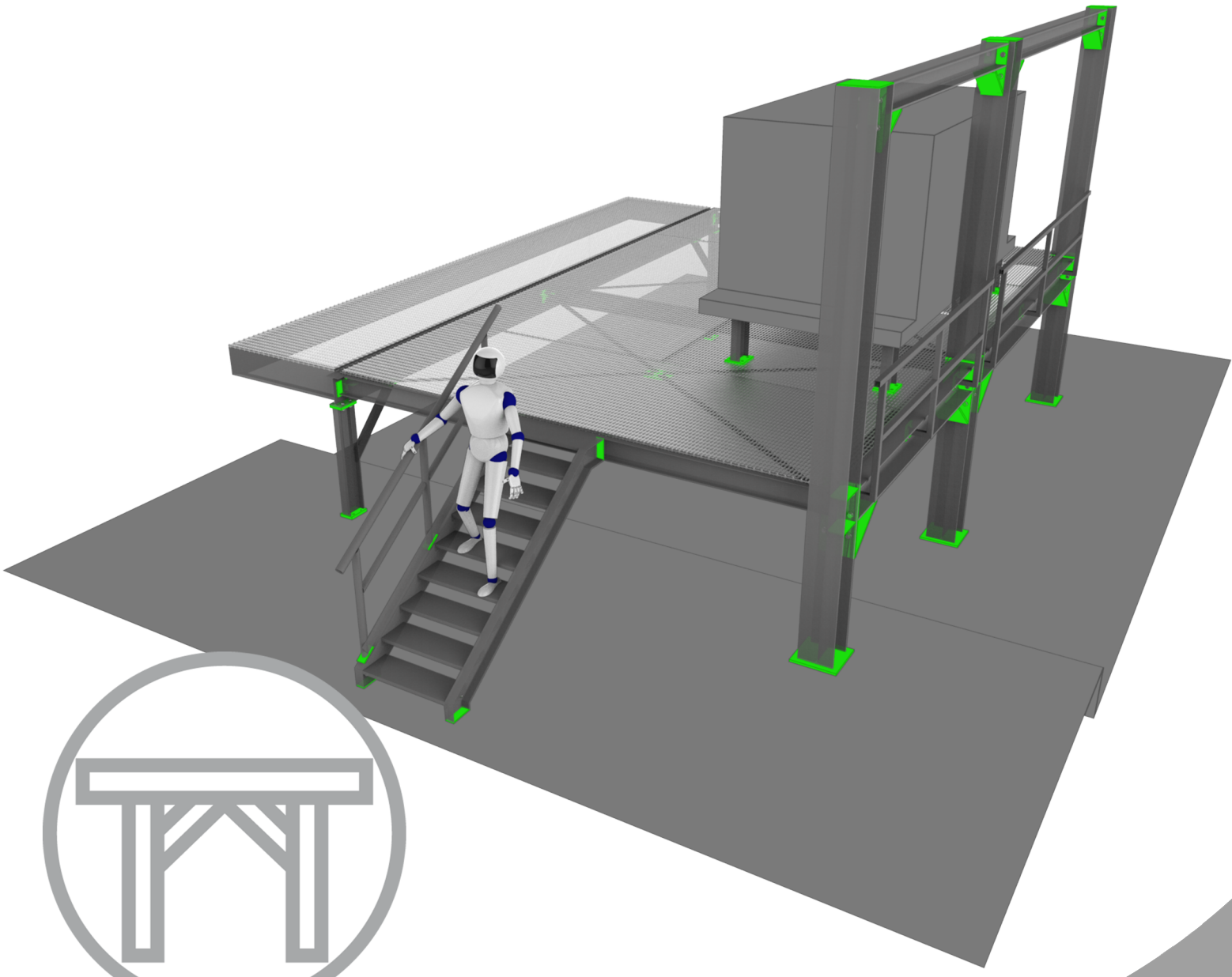
Advanced Framework Extension 4.0

Training Guide: Advanced Tutorial



Build date: 26.04.2017

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Introduction

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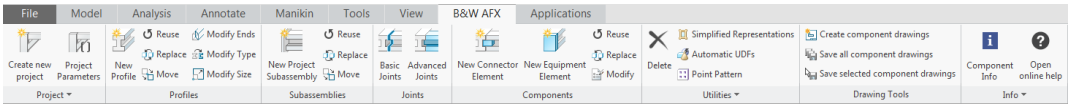
1.1 Objective of this document

The objective of this document is to enable you to create structural steel constructions with b&w **AFX** (formerly b&w EFX).

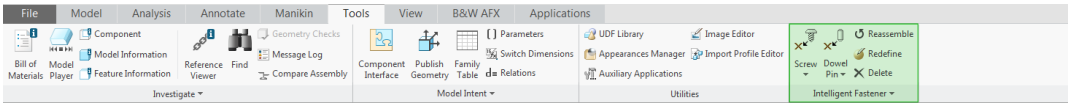
It is assumed that you are familiar with the basic functions of **Creo Parametric**.

1.2 Overview

The **Advanced Framework Extension 4.0** (**AFX**) is an add on software for **Creo Parametric**. It simplifies the design of machine frames, structural steel etc. built up with profiles or plates. All functions of **AFX** are accessible through the ribbon tab **Framework** in **Creo Parametric**.



For this tutorial you will have to use commands of the **Intelligent Fastener 4.0** Ribbon as well. You can find the required commands in the **TOOLS** Ribbon.



1.3 Syntax of this guide

Syntax	Description
Highlight	Important texts
[INPUT]	User input or selections
	Icons of the software

1.4 AFX Ribbon commands

Image	Description
	**Create new project**: Define new **AFX** assembly project
	**Project Parameters**: Project parameter commands
	Rename project: **Rename AFX project**
	Other commands of this group: Various Import and export functions (STAAD; SDNF)
	**New Profiles**: Profile commands
	**Basic Joints**: Basic profile joint commands
	**Advanced Joints**: Advanced profile joint commants
	**New Connector Elements**: Connector element commands

	**New Equipment Elements**: Equipment element commands
	**Project Subassemblies**: Contains functions for working with project subassemblies
	**Delete**: Delete **AFX** element
	**Simplified Representations**: Contains functions for defining weldment groups to structure smaller structural assemblies into different weldments
	**Weld Groups**: Weld group commands
	**Automatic UDFs**: Automatic-UDFs commands
	**Create component drawings**: Create drawings for subcomponents of current assembly
	**Save all component drawings**: Saves all component drawings in session
	**Save selected component drawings**: Save selected component drawings
	**Component Info**: Show component information
	Edit configuration session: Edit the configuration only for the current session
	Edit installation configuration: Edit the global **AFX** configuration
	About AFX: Show version information

1.5 IFX Ribbon commands

Image	Description
	**Assemble on point or axis**: Assemble a screw fastener on an existing point or axis
	**Assemble by mouse click**: Assemble a screw fastener on the selected mouse click position
	**Assemble on point or axis**: Assemble a dowel pin fastener on an existing point or axis
	**Assemble by mouse click**: Assemble a dowel pin fastener on the selected mouse click position
	**Reassemble**: Reassemble an existing screw fastener
	**Redefine**: Redefine a screw fastener
	Delete: Delete a screw fastener
	Check Screw Fasteners: Check all screw fasteners of the current assembly
	Update holes: Update Screw Holes
	Suppress: Suppress all fasteners
	Resume: Resume all suppressed screw fasteners
	**Options**: Edit the configuration options of Intelligent Fastener
	Instance Creator: Create instances of the selected files (*.dat)

Notes:

2

Design Frames with Steel Profiles

Overview
Standardized Connections
Project specific connectors
Automatic UDF's
Review

2.1 Overview

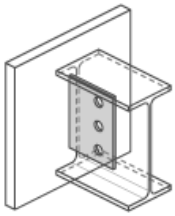
The library of **AFX** contains numerous non-standardized and standardized connectors for structural steel design. Their usage will be explained in the following chapter. Besides this, it will be explained how to create connections which are not contained in the library, by using the ones from the library, modifying them with regular **Creo Parametric** functionality and reusing them on other locations.

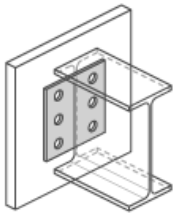
Open `STEEL_CONNECTORS_AND_WELDGROUPS.ASM` from the `connectors_and_weldgroups_start` folder of this tutorial.

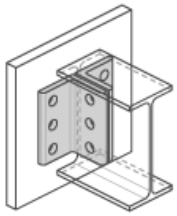
2.2 Standardized Connections

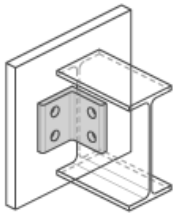
INFO: If you want to use this standardized connections you should be familiar with the corresponding **AISC** or **DAST** standards. If you don't know them try to use nonstandard Connections only! (Further information under www.aisc.org or www.deutscherstahlbau.de)

AISC Connections:

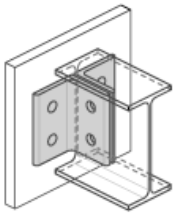

SINGLE SHEAR PLATE

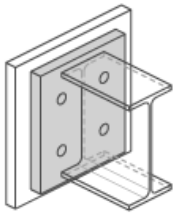

SHEAR END PLATE

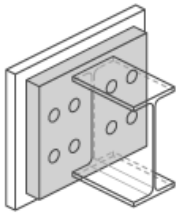

DOUBLE ANGLES

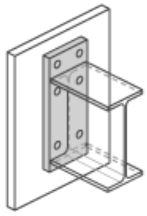

SINGLE ANGLE

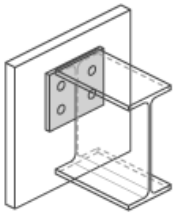
DAST Connections:


ANGLE CONNECTOR IW


END PLATE IH1


END PLATE IH2

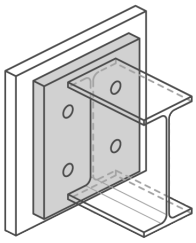

END PLATE IH3


END PLATE IS

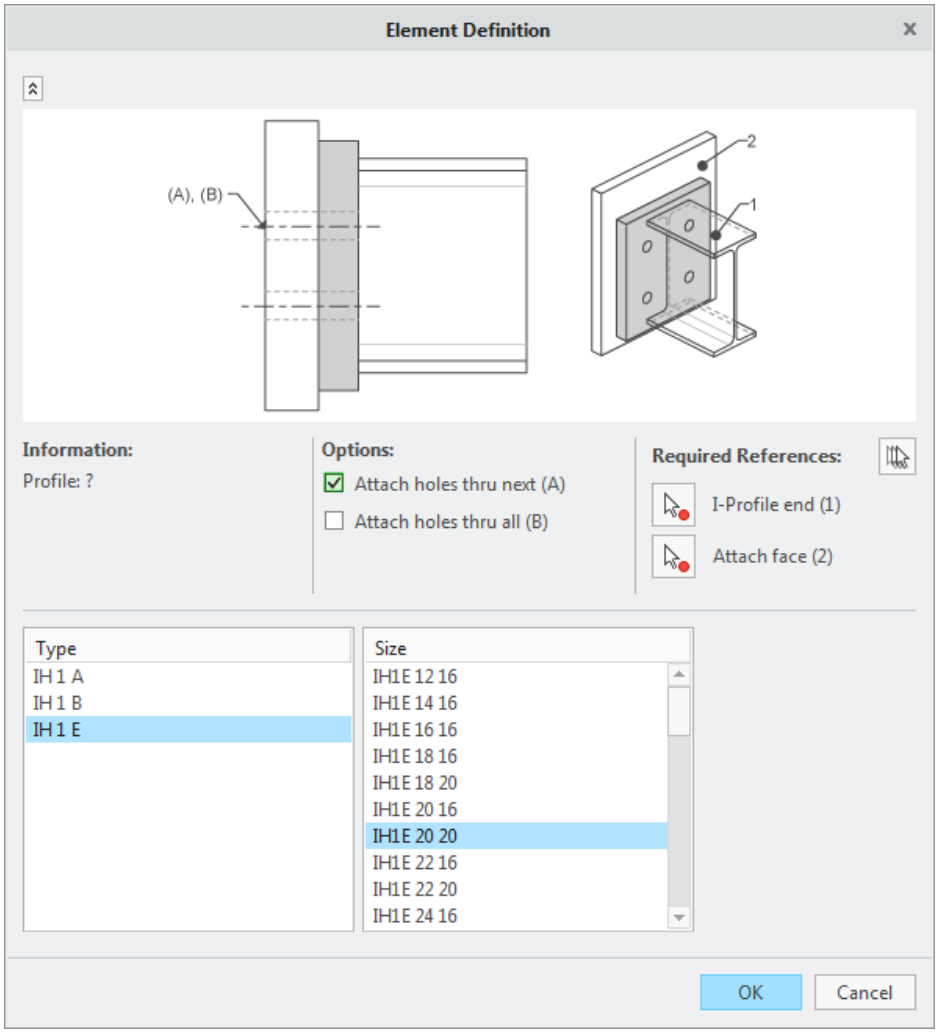
The AFX Library contains standardized connections. They are similar to the **DAST** (German steel design association) and **AISC** (American Institute for Steel Construction). Due to the standardized dimensions of plates and holes, standardized connections can only be used for I-Beams.

Assemble DAST end-plates

To assemble this type of connector proceed as follows.

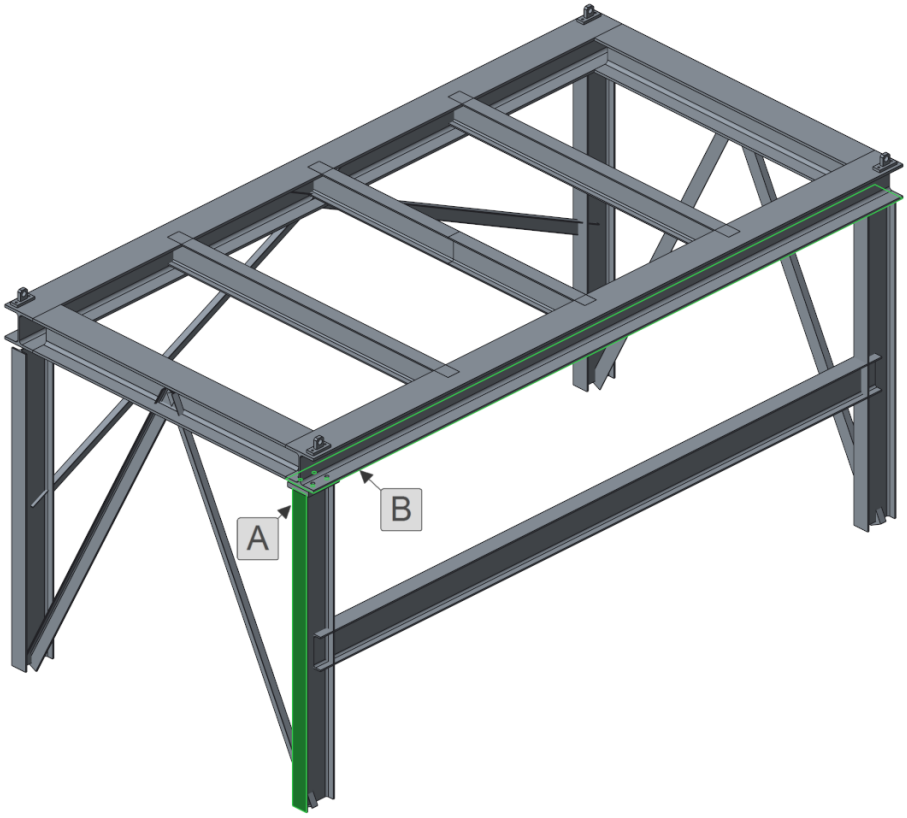


1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [DAST STANDARD] > [END PLATE IH1]** and press **[Next]**.
   - The Element definition dialog box of the end plate opens.



Hint 1 — Detection of sizes. If you select the references before choosing from the tables, the tables will be filtered to show only relevant sizes.

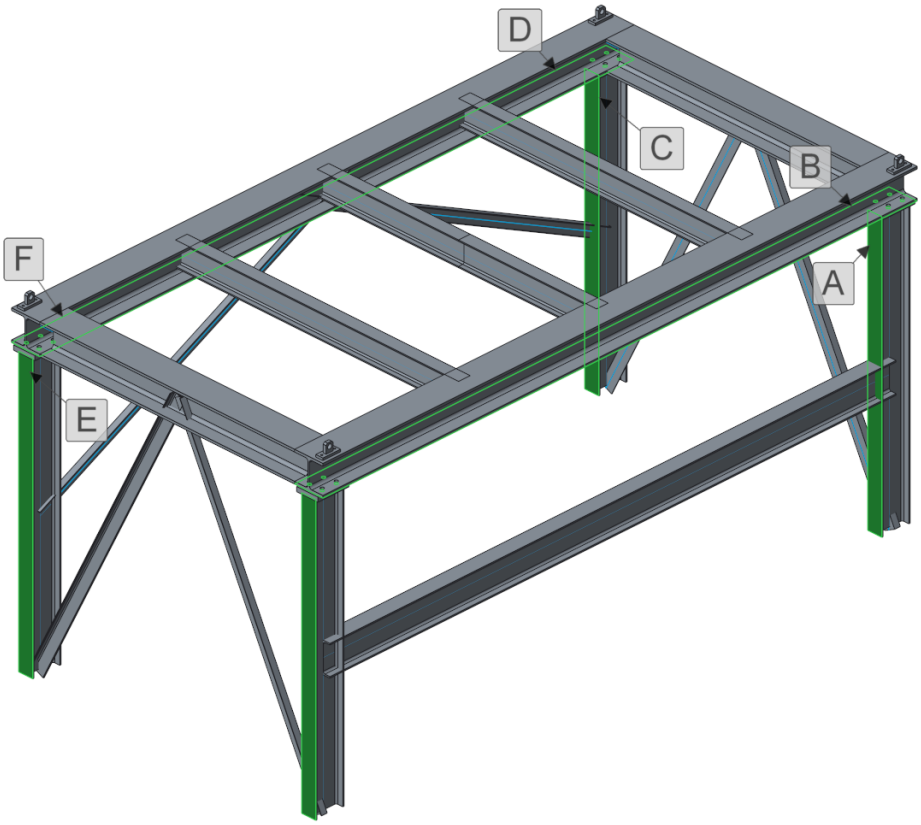
4. Select **[IH1E 20 20]** in the table and make sure to check **[attach holes thru next (A)]**.
5. Click **[I-Profile end (1)]**.
   - The dialog box closes.
6. Select the highlighted surface of the I-Beam near the profile end **[A]**.
   - The dialog box opens again.
7. Click **[Attach face (2)]** and select the bottom surface of the I-beam **[B]**.
8. Check option **[Attach holes thru next (A)]** to create holes in the attaching part.
9. Click **[OK]** to complete the definition.
   - The end-plate is assembled as new part, the holes are created in the upper profile and the vertical profile is adjusted in length.



The next step is reassemble the **end-plate** on the other three vertical I-beams.

1. Click **Reuse** to **reassemble an existing instance of a connector element**.
2. Select the previously assembled connector.

3. Press **Next** to reuse the end-plate.
   - You are now in a selection loop. It is possible to assemble all three end-plates in a row.
4. Select the new references **[A, B]**, **[C, D]**, **[E, F]**.
   - All four plates are now assembled, the required holes are created and the profiles are shortened to make the plates fit in.



Hint 2 — Reassemble preview image. Press in the **ribbon** to show the detailed image, that shows the required references. The required references will also be shown as text in the message area.

For the next exercise it is important to select the a plane on its correct side. For this purpose you should load the `syscol.scl` from the tutorial folder. You will then see the **front sides** of planes in **yellow**.

In the next step another end-plate will be assembled.

