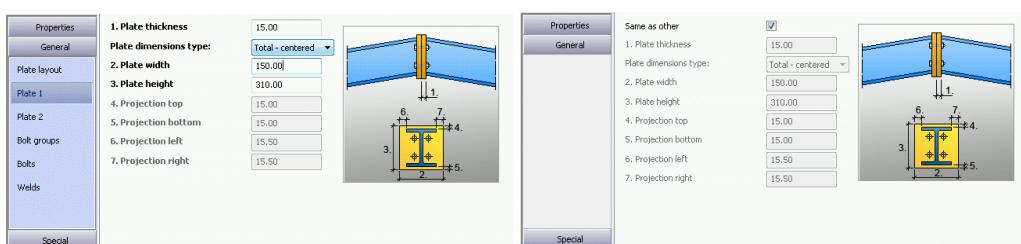
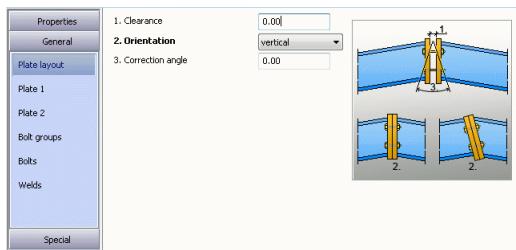


Figure 163: Properties dialog box – **Plate 1** and **Plate 2** tabs

- On the **Plate layout** tab:
 - Define the clearance between the plates: **0**.

Figure 164: Properties dialog box – **Plate layout** tab

- On the **Bolt groups** tab:
 - Define the number of bolts per side: **1**.
 - Define the gauge distance: **90**.
 - Define the number of bolt lines in the first group: **3**.
 - Define the distance to the plate edge for the first group: **60**.
 - Define the distance between bolt lines within the first group: **80**.

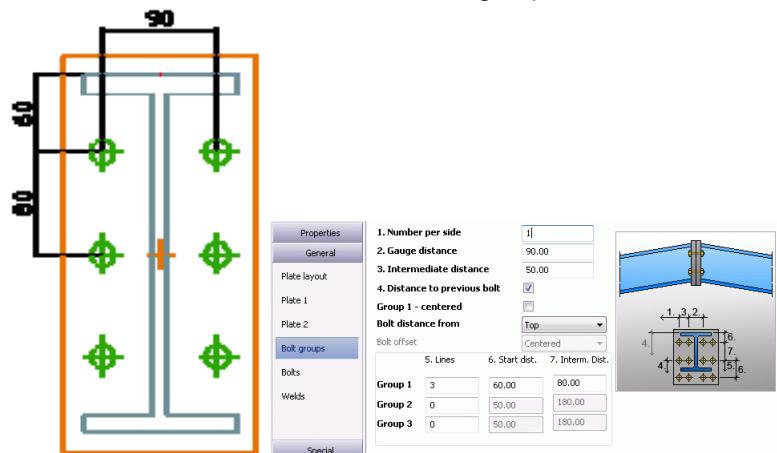
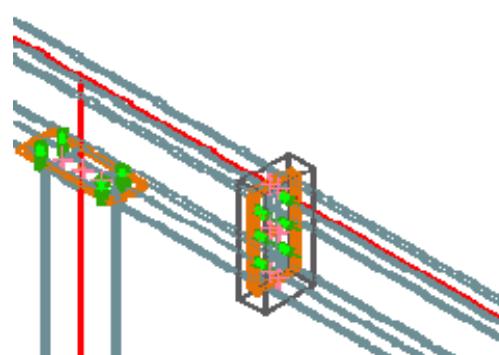
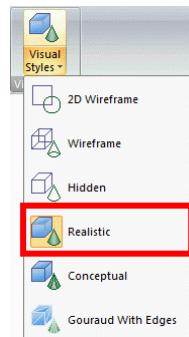
Figure 165: Bolts distances on the **Bolt groups** tab

Figure 166: The front plate splice connection

Next, create by template another front plate splice connection between the **C6-D6** rafter and the **C6-A6** rafter.

For a more realistic presentation of the model, use the **Realistic** visual style.



Note: For a clear view, display the bolts in the solid representation.

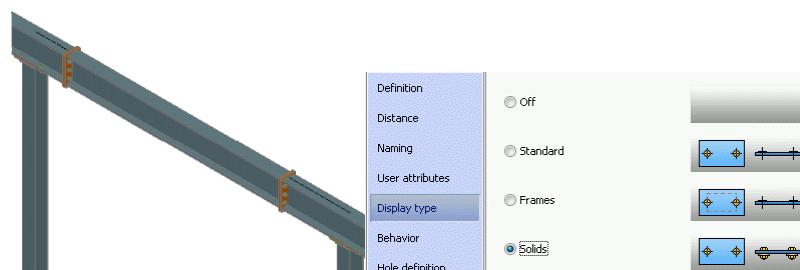
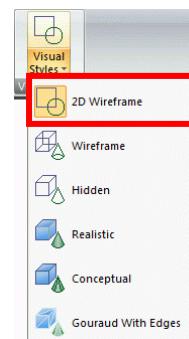


Figure 167: Displaying the connection in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 5: Creating clip angle joints

In this step, connect the **E1 – C1** straight beam to the **E1** and **C1** columns using clip angle connections.

Before starting

Use the **ClipAngle** view defined in the “Lesson 5 – “Defining model views” - for a clearer display of the working area.



Figure 168: Selecting the **ClipAngle** view

Use the buttons on the **Viewpoints** panel to change the view angle.

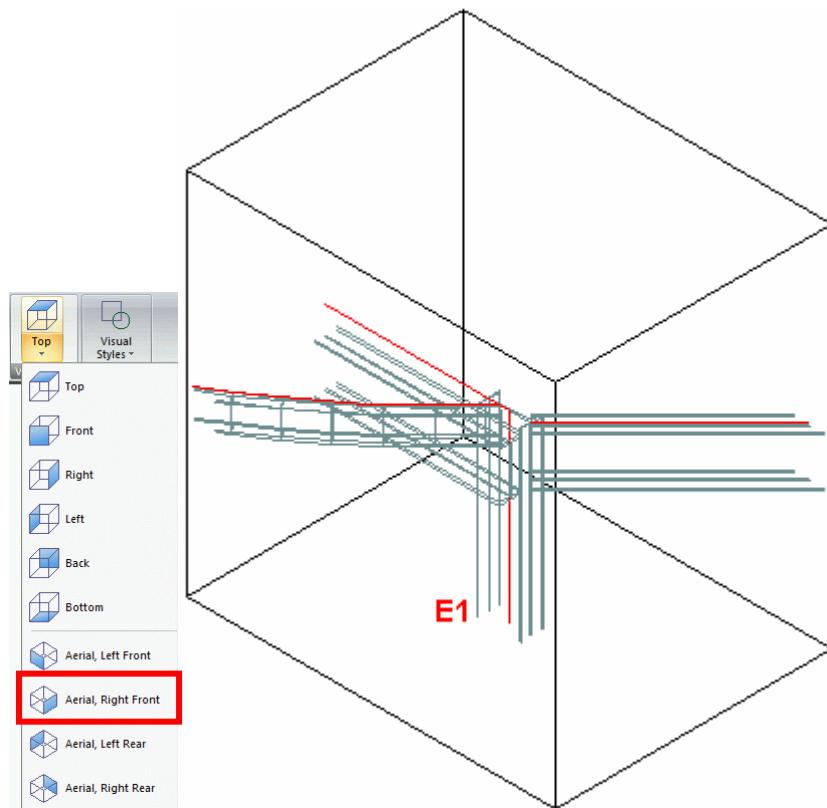


Figure 169: Joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .
2. From the **Clip angle joints** category, click .
3. Select the **E1** column and press **Enter**.
4. Select the **E1-C1** straight beam and press **Enter**.
5. Click **OK** in the “Attention” dialog box.

The joint is created and the beams are connected by two clip angles.

Modifying clip angle joint properties

Next, modify the connection properties so that the two non-identical clip angles are connected with three bolts to the column and the beam.

In the properties dialog box, define the following settings:

- On the **Angle cleat** tab:
 - Select the angle section class: **Angle not identical**.
 - Select the angle section size: **L130X65X12**.
 - From the **Long leg side** drop-down list, select **secondary** to place the long leg on the beam.
 - Define the cut back value: **10**.

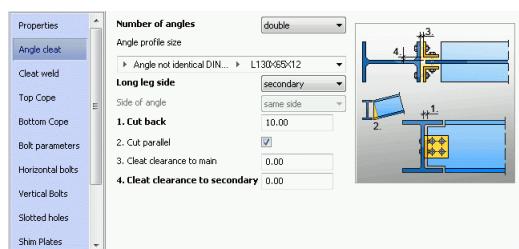


Figure 170: Properties dialog box – Angle cleat tab

- On the **Bolt parameters** tab:
 - Select the diameter of the bolts: **12 mm**.

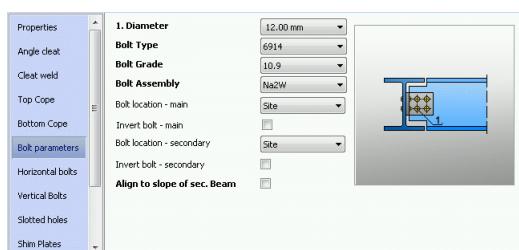


Figure 171: Properties dialog box – Bolt parameters tab

- On the **Horizontal bolts** tab:
 - In both **Number per cleat** fields enter **1** to connect the clip angle to each connection member by one column of bolts.
 - Select the back mark for the main beam: **80**.
 - Select the back mark for the second beam: **85**.

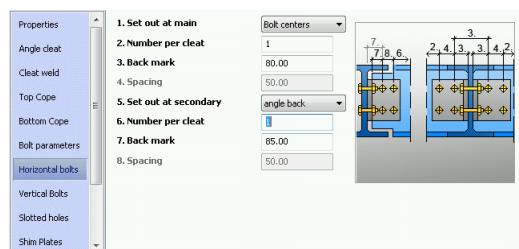


Figure 172: Properties dialog box – Horizontal bolts tab

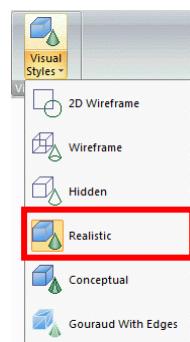
- On the **Vertical bolts** tab:
 - Define the set out distance: **45**.
 - Define the number of bolt lines within the first group: **3**.
 - Define the distance to the plate edge for the first group: **30**.
 - Define the distance of the upper bolt line of the group to the last bolt line of the previous bolt group: **70**.
 - Define the end distance: **30**.

The angle cleat height is adjusted according to the bolt distances.



Figure 173: Bolts vertical distances on the **Vertical bolts** tab

*For a more realistic presentation of the model, use the **Realistic** visual style.*



Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.



Note: For a clear view, display the bolts in the solid representation.

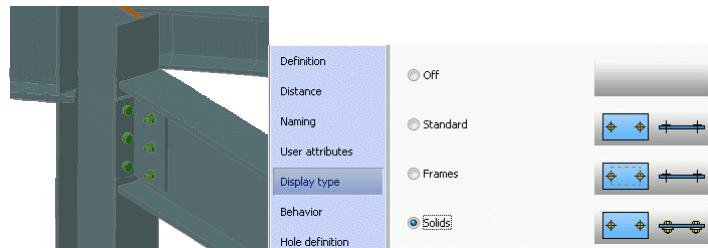
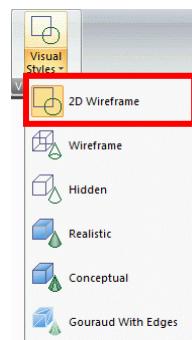


Figure 174: Displaying the joint in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



On the Project Explorer toolbar, click to redisplay all the elements of the model.



Using the same process (or by template), create another connection between the **C1** and **E1 – C1** straight beam.

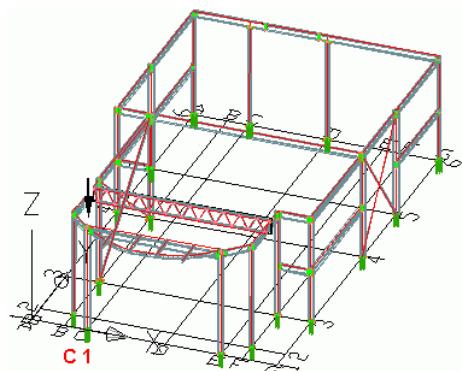


Figure 175: Joints locations in the model

Use the **Real-Time Sphere** to rotate the drawing in 3D and get a suitable position.



Using the same process, create another clip angle connection between the **G6** column and the **G6 – G5** upper beam. Next, change the connection settings so that the clip angles are welded to the column and bolted to the beam.

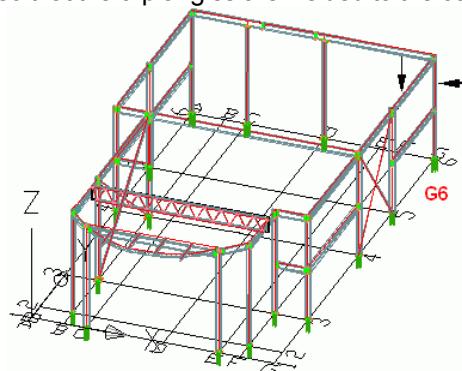


Figure 176: Clip angle locations in the model

In the properties dialog box, define the following settings:

- On the **Angle cleat** tab:
 - From the **Number of angles** drop-down list, select **double**.
 - Select the angle size profile: **L75X55X5**.
 - Define the cut back value: **10**.

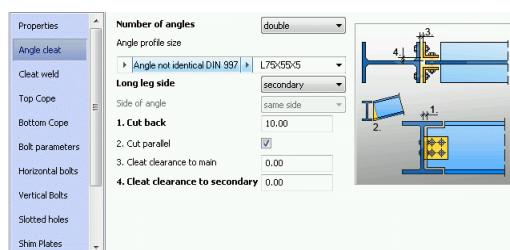


Figure 177: Properties dialog box – Angle cleat tab

- On the **Cleat weld** tab:
 - From the **Welded at** drop-down list, select **main**.



Figure 178: Properties dialog box – Cleat weld tab

- On the **Bolt parameters** tab:
 - Select the bolt diameter: **16**.

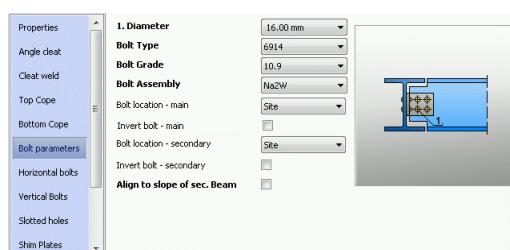


Figure 179: Properties dialog box – Bolt parameters tab

- On the **Horizontal bolts** tab, define the number of bolt lines connecting the clip angle to the beam and their position:
 - Define the back mark at the secondary beam: **45**.
 - In the **6. Number per cleat** field define the number of bolt lines on the secondary member: **1**.

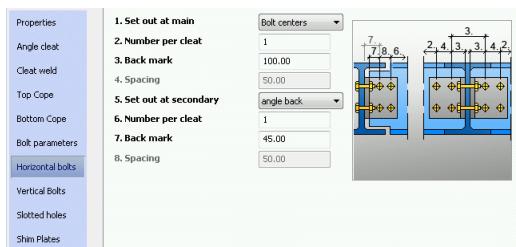


Figure 180: Properties dialog box – **Horizontal bolts** tab

- On the **Vertical bolts** tab:
 - Define the set out distance: **75**, from the clip angle.
 - Define the number of bolt lines in the first group: **3**.
 - Define the distance to the first bolt of the group: **25**.
 - Define the distance between bolt lines within the group: **50**.
 - Define the edge distance: **25**.

The length of the clip angle is automatically calculated according to the bolt distances.

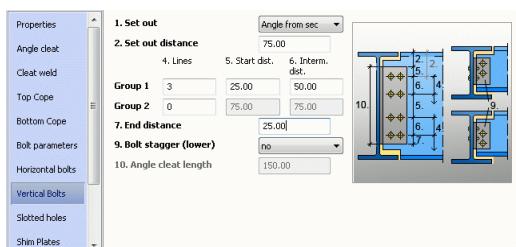


Figure 181: Properties dialog box – **Vertical bolts** tab

For a more realistic presentation of the model, use the **Realistic** visual style.

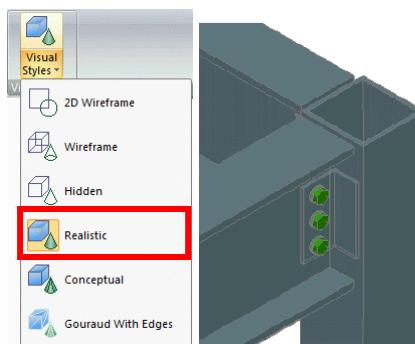
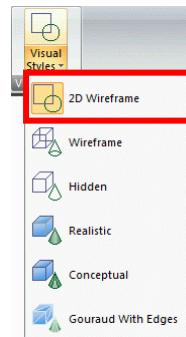


Figure 182: The clip angle joint in shade mode

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Next, create by template all other clip angle connections between:

- The **G6** column and:
 - The **G6-F6** upper beam
 - The **G6-G5** middle beam
- The **A6** column and:
 - The **A6-A5** upper beam
 - The **A6-B6** upper beam
 - The **A6-A5** middle beam

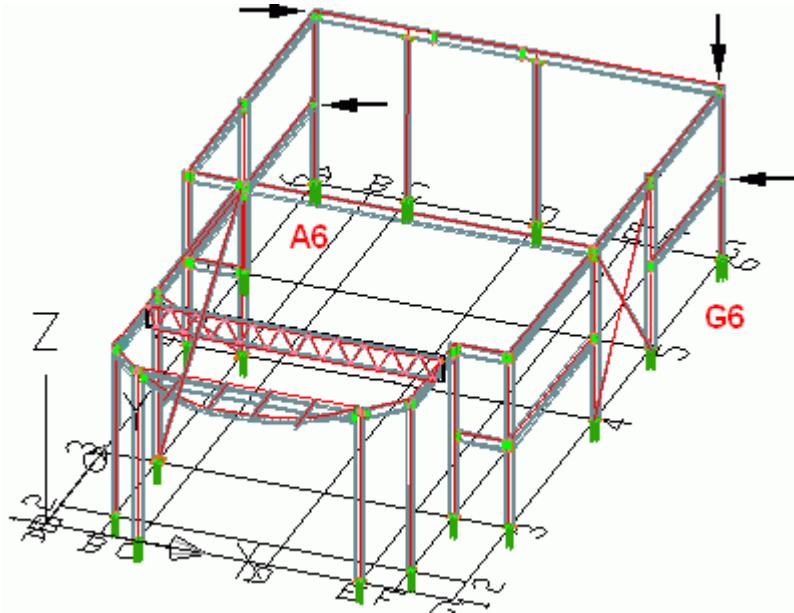


Figure 183: Clip angle locations in the model

Step 6: Creating a clip angle skewed joint

In this step, create a connection with a folded clip angle. The attached beam is skewed to the main one.

Before starting

Use the **ClipAngleSkewed** view from Lesson 5 – “Defining model views” - for a clearer display of the working area.



Figure 184: Selecting the “ClipAngleSkewed” view

Use the buttons on the **Viewpoints** panel to change the view angle.

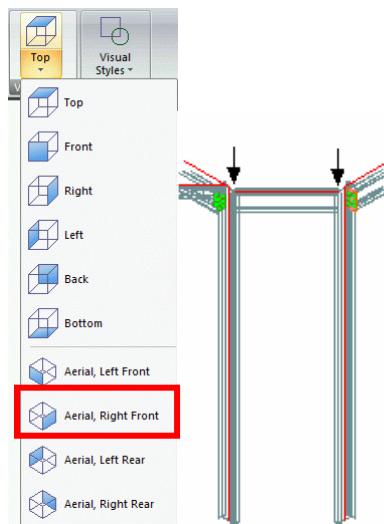


Figure 185: Clip angle skewed location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .
2. From the **Clip angle joints** category, click .
3. Select the **E1** column and press **Enter**.
4. Select the **E1-F2** beam and press **Enter**.
5. Click **OK** in the “Attention” dialog box.



The joint is created and the beams are connected. The properties dialog box appears.

Modifying clip angle skewed joint properties

Next, adjust the cleat position according to the context and create a connection welded to the column and bolted to the skewed rafter (one bolt columns).

In the properties dialog box, define the following settings:

- On the **Cleat weld** tab:
 - Select **Connect on other side** to create the clip angle on the correct side.
 - From the **Welded at** drop-down list, select **main**.

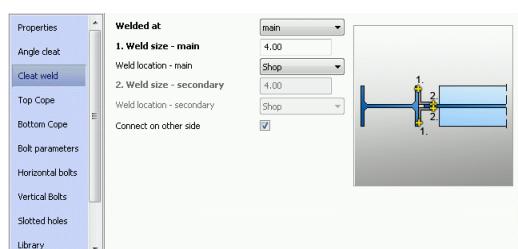


Figure 186: Properties dialog box – **Cleat weld** tab

- On the **Angle cleat** tab:

- Select **7. Cut parallel** option to cut the secondary beam parallel with the main beam section.
- Select the number of angles: **double**.
- Define the thickness of the angles: **10**.
- Define the clip leg length at column: **60**.
- Define the leg length at beam: **150**.
- Define the cut back distance: **10**.

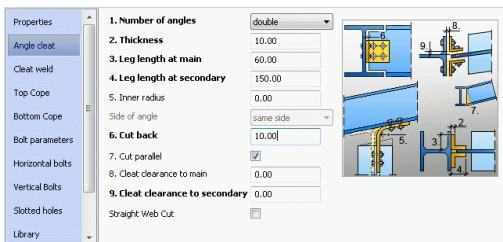


Figure 187: Properties dialog box – Angle cleat tab

- On the **Horizontal bolts** tab, define the number of bolt lines connecting the clip angle to the beam and their position:

- From the **Set out at main** drop-down list, select **total**.
- Define back mark at secondary member: **80**.
- In the **6. Number per cleat** field define the number of bolt columns on the secondary member: **1**.

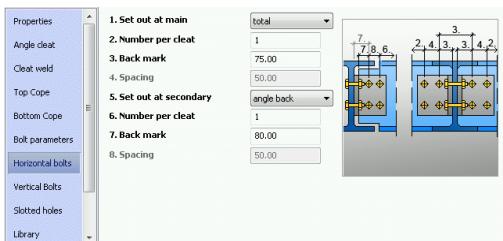


Figure 188: Properties dialog box – Horizontal bolts tab

- On the **Vertical bolts** tab, adjust the clip angle position relative to the secondary beam, then set the number of bolts and the distances between them:

- From the **Set out** drop-down list, select **Angle from sec.**
- Define the set out distance: **50**.
- Define the number of bolt lines in the first group: **3**.
- Define the distance to the plate edge from the first bolt of the group: **25**.
- Define the distance between bolt lines within the group: **70**.
- Define the end distance: **25**.

The length of the folded cleat is automatically calculated according to the bolt distances.