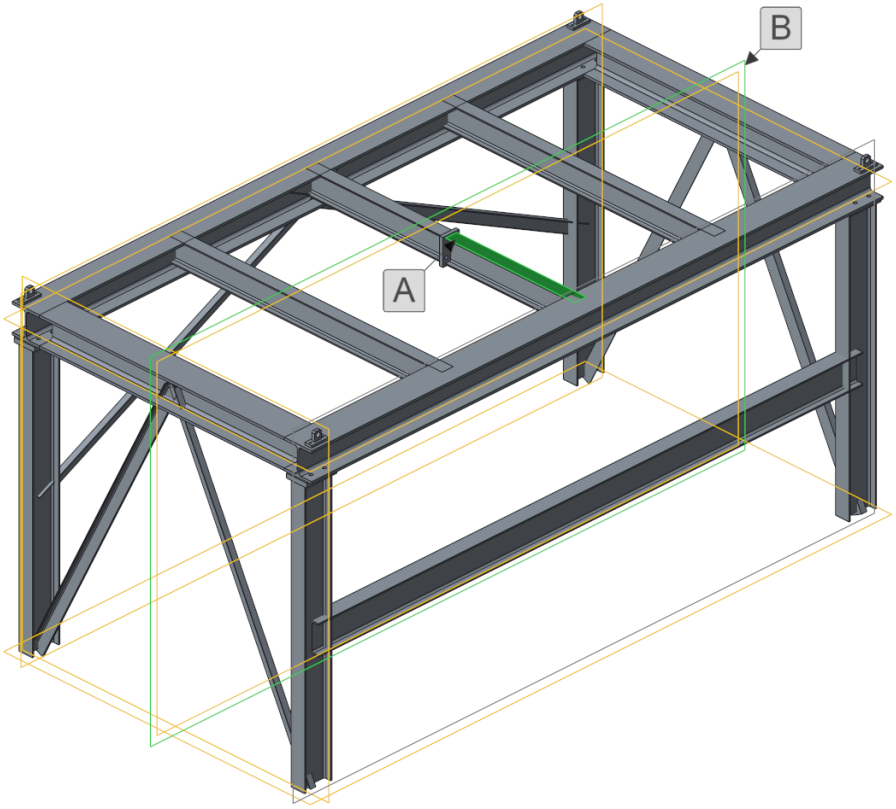
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [DAST STANDARD] > [END PLATE IH1]**.
   - The Element definition dialog box of the end plate opens.
4. Select **[IH1E] [16x16]** from the table.
5. Click **[I-Profile end (1)]** and select the surface **[A]** of the I-Beam near the profile end.

6. Click **[Attach face (2)]** and select the center plane A_YZ **[B]**.

   OR

7. Click **Select all required references** to select the two required references in a row.
8. Select the surface **[A]** of the I-Beam and the center plane A_YZ **[B]**.
9. Click **[OK]** to complete the definition.

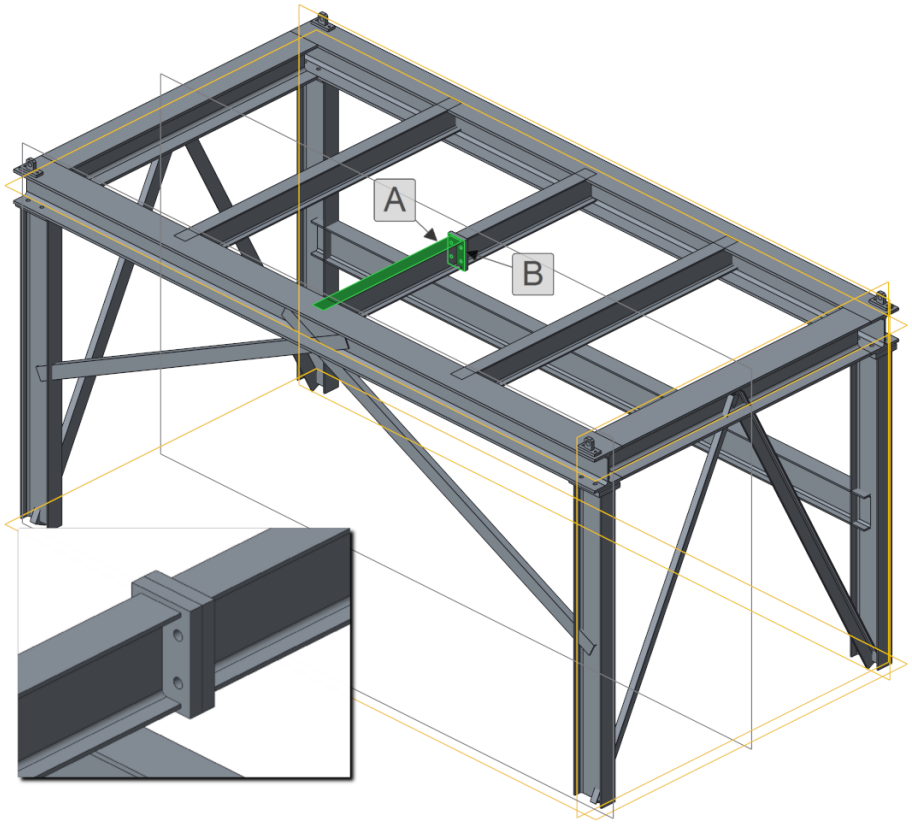
- The plate is assembled and the profile is adjusted in length.



To reassemble the end-plate on the other side proceed as follows

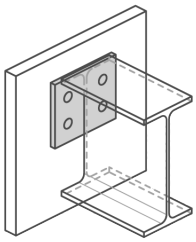
1. Select the previously assembled end-plate IH1E_16_16.PRT
2. Press the right mouse button and select **Framework > Components > Reuse** and then **Next**.
3. Select the surface **[A]** of the I-Beam.

4. Select the highlighted surface of the just assembled end plate **[B]**.
5. Click **[OK]** to complete the definition.
   - The second end-plate is assembled.

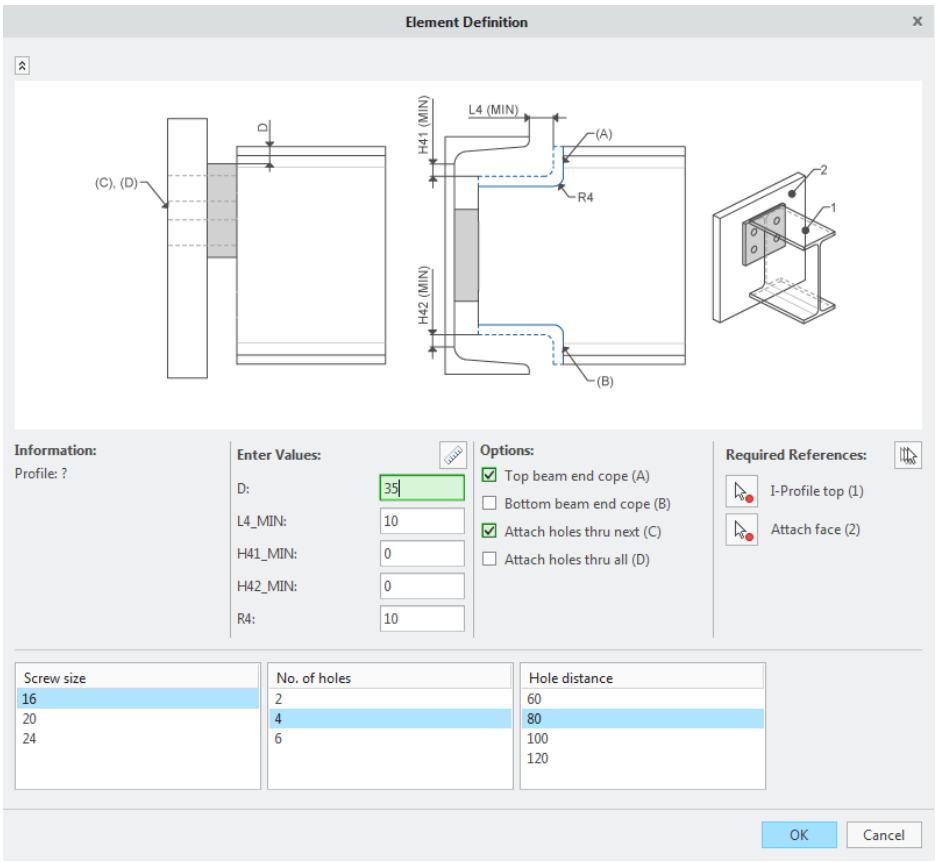


Assemble DAST shear end-plates

To assemble shear end-plates proceed as follows.



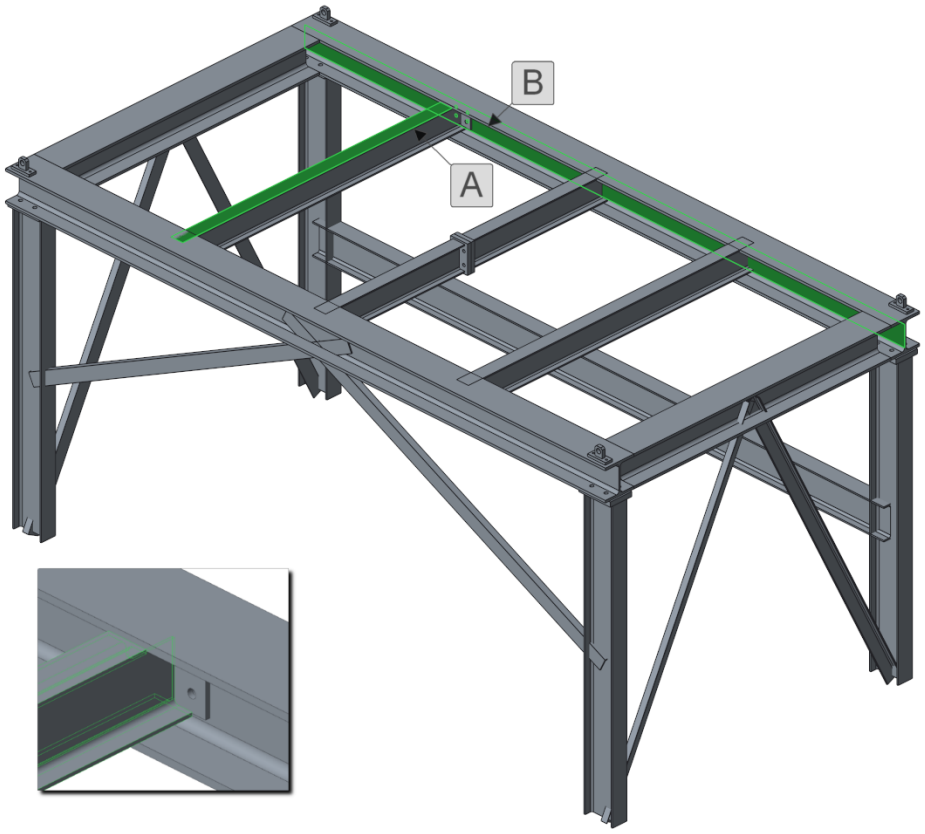
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [DAST STANDARD] > [END PLATE IS]** and then click **Next**.
   - The Element definition dialog box of the end plate opens.



Define the connector as shown in the dialog above.

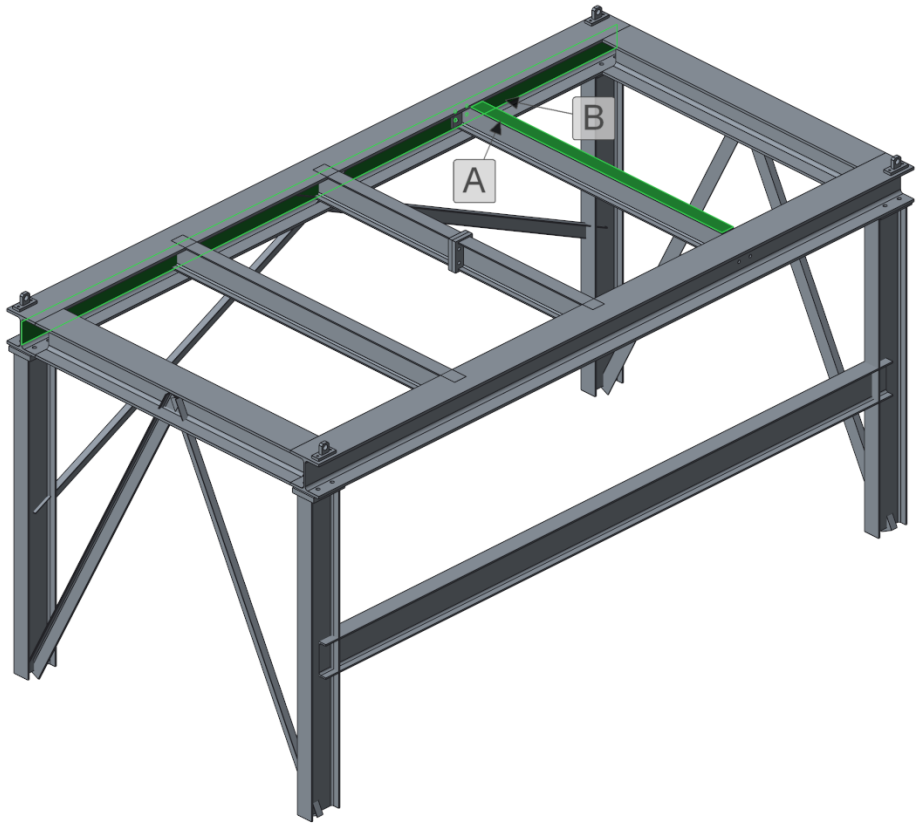
Hint 3 — Beam end cope dimensions. The values **L4_MIN**, **H41_MIN** and **H42_MIN** should be left with default values in most cases. **AFX** searches in the configuration folder for `beam_end_cope_dims.cfg` and will round the dimension from profile end or top to end face of the cope. The allowed values can be defined in the `beam_end_cope_dims.cfg`.

1. Click **[I-Profile top (1)]** and select the surface **[A]** of the I-Beam near the profile end.
2. Click **[Attach face (2)]** and select the face **[B]**.
3. Click **[OK]** to complete the definition.
   - The end-plate is assembled and the profile receives a cut-out to avoid interference.



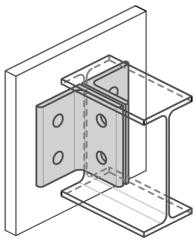
Now reassemble the **end-plate** on the opposing side.

1. Select the previously assembled end-plate `IS_16_4_80.PRT`
2. Press the right mouse button and select **Framework** > **Reuse** and then press **Next**.
3. Select the surface **[A]** of the I-Beam.
4. Select face **[B]** as **Attach face**.
   - The second end-plate will be assembled.
5. Press **[Middle mouse button]** to abort the selection loop and close the dialog.

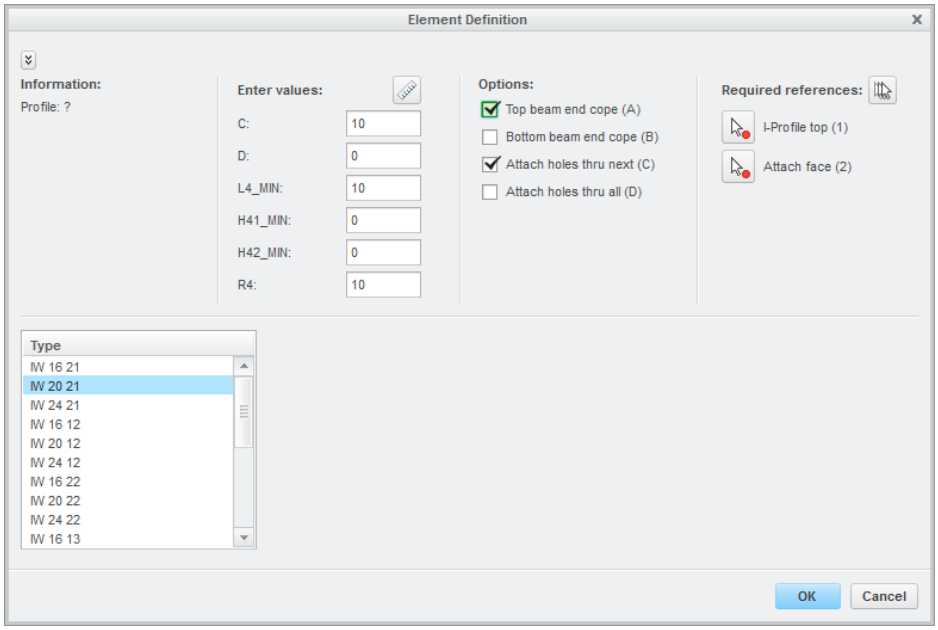


Assemble DAST double angle

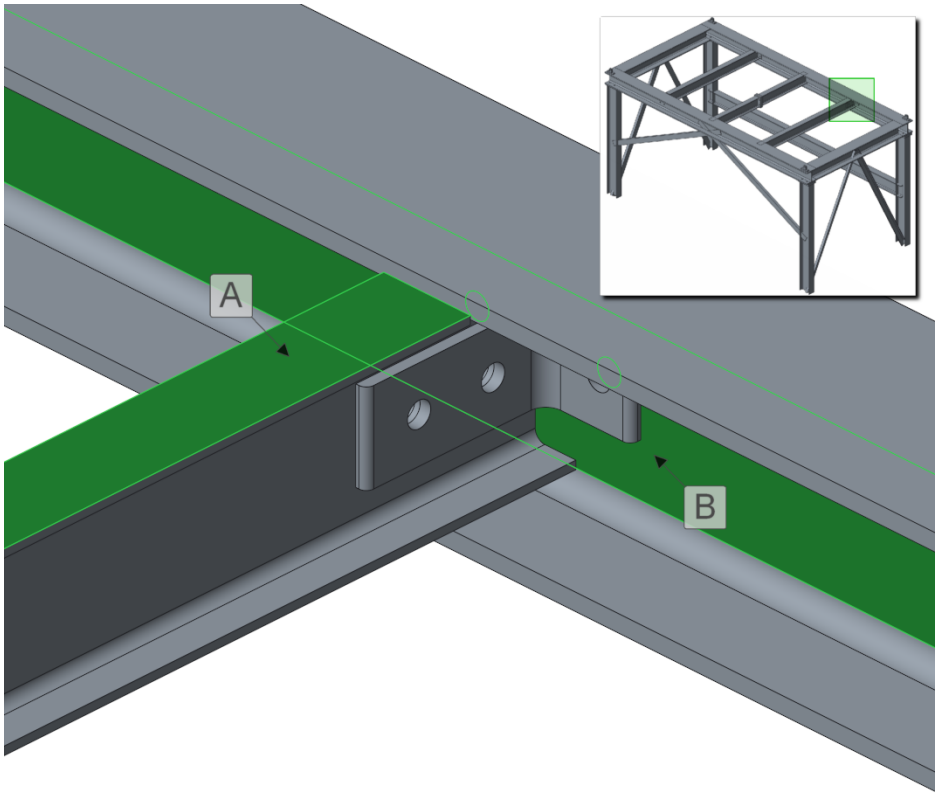
To assemble a DAST double angle proceed as follows.



1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [DAST STANDARD] > [ANGLE CONNECTOR IW]** and **Next**.
   - The Element definition dialog box of the end plate opens.



Define the connector as shown in the dialog above.

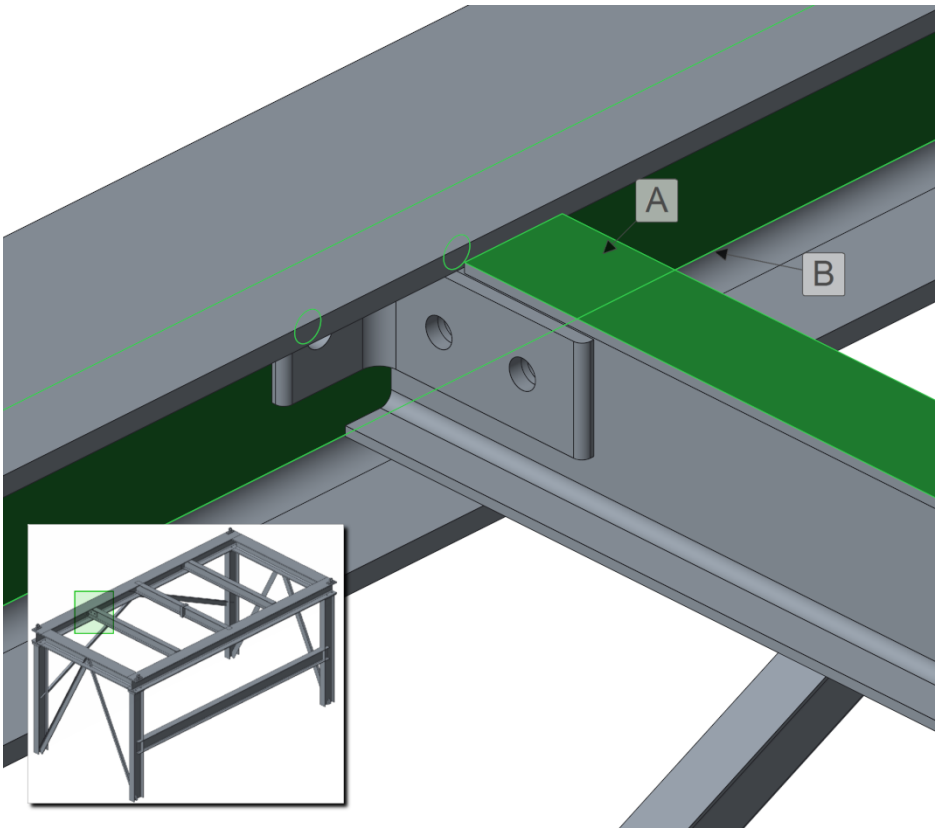


4. Click **[I-Profile top (1)]** and select the surface **[A]** of the I-Beam near the profile end.

5. Click **[Attach face (2)]** and select the face **[B]**.
6. Click **[OK]** to complete the definition.
   - The double angle connector is assembled and the profile is cut-out to prevent interference.

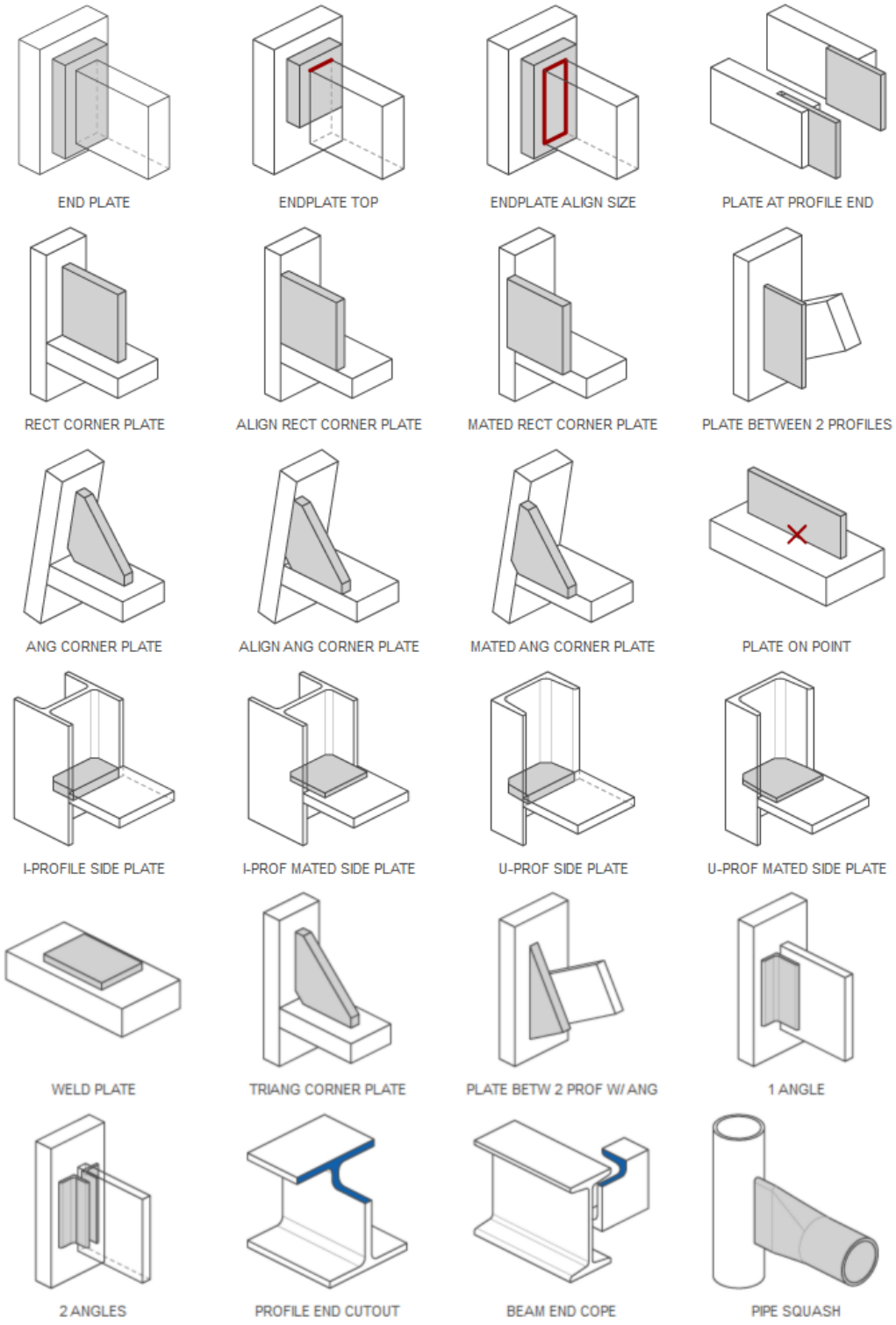
Now reassemble the **double angle** on the opposing side.

1. Select the previously assembled double angle `SC_ANGLE_CON_IW_1.ASM`
2. Press the right mouse button and select **Framework** > **Reuse** and **Next**.
3. Select the surface **[A]** of the I-Beam.
4. Select face **[B]** as **Attach face**.
   - The second double angle is assembled.
5. Press **[Middle mouse button]** to abort the loop. Then close the dialog.



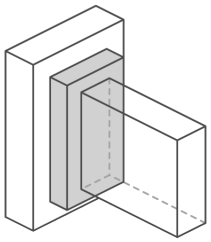
2.3 Project specific connectors

With **AFX** you can define various project specific connector elements. In the next chapters the use of these elements will be described.

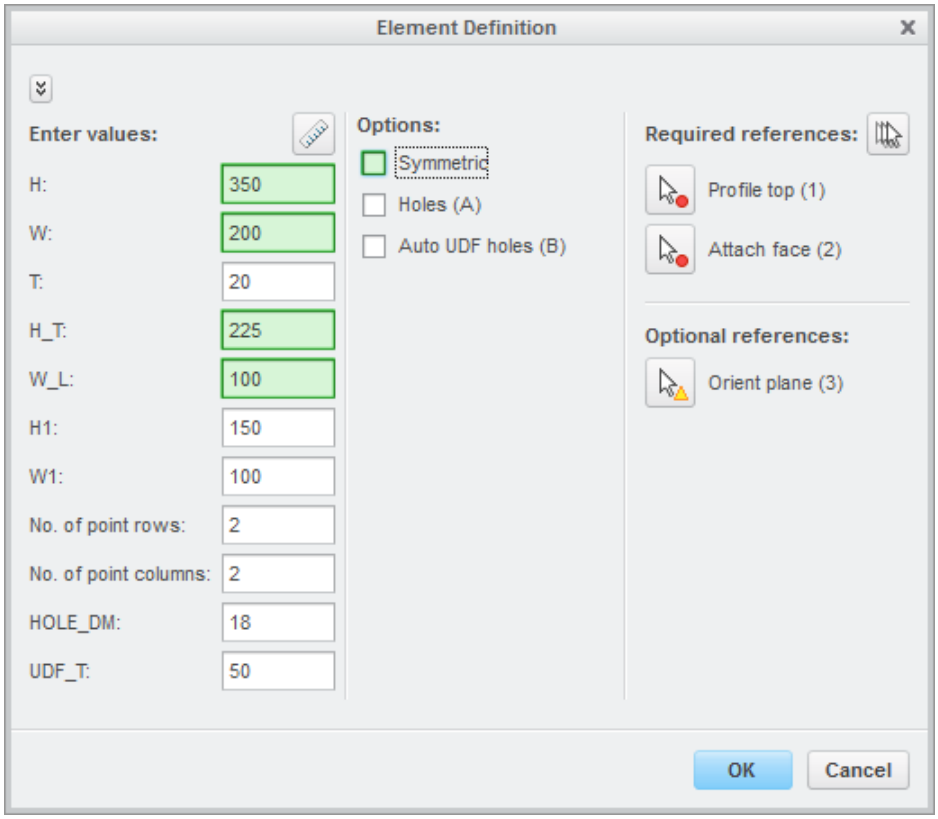


Non-standard end-plates

The first element to assemble is the **end-plate**. This type is the most generic and allows the most configuration.

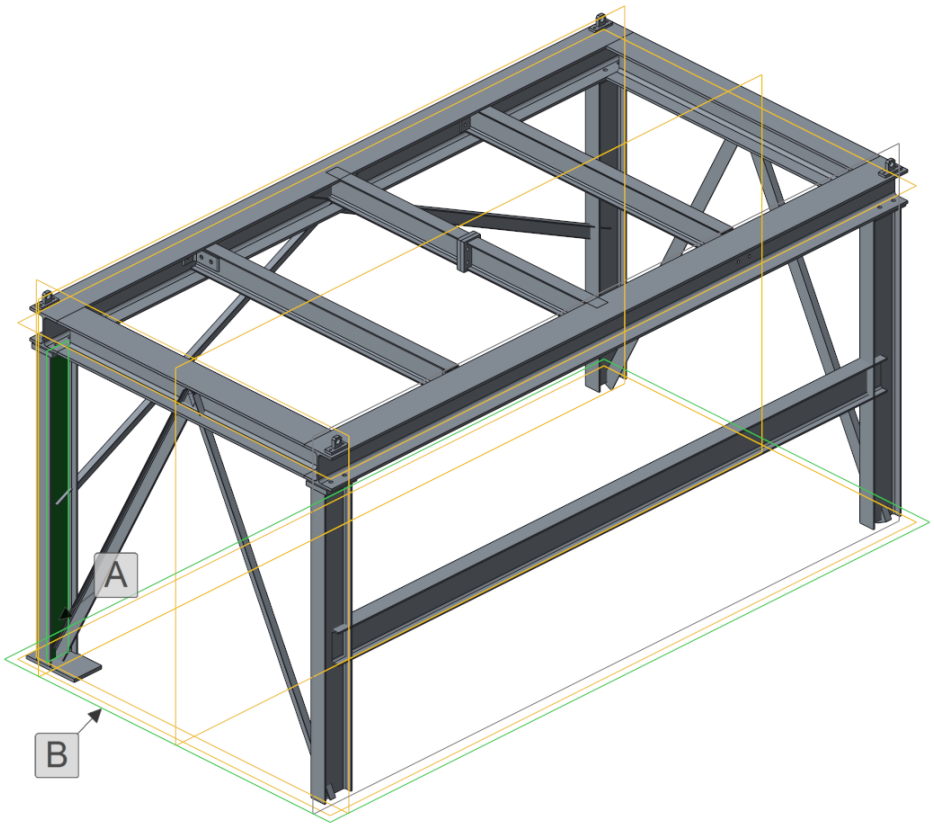


1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE]** and press **Next**.
   - The Element definition dialog box of the end plate opens.

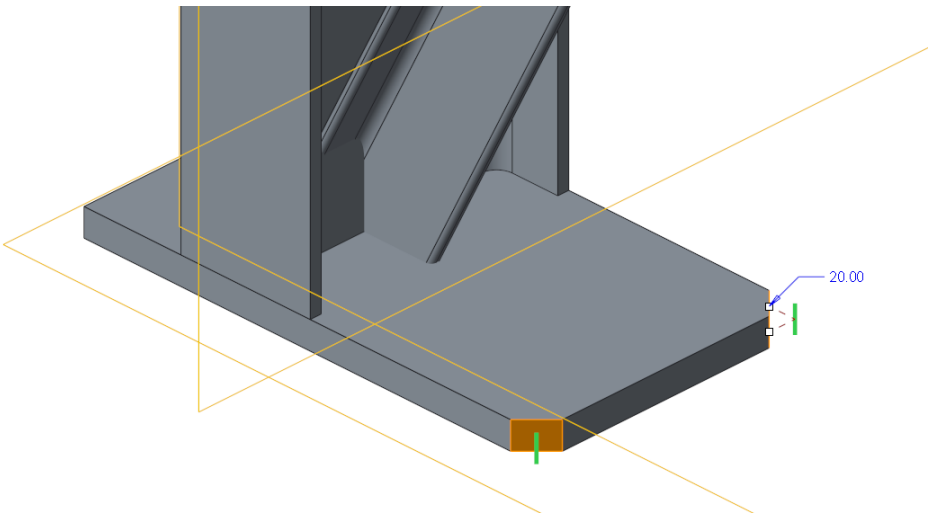


   Define the connector as shown in the dialog above.

4. Click left to **[Profile top (1)]**.
5. Select the highlighted surface of the I-Beam near the profile end **[A]**.
6. Click left to **[Attach face (2)]** and select the plane A_XZ **[B]**.
7. Click **[OK]** to complete the definition.
   - The connector is assembled and the I-beam gets shortened.



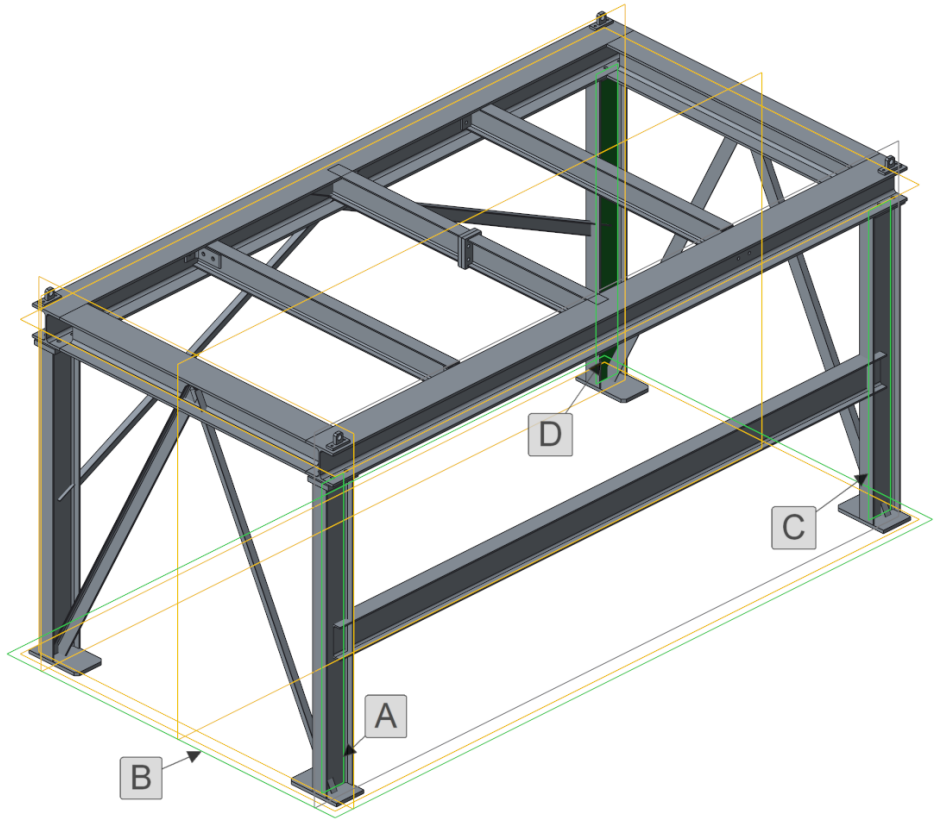
8. Activate the **end-plate** and add a chamfer **[20]** with **Creo Parametric** standard functionality.



Now reassemble the end-plate element to the other three vertical I-beams.

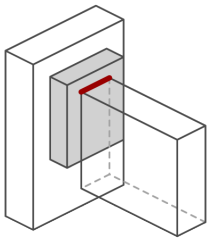
1. Select the end-plate SC_ENDPLATE_1.
2. Press the right mouse button and select **Framework >** **Reuse** and press **Next**.

3. Select the required references **[A, B]**, **[C, B]**, **[D, B]** as shown in the next picture. Make sure to select the **inner web-surface** to assure the correct orientation of the end-plates.
   - The three end-plates are assembled.
4. Press **[Middle mouse button]** and close the dialog.

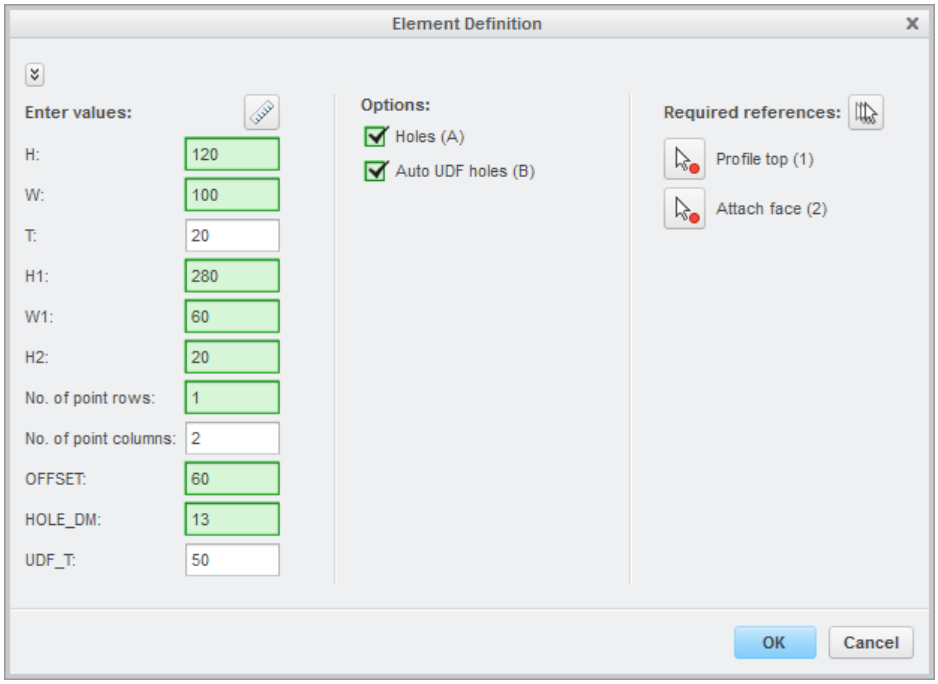


Hint 4 — Modify AFX parts with Creo Parametric. Always keep in mind, that you can modify connectors and equipment with standard **Creo Parametric** functionality!

The second element to assemble is the **end-plate top**. This type is referenced to the selected profile surface, therefore you can easily control the offset to the surface.