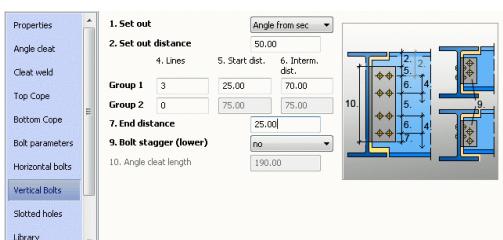
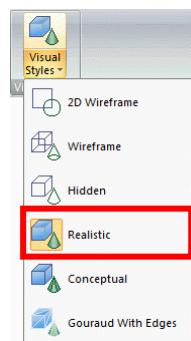


Figure 189: Properties dialog box – Vertical bolts tab

For a more realistic presentation of the model, use the **Realistic** visual style.



Use the **Real-Time Sphere** to rotate the drawing in 3D and get a suitable position.



Note: For a clear view, display the bolts in the solid representation.

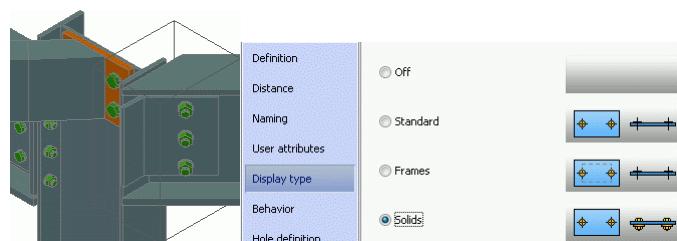
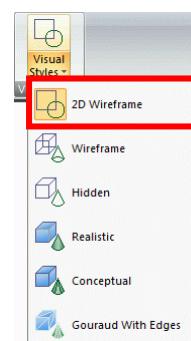


Figure 190: Displaying the connection in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



On the Project Explorer toolbar, click to redisplay all the elements of the model.



Creating by template clip angle skewed joints

Next, create by template all other clip angle skewed connections between:

- The **F2** column and the **F2 – F1** beam
- The **C1** column and the **C1-B2** beam
- The **B2** column and the **C1-B2** beam

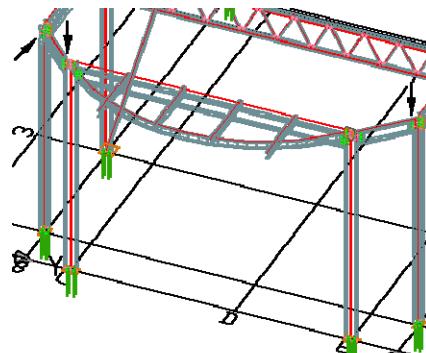
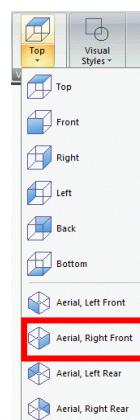


Figure 191: Clip angle skewed location in the model

Step 7: Creating copes

In this step, create copes on each purlin according to the **C1-E1** curved beam. The purlins should be welded to the beam.

*Use the buttons on the **Viewpoints** panel to change the view angle.*



Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.

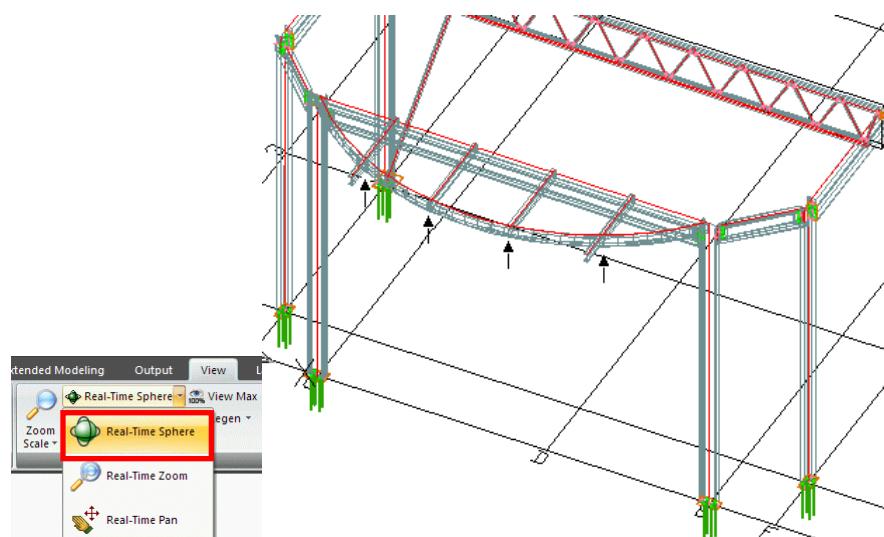


Figure 192: The cope locations in the model

1. On the **Beam features** tool palette, click  P.
2. Select the C1-E1 curved beam and press **Enter**.
3. Select a purlin (for example, the first purlin) and press **Enter**.
4. Click **OK** in the “Attention” dialog box.

The purlin is copied.

5. In the properties dialog box, on the **Welds** tab, select the **Create weld** option.

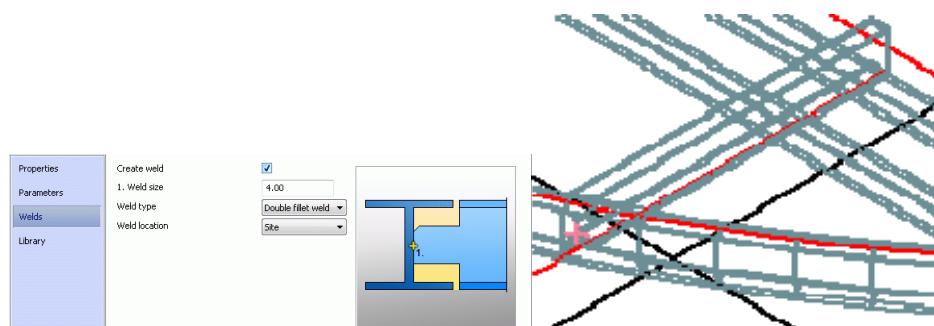


Figure 193: Creating a welded connection

Using a similar process, create a processing for each purlin.

*For a more realistic presentation of the model, use the **Realistic** visual style.*

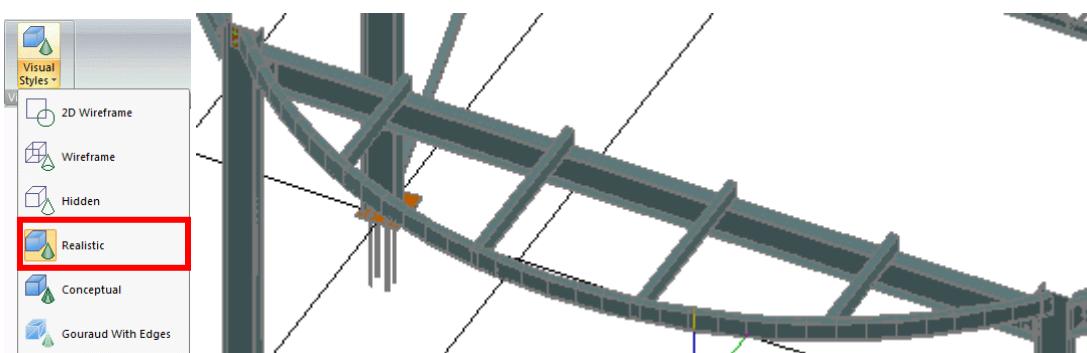
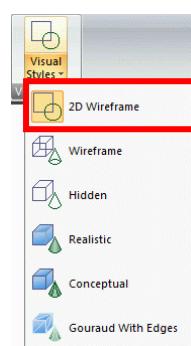


Figure 194: The model in shade mode

*To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.*



Step 8: Creating purlin connections

In this step, connect each purlin to the **C1-E1** straight beam.

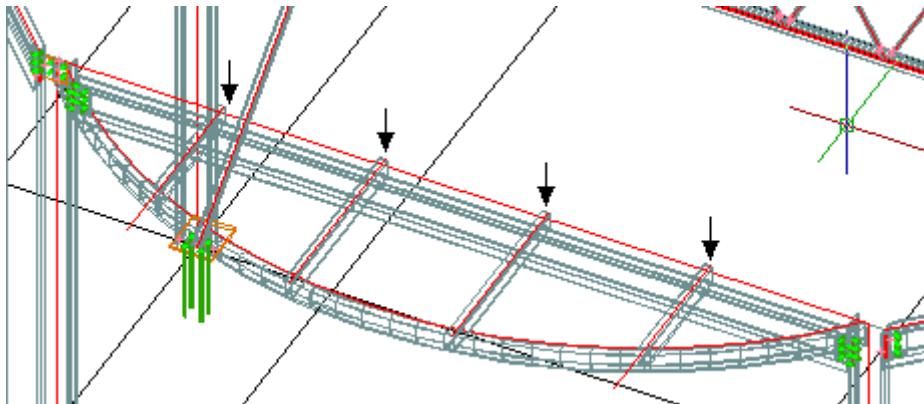
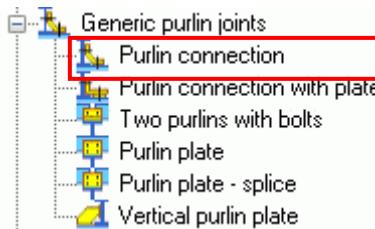


Figure 195: The joints locations in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling panel**, click .
2. From the **Generic purlin joints** category, click .



3. Select the **C1-E1** straight beam and press **Enter**.
4. Select a purlin (e.g., the first purlin) and press **Enter**.
5. Click **OK** in the “Attention” dialog box.

The purlin is connected and the properties dialog box appears.

Modifying purlin connection elements properties

In the properties dialog box, define the following settings:

- On the **Structural parts** tab, select the **Use folded beam as purlin shoe** option to use a folded beam as purlin shoe.

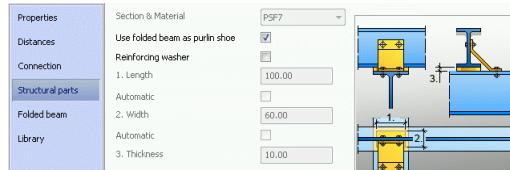


Figure 196: Properties dialog box – Structural parts tab

- On the **Connection** tab, select the diameter of the bolts: **12 mm**.



Figure 197: Properties dialog box – Connection tab

- On the **Distances** tab:

- From the **Purlin shoe position** drop-down list, select **other side** to place the connection element correctly.
- Define the distance from the section origin to the bolt connecting the element with the purlin: **70**.
- Define the bolt distance to the purlin shoe edge: **35**.
- Define the distance between the two bolts at purlin: **70**.
- Define the distance between the two bolts at main beam: **80**.

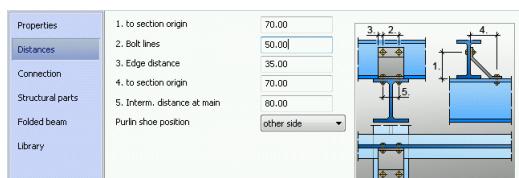


Figure 198: Properties dialog box – **Distances** tab

Note: For a clear view, display the bolts in the solid representation.

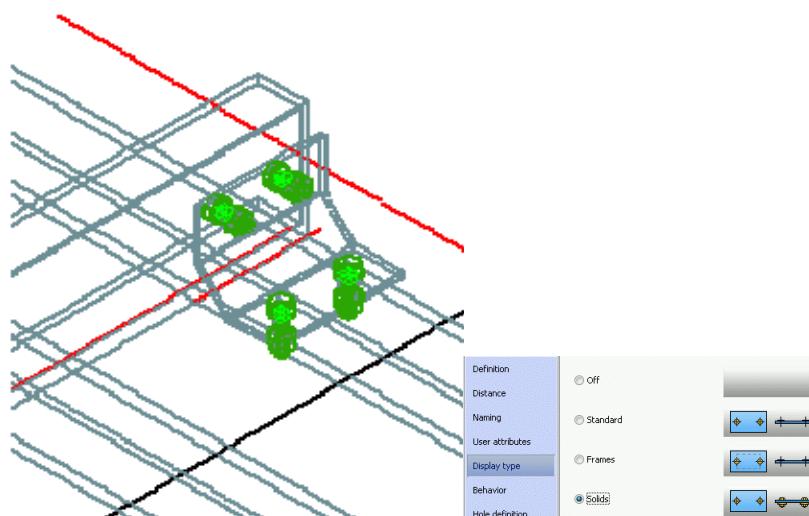


Figure 199: The purlin connections with bolts in the solid representation

Next, create by template all other purlin connections.

For a more realistic presentation of the model, use the **Realistic** visual style.

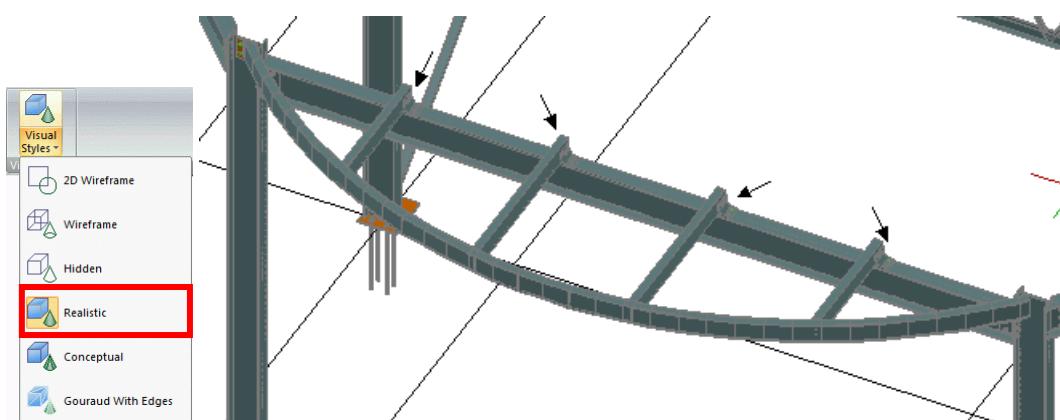
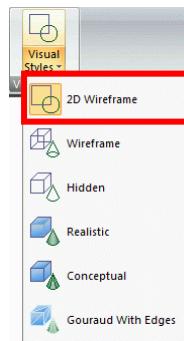


Figure 200: Purlin connections

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 9: Creating a flat bracing with tension bolt for a base plate

In this step, create flat bracings with tension bolts at the base plate of the **G4** and **G5** columns in the location indicated in Figure 201.

Use the buttons on the **Viewpoints** panel to change the view angle.

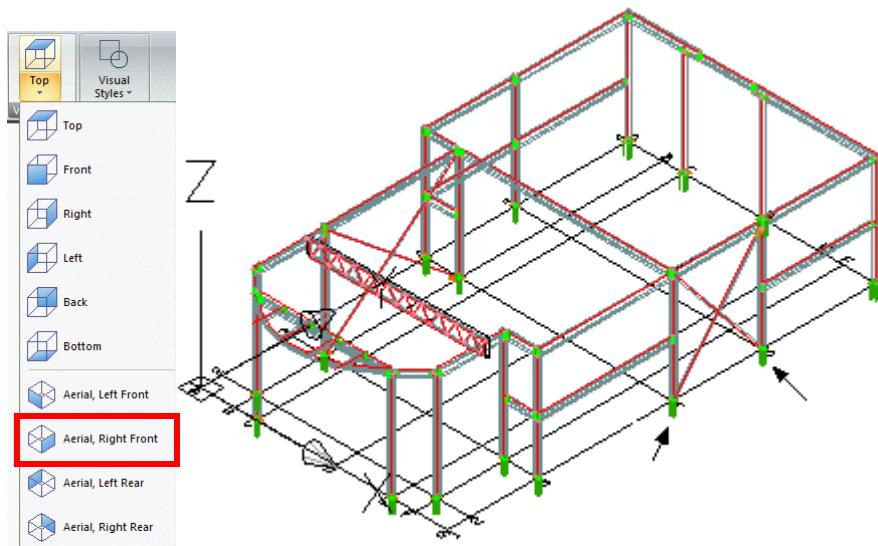
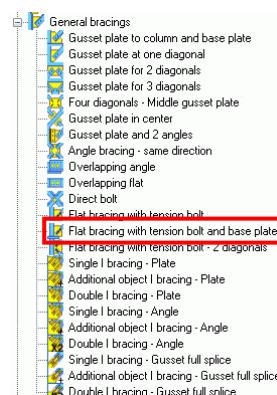


Figure 201: Joint locations in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click

2. From the **General bracings** category, click



3. Select the G4 column and press **Enter**.
4. Select the base plate and press **Enter**.
5. Select the bracing and press **Enter**.

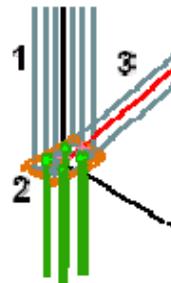


Figure 202: Selecting the elements to connect

6. Click **OK** in the “Attention” dialog box.

The joint is created and the properties dialog box appears.

Modifying flat bracing with tension bolt and base plate properties

In the properties dialog box, select the **General** category and define the following settings:

- On the **Gusset plate shape** tab:
 - From the **Shape** drop-down list, select the shape of the gusset plate: **two chamfer**.
 - Select the **Aligned to base plate** option to align the gusset plate to the base plate.

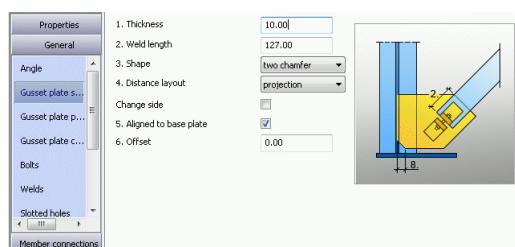


Figure 203: Properties dialog box – **Gusset plate shape** tab

- On the **Gusset plate contour** tab:

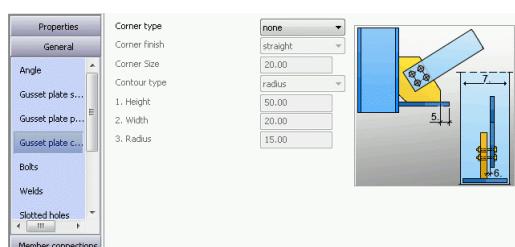


Figure 204: Properties dialog box – **Gusset plate contour** tab

- On the **Angle** tab:
 - Select the angle section class: **Angle identical**.
 - Select the angle section size: **L100X10**.

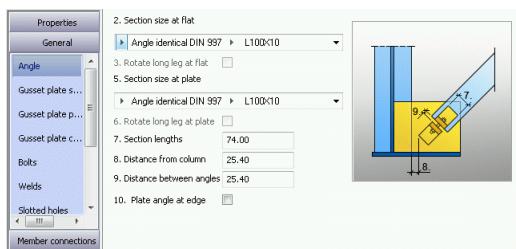


Figure 205: Properties dialog box – **Angle** tab

*For a more realistic presentation of the model, use the **Realistic** visual style.*

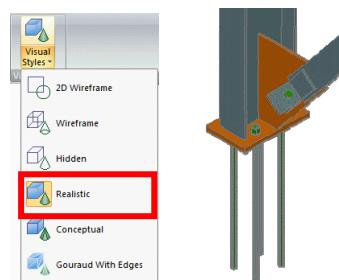


Figure 206: The flat bracing connection in shade mode

Next, create by template another flat bracing with tension bolt connection between the **G5** column, the base plate and the **G5 – G4** bracing.

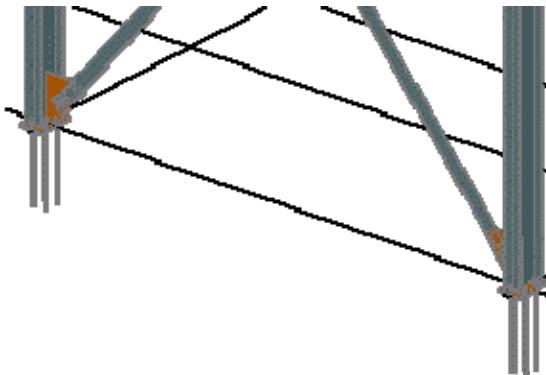
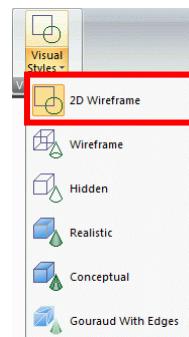


Figure 207: The two connections

*To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.*



Step 10: Creating a flat bracing with tension bolt for a beam

Use the same joint (flat bracing with tension bolts) to connect the **G4** and **G5** columns with the bracings and the **G4 – G5** beam.

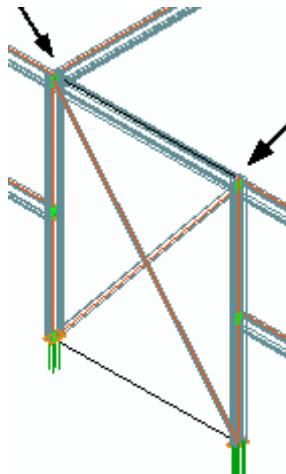
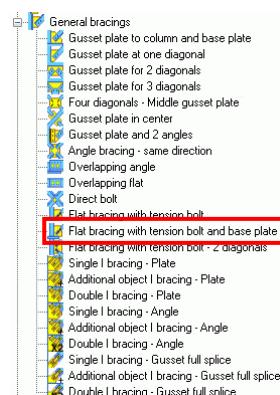


Figure 208: Joints locations in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling panel**, click  **Connection vault**.
2. From the **General bracings** category, click .



3. Select the **G4** column and press **Enter**.
4. Select the beam and press **Enter**.
5. Select the bracing and press **Enter**.

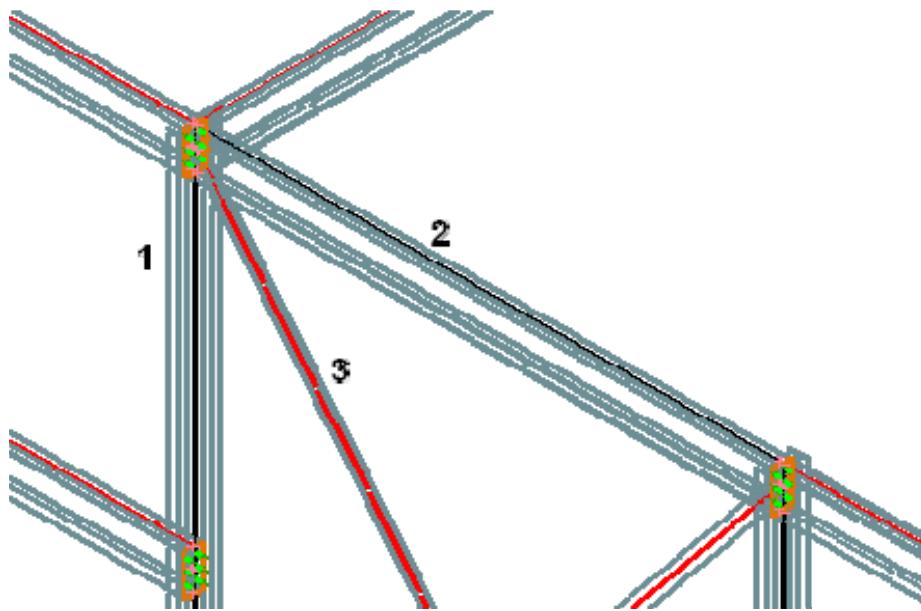


Figure 209: Selecting the elements to connect

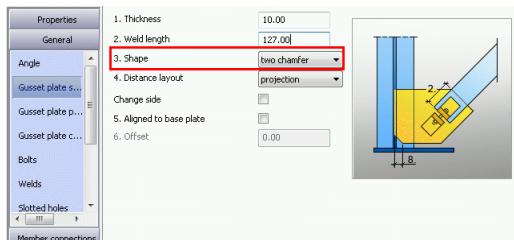
6. Click **OK** in the “Attention” dialog box.