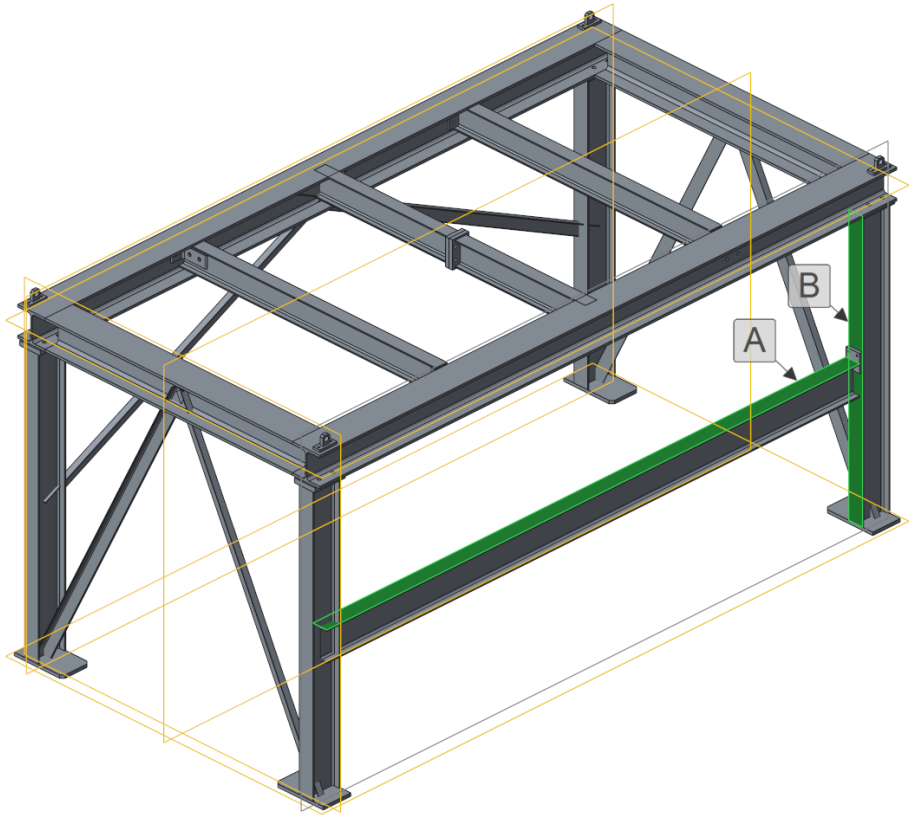
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE TOP]** and press **Next**.
   - The Element definition dialog box of the end-plate top opens.



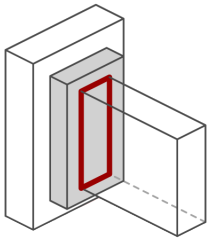
Define the connector as shown in the dialog above.

4. Click **[Profile top (1)]**.
5. Select the highlighted surface of the I-Beam near the profile end **[A]**.
6. Click **[Attach face (2)]**.

7. Select the highlighted surface of the I-Beam **[B]**.
8. Click **[OK]** to complete the definition.
   - The connector is assembled and the I-beam gets adjusted in length.

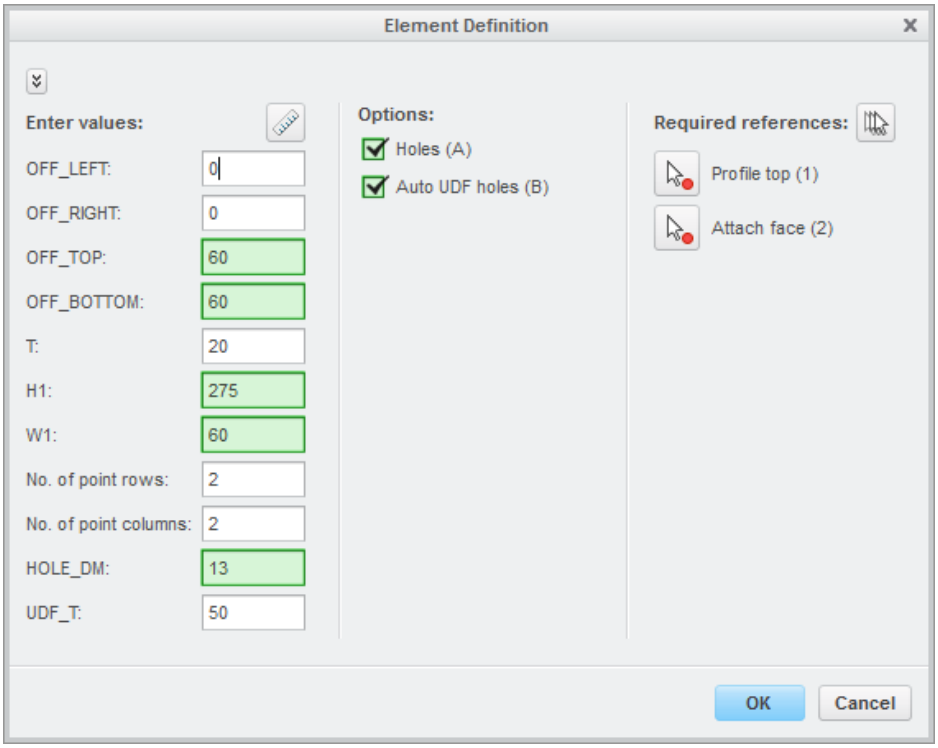


The third and last end-plate type will be **end-plate align size**. This type on referenced on all four sides of the selected profile, therefore you can control all offsets to the profile.



1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE ALIGN SIZE]** and press **Next**.

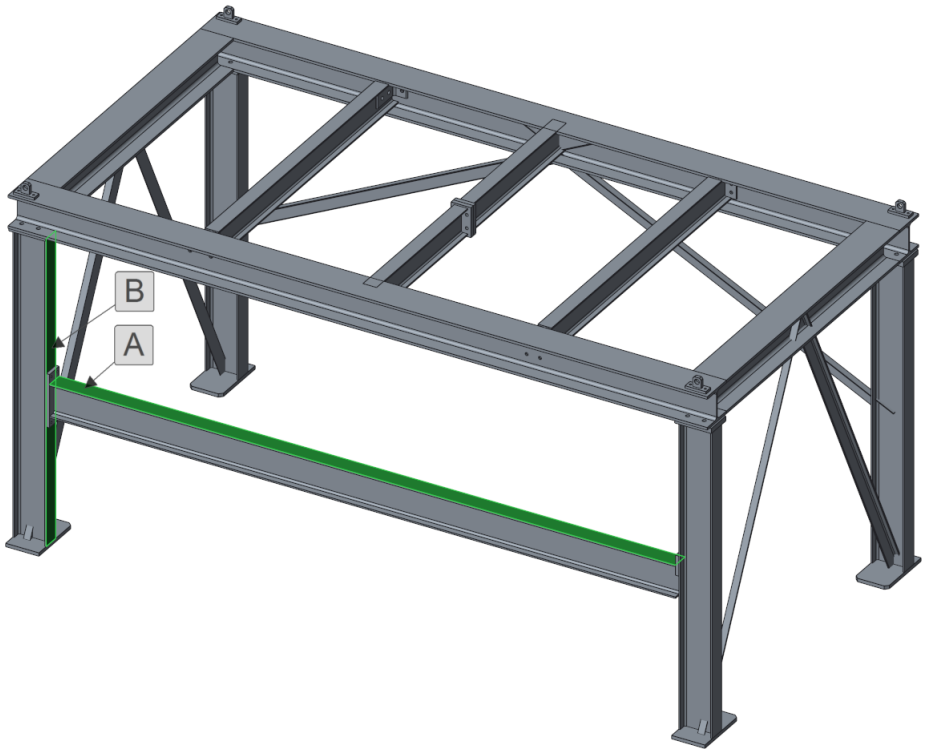
   The Element definition dialog box of the endplate top opens.



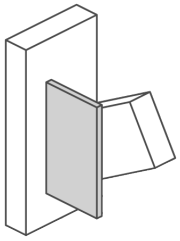
Define the connector as shown in the dialog above.

4. Click **[Profile top (1)]**.
5. Select the highlighted surface of the I-Beam near the profile end **[A]**.
6. Click **[Attach face (2)]**.
7. Select the highlighted surface of the I-Beam **[B]**.
8. Click **[OK]** to complete the definition.

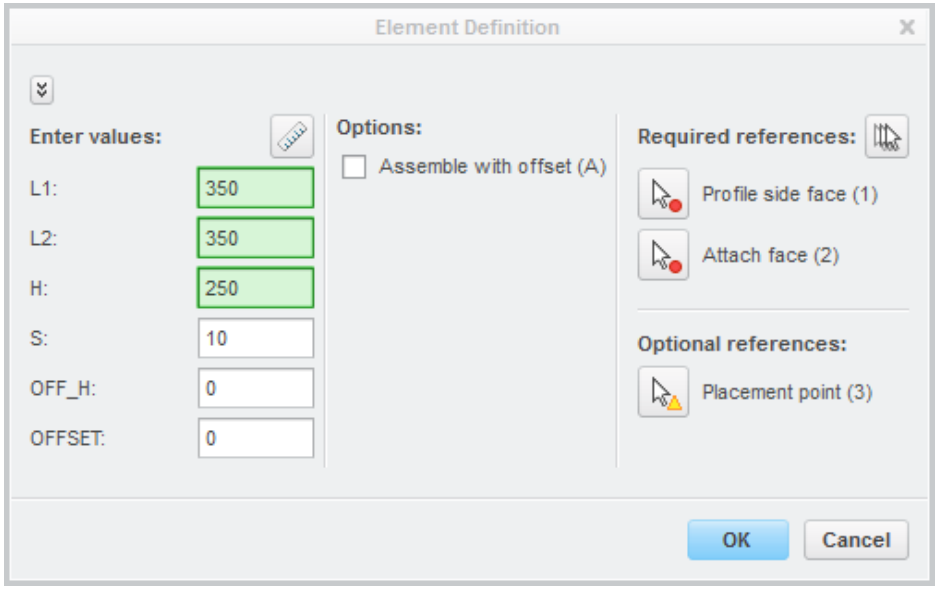
- The connector is assembled and the I-beam gets adjusted in length.



Brackets between two Profiles



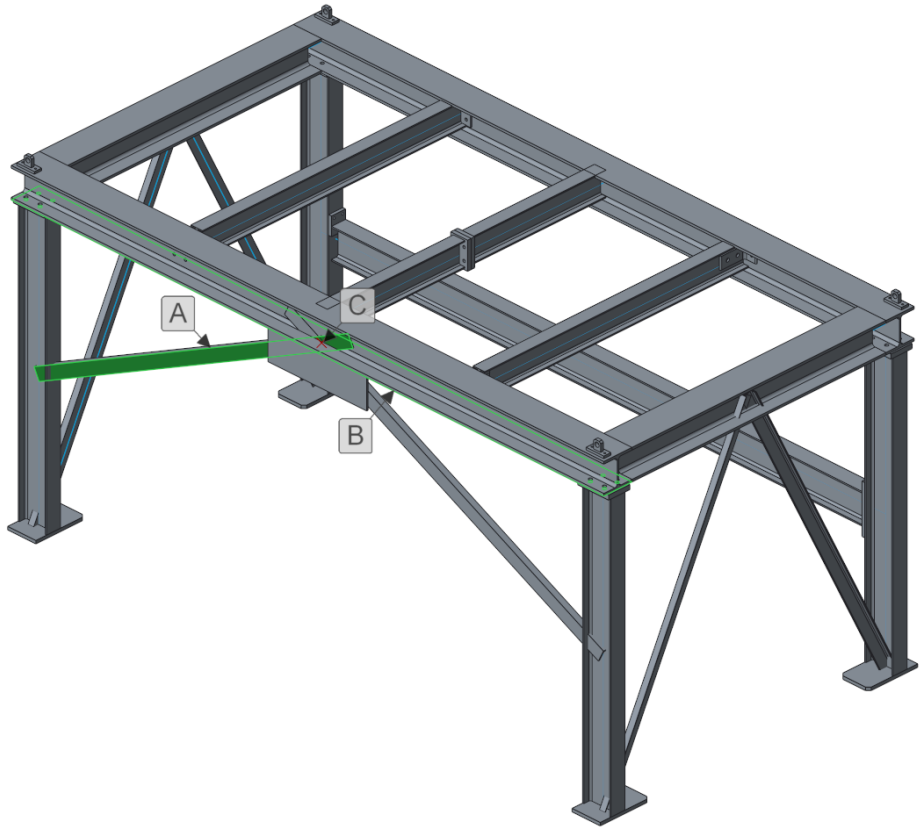
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [PLATE BETWEEN 2 PROFILES]** and press **Next**.
   - The **Element definition** dialog box of the **PLATE BETWEEN 2 PROFILES** opens.



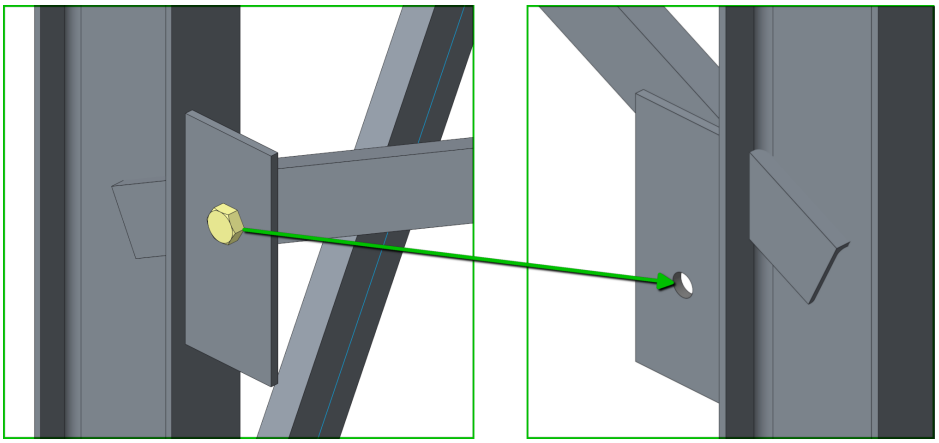
Define the connector as shown in the dialog above.

4. Click **[Profile side face (1)]**.
5. Select the highlighted surface of the I-Beam near the profile end **[A]**.
6. Click **[Attach face (2)]**.
7. Select the highlighted surface of the horizontal I-beam **[B]**.
8. Click **[Placement point (3)]**.

9. Select the datum point **[C]**.
10. Click **[OK]** to complete the definition.
    - The connector is assembled.

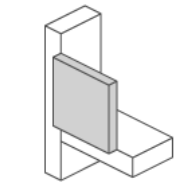


The next plates will be left and right Brackets. We could create them with the same connector, but this would cause a problem, when it is reassembled on the other side:

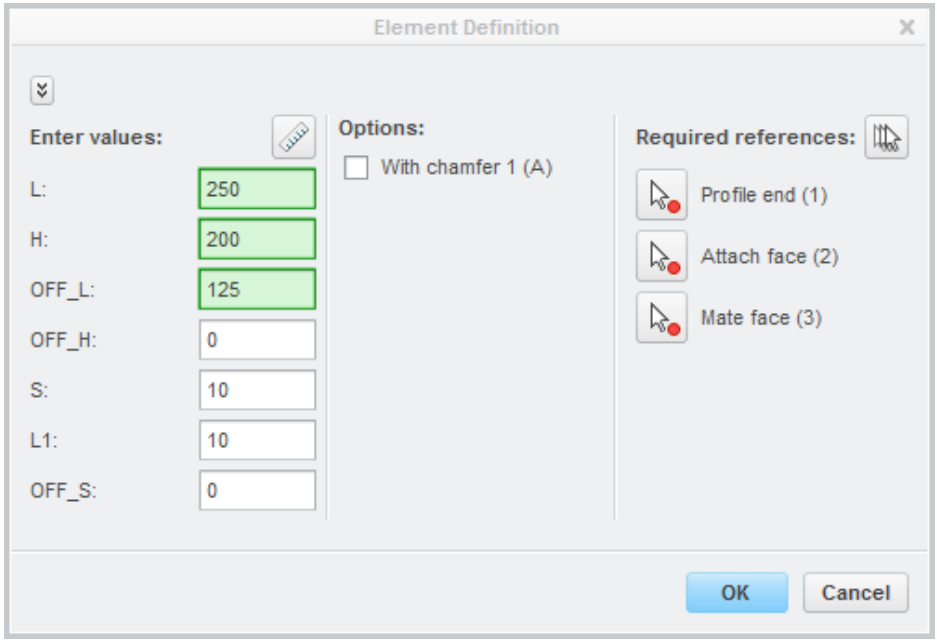


The **PLATE BETWEEN 2 PROFILES** lacks the third position information. To avoid this behavior we take the **MATED RECT CORNER PLATE** instead.

Mated rect corner plate



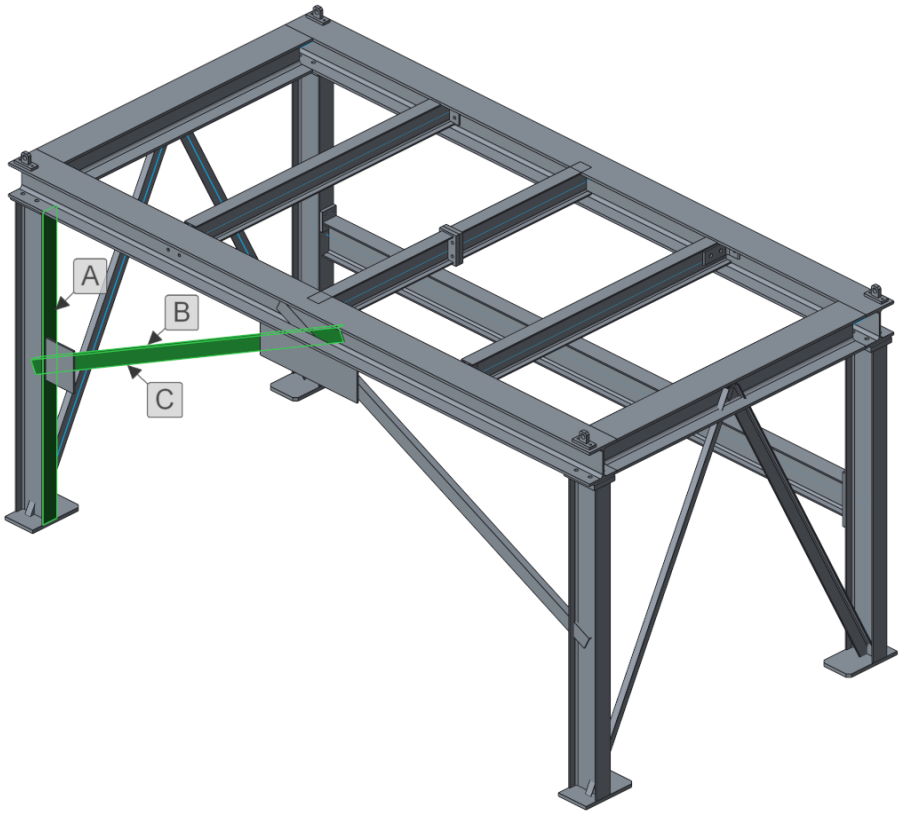
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [MATED RECT CORNER PLATE]** and press **Next**.
   - The **Element definition** dialog box of the **MATED RECT CORNER PLATE** opens.



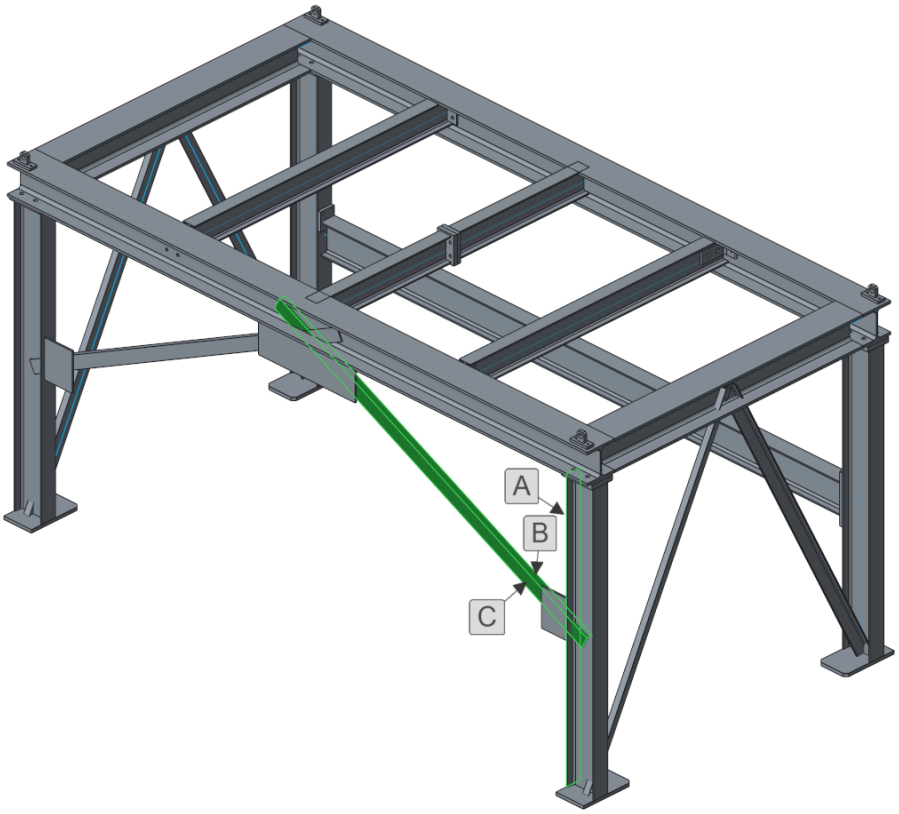
   Define the connector as shown in the dialog above.

4. Click **[Profile top (1)]**.
5. Select the highlighted surface of the I-Beam **[A]**.
6. Click **[Attach face (2)]**.
7. Select the upper highlighted surface of the profile **[B]**.
8. Click **[Mate face (3)]**.

9. Select the frontal highlighted surface of the profile **[C]**.
10. Click **[OK]** to complete the definition.
    - The connector is assembled.

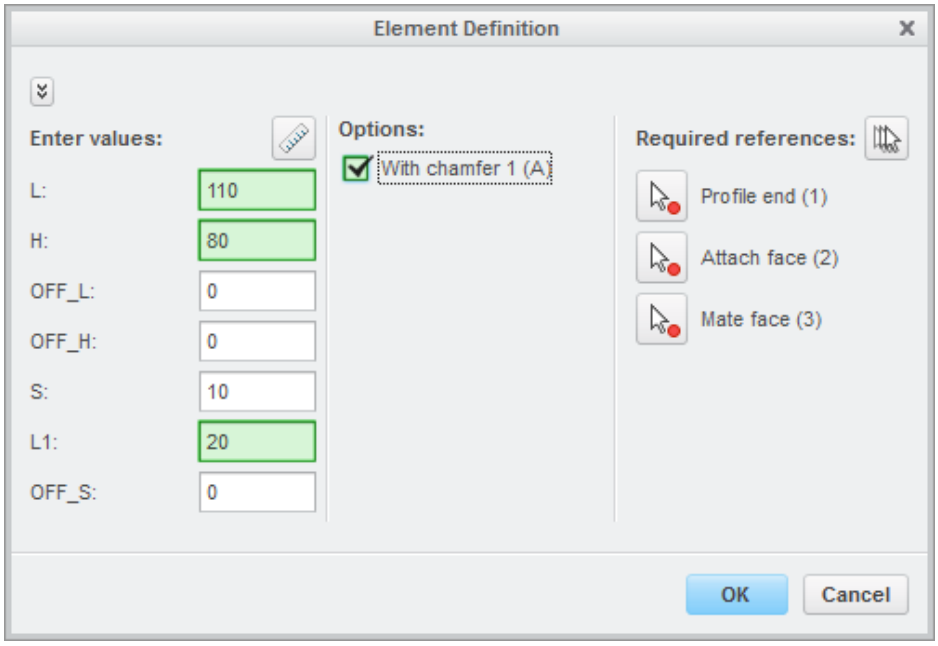


This plate can be **reassembled** via ↺. To do so select the previously assembled plate and the references **[A]**, **[B]** and **[C]** as shown in the next picture. OFF_S is 0.



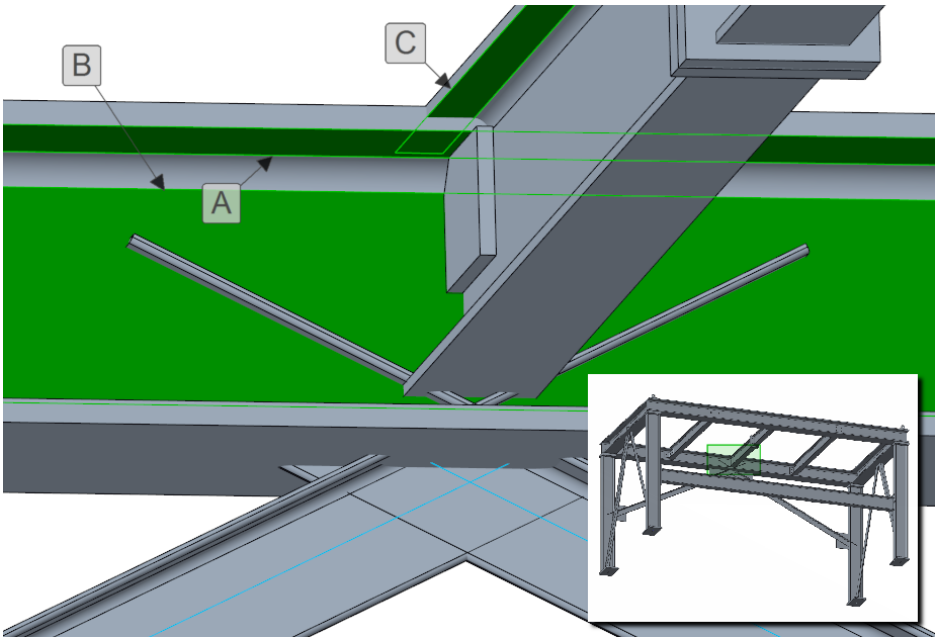
Another example:

1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [MATED RECT CORNER PLATE]** and press **Next**.
   - The **Element definition** dialog box of the **PLATE BETWEEN 2 PROFILES** opens.



Define the connector as shown in the dialog above.

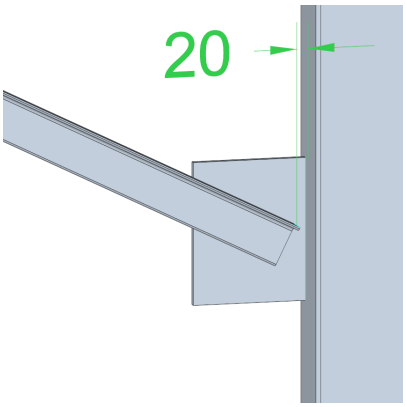
4. Click **[Profile end (1)]**.
5. Select the upper highlighted surface of the I-Beam **[A]**.
6. Click **[Attach face (2)]**.
7. Select the frontal highlighted surface of the profile **[B]**.
8. Click **[Mate face (3)]**.
9. Select the highlighted surface of the profile **[C]**.
10. Click **[OK]** to complete the definition.
    - The plate is assembled.



Apply joints

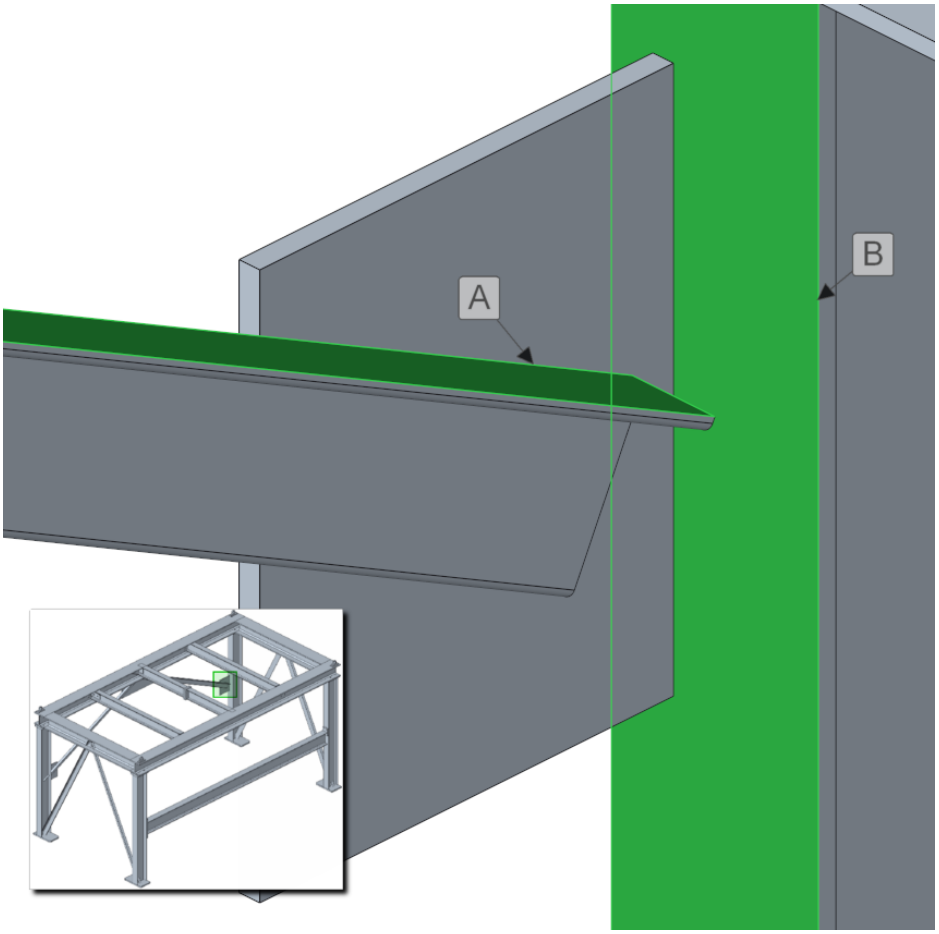
In this section we want to create a special kind of joint. We could use a standard joint to make the angular mounted profiles fit, however we want them to be dimensioned in a special way - this makes it easier to ensure, that a certain offset is maintained (important for bevel seam). Once it is set up, it will even regenerate to the desired distance after modifications.

To apply this joint type proceed as follows.

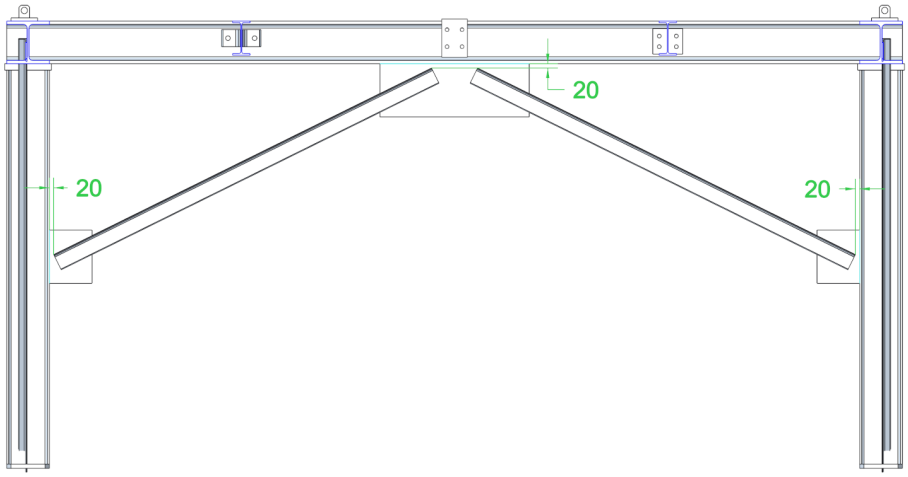


1. Click **Basic Joints** to open the **Basic Joints** dialog box.
2. Mark the checkbox **With offset**.
3. Click  in **basic Joints** dialog box, to create a joint of type **selected surface with normal cut and offset along intersection line** between the upper left end of the diagonal profile and the upper horizontal rectangular tube.
4. Enter the desired offset **[20]** into the input panel.

5. Select the diagonal profile at the upper side surface near the upper left end **[A]**.
6. Select highlighted surface of the I-beam **[B]** and press repeat.
    - The diagonal profile is cut normal to its length axis along the intersection line between the 2 selected surfaces.



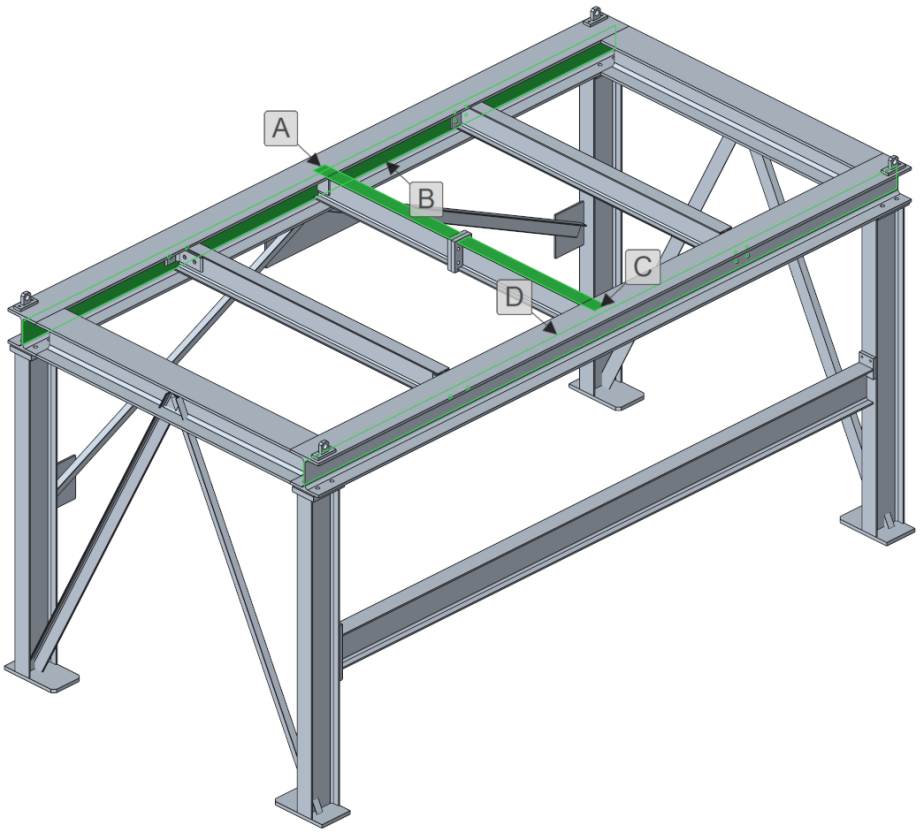
Complete the joint definition on the remaining profile ends.



Prepare more joints for next section:

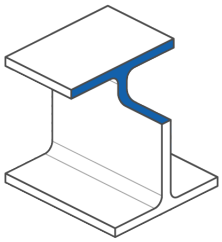
1. Click **Basic Joints** to open the **Basic Joints** dialog box.
2. Click in 

Basic Joints dialog box, to create a joint of type **Define joint to selected surface** between both sides of the center I-beam and the corresponding adjacent horizontal beams.
3. Select the highlighted surface **[A]** of the center beam.
4. Select the inner web surface **[B]** of the adjacent beam and press repeat.
5. Select the highlighted surface **[C]** of the center beam.
6. Select the inner web surface **[D]** of the adjacent beam and press ok.
   - The two center beams will be justified to the inner web surfaces.



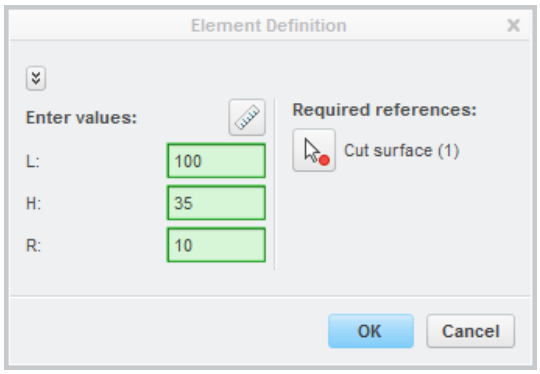
Define Profile End Cutouts

Define a **PROFILE END CUTOUT** to the left center profile.

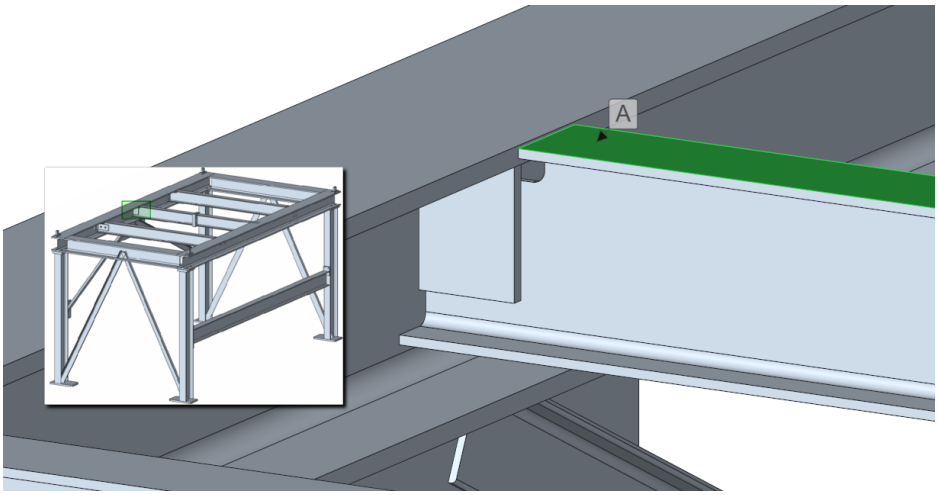


1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [PROFILE END CUTOUT]**.
   - The **Element definition** dialog box of the **PROFILE END CUTOUT** opens.

   Define the connector as shown in the dialog.

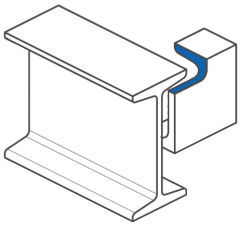


4. Click **[Cut surface (1)]**.
5. Select the Profile top **[A]**.
6. Click **[OK]** to complete the definition.
   - The profile is cut out.



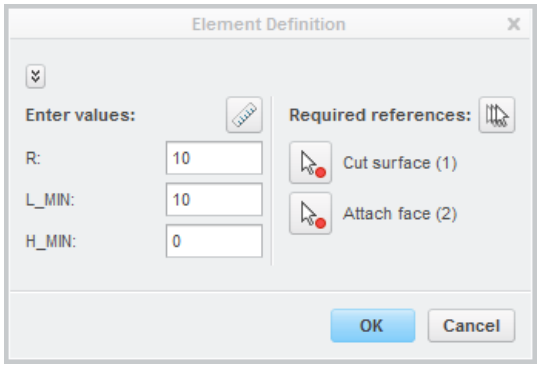
Hint 5 — Delete End Cutouts. If you want to delete these kind of connector elements you will have to select the surface that has been modified by the connector. It is also possible to open the component and delete the feature.

Now we will fix the intersection on the other side with a different connector. Define a **PROFILE END CUTOUT** to the two center profiles.