The properties dialog box appears.

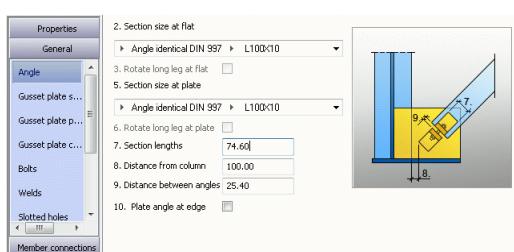
Modifying flat bracing with tension bolt and base plate properties

In the properties dialog box, select the **General** category and define the following settings:

- On the **Gusset plate shape** tab, select the shape of the gusset plate: **two chamfer**.

Figure 210: Properties dialog box – **Gusset plate shape** tab

- On the **Angle** tab, define the angle distance from the column: **100**.

Figure 211: Properties dialog box – **Angle** tab

Create by template another flat bracing with tension bolt between the G5 column, the G5 - G4 beam and the bracing.

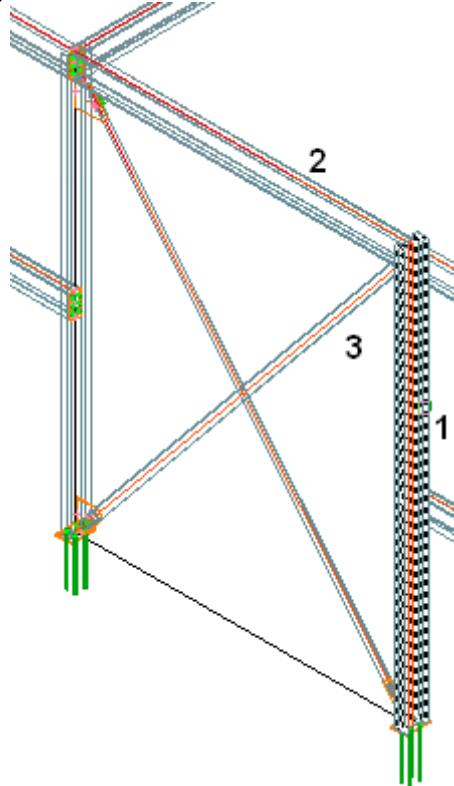


Figure 212: Selecting the elements to connect

For a more realistic presentation of the model, use the **Realistic** visual style.

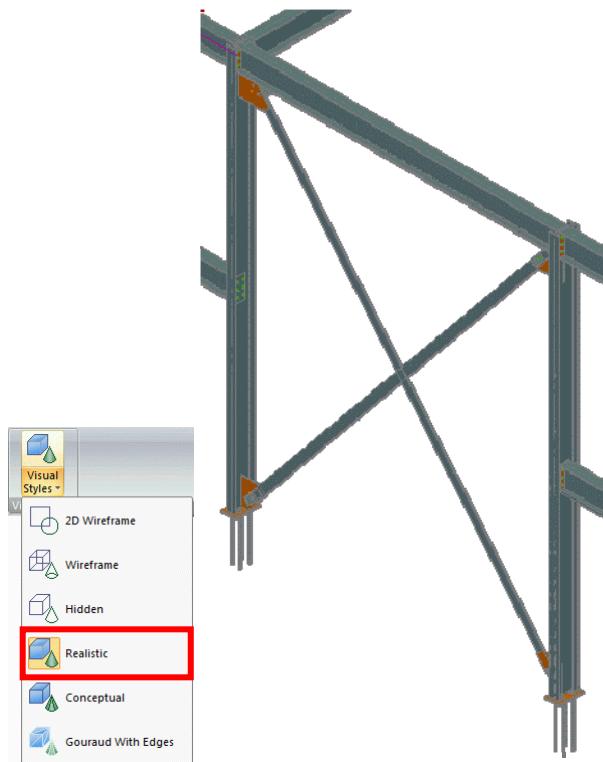
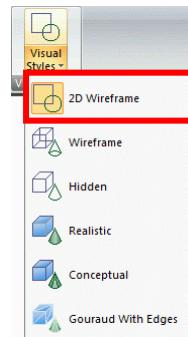


Figure 213: The joints in shade mode

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 11: Creating middle gusset plates

In this step, connect the bracings between the **G4** and **G5** columns by a middle gusset plate.

Before starting

Use the buttons on the **Viewpoints** panel to change the view angle.

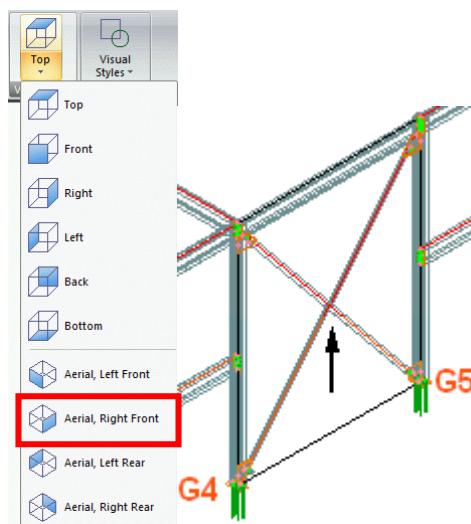
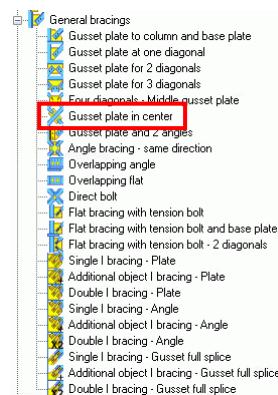


Figure 214: The joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click
2. From the **General bracings** category, click



3. Select the continuous diagonal and press **Enter**.

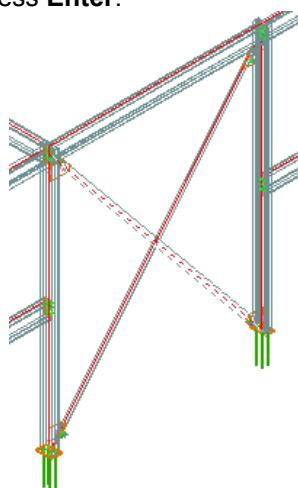


Figure 215: Selecting the continuous bracing

4. Select one of the two split beams and press **Enter**.
5. Select the other split beam and press **Enter**.

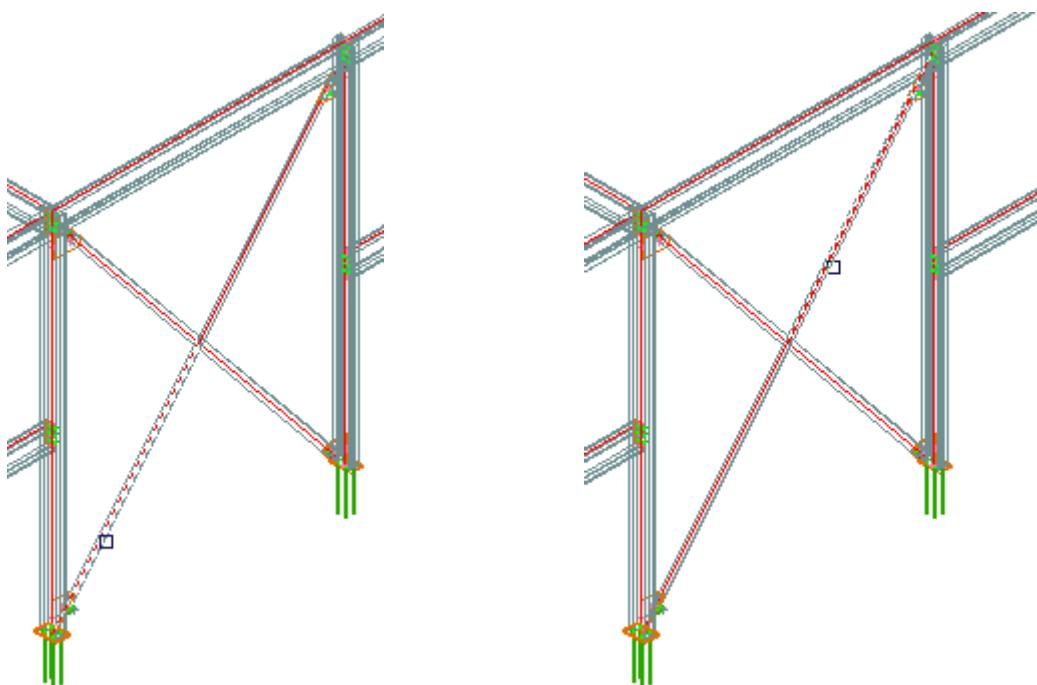


Figure 216: Selecting the elements to connect

6. Click **OK** in the “Attention” dialog box.

The joint is created and the properties dialog box appears.

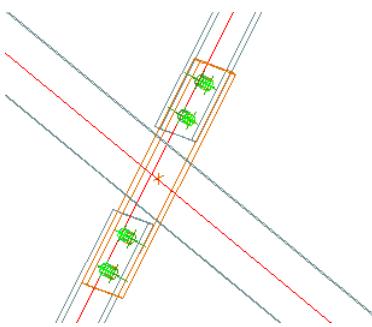


Figure 217: The middle gusset plate connection

Step 12: Creating a tube connection with sandwich plates for the base plate

In this step, create a tube connection with sandwich plates to connect the **B3** and **B4** columns with the base plates and the bracings. A similar joint connects the **B3 – B4** beam with the end plate (from the double side connection) and the bracing.

Before starting

Use the buttons on the **Viewpoints** panel to change the view angle.

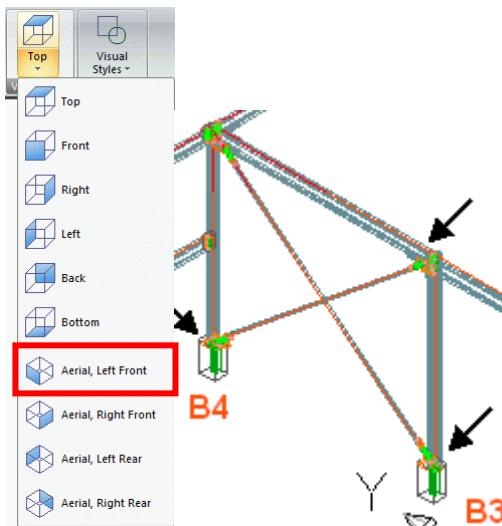
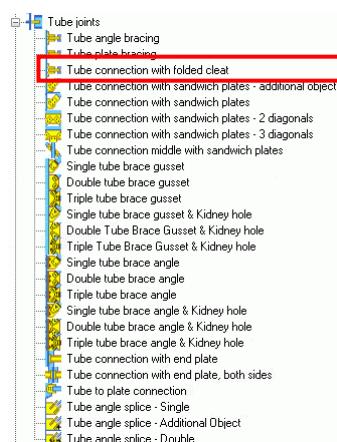


Figure 218: Joints locations in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .

2. From the **Tube joints** category, click .



3. Select the **B4** column and press **Enter**.
4. Select the base plate and press **Enter**.
5. Select the bracing and press **Enter**.

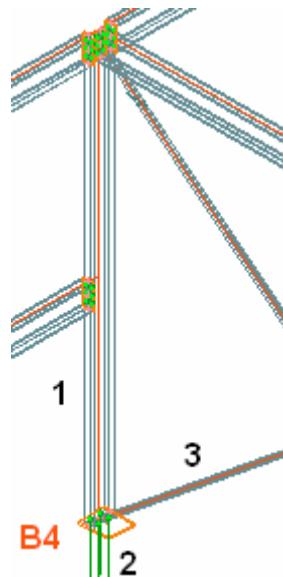


Figure 219: Selecting the elements to connect

6. Click **OK** in the “Attention” dialog box.

The properties dialog box appears.

Modifying the tube connection with sandwich plates properties

In the properties dialog box, select the **General** category and define the following settings:

- On the **Gusset plate shape** tab:
 - From the **1. Shape at column** drop-down list select **perpendicular** to create the top edge of the plate perpendicular on the column.
 - Select the **Aligned to base plate** option to align the gusset plate to the base plate.

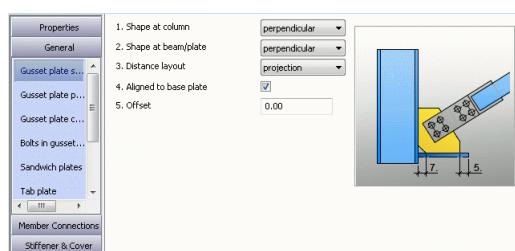


Figure 220: Properties dialog box – **Gusset plate shape** tab

- On the **Gusset plate contour** tab:
 - From the **Corner type** drop-down list, select **Corner finish**.
 - From the **Corner finish** drop-down list, select **straight**.
 - Define the corner size: **10**.

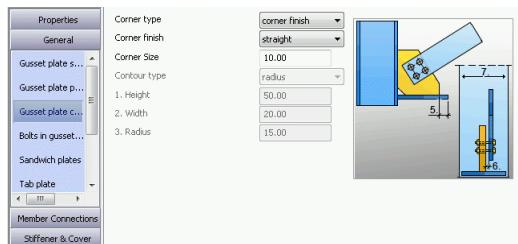


Figure 221: Properties dialog box – **Gusset plate corner** tab

- On the **Tab plate** tab define the tab plate size and the slot length:
 - Define the cut back layout as **100** from the last bolt.
 - From the **Width layout** drop-down list, select **total**.
 - Define the width value: **120**.

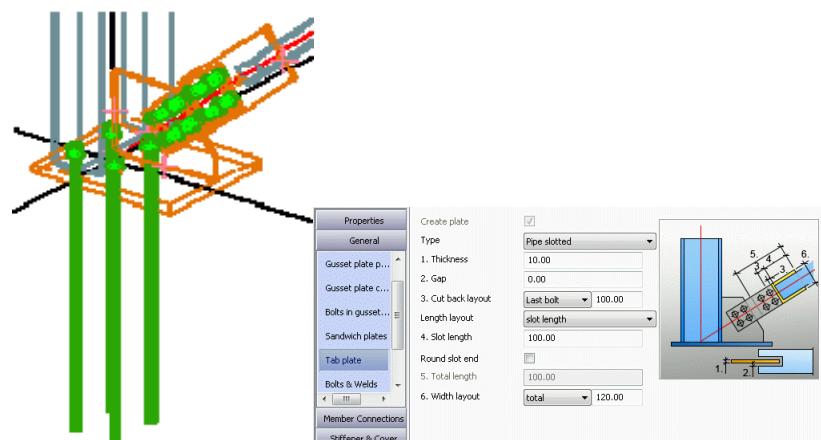


Figure 222: Properties dialog box – **Tab plate** tab

Next, create by template another Tube connection with sandwich plates connection for the **B3** column with the base plate and the bracing.

*For a more realistic presentation of the model, use the **Realistic** visual style.*

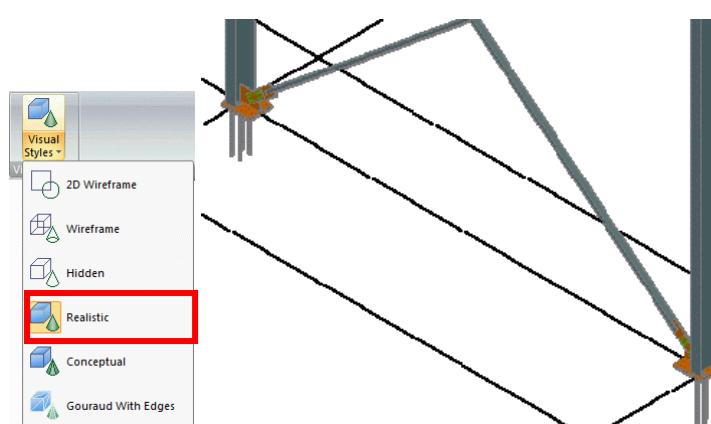
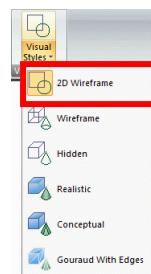


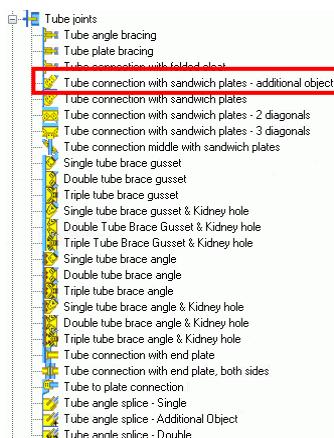
Figure 223: Tube connection with sandwich plates joint – shade mode

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Using a similar process, create another tube connection with sandwich plate connecting the **B3 – B4** column with the end plate (from the double side connection) and the bracing.

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .
2. From the **Tube joints** category, click .



3. Select the **B3-B4** rafter and press **Enter**.
4. Select the side end plate and press **Enter**.
5. Select the bracing and press **Enter**.

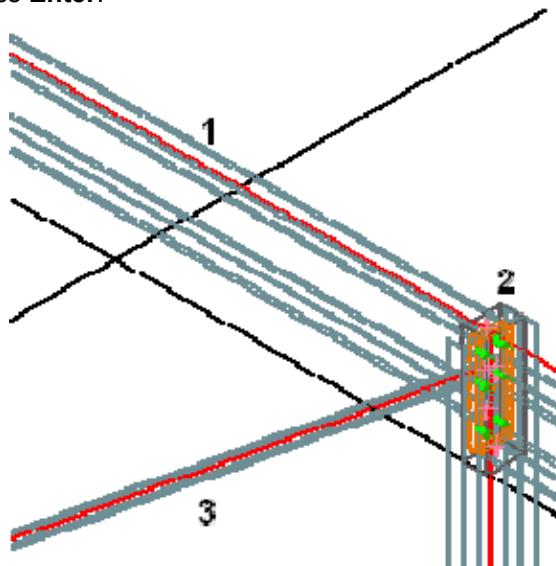


Figure 224: Selecting the elements to connect

6. Click **OK** in the “Attention” dialog box.

The joint is created and the properties dialog box appears.

Modifying the properties

In the properties dialog box, select the **General** category and define the following settings:

- On the **Gusset plate shape** tab:
 - From the **Shape at column** drop-down list, select **variable**.
 - From the **Shape at beam/plate** drop-down list, select **variable**.
 - Select the **Align to base plate** option to align the gusset plate edge to the side end plate.

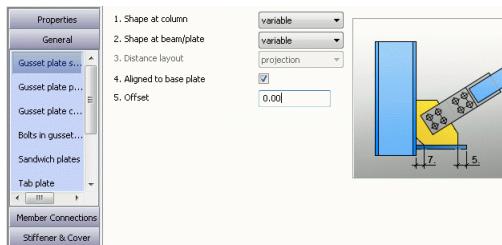


Figure 225: Properties dialog box – **Gusset plate shape** tab

- On the **Gusset plate contour** tab:

- From the **Corner type** drop-down list, select **Corner finish**.
- From the **Corner finish** drop-down list, select **straight**.
- Define the corner size: **10**.

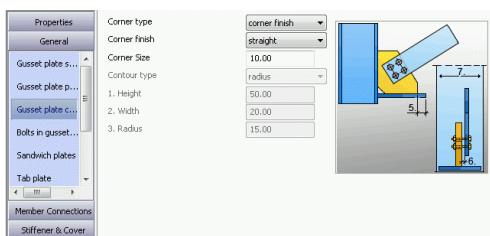


Figure 226: Properties dialog box – **Gusset plate contour** tab

- On the **Gusset plate parameter** tab:

- For the **Projection 2** parameter enter **50** to modify the gusset plate shape at sandwich plates.

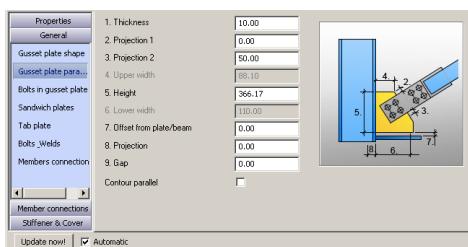


Figure 227: Properties dialog box – **Gusset plate parameter** tab

- On the **Tab plate** tab:

- Define a cut back length of **50** from the last bolt.

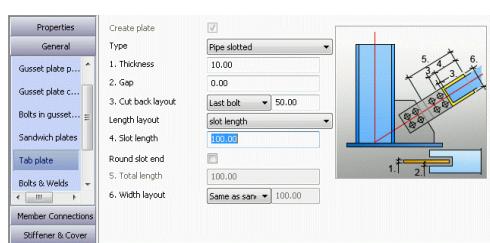
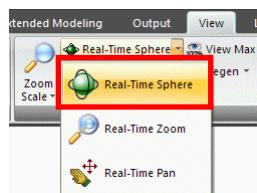


Figure 228: Properties dialog box – **Tab plate** tab

Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.



For a more realistic presentation of the model, use the **Realistic** visual style.

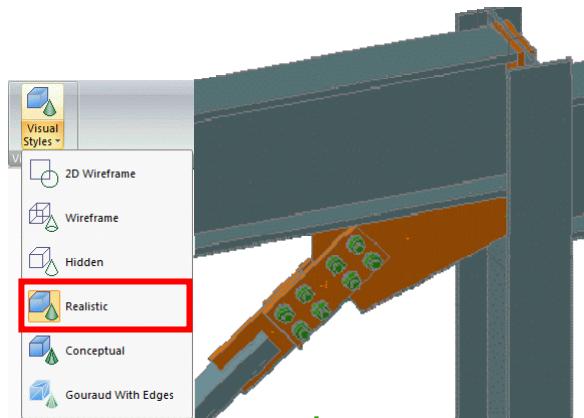
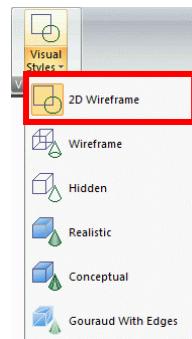


Figure 229: The joint at B3-B4 – in shade mode

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 13: Creating a tube connection with sandwich plates for a beam as an additional object

In this step, create a tube connection with sandwich plate to connect the **B3 – B4** beam with the **B4** column and the bracing.

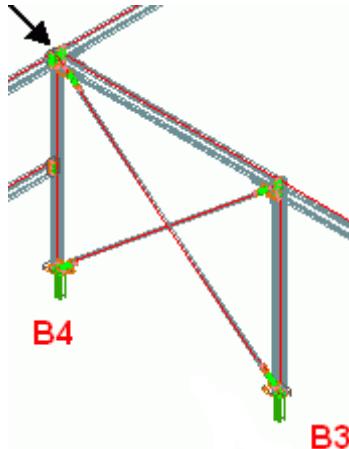
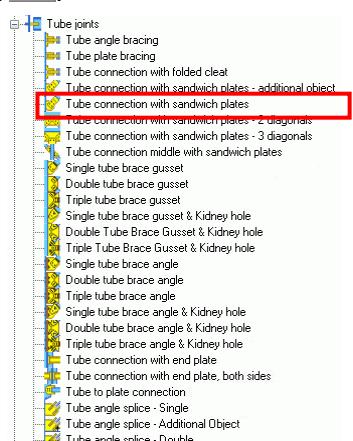


Figure 230: Joints locations in the model



1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click .
2. From the **Tube joints** category, click .



3. Select the **B4** column and press **Enter**.
4. Select the **B4 – B3** rafter and press **Enter**.
5. Select the bracing and press **Enter**.

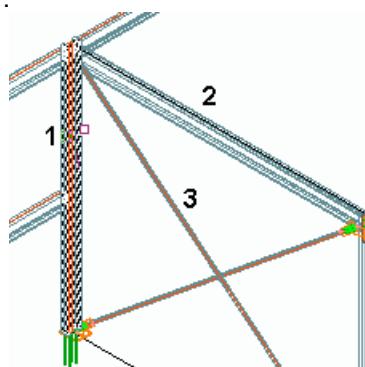


Figure 231: Selecting the elements to connect

In the properties dialog box, select the **General** category and define the following settings:

- On the **Gusset plate parameter** tab:
 - Define the gusset plate thickness: **10**.

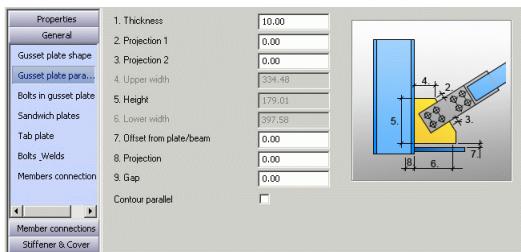


Figure 232: Properties dialog box – **Gusset plate parameter** tab

- On the **Gusset plate contour** tab:
 - From the **Corner type** drop-down list, select **Corner finish**.
 - From the **Corner finish** drop-down list, select **straight**.
 - Define the corner size: **10**.

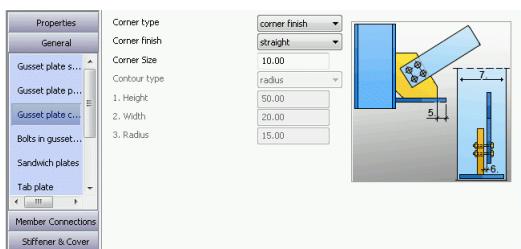


Figure 233: Properties dialog box – **Gusset plate contour** tab

- On the **Bolts in gusset plate** tab set the number of bolts:
 - Define a cut back value of **200** from the column.

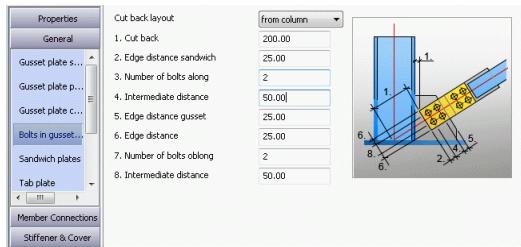


Figure 234: Properties dialog box – **Bolts in gusset plate** tab

- On the **Tab plate** tab:
 - Define a back length of **50** from the last bolt.

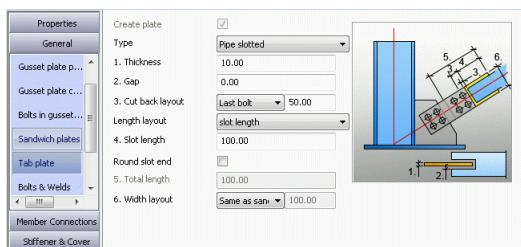


Figure 235: Properties dialog box – **Tab plate** tab

For a more realistic presentation of the model, use the **Realistic** visual style.

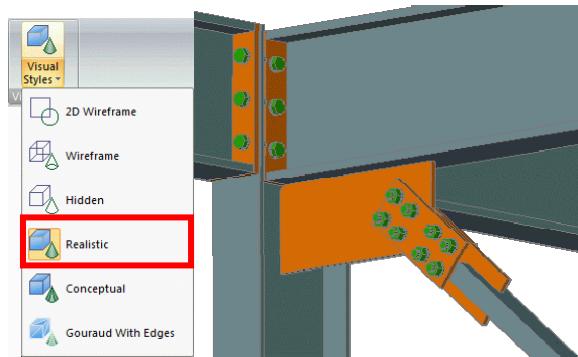
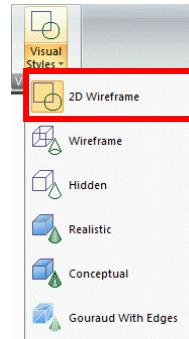


Figure 236: The joint at **B4** column in shade mode

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.



Step 14: Creating a tube connection middle with sandwich plates

In this step, connect the bracings between the **B3** and **B4** columns by a tube connection middle with sandwich plates.

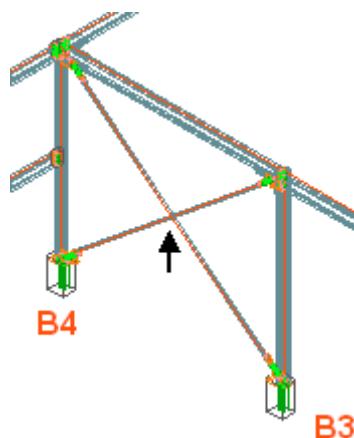
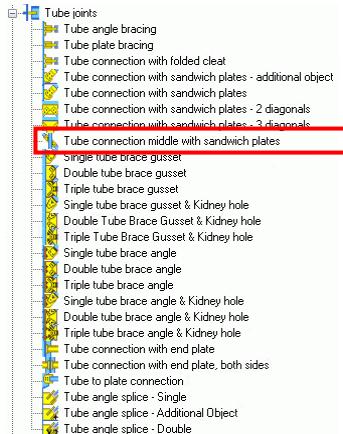


Figure 237: Joint location in the model

1. Open the Connection vault: On the **Home** tab, **Extended modeling** panel, click 
2. From the **Tube joints** category, click 



2. Select the continuous diagonal and press **Enter**.

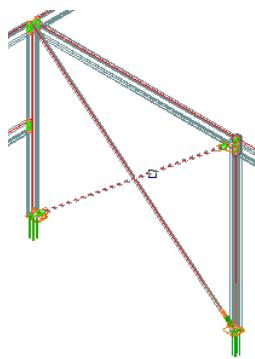


Figure 238: Selecting the continuous bracing

3. Select one of the two split bracings and press **Enter**.
4. Select the other split bracing and press **Enter**.

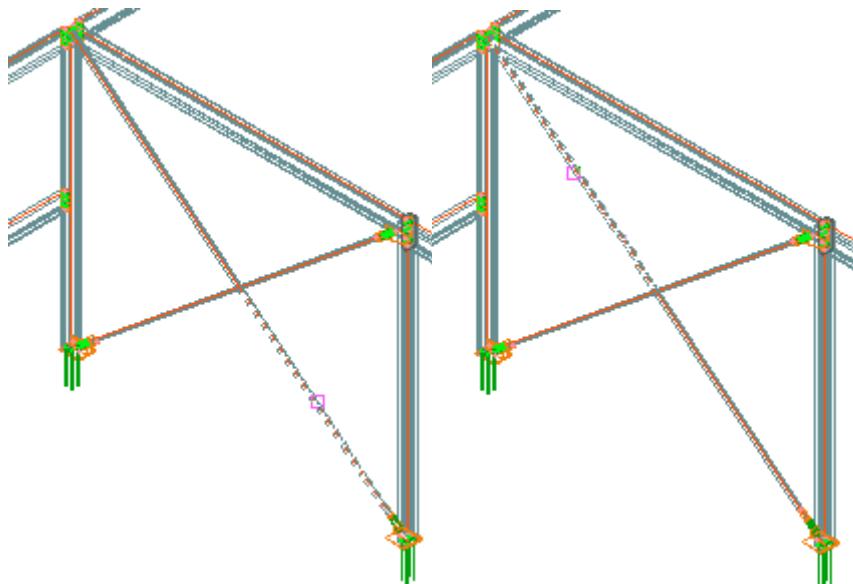


Figure 239: Selecting the elements to connect

5. Click **OK** in the “Attention” dialog box.

The joint is created and the properties dialog box appears.

Modifying the tube connection properties

In the properties dialog box, define the following settings:

- On the **Gusset plate basic** tab:
 - Define the thickness of the gusset plate: **10**.

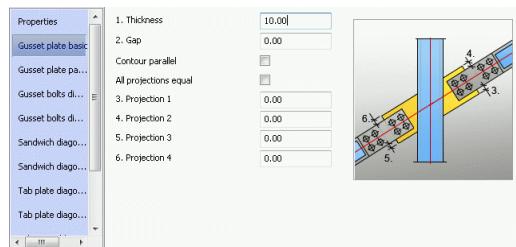


Figure 240: Properties dialog box – **Gusset plate basic** tab

- On the **Gusset plate parameter** tab, from the “Shape” drop-down list, select **2 corners**.

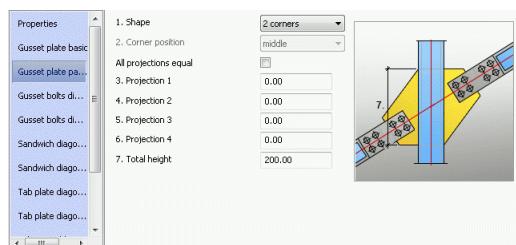
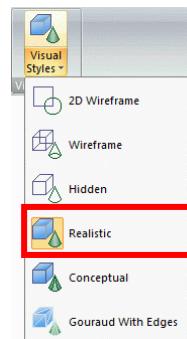


Figure 241: Properties dialog box – **Gusset plate parameter** tab

*For a more realistic presentation of the model, use the **Realistic** visual style.*



Note: For a clear view, display the bolts in the solid representation.

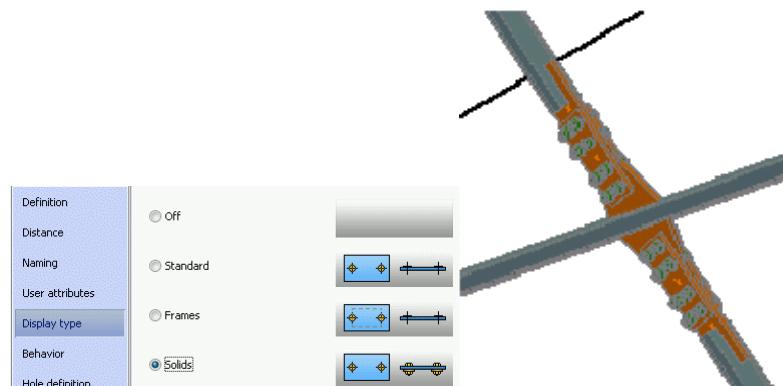


Figure 242: Tube connection middle with sandwich plate in shade mode, with bolts in the solid representation

To cancel the shading, click the **2D Wireframe** option on the **Visual styles** panel.

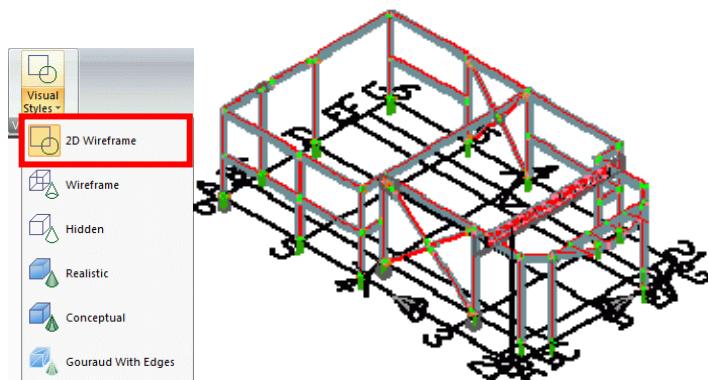


Figure 243: The model with all connections

Lesson 8: Collision check

With Advance Steel you can check for collisions in beams, plates, and bolts for either the entire model or a selection set.

The tool compensates for geometric inaccuracies with a given minimum volume of the collision solids set in the defaults. ACIS-solids are considered in the collision and collisions with a small volume are ignored.

In this lesson, you will learn how to:

- Check for collisions
- Display checking results
- Mark collision objects
- Repair collisions

Step 1: Checking for collisions

Once the model is finished, check it for collisions.

1. On the **Home** tab, **Utilities** panel, click .

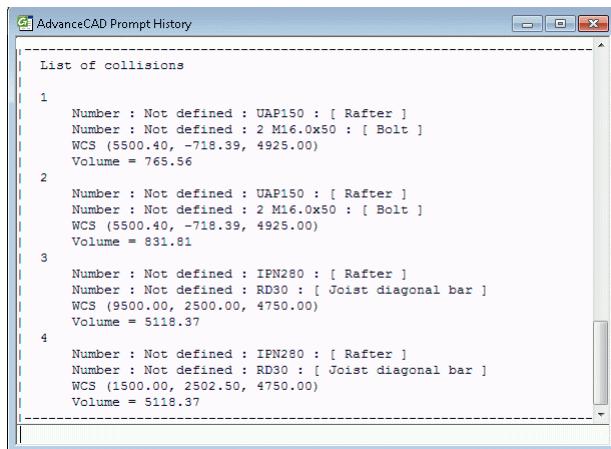


The collisions are listed in the text window with continuous index numbers.

2. Press **F2** to see the list of collisions in the text window.

There are 4 collisions in the model:

- Two between the joist diagonal bars and the rafters
- Two at the moment end plate connections



```

AdvancedCAD Prompt History

List of collisions

1
Number : Not defined : UAP150 : [ Rafter ]
Number : Not defined : 2 M16.0x50 : [ Bolt ]
WCS (5500.40, -718.39, 4925.00)
Volume = 765.56

2
Number : Not defined : UAP150 : [ Rafter ]
Number : Not defined : 2 M16.0x50 : [ Bolt ]
WCS (5500.40, -718.39, 4925.00)
Volume = 831.81

3
Number : Not defined : IPN280 : [ Rafter ]
Number : Not defined : RD30 : [ Joist diagonal bar ]
WCS (9500.00, 2500.00, 4750.00)
Volume = 5118.37

4
Number : Not defined : IPN280 : [ Rafter ]
Number : Not defined : RD30 : [ Joist diagonal bar ]
WCS (1500.00, 2502.50, 4750.00)
Volume = 5118.37
  
```

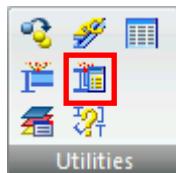
Figure 244: Text window – List of collisions

Note: Collision solids are not created.

Step 2: Displaying a collision

For each collision, the collision solid can be displayed.

1. On the **Home** tab, **Utilities** panel, click  to display a certain collision.



2. On the command line, enter the index number of a particular collision, for example **3** (the collision between the rafter and the joist diagonal bar).

```
Command: _AstM4CommBatchErrorsMarkObject
Please enter the index number of the collision:3
```

Figure 245: Collision index number on the command line

The collision solid is created and displayed in red.

In complex models, collision solids are hard to find. The “Search highlighted objects” tool creates a red arrow that shows the location of the collision solids.

3. On the **Selection** tool palette, click  to display the arrow pointing to the collision.

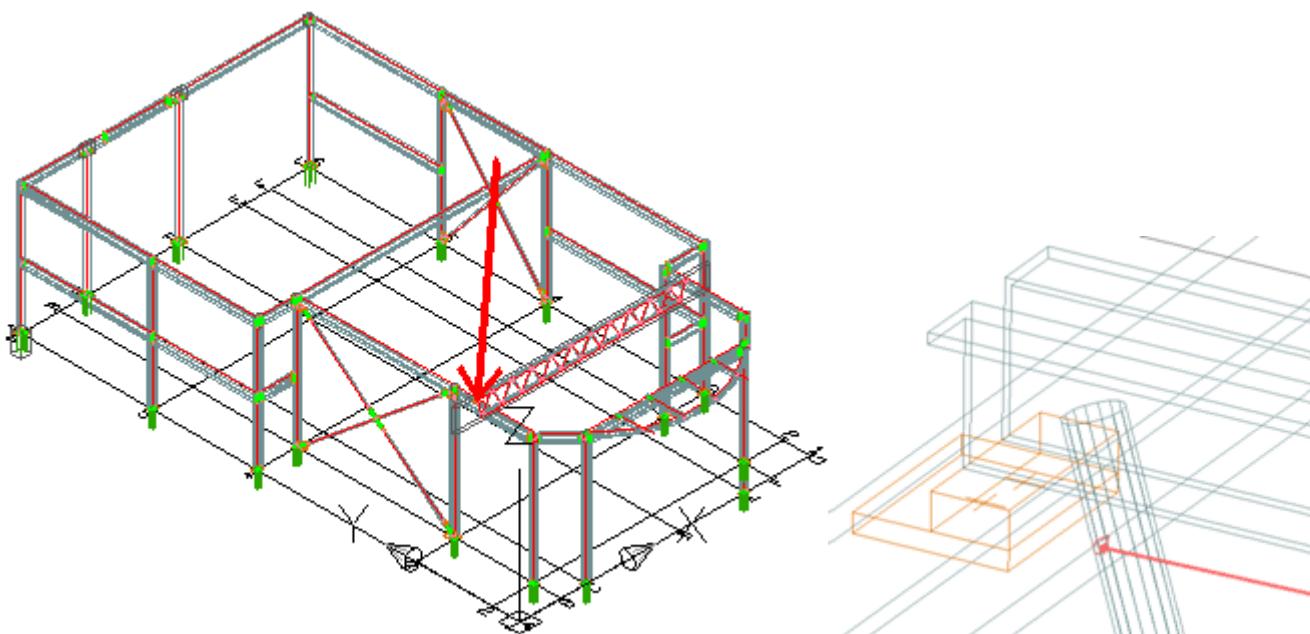


Figure 246: Locating the collision

Unmarking collisions



On the **Standard** tool palette, click  to unmark the found collisions.

Step 3: Repairing the collisions

In this step, repair the collisions found at the joist and the moment end plate connections of the **C6** and **D6** columns.

Collision number 4 and 3

This collision appears between the joist diagonal bars and the **B2 – B3** and **F3 – F3** rafters.



Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.



To repair the collision, modify the Opening P distance.

1. Select a joist element.
2. Right click and select **Advance Joint Properties** from the context menu.
3. In the properties dialog box, on the **Geometry** tab, in the **5. Opening P** field, enter **360**.

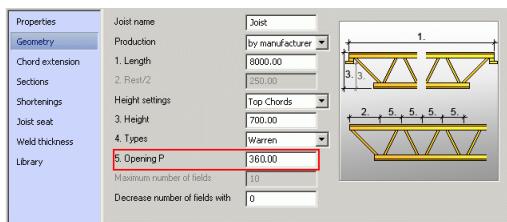


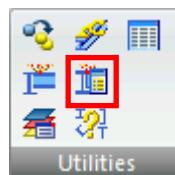
Figure 247: Properties dialog box – **Geometry** tab

Both collisions are repaired.

Collision number 1 and 2

This collision appears between the lower bolts of the moment end plate connections and the **E1 – C1** curved beam.

1. On the **Home** tab, **Utilities** panel, click to display a certain collision.



2. On the command line, enter **1**.

The collision solid is created.

3. On the **Selection** tool palette, click  to display the arrow pointing to the collision.

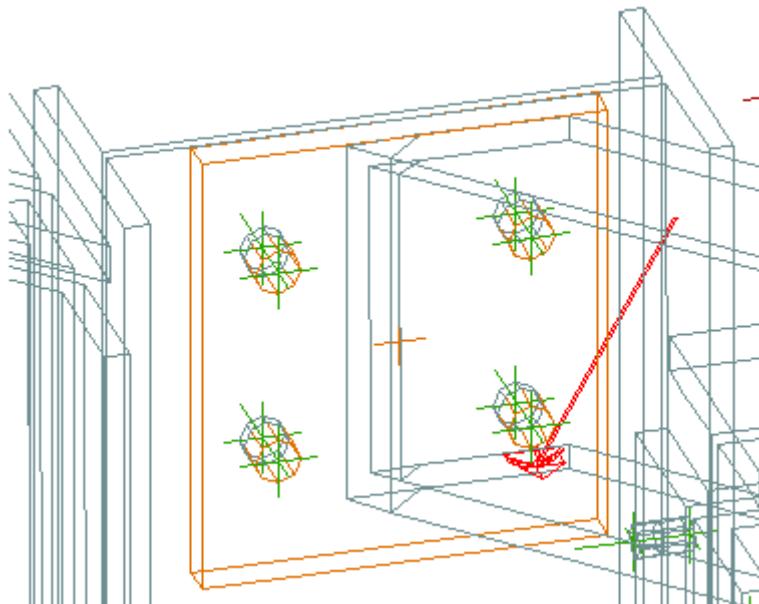


Figure 248: Locating the collision

To repair the collision, modify the distance between the bolts.

4. Select an element of the moment end plate connection, for example, the plate.
5. Right click and select **Advance Joint Properties** from the context menu.
6. In the properties dialog box, on the **Vertical bolts** tab, enter **70** in the **Intermediate distance** field.

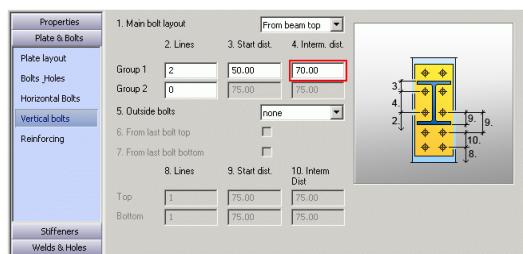


Figure 249: Properties dialog box – **Vertical bolts** tab

As Collision 2 is similar, use the same procedure to modify the joint properties of the moment end plate joint connecting the **E1** column with the curved beam.

Recheck the model using the “Clash check” tool  to make sure there is no additional collision found.

```
| -----
|       No collisions found
| -----
```

Lesson 9: Displaying connected objects

In this lesson, you will learn to:

- Display objects connected in workshop
- Display connected Advance objects
- Check the objects in relation to their connecting elements

Step 1: Displaying objects connected in shop

The “Display objects connected in shop” tool, displays the objects connected in the workshop, including beams, plates, bolt patterns and welds.

In this step, display objects connected in the workshop to the **A6** column.

*Use the buttons on the **Viewpoints** panel to change the view angle.*

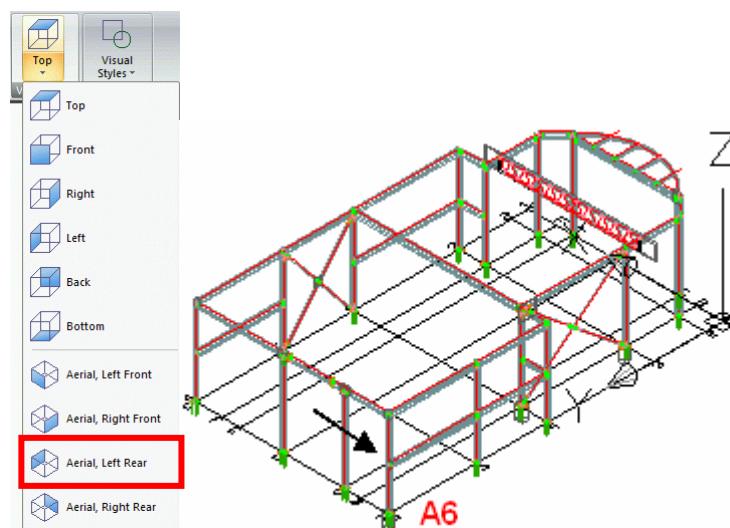


Figure 250: **A6** column

1. On the **Selection** tool palette, click .

Note: To remove previous marking and simultaneously display objects connected in the shop, on the **Selection** tool palette, click .

2. Select the **A6** column and press **Enter**.

The objects connected in the workshop are highlighted in red.

3. On the Selection tool palette, click  to select the marked objects (column, clip angles and base plate).

Use the Real-Time Sphere to rotate the drawing in 3D and get a suitable position.