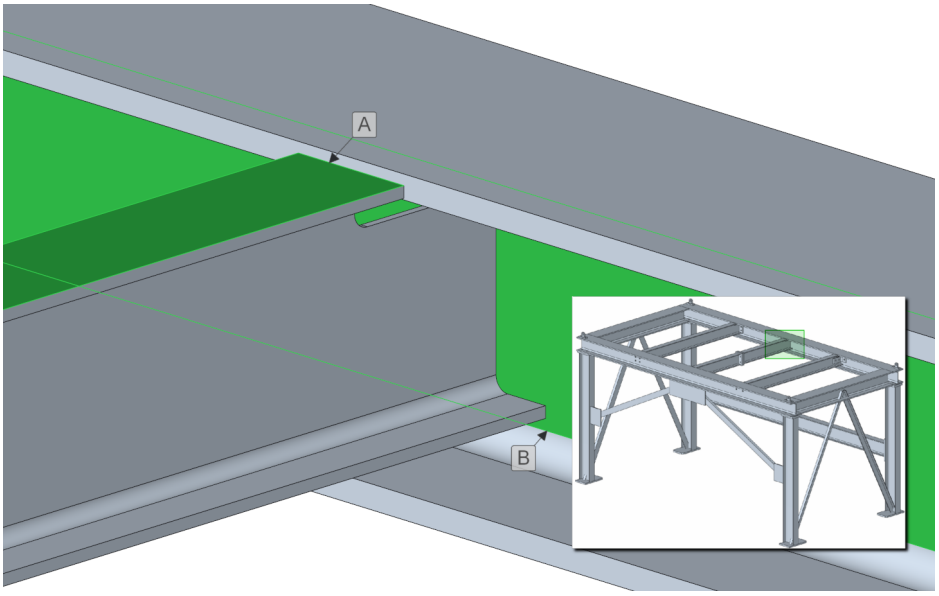
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [BEAM END COPE]**.
   - The **Element definition** dialog box of the **BEAM END COPE** opens.

     Keep **default settings** in the dialog.



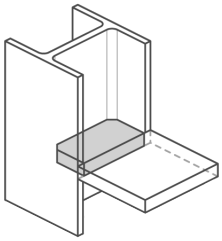
4. Click **[Cut surface (1)]**.
5. Select the Profile top **[A]**.
6. Click **[Attach face (2)]**.

7. Select the inner web surface **[B]**.
8. Click **[OK]** to complete the definition.
   - The profile will be cut out.



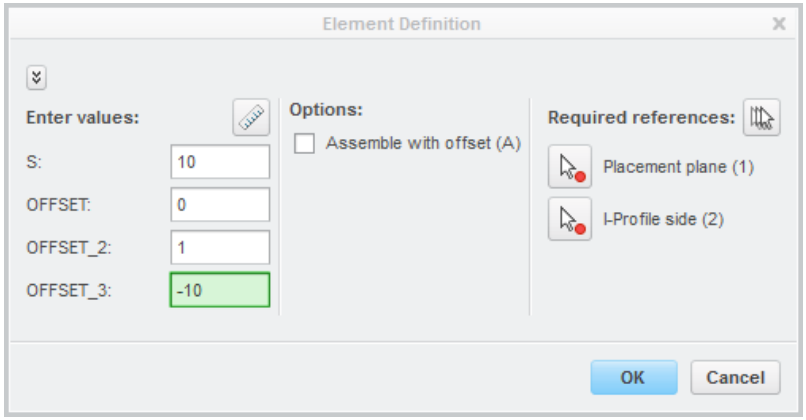
I-Profile Side Plates

Type 1: I-Profile Side Plate

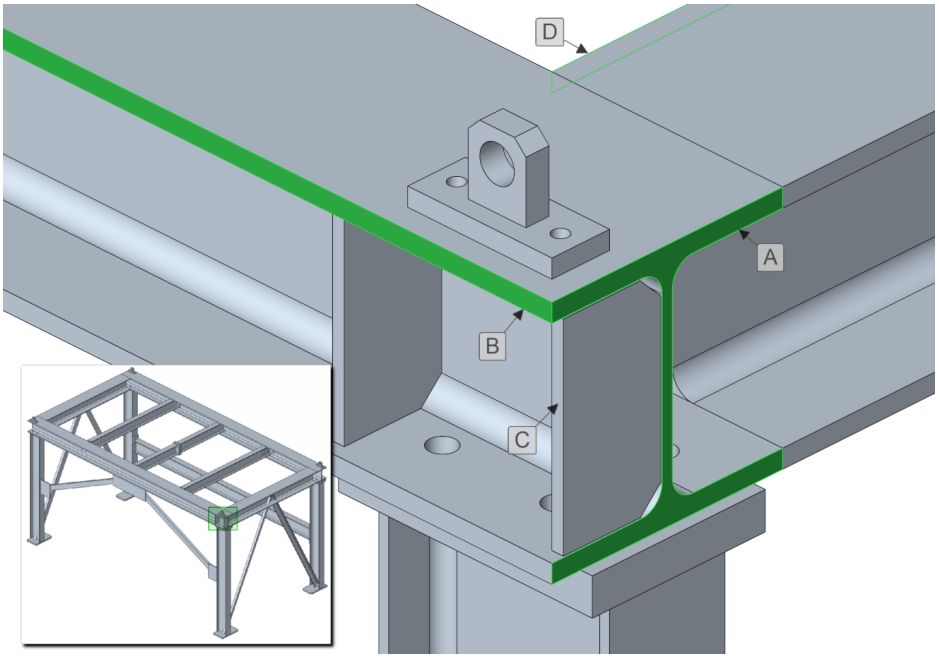


1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [I-PROFILE SIDE PLATE]**.
   - The **Element definition** dialog box of the **IPROFILE SIDE PLATE** opens.

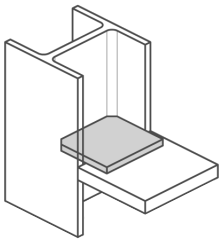
     Define the connector as shown in the dialog above.



4. Click **[Placement surface (1)]**.
5. Select highlighted surface of the I-Beam **[A]**.
6. Click **[I-profile side (2)]**.
7. Select the highlighted surface of the profile **[B]**.
8. Click **[OK]** to complete the definition.
   - The side plate is assembled.
9. Select and select the **I-PROFILE SIDE PLATE [C]** and press next.
10. Select highlighted surface of the I-Beam **[D]**.
11. Select the highlighted surface of the profile **[E]**.
    - The side plate is assembled reassembled.

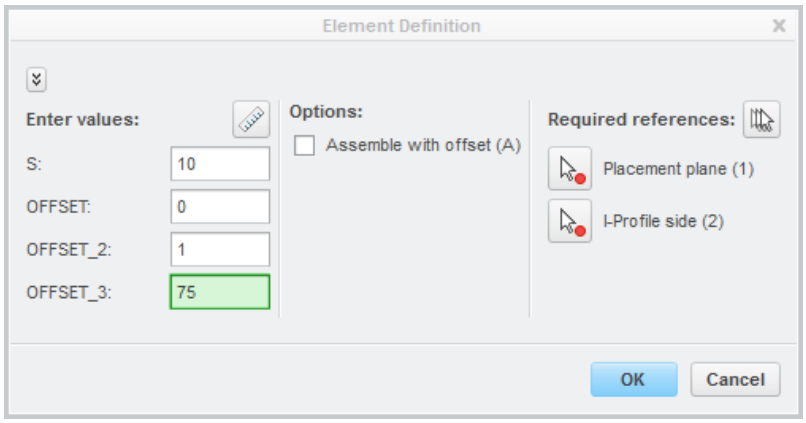


Type 2: I-Profile Mated Side Plate



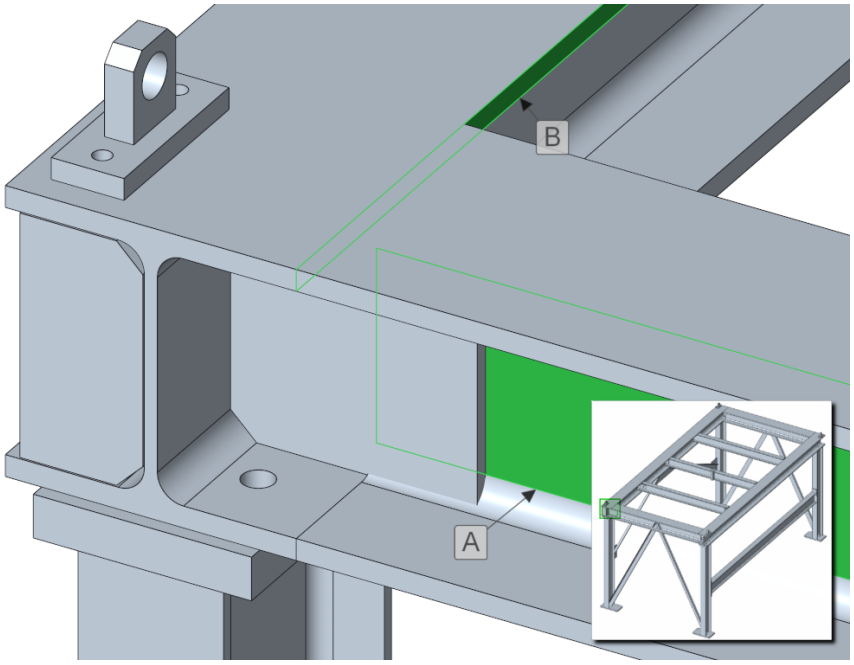
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [I-PROFILE MATED SIDE PLATE]**.
   - The **Element definition** dialog box of the **IPROFILE MATED SIDE PLATE** opens.

     Define the connector as shown in the dialog.

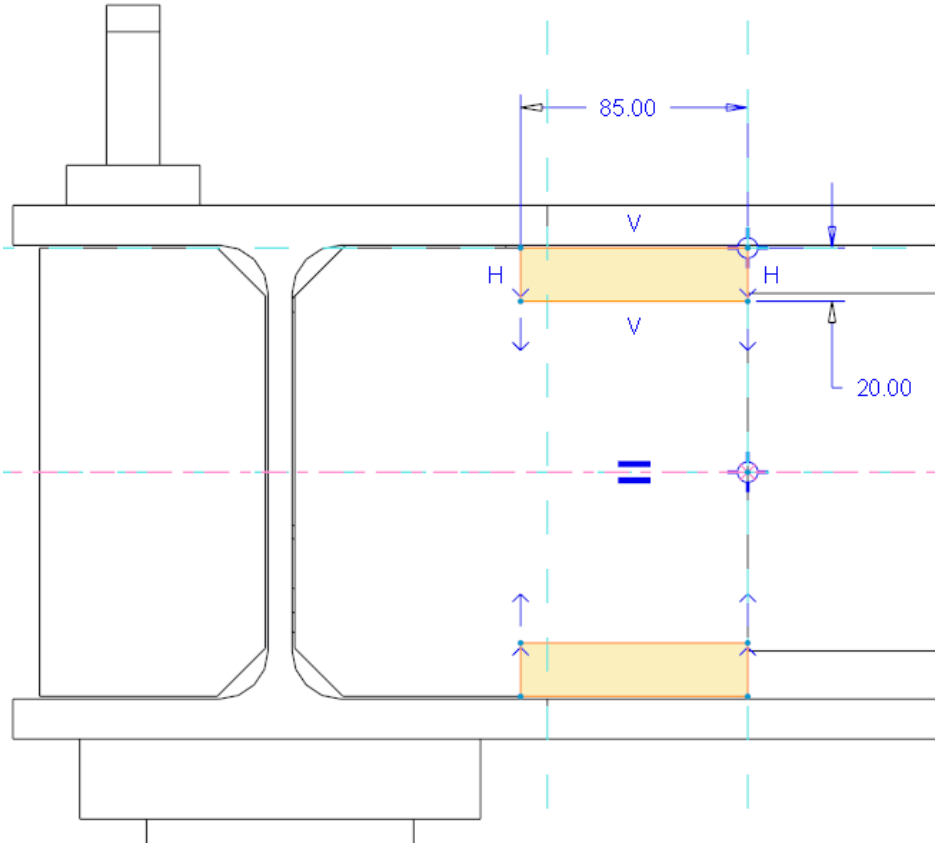


4. Click **[Placement plane (1)]**.
5. Select highlighted surface of the I-Beam **[A]**.
6. Click **[I-Profile side (2)]**.

7. Select the highlighted surface of the I-Beam **[B]**.
8. Click **[OK]** to complete the definition.
   - The side plate is assembled.



Unfortunately the plate interferes with the I-beam. To resolve that problem, simply activate the plate and modify it with a standard **Creo Parametric** extrude feature as shown below.



Now reassemble the connector on the other three corners of the frame.

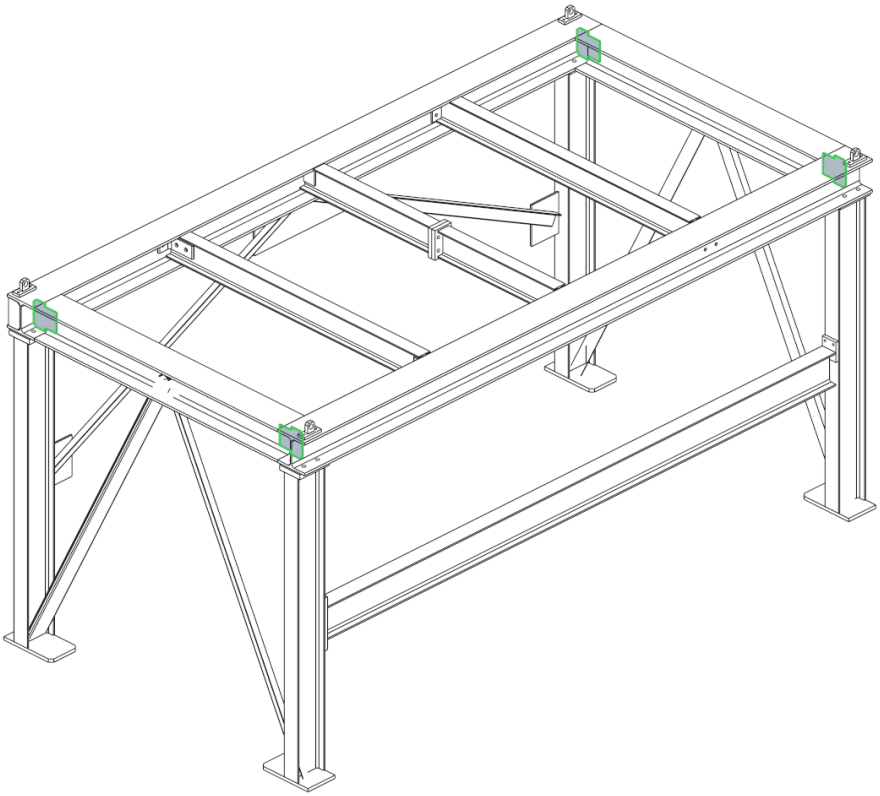
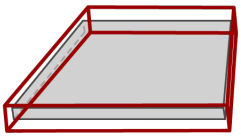


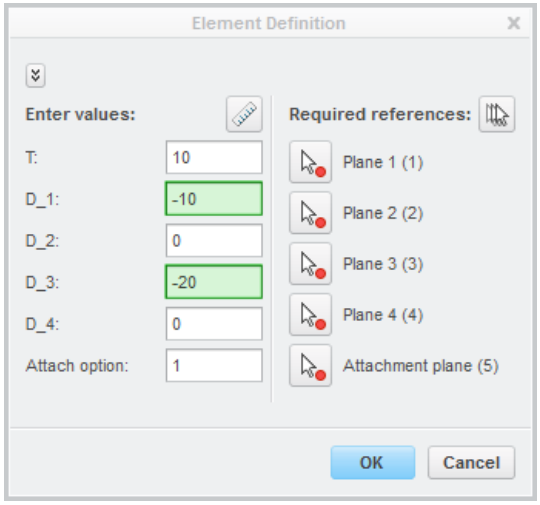
Plate between four planes

This technique is an alternative to the mated rectangular corner plate from chapter 2.2.3.

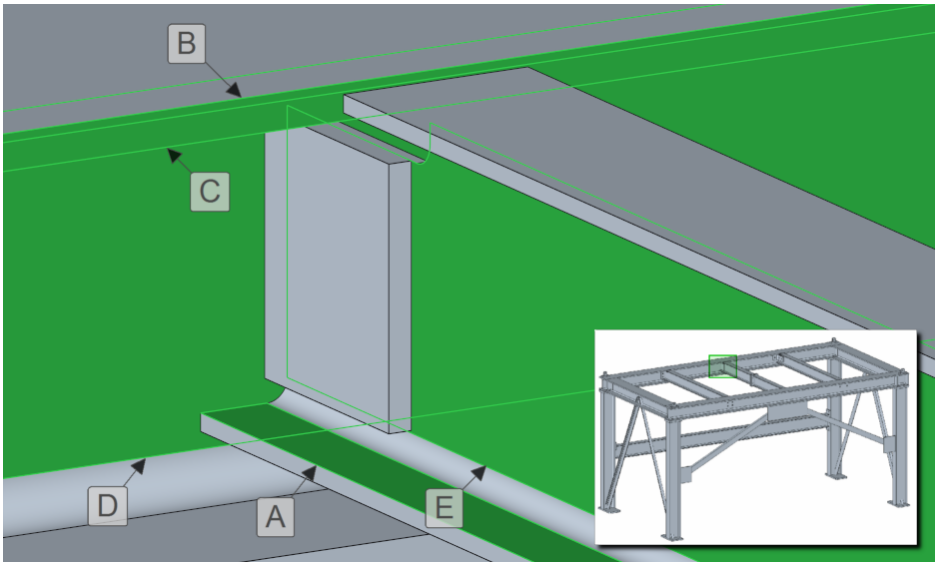


1. Click **New Equipment Elements** to open the **Equipment Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[PLATES MM] > [PLATE 4 PLANES]**.
   - The **Element definition** dialog box of the **PLATE 4 PLANES** opens.

     Define the connector as shown in the dialog.

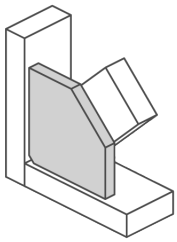


4. Click **[Plane (1)]** and select the highlighted surface **[A]**.
5. Click **[Plane (2)]** and select the highlighted surface **[B]**.
6. Click **[Plane (3)]** and select the highlighted surface **[C]**.
7. Click **[Plane (4)]** and select the highlighted surface **[D]**.
8. Click **[Attachment plane (5)]** and select the highlighted surface **[E]**.
9. Click **[OK]** to complete the definition.
   - The plate is assembled.



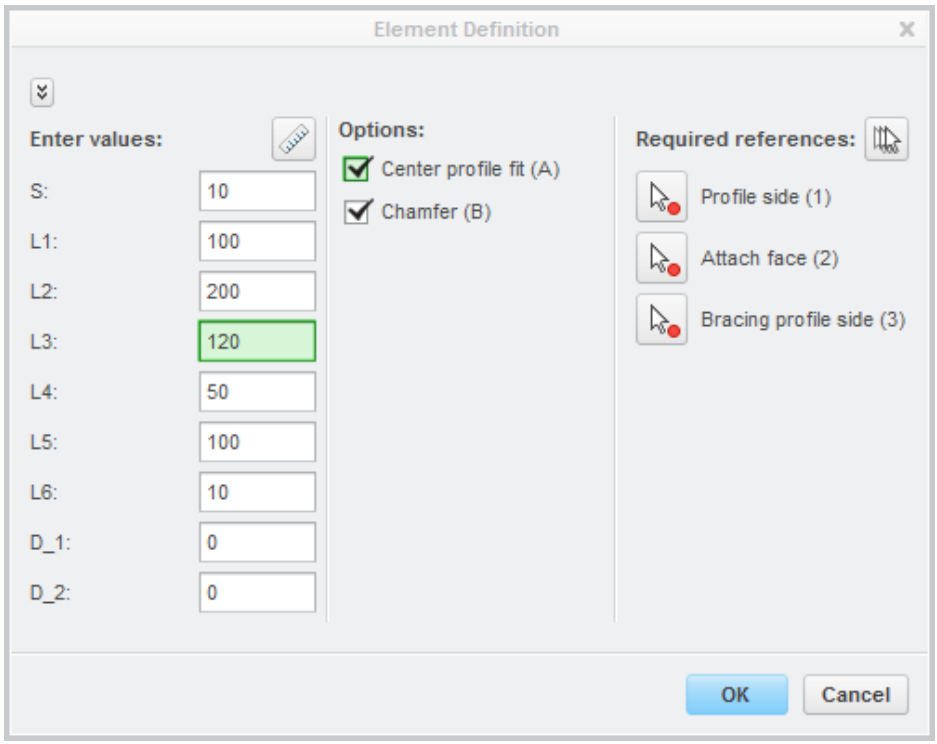
Bracing plates

Type 1: Bracing plate rectangular



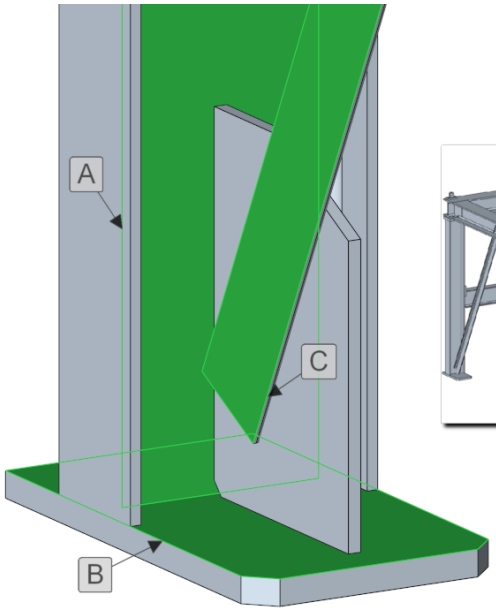
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [BRACING PLATES] > [BRACING PLATE RECTANGULAR]**.
   - The **Element definition** dialog box of the **BRACING PLATE RECTANGULAR** opens.