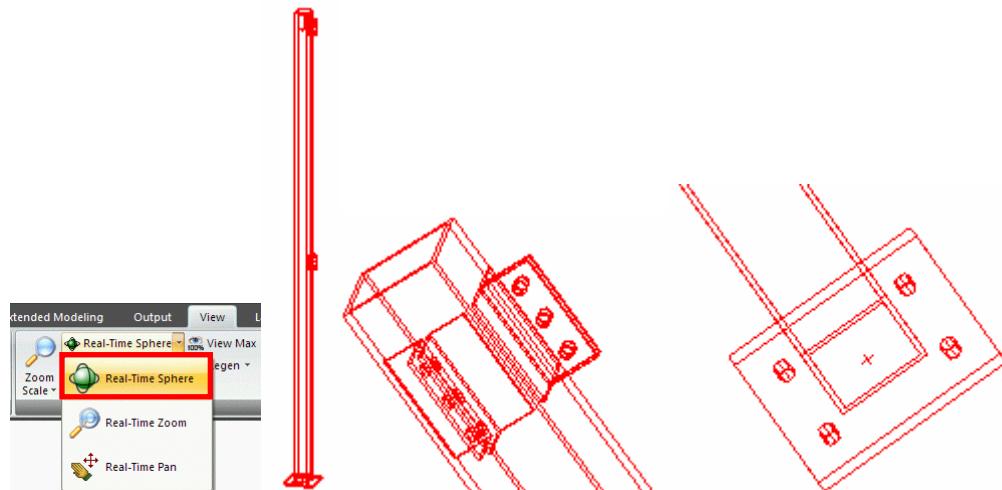


Figure 251: Viewing the elements connected in shop

Step 2: Displaying connected objects

The “Display connected objects” tool highlights the objects with both workshop and site connections, including beams, plates, bolt patterns and welds. This tool is useful to check if the elements of a model are connected.

Displaying all connected objects

1. On the **Selection** tool palette, click .
2. Select an element from the model, for example, the **A6** column and press **Enter**.

The connected objects are highlighted in red.

Notice that the joist is not highlighted, as it is not connected to the rafters. To fix this, the plates must be welded to the rafter.

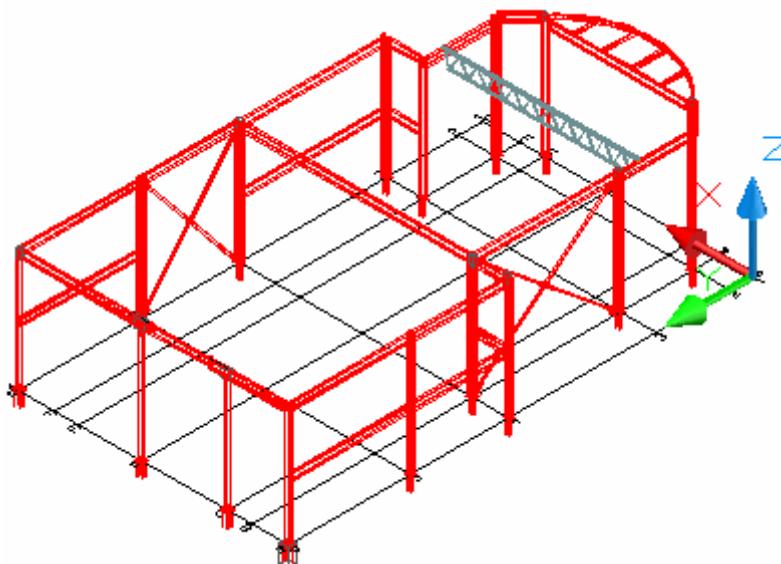
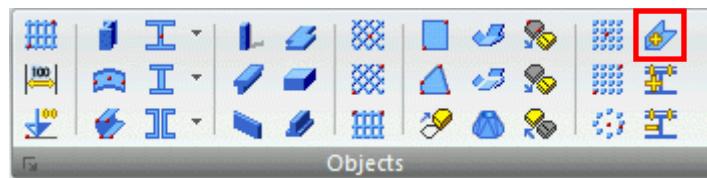


Figure 252: Connected elements

Creating the necessary connection

1. On the **Home** tab, **Objects** panel, click .



2. Select the elements to connect: the rafter and the joist seat and press **Enter**.
3. Click the weld point.

The weld is created.

Using the same process, create another weld between the other plate and the rafter.

Redisplaying all connected objects

1. On the **Selection** tool palette, click  to remove the marking.

2. On the **Selection** tool palette, click again .

The complete model is highlighted in red, provided that all connections were applied correctly.

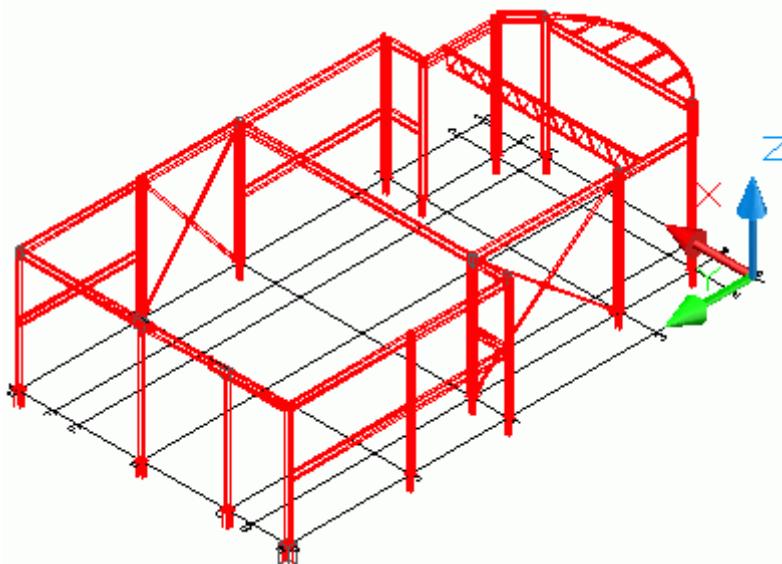


Figure 253: Completed model

Note: To remove previous markings and to simultaneously display connected objects, on the **Selection** tool palette, click .

Step 3: Displaying connection means

The “Display connection means” tool, checks the objects in relation to their connecting elements and identifies them. For example, it can determine if weld points are placed far away from the objects to be welded.

In this step, display the connection means for the **A6** column.

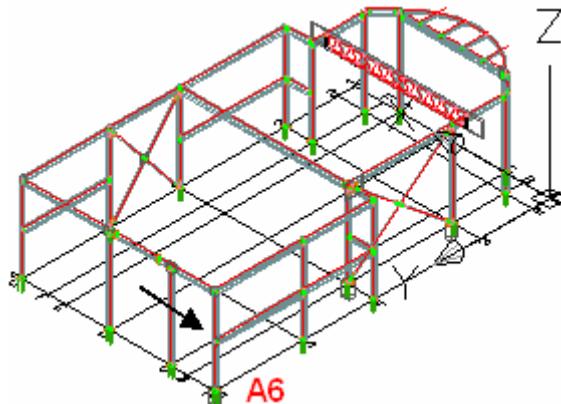


Figure 254: **A6** column



1. On the **Selection** tool palette, click .

Note: To remove previous marking and to simultaneously display connection means, on the **Selection** tool palette, click .

2. Select the **A6** column and press **Enter**.

The connected objects are highlighted in red and their number is displayed on the command line.



Figure 255: Command line

The highlighted objects can be pointed with arrows using the tool “Search marked objects” 

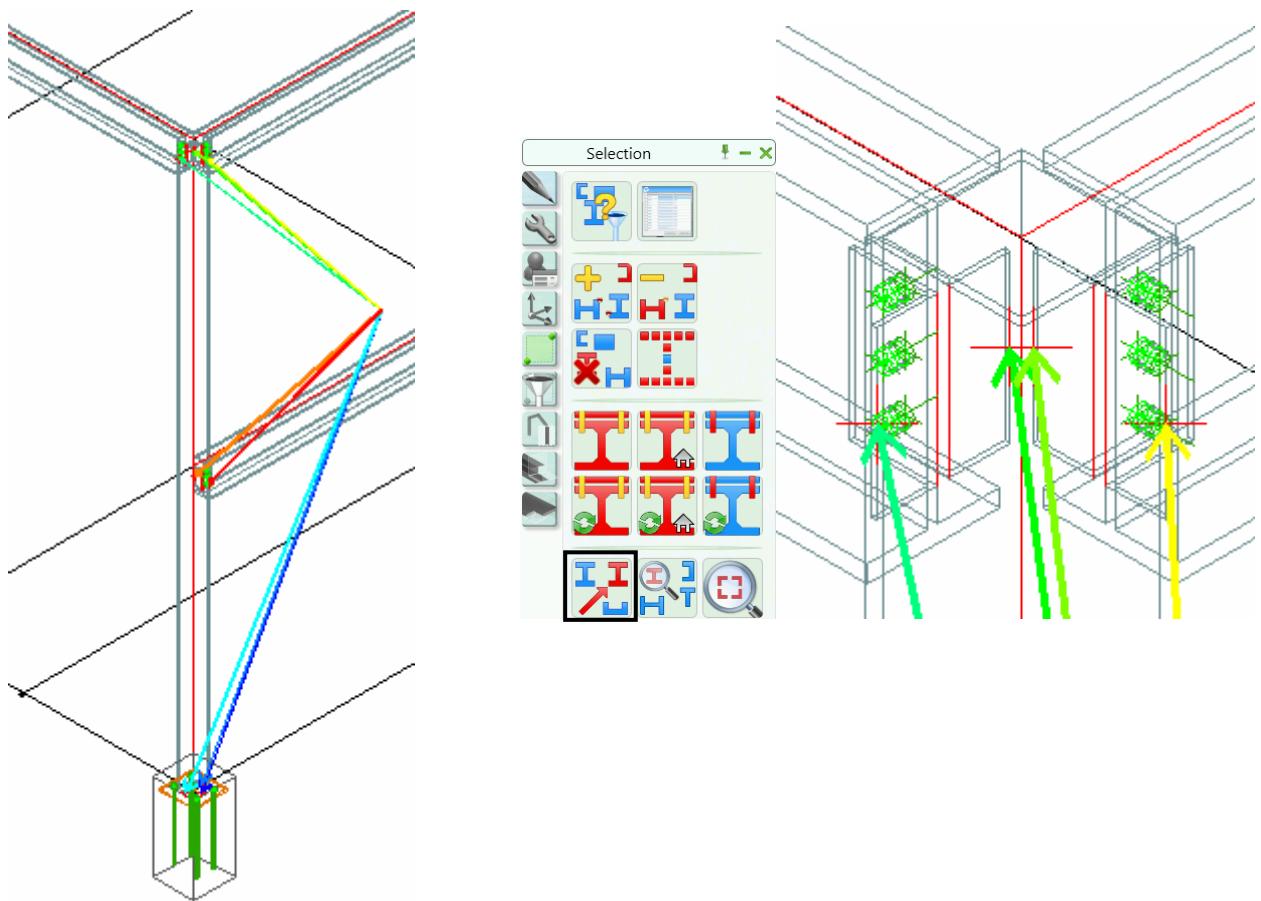


Figure 256: Locating connection elements in the model

3. On the **Selection** tool palette, click .

The marking is removed.

Lesson 10: Creating cameras

In this lesson, you will learn how to:

- Create a camera
- Modify camera properties
- Modify the position of an elevation view

Concept

The camera is an object representing the location, angle, and depth of a view, allowing individual focus for automated detail drawing creation. The camera's model views, elevations or overview settings and node details can be presented, named, and saved.

Before starting

Use the buttons on the **Viewpoints** panel to change the view angle.

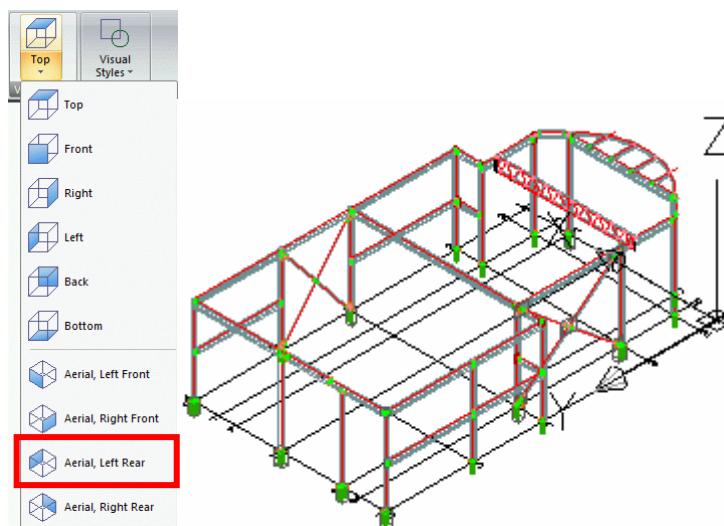


Figure 257: The model from the suitable view angle

Step 1: Inserting a camera according to the UCS

Insert a camera and get an elevation detail for the rear of the structure. The camera is created using the current user coordinate system. The view direction runs against the Z-axis.

For the first camera, use the lower end of the **G6** column.

Place the UCS in a suitable position as in the Figure 258. The XY plane is in the elevation plane.



1. On the **UCS** tool palette, click .
2. Specify the new origin point in the lower system line end point of the **G6** column.

3. Next, rotate the UCS so that it is placed as in the Figure 258.

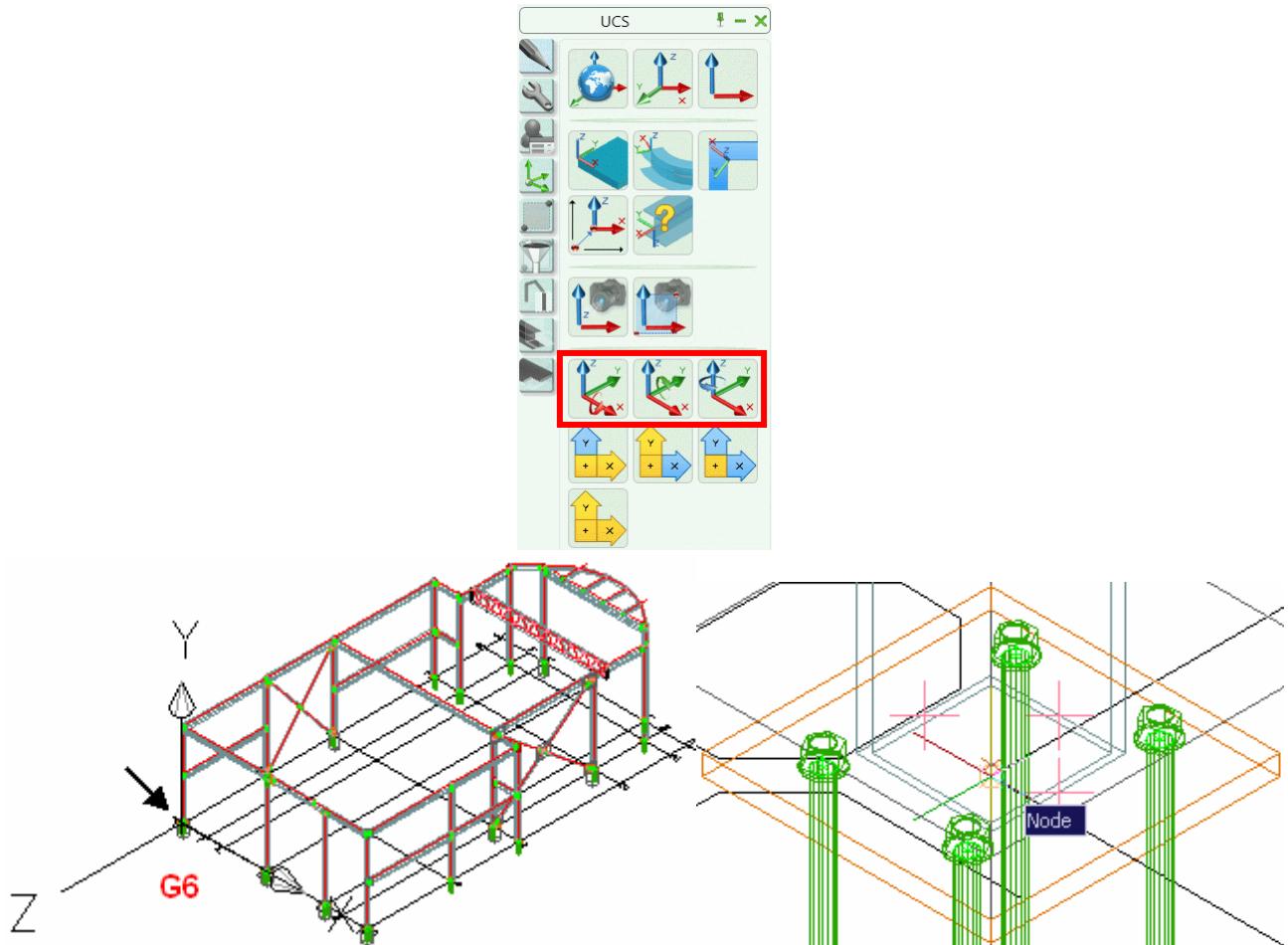


Figure 258: Placing the UCS

4. On the **Tools** tool palette, click to create a camera according to the current user coordinate system.

5. On the command line, enter the origin of the camera: **0,0,0**.

The camera is created and the properties dialog box appears.

6. In the camera properties dialog box, make the following settings:

On the **Properties** tab:

- From the **Type** drop-down list, select **Overview**.

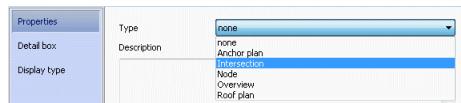


Figure 259: Camera properties dialog box – Selecting the camera type

- In the **Description** area enter **Elevation View 1**.



Figure 260: Camera properties dialog box – Adding a description

On the **Detail box** tab define the camera size so that all necessary elements are included:

- Select the **Front** and **Rear** options and enter the Z values to define the clipping distance in front and behind the grid on the UCS plane (depth).
- Select the XY Viewport: **Fixed**.
- Define the elevation length (in the X direction): **12000**.
- Define the elevation height (in the Y direction): **6000**.

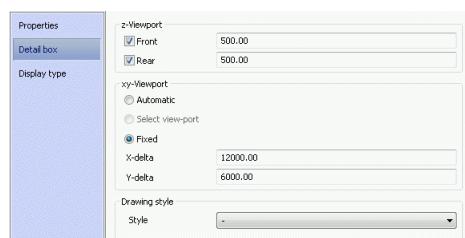


Figure 261: Camera properties dialog box – Setting the detail box size

The grid is stretched around the origin. The pencil-top shows the view direction against the positive Z-axis.

On the **Display type** tab, select the **Standard** option.



Figure 262: Camera properties dialog box – Display type tab

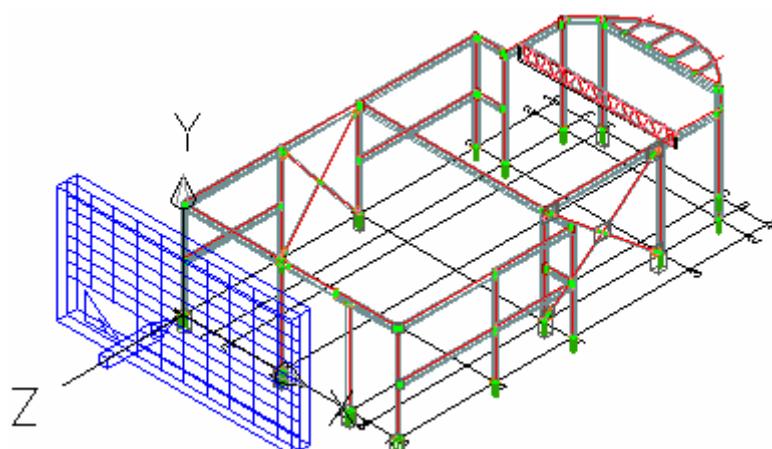


Figure 263: The camera

Use the buttons on the **Viewpoints** panel to change the view angle.

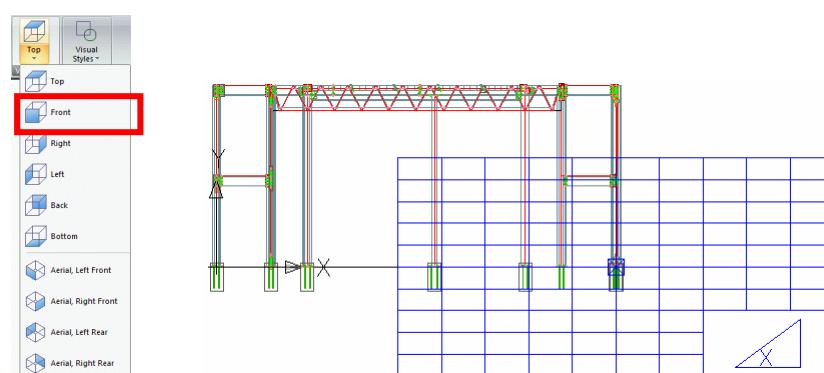
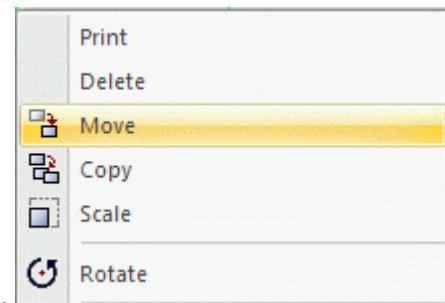


Figure 264: Camera – Front view

Step 2: Moving the camera

1. Select a grip point of the camera.



2. Right click and select **Move** from the context menu.

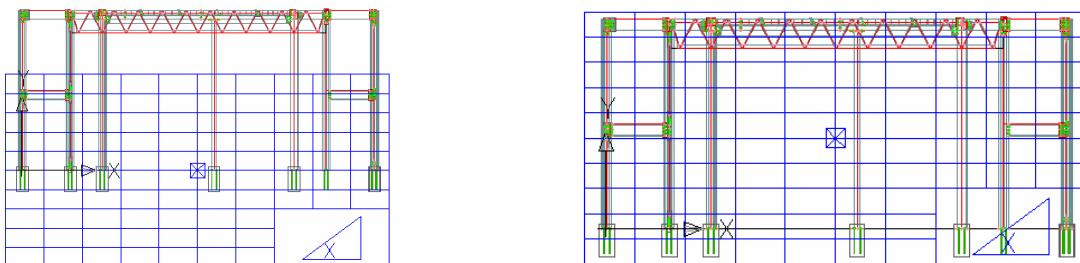


Figure 265: Moving the camera

*Use the buttons on the **Viewpoints** panel to change the view angle.*

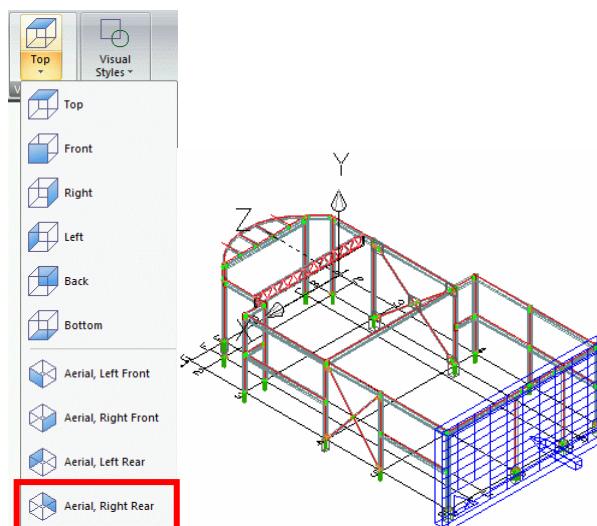


Figure 266: The camera in 3D view

To create a second camera at the lower system line end point of the **G3** column, place first the UCS in a suitable position as in the Figure 267. The XY plane is in the elevation plane.