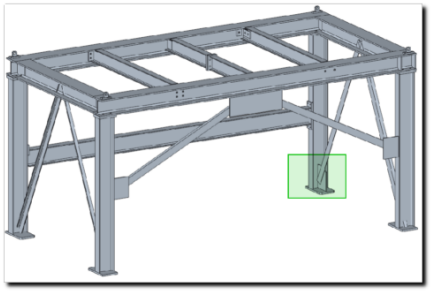
     Define the connector as shown in the dialog.



4. Click **[Profile side (1)]**.
5. Select highlighted surface of the I-Beam **[A]**.
6. Click **[Attach face (2)]**.
7. Select the highlighted surface of the plate **[B]**.
8. Click **[Bracing profile side (3)]**.

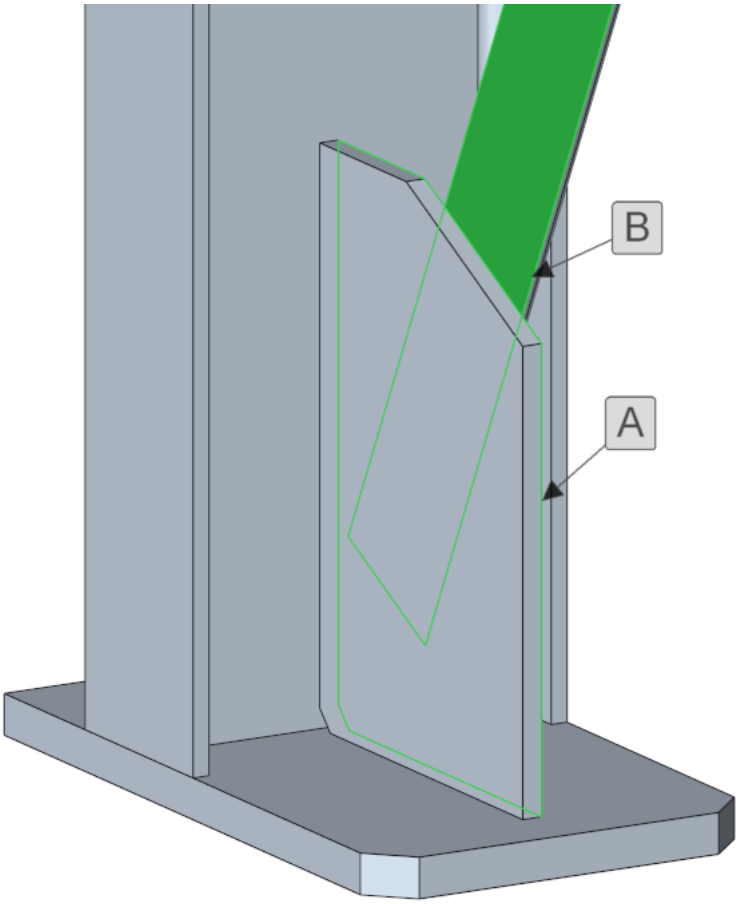
9. Select the highlighted surface of the profile **[C]**.
10. Click **[OK]** to complete the definition.
    - The bracing plate is assembled and the beam gets adjusted in length.





Hint 6 — Bracing plate options. If you check the Center profile fit (A), then the bracing plate will be aligned in the center of the selected **Profile side (1)** and not attached to the **bracing profile side (3)**.

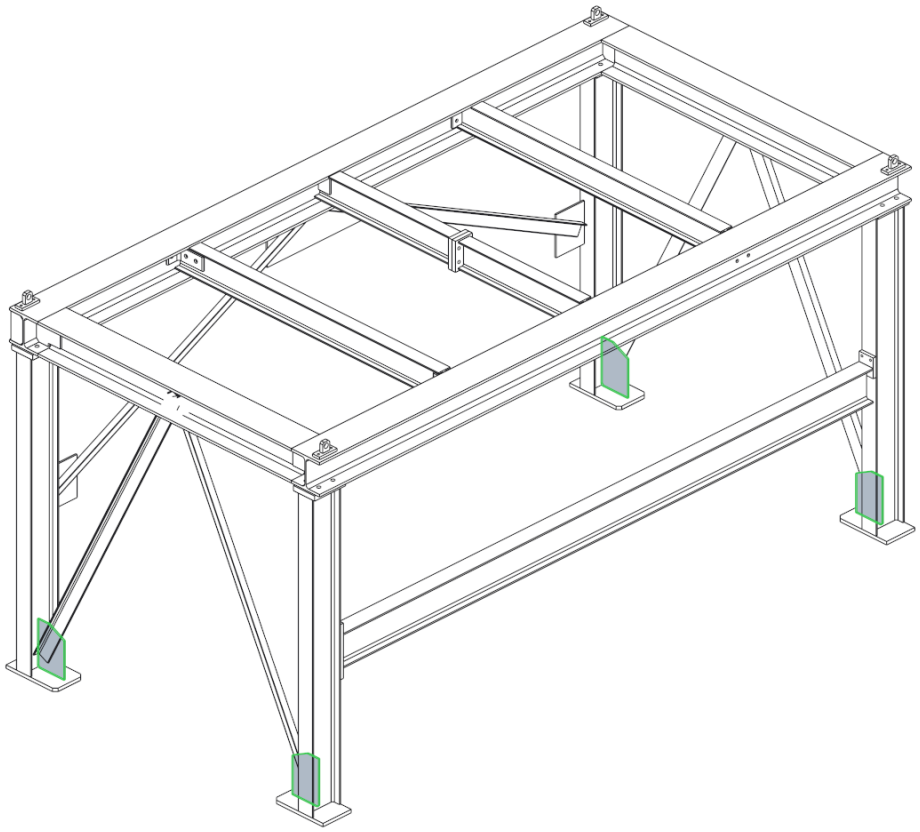
Use the  **Align profile with selected entities from profile and another part** command in Profiles group to fix the intersection of the bracing plate and the profile.



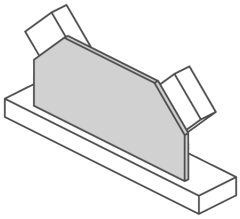
1. Open the **Move** dialog in the profiles group of the ribbon.
2. Click the highlighted surface of the bracing plate **[A]**.
3. Select **Align profile with selected entities from profile and another part**.
4. Click the highlighted surface of the angle beam **[B]**.
   - The profile is attached to the bracing plate.

Now use the **Reuse** command in the Components group to reassemble the bracing plate.

Once more align the angle beams correctly by using the **Align profile with selected entities from profile and another part** command in **Profiles** group.

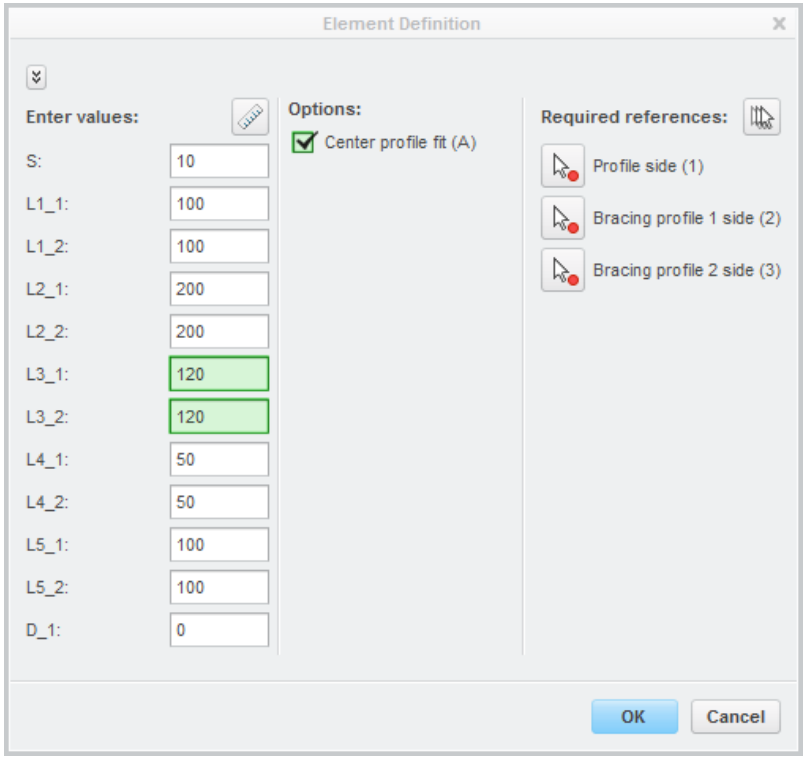


Type 2: Bracing plate double



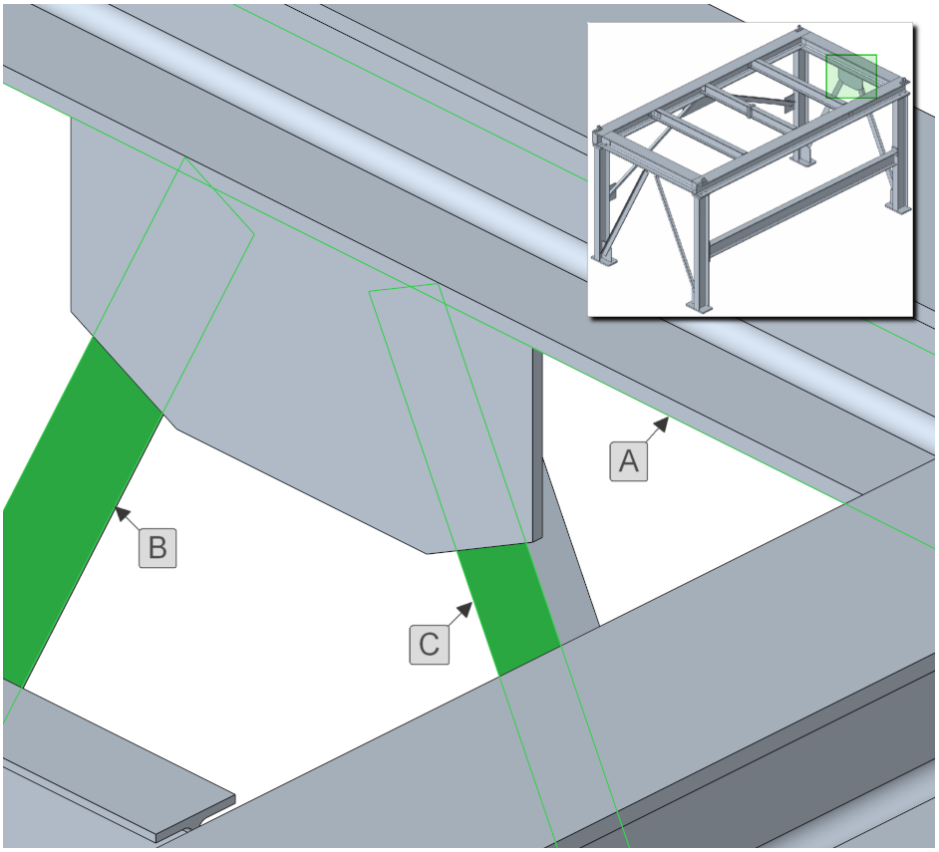
1. Click **New Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [BRACING PLATES] > [BRACING PLATE DOUBLE]**.
   - The **Element definition** dialog box of the **BRACING PLATE DOUBLE** opens.

     Define the connector as shown in the dialog.



4. Click **[Profile side (1)]**.
5. Select highlighted surface of the beam **[A]**.
6. Click **[Bracing profile 1 side (2)]**.
7. Select the highlighted surface of the angle beam **[B]**.
8. Click **[Bracing profile 2 side (3)]**.
9. Select the highlighted surface of the angle beam **[C]**.
10. Click **[OK]** to complete the definition.

- The bracing plate is be assembled and the angle beams are shortened.



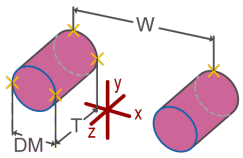
Use **Reuse** and select the instance method **Copy** to reassemble the plate at the front location.

2.4 Automatic UDF's

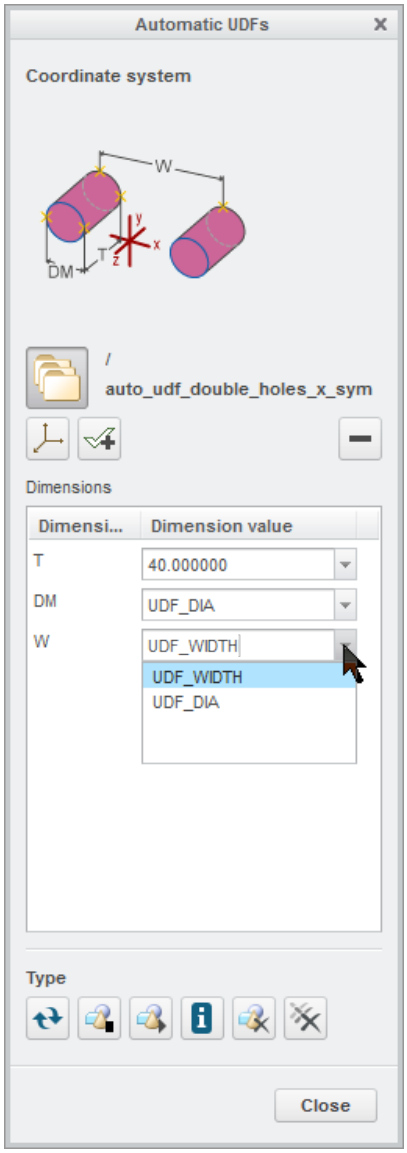
The **Automatic UDF**function is a very powerful utility of **AFX** and allows you to automate certain repeating and boring tasks. You simply define **Automatic UDF** information to a coordinate system of a part. Afterwards you can tell **AFX** to look for the **Auto UDF's** and to create the specified definitions (e.g. holes, cut-outs, nc-data, etc.) with just a click of a button.

Define Automatic UDF's

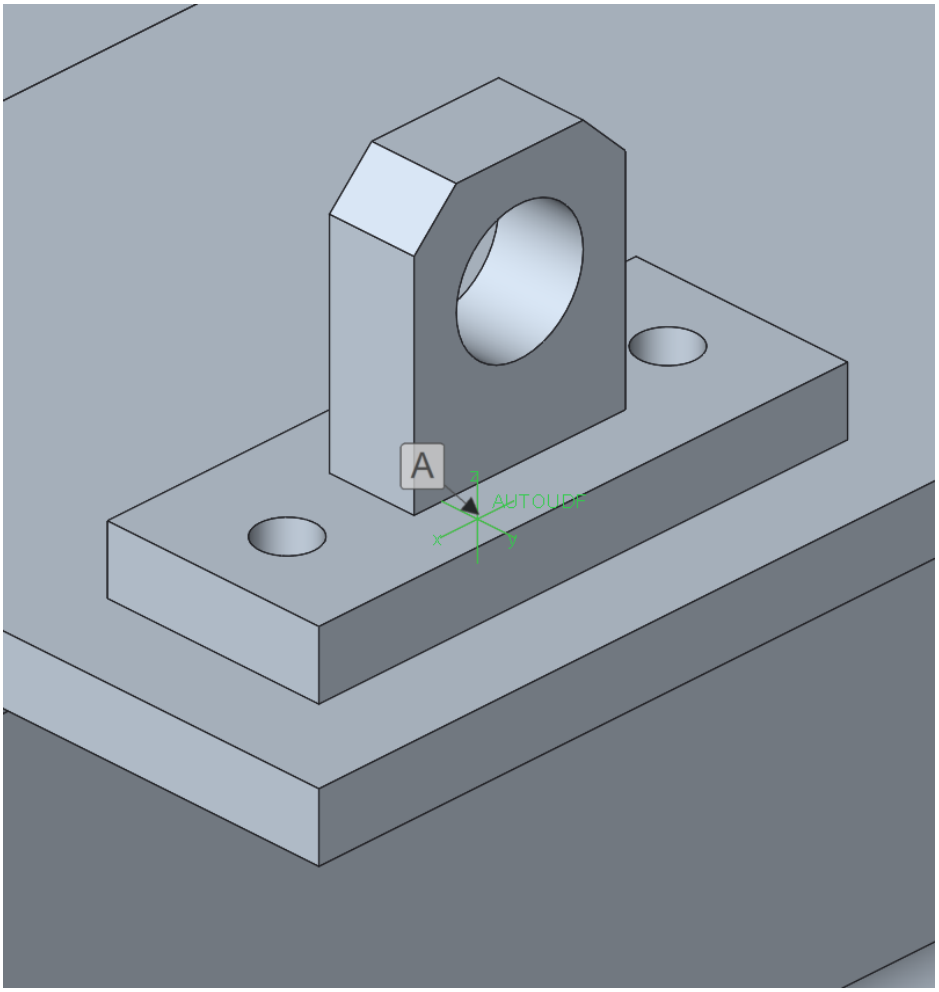
To define a **Automatic UDF** proceed as follows.



1. Click **Automatic UDFs** to open the **Automatic UDF's** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[AUTO_UDF_DOUBLE_HOLES_X_SYM]**.
4. Select the coordinate system AUTOUDF of one of the AUTOUDF.PRT **[A]**.
   - While modeling the AUTOUDF.PRT the creator already renamed some of the dimensions required for the **Automatic UDF**. **AFX** allows you to select them in the Dialog.