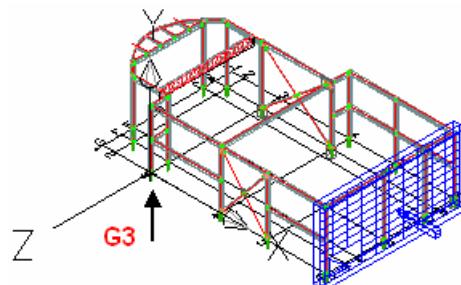
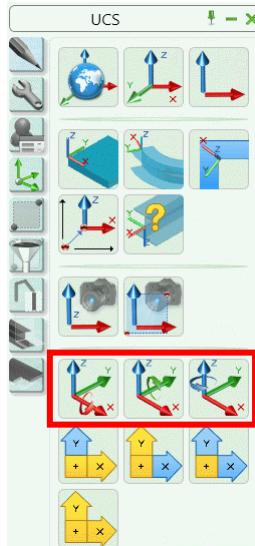


Figure 267: The camera in 3D view

Next, move the UCS with the origin in the lower system line end point of the **G3** column.

Using the same process, create an overview camera with the origin in the origin of the new coordinate system.

1. On the **UCS** tool palette, click .
2. Specify the new origin point in the lower system line end point of the **G3** column.
3. Next, rotate the UCS by Y to place the coordinate system in a suitable position (see Figure 272).



4. On the **Tools** tool palette, click  to create a camera according to the current user coordinate system.
5. On the command line, enter the origin for the camera: **0,0,0**.

The camera is created and the properties dialog box appears.

In the camera properties dialog box, make the following settings:

- On the **Properties** tab:
 - Select the type of the camera: **Overview**.

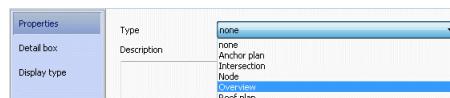


Figure 268: Camera properties dialog box – Selecting the camera type

- In the **Description** area enter **Elevation View 2**.

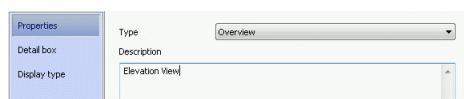


Figure 269: Camera properties dialog box – Adding a description

- On the **Detail box** tab define the camera size so that the necessary elements are included:
 - Select the **Front** and **Rear** options; the Z values refer to the clipping distance in front and behind the grid on the UCS plane (depth).
 - Select the XY Viewport: **Fixed**.
 - Define the elevation length: **16000**.
 - Define the elevation height: **6000**.

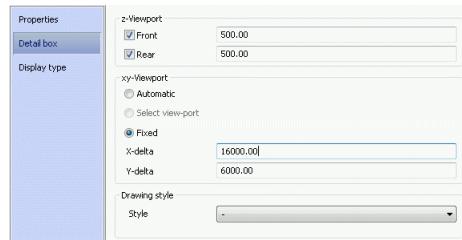


Figure 270: Camera properties dialog box – Setting the detail box size

The grid is stretched around the origin. The pencil-top shows the view direction against the positive Z-axis.

- On the **Representation type** tab, select the **Standard** option.



Figure 271: Camera properties dialog box – Representation type tab

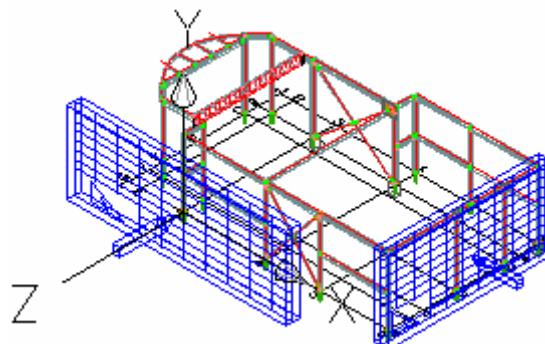


Figure 272: The model and the two cameras

*Use the buttons on the **Viewpoints** panel to change the view angle.*

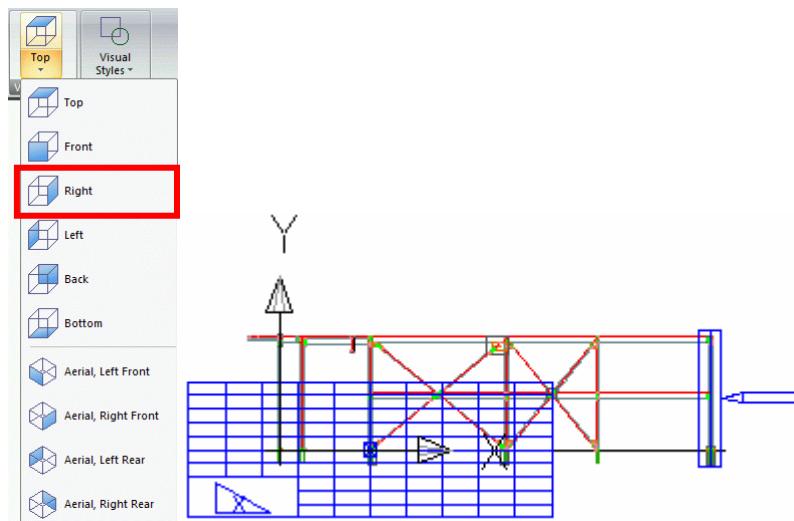
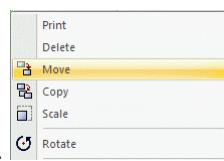


Figure 273: The second camera from the side view

Use the Move tool to move the second camera:

1. Select a grip point of the camera.



2. Right click and select **Move** from the context menu.



Figure 274: Moving the camera

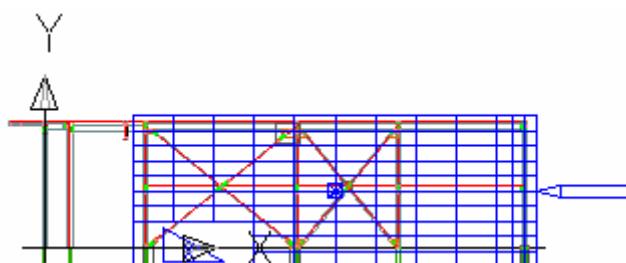


Figure 275: Moving the camera – new position

Use the buttons on the **Viewpoints** panel to change the view angle.

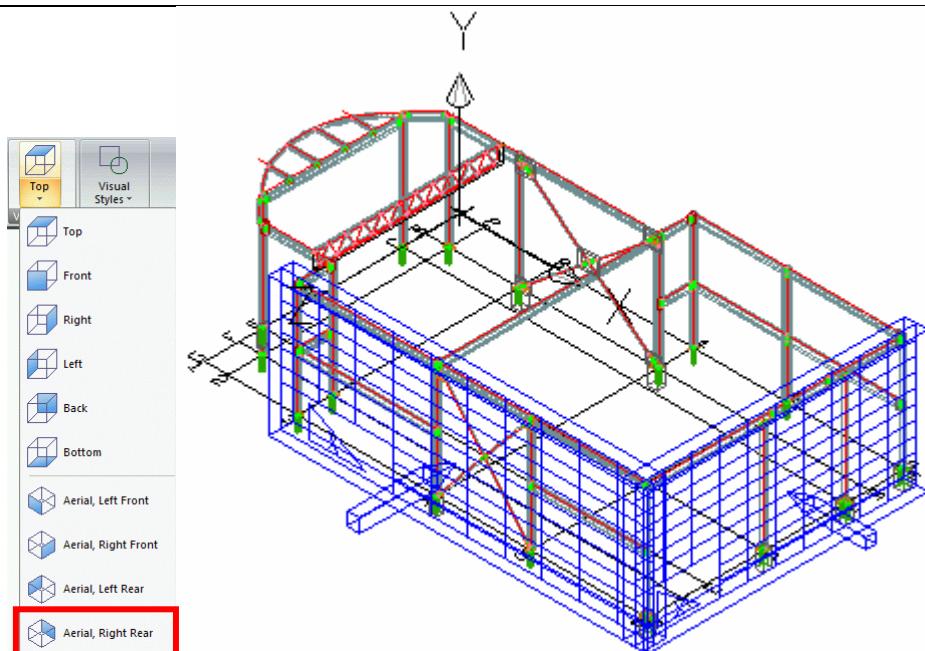


Figure 276: The model and the two cameras in 3D view

Step 3: Changing the camera color

In this step, change the color of a camera to distinguish it from the other one.

1. Select the second camera.
2. On the **Utilities** tab, **Properties** panel, from the first drop-down list, select **Cyan**.

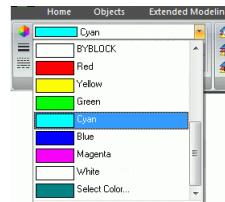


Figure 277: Selecting the new camera color

The camera is reassigned to another color.

The cameras created in this lesson are used for the drawing creation, in Lesson 12: Drawings.

To turn the camera representation off, hide the Cameras Layer.

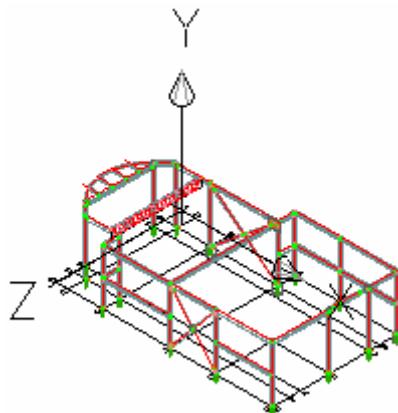


Figure 278: The model

Lesson 11: Numbering

In this lesson you will learn how to:

- Configure automatic prefixes for single part and assembly numbering
- Number the elements

Concept

The **Advance numbering** tool automatically numbers **Single Parts** and **Assemblies** for the entire model. The basis for the numbering is finding **identical parts** that should have the same mark.

The elements are compared by geometry, material properties, coating, and commodity (and behavior). The properties name and lot/phase are not considered for numbering. The model role is used by the automatic prefix tool to assign prefixes but not used directly for numbering.

If nothing is selected, the entire model is numbered. When Advance objects are selected, only these items are numbered. The results in either case are recorded in the text window (or by a default setting, they can be recorded in a text file that can be saved).

*Use the buttons on the **Viewpoints** panel to change the view angle.*

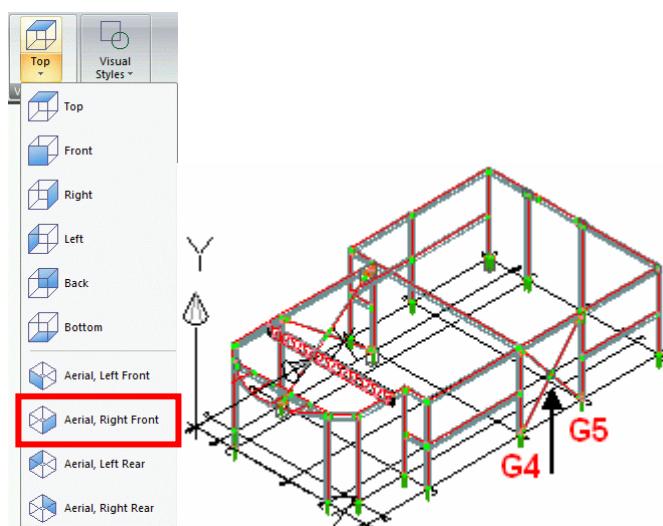
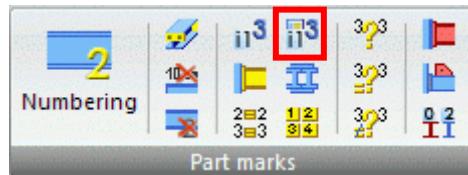


Figure 279: The gusset plate location in the model

Step 1: Setting prefixes

In this step, configure the automatic prefixes used for single part and assembly numbering. The prefix is based on the model role.

1. On the **Output** tab, **Part marks** panel, click .



In the “Prefix configuration” dialog box set different profiles for different jobs.

2. Select the **Use prefix** option to activate the prefix settings.

Note: The **User tree** includes a copy of all the prefix profiles from the **Advance tree** and can be modified.

3. Expand the **User** branch and select **Default\Plates\Any\Gusset Plate** to define the prefixes for gusset plates.
4. In the right side of the window enter the prefixes for single parts and assemblies.

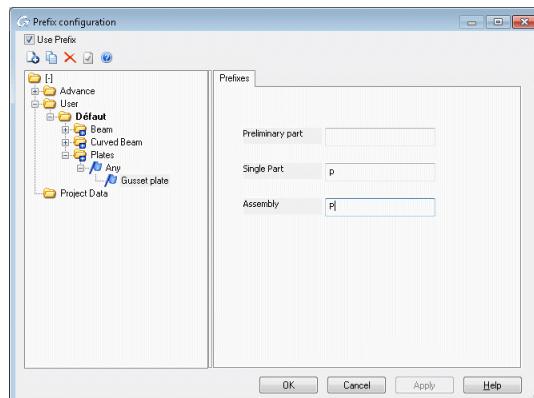


Figure 280: Setting the prefixes for the gusset plates

5. Right click **Default** and select **Set as current profile** from the context menu.

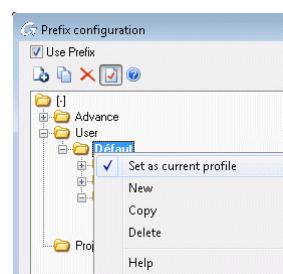


Figure 281: Setting the current profile

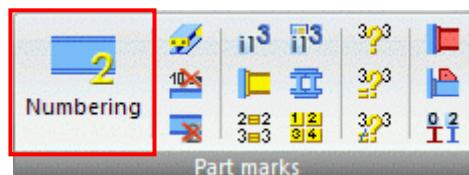
6. Click **OK**.

Step 2: Numbering

Once the prefixes are set, perform the numbering process for single parts and assembly parts in one step.

The biggest part of an assembly is the main part and will get an assembly number; all the other parts are considered attached and will have single part numbers.

1. On the **Output** tab, **Part marks** panel, click .



2. In the “Numbering” dialog box, define the following settings:

- On the **General** tab, select the **Process single parts** and **Process assemblies** options to number the single parts and assemblies in one step.

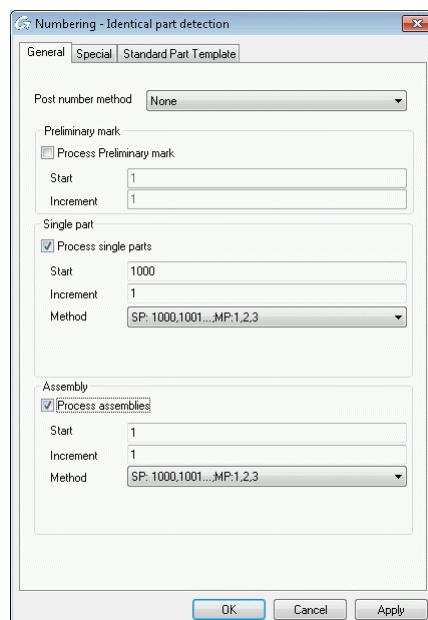
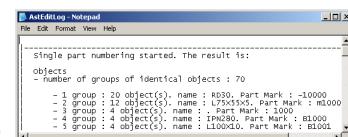


Figure 282: Numbering dialog box

Note: The numbering for single parts and assembly parts can be performed separately. In this case, the single part numbering **must** be done first.

3. Click **OK**.



The results appear in the command line. To display the text window, press **F2**.

Step 3: Displaying the results

For each element of the model the part marks are displayed in the properties dialog box.

1. Select the middle gusset plate between the **G4** and **G5** columns.
2. Right click and select **Advance Properties** from the context menu.

The single part and assembly numbers are shown with their prefixes.

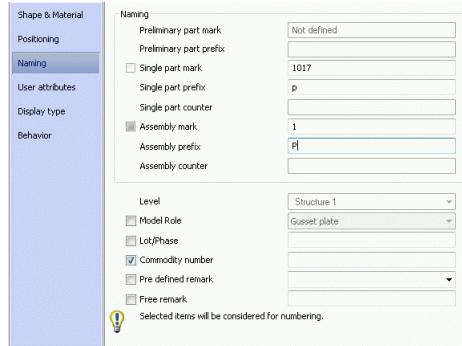


Figure 283: Properties dialog box – single part and assembly numbers

Lesson 12: Drawings

After building the 3D model, you can automatically create dimensioned and labeled 2D general arrangement and shop drawings. The linked drawings are created in separate DWG files from the model, but they are linked to track changes.

In this lesson, you learn how to:

- Create automatic detailing of all the single parts in the model using processes
- Modify the scale of a detail
- Insert a label in the linked drawings using the model information
- Insert a linear vertical dimension
- Create automatic detailing of all the main parts in the model using processes
- Insert a weld symbol
- Insert a structured list in a drawing

Concepts

Drawings: A drawing may consist of several linked details, which are individual Advance objects having their own properties. The model knows which drawings have been linked and checks if these drawings still correlate. Thus, the drawings can be updated after any model modifications. This link is ‘one way’ and changing a drawing cannot modify the model.

Processes: In addition to drawing styles, Advance has *Processes*, which automatically create drawings (using appropriate drawing styles) and arrange the linked details within the drawing (DWG) or across several drawings.

A process includes a number of sub-processes: selection of parts, sorting, drawing style selection, rules for arranging details on the sheet, and rules to attach new sheets.

Before starting

If necessary, change the color of the visible lines in the **GRAITEC Advance Manager** - the configuration tool used by Advance to modify default information for modeling, drawing creation, database management as well as many other management settings.

1. From the Windows start menu: **All Programs > Graitec > Advance Manager 2013 > Advance Manager 2013**
2. From the **Settings** category, select **Defaults**.
3. On the “**Defaults**” tab, **AdvanceSteel** category, from the **Drawing-presentation** category, **Color** group, select **Color of visible lines** to modify the default color of the visible lines.
4. In the right side of the window select the desired color.



Figure 284: Defining the default color of the visible lines in details

Step 1: Automatic detailing of all the single parts in the model

Using processes, create drawings for all the single parts in the model.

1. On the **Home** tab, **Document creation** panel, click Quick documents.
2. In the Quick documents window, select **All Sp PageFull A0**.

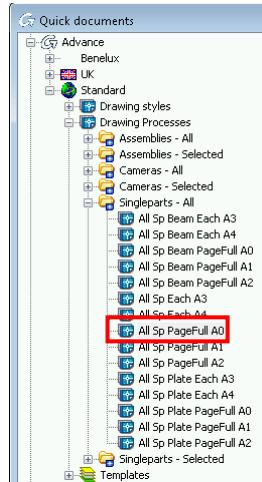


Figure 285: Selecting the process

3. Click **Use**.
4. In the process properties dialog box, from the **Selection** drop-down list, select **All Sp**.

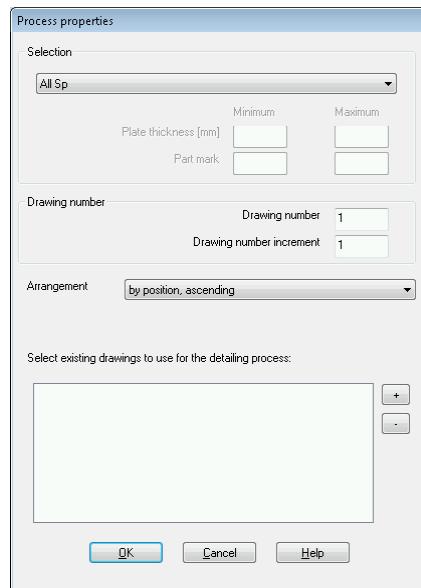


Figure 286: Process properties dialog box

A new folder is created with the same name and at the same location as the DWG model. The linked details are saved in .dwg files in the **Details** sub-folder.

5. To see the details, from the Quick start toolbar, click **Open**.



6. In the **Details** folder, open the desired .dwg file, for example, the **001_A0-Detail** dwg file.

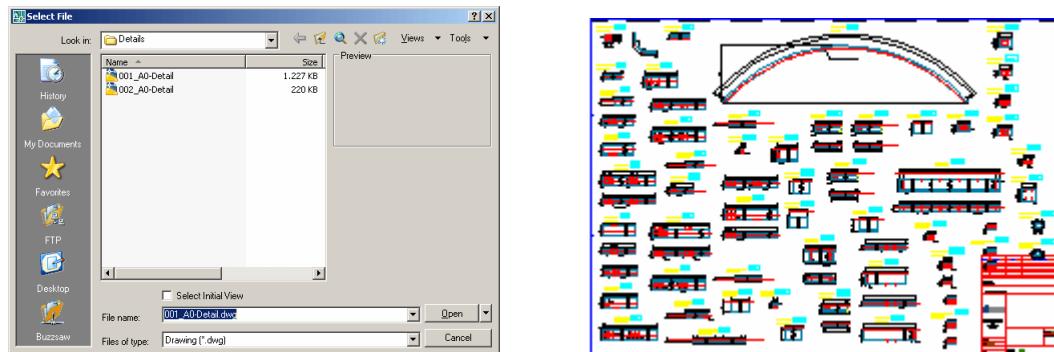


Figure 287: The **001_A0-Detail** file

Warning! If a DWG model or a DWG drawing is renamed, the relationship is broken, but can be recreated (registered). Drawings can also be separated (unlinked) from the model.

Step 2: Modifying the properties of a detail

Next, modify the scale, and manually add a label and a dimension.

Modifying the scale of a detail

1. Select the green frame of a detail, e.g., a detail of a beam.

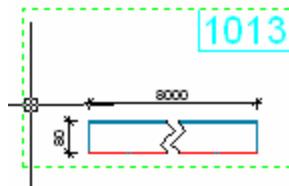


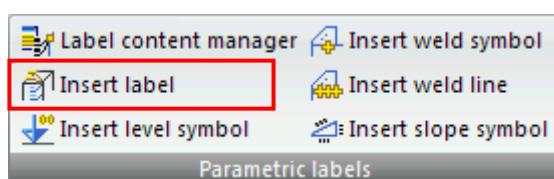
Figure 288: Selecting the detail frame

2. Right click and select **Advance Detail Properties** from the context menu.
3. In the details properties dialog box, from the **Scale** drop-down list, select **1:20**.
4. Click **OK**.

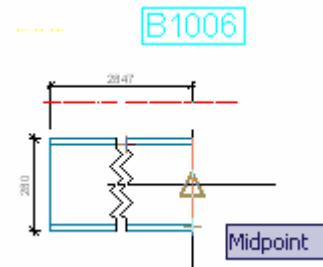
Inserting a label

In Advance, the 3D model and the associated drawings are linked to each other; therefore, it is possible to create additional labels in the linked drawings using the model information.

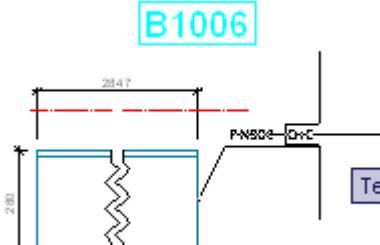
1. On the **Labels & Dimensions** tab, **Parametric labels** panel, click



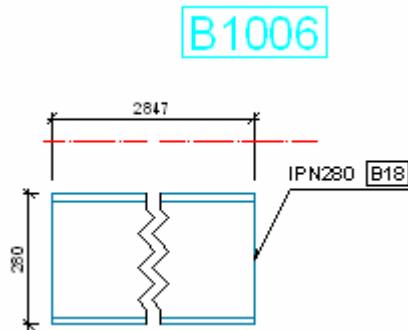
2. Select the detail on which to add a label.



3. Select the label start point.



4. Select the text start point.
 5. Define the text angle using the mouse pointer.
 6. Press **Enter** to finish.

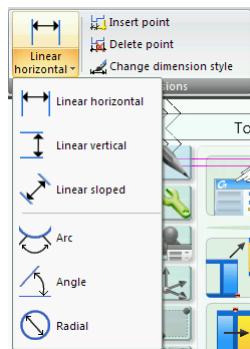


The label is placed.

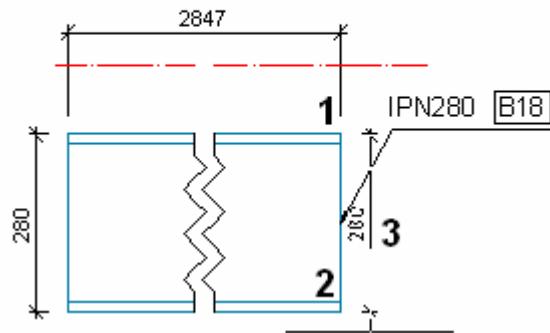
Inserting a linear vertical dimension

Insert a linear vertical dimension on the same detail.

1. On the **Labels & Dimensions** tab, **Parametric dimensions** panel, click .



2. Select the detail on which to add a label.
 3. Select the first dimension point.
 4. Select the second dimension point.

B1006

5. Select a point to define the dimension line position.
6. Press **Enter** to finish.

The linear vertical dimension is created.

7. Press **Esc** to close the command.

Close the dimension properties dialog box.

B1006

Figure 289: The dimension properties

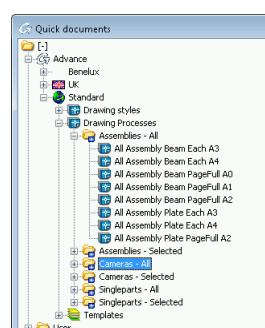
Step 3: Automatic detailing of all the main parts in the model

Using a process, create automatic drawings for all the main parts in the model.

Go back to the model DWG.



1. On the **Home** tab, **Document creation** panel, click **Quick documents**.
2. In the Quick documents window, select **All Assembly Beam PageFull A0**.



3. Click **Use**.
4. In the process properties dialog box, set the drawing number options for the drawings created by the process: the first number and the increment number.
5. From the **Selection** drop-down list, select the **All Mp** option.



Figure 290: Process properties dialog box

6. Click **OK**.

The linked details are created in the **Details** folder.

7. To see the details, from the Quick start toolbar, click **Open**.



8. From the **Details** folder, open the desired .dwg file, for example, the **003_A0-Beam.dwg** file.

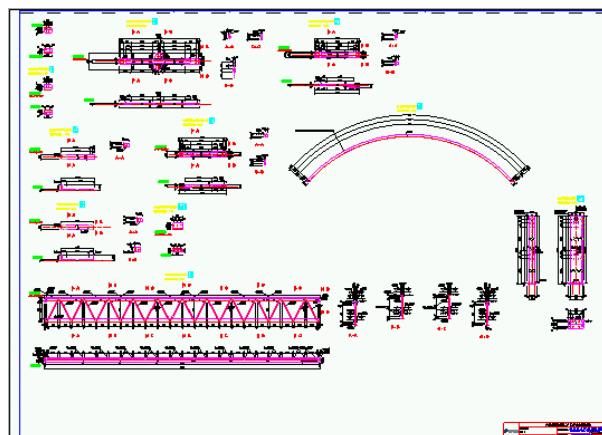
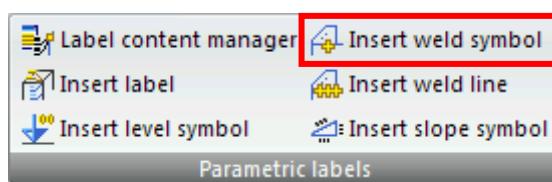


Figure 291: The 003_A0-Detail.dwg file

Inserting a weld symbol

In Advance, the 3D model and the associated drawings are linked to each other; therefore, it is possible to create additional weld symbols in the linked drawings using the model information.

1. On the **Labels & Dimensions** tab, **Parametric labels** panel, click .



2. Select the detail on which to add the weld symbol. For example, select the detail of a beam
3. Define the symbol reference point on the detail.

4. Define the symbol position point.

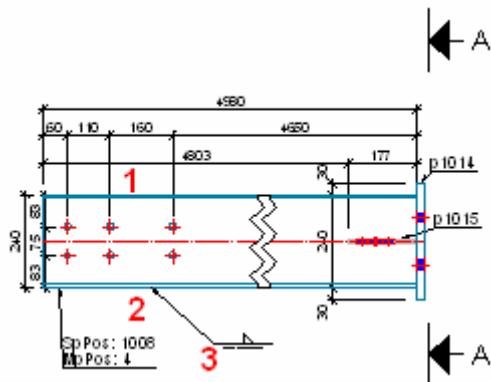


Figure 292: Inserting a weld symbol

The weld symbol is created.

5. In the weld symbol properties dialog box, define the following settings:

- On the **Upper weld** tab, define the weld thickness: **10**.

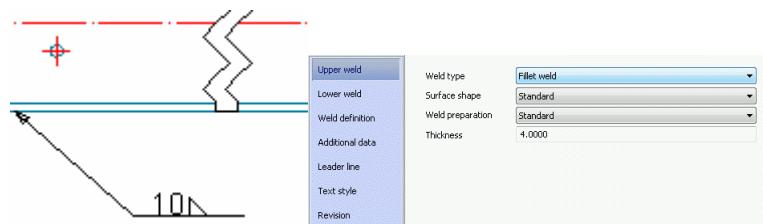


Figure 293: Defining the weld thickness

- On the **Lower weld** tab, from the **Weld type** drop-down list, make sure to select the **None** option.

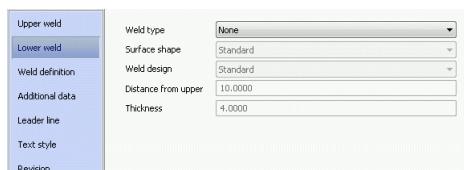


Figure 294: Weld properties – Lower weld tab

- On the **Weld definition** tab, from the **Location** drop-down list, make sure to select the **Shop** option.

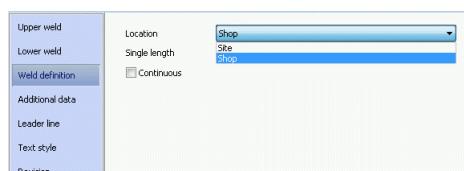
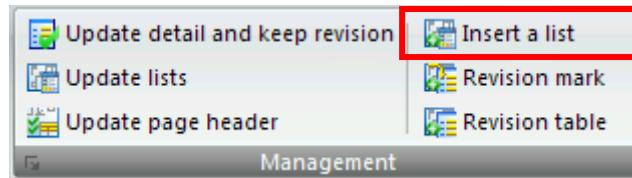


Figure 295: Weld properties – Weld definition tab

Step 4: Inserting a structured list in a drawing

Next, insert a structured list in the detail drawing.

- On the **Labels & Dimensions** tab, **Management** panel, click .



- Select a point for the upper right point of the list frame.
- Select the second point for the lower left point of the list frame.
- In the list properties dialog box, make the following settings:
 - On the **Layout** tab, do not define fixed points to specify a specific width and height of the list.

Note: This way, the list is not limited from extending as more material is added.

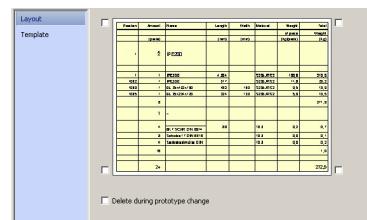


Figure 296: List properties dialog box

- On the **Templates** tab, from the **Advance Templates** group, select the **Drawing – Structured list** layout.

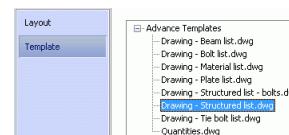
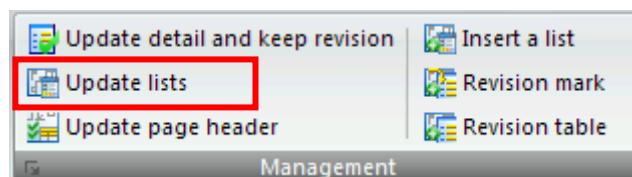


Figure 297: List properties dialog box – **Template** tab

- Close the dialog box.

The structured list template is added on the drawing. The list must be updated using the information from the model.

- On the **Labels & Dimensions** tab, **Management** panel, click to update the list.



The structured list is automatically updated.

Index	Component	Length	Weight	Unit weight
1	Flange bolt 10.9	1800	1.25	0.6
2	Flange bolt 10.9	1800	1.25	0.6
3	Flange bolt 10.9	1800	1.25	0.6
4	Flange bolt 10.9	1800	1.25	0.6
5	Flange bolt 10.9	1800	1.25	0.6
6	Flange bolt 10.9	1800	1.25	0.6
7	Flange bolt 10.9	1800	1.25	0.6
8	Flange bolt 10.9	1800	1.25	0.6
9	Flange bolt 10.9	1800	1.25	0.6
10	Flange bolt 10.9	1800	1.25	0.6
11	Flange bolt 10.9	1800	1.25	0.6
12	Flange bolt 10.9	1800	1.25	0.6
13	Flange bolt 10.9	1800	1.25	0.6
14	Flange bolt 10.9	1800	1.25	0.6
15	Flange bolt 10.9	1800	1.25	0.6
16	Flange bolt 10.9	1800	1.25	0.6
17	Flange bolt 10.9	1800	1.25	0.6
18	Flange bolt 10.9	1800	1.25	0.6
19	Flange bolt 10.9	1800	1.25	0.6
20	Flange bolt 10.9	1800	1.25	0.6
21	Flange bolt 10.9	1800	1.25	0.6
22	Flange bolt 10.9	1800	1.25	0.6
23	Flange bolt 10.9	1800	1.25	0.6
24	Flange bolt 10.9	1800	1.25	0.6
25	Flange bolt 10.9	1800	1.25	0.6
26	Flange bolt 10.9	1800	1.25	0.6
27	Flange bolt 10.9	1800	1.25	0.6
28	Flange bolt 10.9	1800	1.25	0.6
29	Flange bolt 10.9	1800	1.25	0.6
30	Flange bolt 10.9	1800	1.25	0.6
31	Flange bolt 10.9	1800	1.25	0.6
32	Flange bolt 10.9	1800	1.25	0.6
33	Flange bolt 10.9	1800	1.25	0.6
34	Flange bolt 10.9	1800	1.25	0.6
35	Flange bolt 10.9	1800	1.25	0.6
36	Flange bolt 10.9	1800	1.25	0.6
37	Flange bolt 10.9	1800	1.25	0.6
38	Flange bolt 10.9	1800	1.25	0.6
39	Flange bolt 10.9	1800	1.25	0.6
40	Flange bolt 10.9	1800	1.25	0.6
41	Flange bolt 10.9	1800	1.25	0.6
42	Flange bolt 10.9	1800	1.25	0.6
43	Flange bolt 10.9	1800	1.25	0.6
44	Flange bolt 10.9	1800	1.25	0.6
45	Flange bolt 10.9	1800	1.25	0.6
46	Flange bolt 10.9	1800	1.25	0.6
47	Flange bolt 10.9	1800	1.25	0.6
48	Flange bolt 10.9	1800	1.25	0.6
49	Flange bolt 10.9	1800	1.25	0.6
50	Flange bolt 10.9	1800	1.25	0.6
51	Flange bolt 10.9	1800	1.25	0.6
52	Flange bolt 10.9	1800	1.25	0.6
53	Flange bolt 10.9	1800	1.25	0.6
54	Flange bolt 10.9	1800	1.25	0.6
55	Flange bolt 10.9	1800	1.25	0.6
56	Flange bolt 10.9	1800	1.25	0.6
57	Flange bolt 10.9	1800	1.25	0.6
58	Flange bolt 10.9	1800	1.25	0.6
59	Flange bolt 10.9	1800	1.25	0.6
60	Flange bolt 10.9	1800	1.25	0.6
61	Flange bolt 10.9	1800	1.25	0.6
62	Flange bolt 10.9	1800	1.25	0.6
63	Flange bolt 10.9	1800	1.25	0.6
64	Flange bolt 10.9	1800	1.25	0.6
65	Flange bolt 10.9	1800	1.25	0.6
66	Flange bolt 10.9	1800	1.25	0.6
67	Flange bolt 10.9	1800	1.25	0.6
68	Flange bolt 10.9	1800	1.25	0.6
69	Flange bolt 10.9	1800	1.25	0.6
70	Flange bolt 10.9	1800	1.25	0.6
71	Flange bolt 10.9	1800	1.25	0.6
72	Flange bolt 10.9	1800	1.25	0.6
73	Flange bolt 10.9	1800	1.25	0.6
74	Flange bolt 10.9	1800	1.25	0.6
75	Flange bolt 10.9	1800	1.25	0.6
76	Flange bolt 10.9	1800	1.25	0.6
77	Flange bolt 10.9	1800	1.25	0.6
78	Flange bolt 10.9	1800	1.25	0.6
79	Flange bolt 10.9	1800	1.25	0.6
80	Flange bolt 10.9	1800	1.25	0.6
81	Flange bolt 10.9	1800	1.25	0.6
82	Flange bolt 10.9	1800	1.25	0.6
83	Flange bolt 10.9	1800	1.25	0.6
84	Flange bolt 10.9	1800	1.25	0.6
85	Flange bolt 10.9	1800	1.25	0.6
86	Flange bolt 10.9	1800	1.25	0.6
87	Flange bolt 10.9	1800	1.25	0.6
88	Flange bolt 10.9	1800	1.25	0.6
89	Flange bolt 10.9	1800	1.25	0.6
90	Flange bolt 10.9	1800	1.25	0.6
91	Flange bolt 10.9	1800	1.25	0.6
92	Flange bolt 10.9	1800	1.25	0.6
93	Flange bolt 10.9	1800	1.25	0.6
94	Flange bolt 10.9	1800	1.25	0.6
95	Flange bolt 10.9	1800	1.25	0.6
96	Flange bolt 10.9	1800	1.25	0.6
97	Flange bolt 10.9	1800	1.25	0.6
98	Flange bolt 10.9	1800	1.25	0.6
99	Flange bolt 10.9	1800	1.25	0.6
100	Flange bolt 10.9	1800	1.25	0.6
101	Flange bolt 10.9	1800	1.25	0.6
102	Flange bolt 10.9	1800	1.25	0.6
103	Flange bolt 10.9	1800	1.25	0.6
104	Flange bolt 10.9	1800	1.25	0.6
105	Flange bolt 10.9	1800	1.25	0.6
106	Flange bolt 10.9	1800	1.25	0.6
107	Flange bolt 10.9	1800	1.25	0.6
108	Flange bolt 10.9	1800	1.25	0.6
109	Flange bolt 10.9	1800	1.25	0.6
110	Flange bolt 10.9	1800	1.25	0.6
111	Flange bolt 10.9	1800	1.25	0.6
112	Flange bolt 10.9	1800	1.25	0.6
113	Flange bolt 10.9	1800	1.25	0.6
114	Flange bolt 10.9	1800	1.25	0.6
115	Flange bolt 10.9	1800	1.25	0.6
116	Flange bolt 10.9	1800	1.25	0.6
117	Flange bolt 10.9	1800	1.25	0.6
118	Flange bolt 10.9	1800	1.25	0.6
119	Flange bolt 10.9	1800	1.25	0.6
120	Flange bolt 10.9	1800	1.25	0.6
121	Flange bolt 10.9	1800	1.25	0.6
122	Flange bolt 10.9	1800	1.25	0.6
123	Flange bolt 10.9	1800	1.25	0.6
124	Flange bolt 10.9	1800	1.25	0.6
125	Flange bolt 10.9	1800	1.25	0.6
126	Flange bolt 10.9	1800	1.25	0.6
127	Flange bolt 10.9	1800	1.25	0.6
128	Flange bolt 10.9	1800	1.25	0.6
129	Flange bolt 10.9	1800	1.25	0.6
130	Flange bolt 10.9	1800	1.25	0.6
131	Flange bolt 10.9	1800	1.25	0.6
132	Flange bolt 10.9	1800	1.25	0.6
133	Flange bolt 10.9	1800	1.25	0.6
134	Flange bolt 10.9	1800	1.25	0.6
135	Flange bolt 10.9	1800	1.25	0.6
136	Flange bolt 10.9	1800	1.25	0.6
137	Flange bolt 10.9	1800	1.25	0.6
138	Flange bolt 10.9	1800	1.25	0.6
139	Flange bolt 10.9	1800	1.25	0.6
140	Flange bolt 10.9	1800	1.25	0.6
141	Flange bolt 10.9	1800	1.25	0.6
142	Flange bolt 10.9	1800	1.25	0.6
143	Flange bolt 10.9	1800	1.25	0.6
144	Flange bolt 10.9	1800	1.25	0.6
145	Flange bolt 10.9	1800	1.25	0.6
146	Flange bolt 10.9	1800	1.25	0.6
147	Flange bolt 10.9	1800	1.25	0.6
148	Flange bolt 10.9	1800	1.25	0.6
149	Flange bolt 10.9	1800	1.25	0.6
150	Flange bolt 10.9	1800	1.25	0.6
151	Flange bolt 10.9	1800	1.25	0.6
152	Flange bolt 10.9	1800	1.25	0.6
153	Flange bolt 10.9	1800	1.25	0.6
154	Flange bolt 10.9	1800	1.25	0.6
155	Flange bolt 10.9	1800	1.25	0.6
156	Flange bolt 10.9	1800	1.25	0.6
157	Flange bolt 10.9	1800	1.25	0.6
158	Flange bolt 10.9	1800	1.25	0.6
159	Flange bolt 10.9	1800	1.25	0.6
160	Flange bolt 10.9	1800	1.25	0.6
161	Flange bolt 10.9	1800	1.25	0.6
162	Flange bolt 10.9	1800	1.25	0.6
163	Flange bolt 10.9	1800	1.25	0.6
164	Flange bolt 10.9	1800	1.25	0.6
165	Flange bolt 10.9	1800	1.25	0.6
166	Flange bolt 10.9	1800	1.25	0.6
167	Flange bolt 10.9	1800	1.25	0.6
168	Flange bolt 10.9	1800	1.25	0.6
169	Flange bolt 10.9	1800	1.25	0.6
170	Flange bolt 10.9	1800	1.25	0.6
171	Flange bolt 10.9	1800	1.25	0.6
172	Flange bolt 10.9	1800	1.25	0.6
173	Flange bolt 10.9	1800	1.25	0.6
174	Flange bolt 10.9	1800	1.25	0.6
175	Flange bolt 10.9	1800	1.25	0.6
176	Flange bolt 10.9	1800	1.25	0.6
177	Flange bolt 10.9	1800	1.25	0.6
178	Flange bolt 10.9	1800	1.25	0.6
179	Flange bolt 10.9	1800	1.25	0.6
180	Flange bolt 10.9	1800	1.25	0.6
181	Flange bolt 10.9	1800	1.25	0.6
182	Flange bolt 10.9	1800	1.25	0.6
183	Flange bolt 10.9	1800	1.25	0.6
184	Flange bolt 10.9	1800	1.25	0.6
185	Flange bolt 10.9	1800	1.25	0.6
186	Flange bolt 10.9	1800	1.25	0.6
187	Flange bolt 10.9	1800	1.25	0.6
188	Flange bolt 10.9	1800	1.25	0.6
189	Flange bolt 10.9	1800	1.25	0.6
190	Flange bolt 10.9	1800	1.25	0.6
191	Flange bolt 10.9	1800	1.25	0.6
192	Flange bolt 10.9	1800	1.25	0.6
193	Flange bolt 10.9	1800	1.25	0.6
194	Flange bolt 10.9	1800	1.25	0.6
195	Flange bolt 10.9	1800	1.25	0.6
196	Flange bolt 10.9	1800	1.25	

Lesson 13: Generating details using cameras

In this lesson you will learn how to:

- Insert level symbols in the model
- Create details generated by automatic detailing processes using cameras
- Modify level symbols properties in drawings
- Insert level symbols in drawings

Step 1: Inserting a level symbol in the model

In this step, create a level symbol on top of the base plate of the **G6** column.

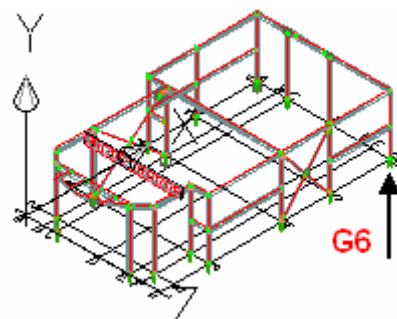
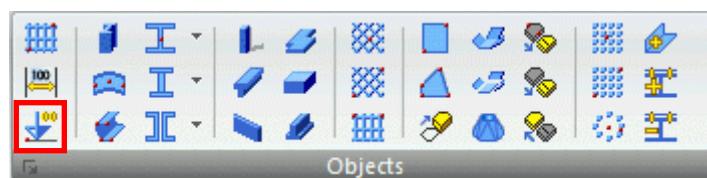


Figure 299: The **G6** column

1. On the **Home** tab, **Objects** panel, click .



2. Select the insertion point on top of the base plate from the **G6** base plate joint, as in Figure 300.

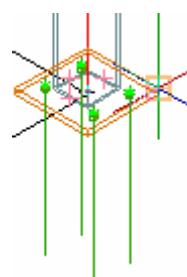


Figure 300: Level symbol insertion point

3. In the level symbol properties dialog box, on the **Default** tab, define the symbol size: **200**.



Figure 301: Properties dialog box - Defining the symbol size

Note: On the **Current** tab, notice the height notations in reference to a zero level (absolute) and to the WCS (relative): **20**.



Figure 302: Properties dialog box – **Current** tab

The level symbol is created.

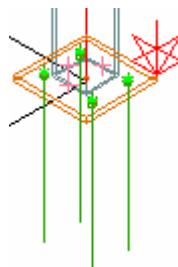


Figure 303: The level symbol

Using the same process, create level symbols on top of the **G5-G6** middle beam and on top of the **G6** column.

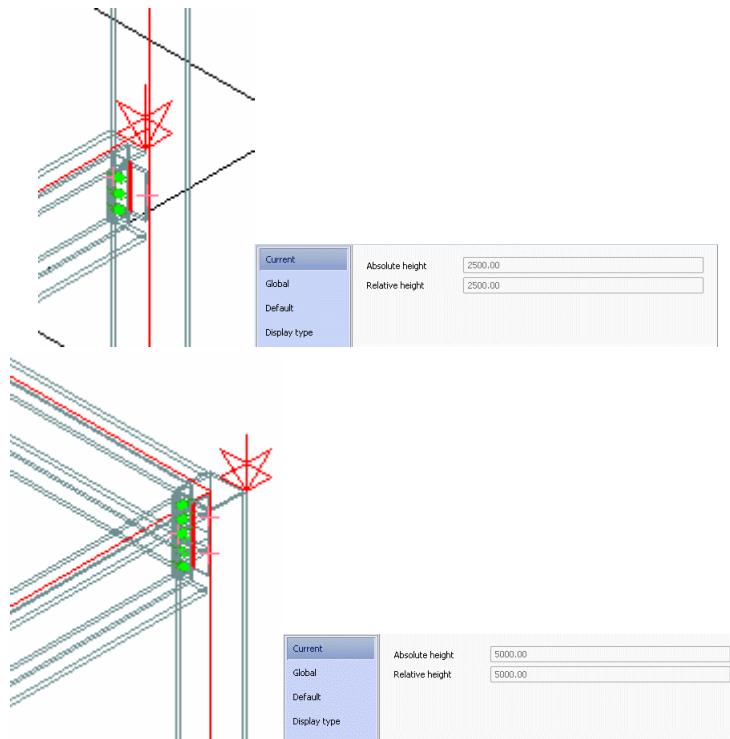


Figure 304: Level symbol properties

Step 2: Creating details using cameras

The cameras can be used in processes for automatic creation of details.

In this step, create detail drawings from the cameras created in Lesson 10: Create cameras.

Before starting

Turn on the **Cameras** layer to display the cameras.

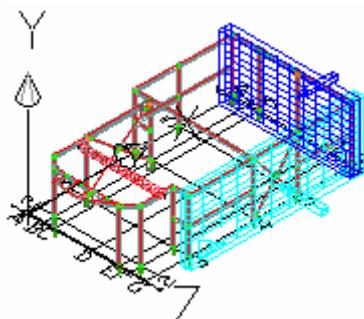
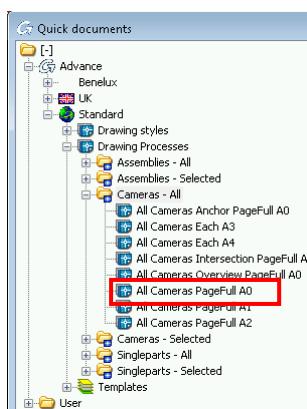


Figure 305: The model and the created cameras

1. On the **Home** tab, **Document creation** panel, click  **Quick documents**.
2. In the Quick documents window, select **All Cameras PageFull A0**.



3. Click **Use**.
4. In the process properties dialog box, from the **Selection** drop-down list, select **All Cameras**.

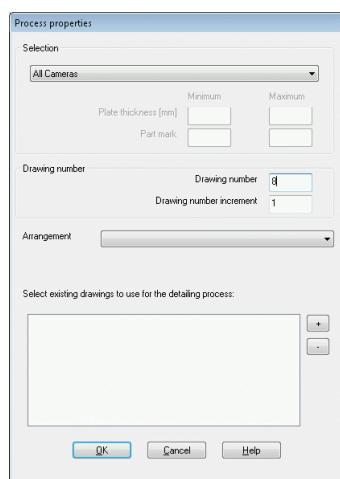


Figure 306: Process properties dialog box

5. Click **OK**.

The linked details are created in the **Details** folder.

6. To see the details, from the Quick start toolbar, click **Open**.



7. From the **Details** folder, open the **008_A0-Camera.dwg** file.

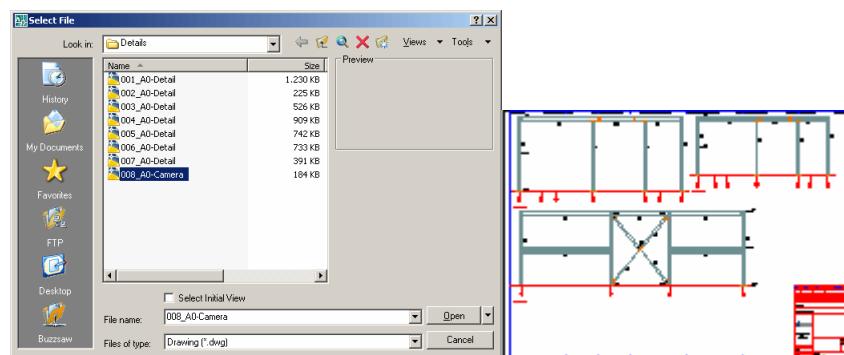


Figure 307: Selecting the dwg to display

Note: Notice that the level symbols are represented in the drawing!

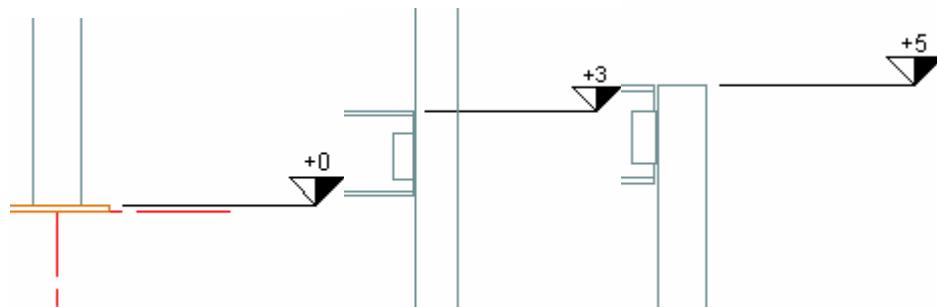
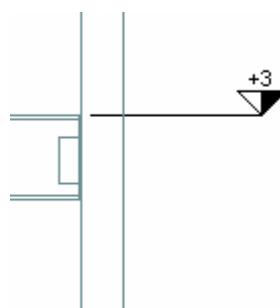


Figure 308: Level symbols

Modifying level symbol representations

You can modify the representation of the level symbols directly on the drawing.



1. Select a level symbol.
2. Right click and select **Advance Properties** from the context menu.
3. In the properties dialog box, on the **Presentation mode for Level Symbols** tab, make the following settings:
 - From the **Unit** drop-down list, select **Millimeter** to display the height in millimeters.
 - From the **Precision** drop-down list, select **0.000**.

The value in the level symbol is displayed with 10^{-3} precision and is converted to millimeters.

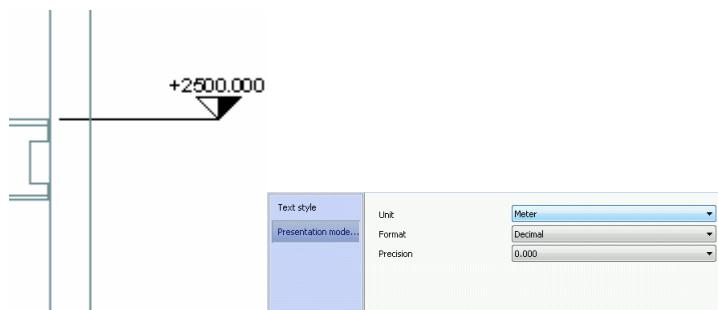


Figure 309: Modifying the value display

Step 3: Inserting a level symbol in the drawing

In Advance, the 3D model and the associated drawings are linked to each other; therefore, it is possible to create additional level symbols in the linked drawings using the model information.

Next, insert a level symbol in the drawing.

1. On the **Labels & Dimensions** tab, **Parametric labels** panel, click .



2. Select the detail on which to add a level symbol.

For example, select the detail shown in the Figure 310.

3. Define the symbol reference point at the detail.
4. Define the symbol position point.

The level symbol is created.

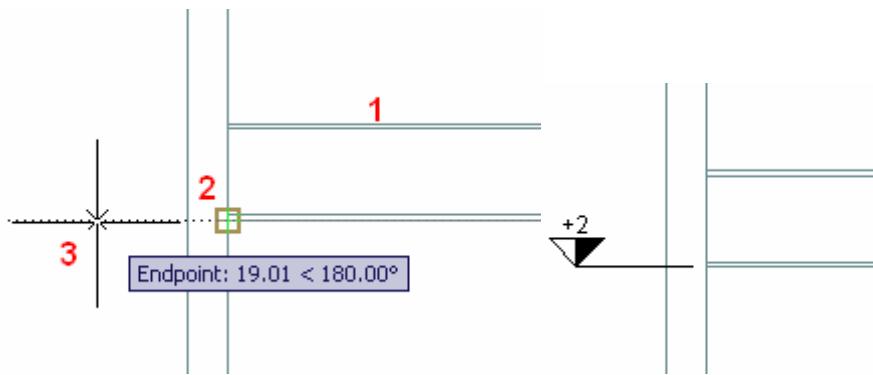


Figure 310: Inserting a level symbol in a detail

Lesson 14: Exploding details

In this lesson you will learn how to:

- Explode a detail
- Batch explode for all selected drawings

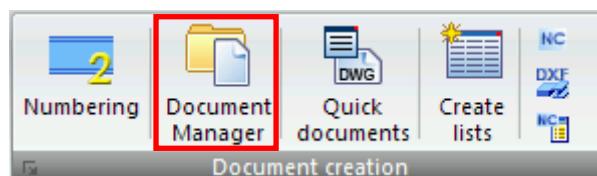
After exploding the details, the elements of the details are seen as individual Advance components and can be adjusted separately.

Before starting

Go back to the model DWG and turn off the **Cameras** layer.

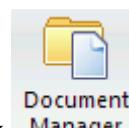
Step 1: Adding to explode details

To explode the details, open the **Document Manager** from the **Document creation** panel.

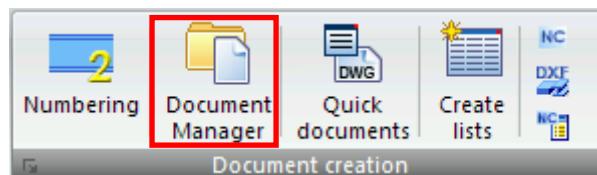


The **Document Manager** manages and presents the connection between the model and the linked drawings: you can display, update, add a revision, flag the issue of drawings, add to batch plot or batch explode, and delete the drawing details.

In this step, add to explode the drawing details.



1. On the **Home** tab, **Document creation** panel, click



Note: The **Document Manager** checks automatically if drawings require an update due to model modifications or you can select the drawings to be checked. Drawing updates are done directly from the **Document Manager**.

The “Document Manager” window appears.

2. From the **Project documents** tree, **Details** group, select the **Current** sub-group.
3. Click **Add to explode**.

The drawings are copied to a new category named **Batch Explode**.

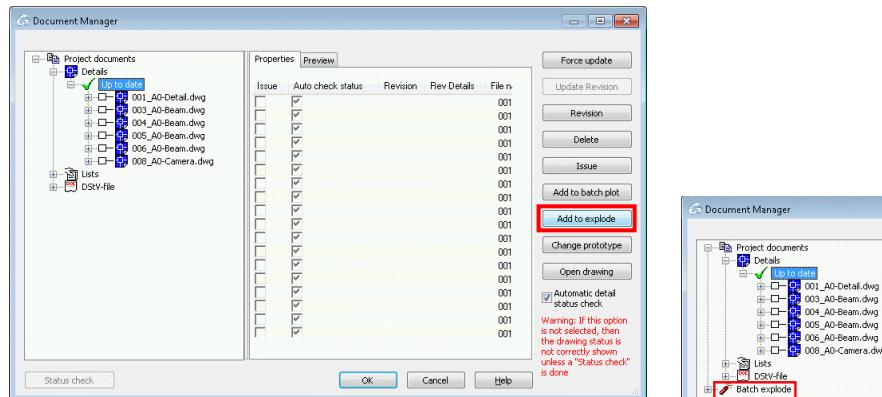


Figure 311: Adding details to Batch explode

4. Select the **Batch explode** tree.



5. On the **Properties** tab, click

The “Detail Explode” window appears.

6. Select the **Enable Detail Explode** option to activate the detail explode settings.

7. Select the **Advance** profile as the current profile.

Note: In the **Advance** profile, expand the **Beam** group and select **Label**. Notice that the detail explode settings cannot be modified.

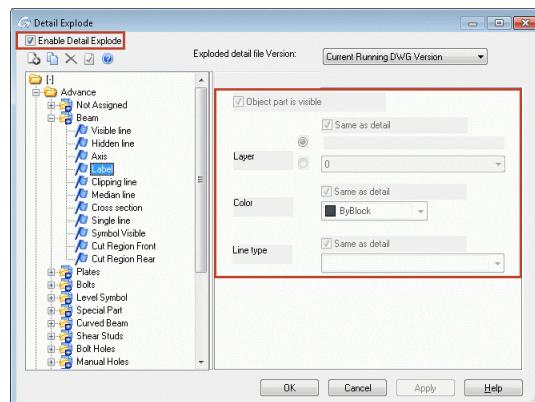


Figure 312: The detail explode dialog box

8. Click **OK** in the “Detail Explode” window.

9. Click **OK** in the “Document Manager” window.

10. Click **OK** in the “Documents to print or delete or explode” dialog box.

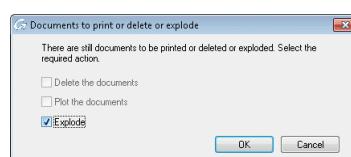


Figure 313: Confirming detail explode

The details are exploded.

A new folder named **DetailsExploded** is created at the same location as the **Details** folder. The linked exploded details appear in different .dwg files in the **DetailsExploded** folder.

- To open the exploded details, from the Quick start toolbar, click **Open**.



- From the **DetailsExploded** folder, select the .dwg file, for example **001_A0-Detail.dwg**.

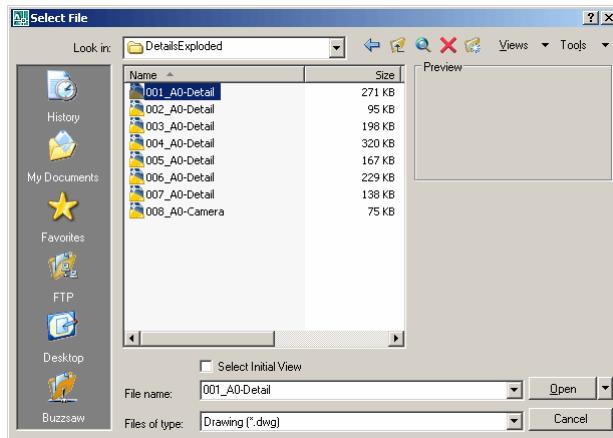


Figure 314: Selecting the dwg file

The elements of a detail are now simple entities and can be adjusted separately.

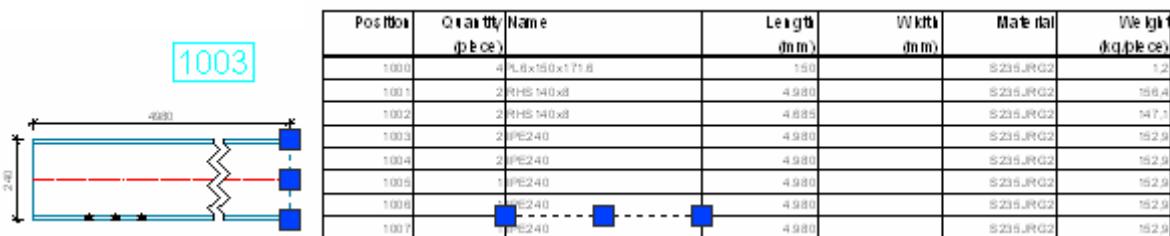


Figure 315: Exploded detail

Lesson 15: Creating external lists

All model objects including their graphic and non-graphic properties, features, joints and connections are stored and managed by Advance. Extracts are created from the numbered and saved models. The Advance List Template Wizard creates structured BOMs (Bills of materials) from the extracts. The created BOMs can be saved, printed, or exported in various formats.

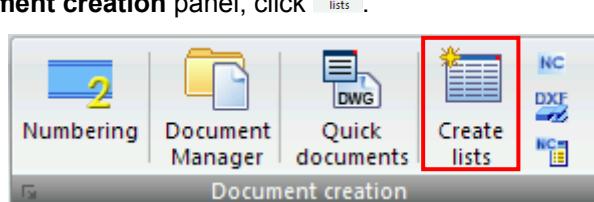
In this lesson you will learn how to:

- Create a model extract
- Create a list
- Export a list

Step 1: Creating a model extract

In this step, a model extract is created based on the model information. Later on, the Advance List Template Wizard creates structured BOMs from the extracts.

1. On the **Home** tab, **Document creation** panel, click  .



3. In the “Selection of model objects for the list” dialog box, select the **Complete Model** option to use the complete model for the list.

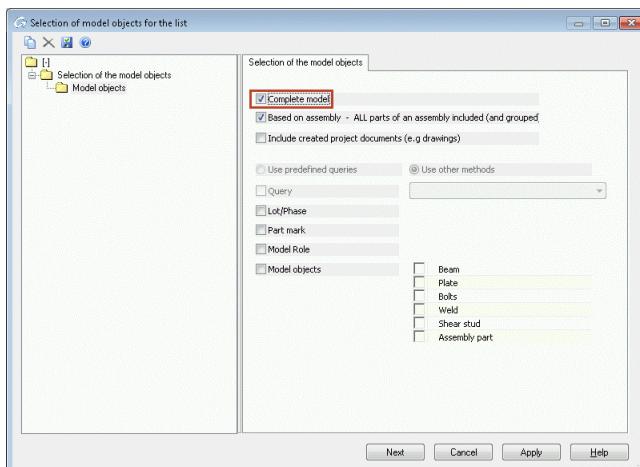


Figure 316: Selecting the complete model

4. Click **Apply**.
5. Click **Next**.
6. In the “Select destination file” dialog box, click **Create lists** to start creating and printing the information list.

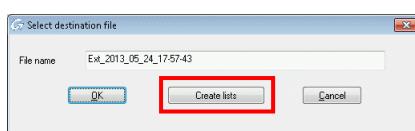


Figure 317: “Select destination file” dialog box

Note: For creating only the extract, click **OK**.

The BOM Editor opens in a new window.

Step 2: Creating a list

In this step, create a list based on the model extract.

- From the Advance Template tree, **Lists** group, **Part List** sub-group, select the **Material list** template.
- Click **Use**.

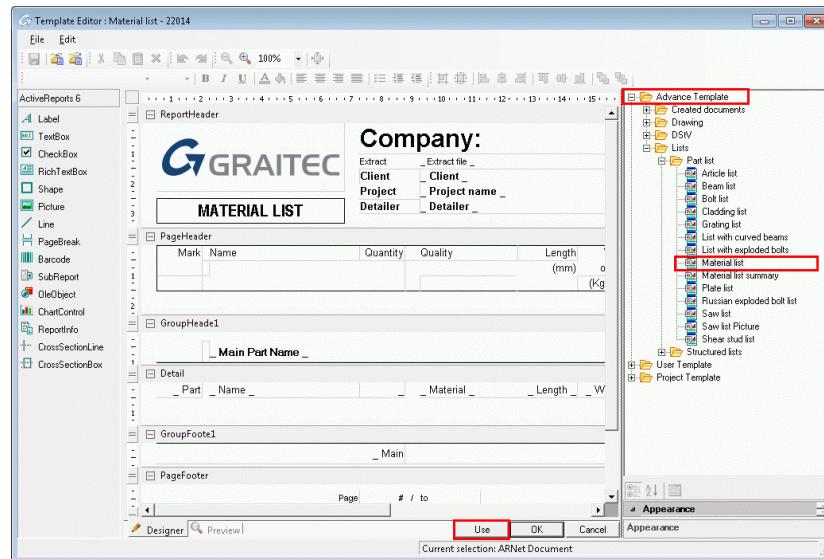


Figure 318: Template editor

- In the “Select the model extract for the BOM” dialog box, select the desired .xml file.

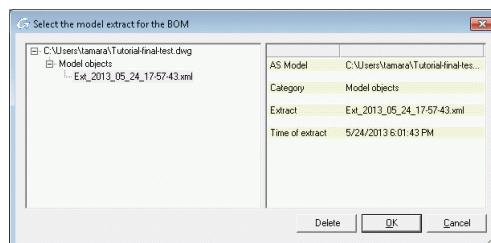


Figure 319: Selecting the model extract

- Click **OK**.

The list containing all the elements of the model appears in a new window.

Item	Description	Quantity	Length (mm)	Weight (kg)	Total length (mm)	Total weight (kg)	Total Surface (m²)
1000 PE200	PE200	4	525x450	190	12	4.8	0.600
1001 PE20	PE20	2	525x450	4,980	156.4	212.7	2.619
1002 RH50x8L	RH50x8L	2	525x450	4,985	147.1	294.2	2.614
1003 PE200	PE200	2	525x450	4,980	152.9	306.8	4.892
1004 PE20	PE20	2	525x450	4,980	152.9	306.8	4.892
1005 PE200	PE200	2	525x450	4,980	152.9	306.8	4.892
1006 PE20	PE20	1	525x450	4,980	152.9	153.9	4.890
1007 PE200	PE200	1	525x450	4,980	152.9	152.9	4.892
1008 PE20	PE20	1	525x450	4,980	152.9	152.9	4.892
1009 PE200	PE200	1	525x450	4,980	152.9	152.9	4.892
1010 PE20	PE20	1	525x450	4,980	152.9	152.9	4.892
1011 PE200	PE200	1	525x450	4,980	152.9	152.9	4.892
1012 U000206	U000206	1	525x450	0.000	43.3	1.872	0.000
1013 U000206	U000206	1	525x450	0.000	43.3	1.872	0.000
1014 U000206	U000206	1	525x450	0.000	43.3	1.872	0.000
1015 FL00010	FL00010	1	525x450	2,775	21.8	21.8	0.611
1016 FL00010	FL00010	2	525x450	2,660	20.9	53.6	0.606
1017 RH50x6x4	RH50x6x4	1	525x450	6,679	36.4	1,242	1.242
1018 RH50x6x4	RH50x6x4	1	525x450	3,118	17	158	1.08
1019 PE20	PE20	1	525x450	4,980	152.9	152.9	4.892

Figure 320: The list

Step 3: Exporting a list

In this step, export a list to PDF format.

1. On the **Material list** toolbar, click **Export**.
2. In the “Report export” dialog box, from the **Export Format** drop-down list, select **Portable Document Format (PDF)**.

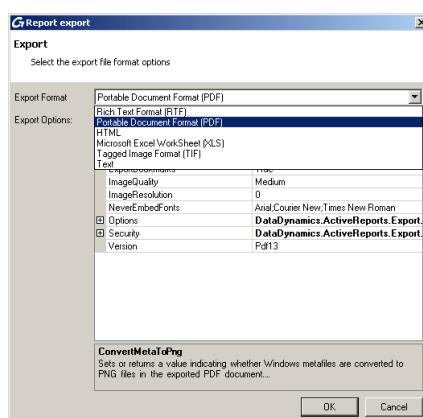


Figure 321: Exporting the list to pdf

3. Click **OK**.
4. In the “Save as” dialog box, enter the name of the created BOM file: **MaterialList**.

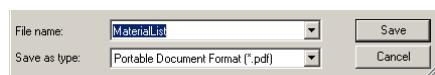


Figure 322: Naming the pdf file

5. Click **Save**.

The list is saved in PDF format.

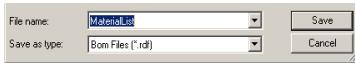
Note: The file is saved by default in the BOM folder corresponding to the model DWG.

6. Close the BOM Editor.

7. In the “Preview selected template” dialog box, click **Yes**.



8. In the “Save as” dialog box, enter the name of the report: **MaterialList**.



9. Click **Save**.

The report is saved in RDF format.

Note: The file is saved by default in the BOM folder corresponding to the model DWG.

10. In the “Template Editor” window, click **OK**.

The list is exported in PDF format.

ADVANCE
GRAITEC



www.graitec.com