5. In the dialog choose UDF_DIA as **DM** and UDF_WIDTH as **W**
6. Click to complete the definition.



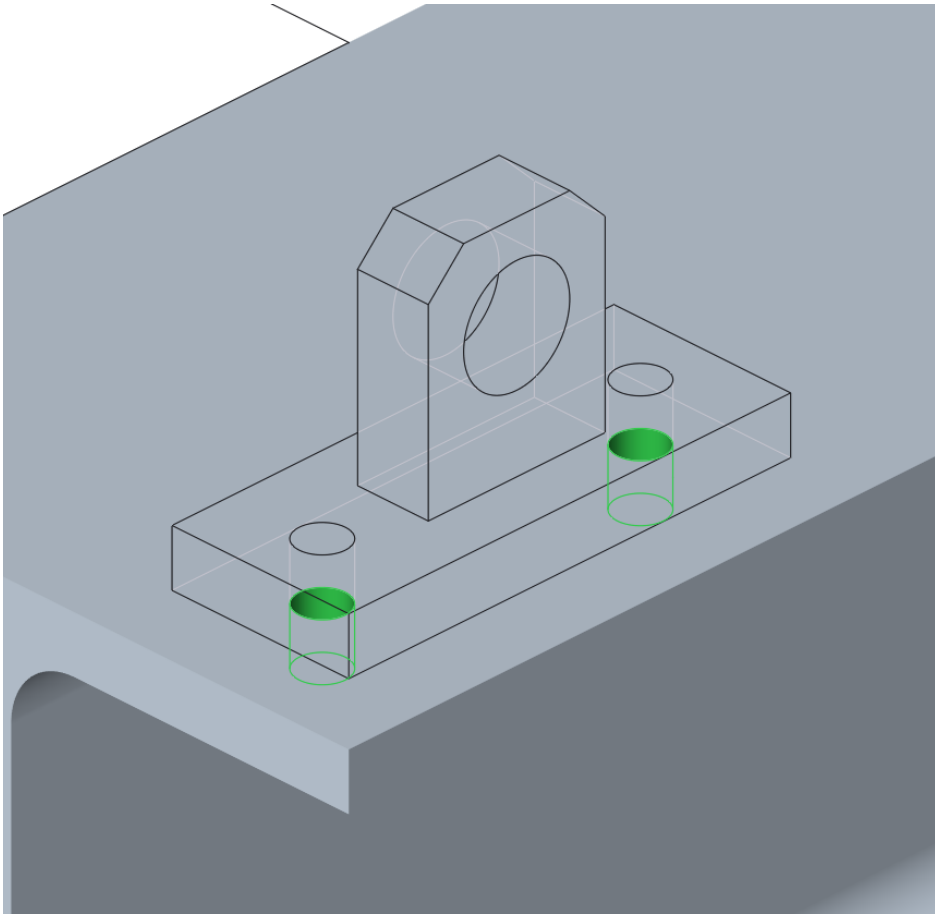
Define Profile End Cutouts

After you successfully created the definition, you of course want to **see** the resulting features in the models.

For that purpose **AFX** offers you the following functionalities:

- Create/Update UDF's
- Suppress all UDF's
- Resume all UDF's
- Shows coordinate systems with automatic UDF's
- Deletes all unrequired UDF's
- Delete all automatic UDF's

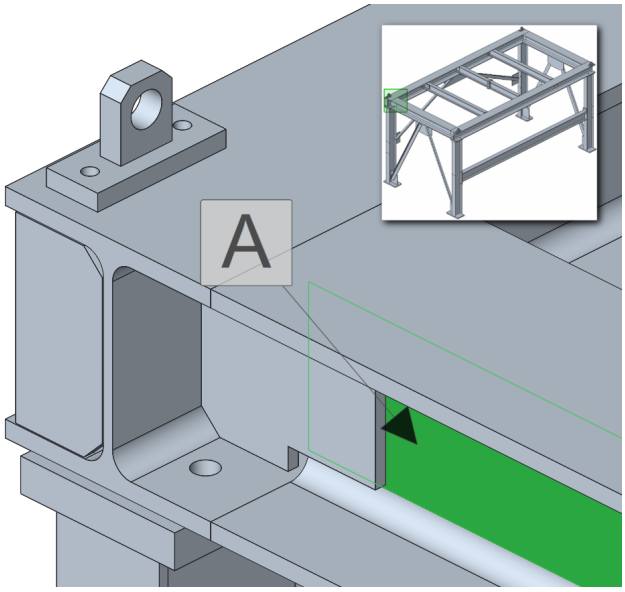
Finally create the just defined **automatic UDF** features via . The required holes will be created in all touching parts of the AUTOUDF.PRT.



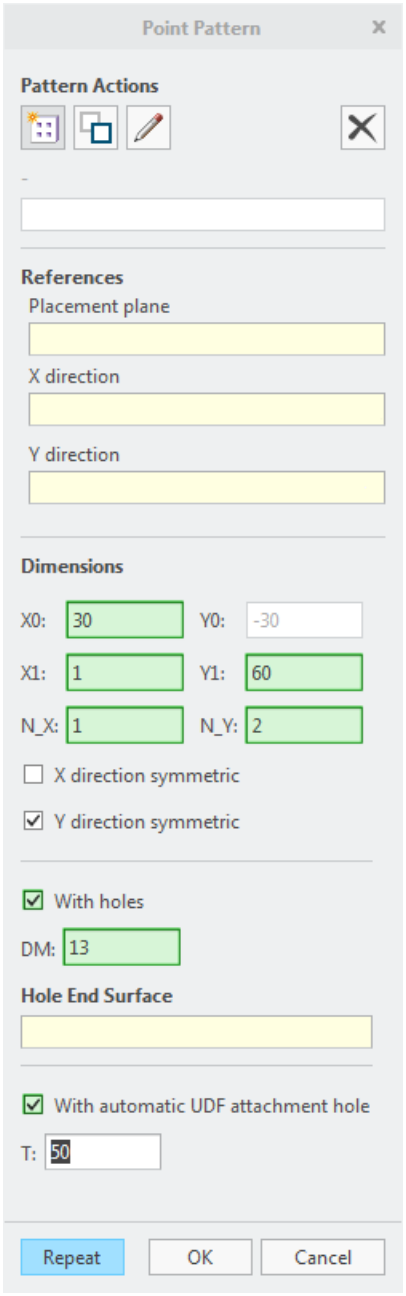
Hint 7—Delete Automatic UDF's. If you delete UDF's via one of the functions described above, **AFX** will only delete the resulting features, but not the definition of the automatic UDF's. To delete the definition you must use the delete function. To do so click and select the coordinate system with the automatic UDF definition.

Profile point patterns

AFX also supports the creation of point patterns. You can use them to just create points on profile ends, but you can also define automatic UDF's. To create profile point patterns proceed as follows.



1. Press **Point Pattern** to open the **Point Pattern** dialog.
2. Switch to the Point Pattern dialog.
3. Click to create a new point pattern.
4. Select the highlighted surface of the I-beam **[A]**.



5. Set the values as shown in the dialog.
6. Press **OK** to complete the definition.
   - The point pattern is now complete and the holes are made.

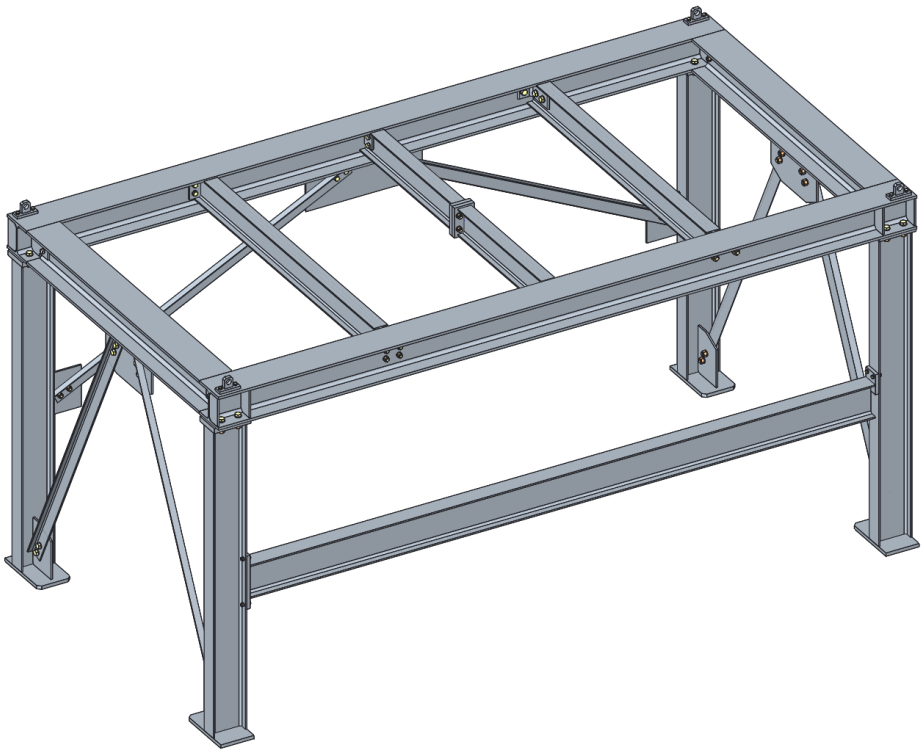
To create the required holes in the touching parts open the **automatic UDFs** dialog and click the button to create them.

The holes are now created in the touching parts.

2.5 Review

In this chapter you learned:

1. How to use various **DAST** and **AISC** connector elements.
   - Make sure you are familiar with **DAST** or **AISC** tables if you use them
   - Read carefully, which references the elements need.
   - Try to define references first, since it will automatically reduce the possible selections in the tables.
2. Working with **nonstandard** and **bracing plate** connectors.
   - You can achieve the same result with different plates.
   - It's up to you which one fits best for your case.
   - Remember, that you can manually create features with basic **Creo Parametric** functionality Plates from the equipment section can often be used as well
3. How to create **Automatic UDF's** and **Point Pattern**.
   - Automatic UDF's have a huge automation potential. Try to use them as often as possible.
   - You can even create your own UDF's and use them.
   - You can create **intelligent** parts, which automatically create their connection features on their touching neighbor parts.
4. **Optional exercise**
   - As exercise try to create more point patterns and also add screw connections to the connectors.



Notes:

3

Weldment groups

Overview
Define weldment groups
Define reuse places
Add components
Drawing creation of Weldment Groups
Create BOMs of weld-groups

3.1 Overview

The **AFX** weld group technique can be used for structuring assemblies. It uses simplified representations in **Creo Parametric**. As you define weld groups, you create simplified representations in the background. Later on you can use these simplified representations to make drawings of the different weld groups and also to create the bill of materials. Using the weld group technique for structuring the assembly in different weldments has some advantages compared to the subassembly technique later in this training guide.

- More flexibility to move components from a top-level assembly to a weld group or from one weld group to the other.
- Less external references from subassembly to top level assembly as all components are in the top level assembly.
- The possibility to design the whole frame at first and afterwards taking care about the structure of the weld groups.

However, especially for bigger structural steel assemblies it also has disadvantages.

- Structure of the structural steel assembly in **Creo Parametric** (model tree) does not correspond to the weld group structure, so it is difficult to keep the overview.
- Management of drawings, BOMs etc. is more difficult, especially if you use **Windchill PDMLink**, as weld groups are not handled as separate objects in these systems.
- Copying of complete units (profile with endplates etc.) is more difficult.

An **AFX** weld group consists of the following elements.

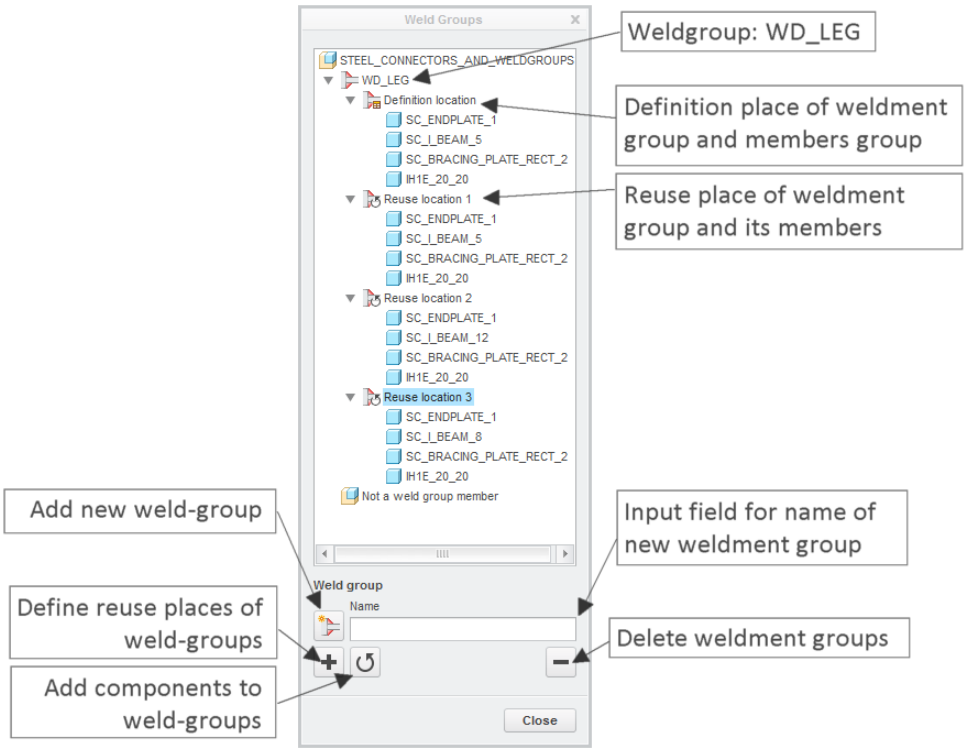
- A **Creo Parametric** simplified representation named `WG_<weldment_group_name>` which represents the definition place of the weld group
- A **Creo Parametric** simplified representation named `WG_<weldment_group_name>_ALL` which represents the definition place and all reuse places of the weld group
- A bulk item model named `WG_<weldment_group_name>`. You can add parameters to this model to show the weld group BOM information later on.

Besides this, a **Creo Parametric** simplified representation named `NOT_WELDMENT_MEMBERS` will be created which contains all components that do not belong to a weld group.

The **weld groups** dialog box contains a tree which shows the already defined weld group structure. To highlight the elements of a weld group in the **Creo Parametric** graphics window, select the weld group item in the tree. To find the weld group information of a component in the **Creo Parametric** graphics window, select the component while the weld group dialog is open.

AFX weld group functions can be accessed by  **Weld Groups**.

The dialog box has the following layout.

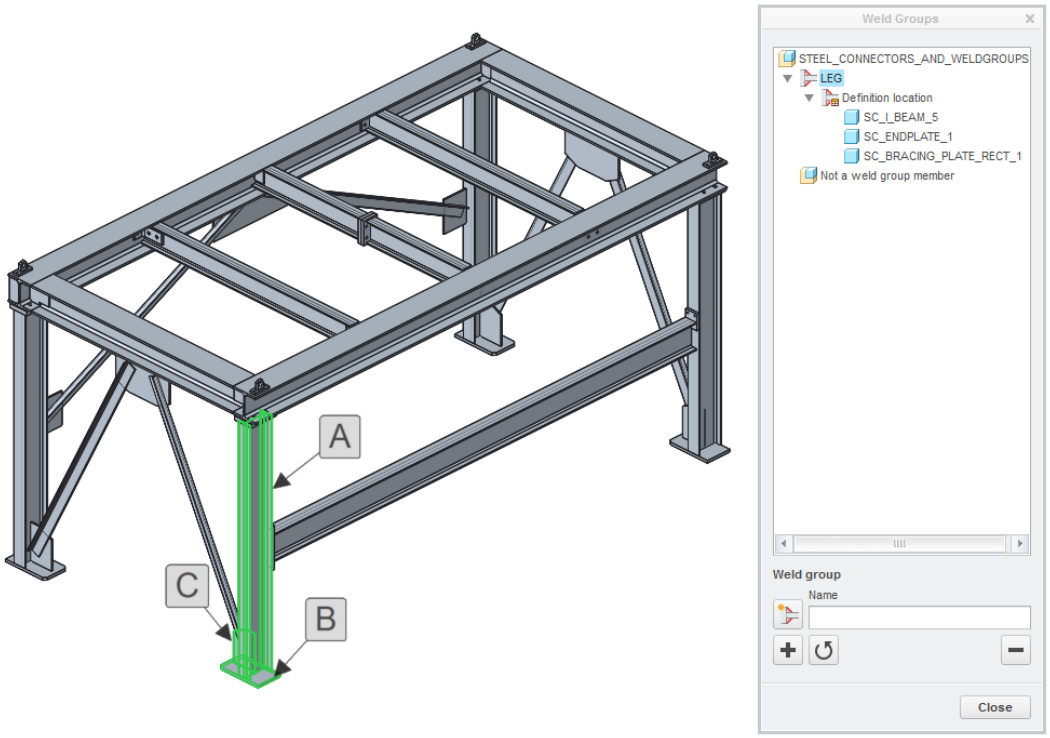


3.2 Define weldment groups

Before getting started we have to delete the automatic UDF holes, because you can only define reused weld groups, if the parts are identical. Open the **Automatic UDFs** dialog and press to delete all automatic UDF holes.

To define a new weldment group for a leg of the frame, in weldment group dialog box proceed as follows.

1. Click **Weld Groups** to open the **Weld Groups** dialog box.
2. Enter the desired name **[LEG]** in the input panel and press to create a new weld group.
3. Select the I-Beam **[A]**, the bottom endplate **[B]** and the bracing plate **[C]** of one leg and click OK or press **[MMB]**.



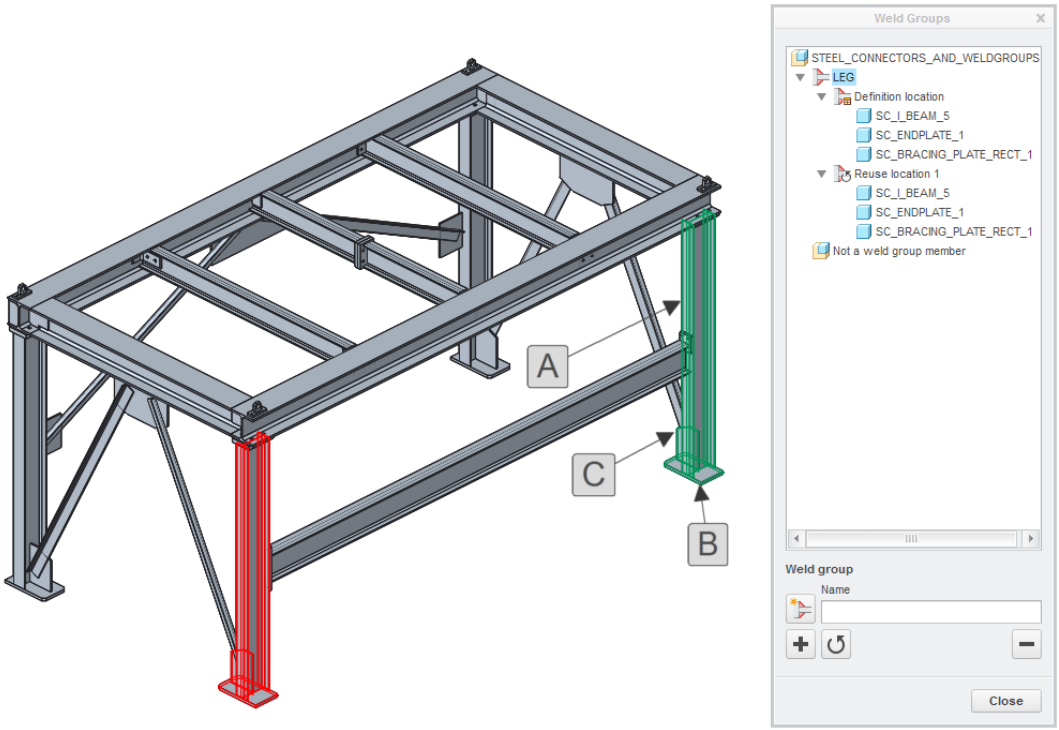
The weldment group is created and can be seen in the weldment group tree. Besides this, the simplified representations are created and a bulk item `WG_LEG.prt` is assembled. This bulk item can be used to define parameter for the specific weld group.

Hint 8 — Visible components in weld-groups. Components that are already members of other weldment groups will be hidden after you press in the dialog box.

3.3 Define reuse places

As the two legs of the frame are identical weldment assemblies, you can define a reuse place in the weldments dialog box.

1. Click on one of the leg weldment group items in the weldment group tree and press ↺.
2. On another corner of the frame select the leg I-Beam **[A]**, the bottom endplate **[B]** and the bracing plate **[C]** of one leg. Make sure to select the components in the correct order!

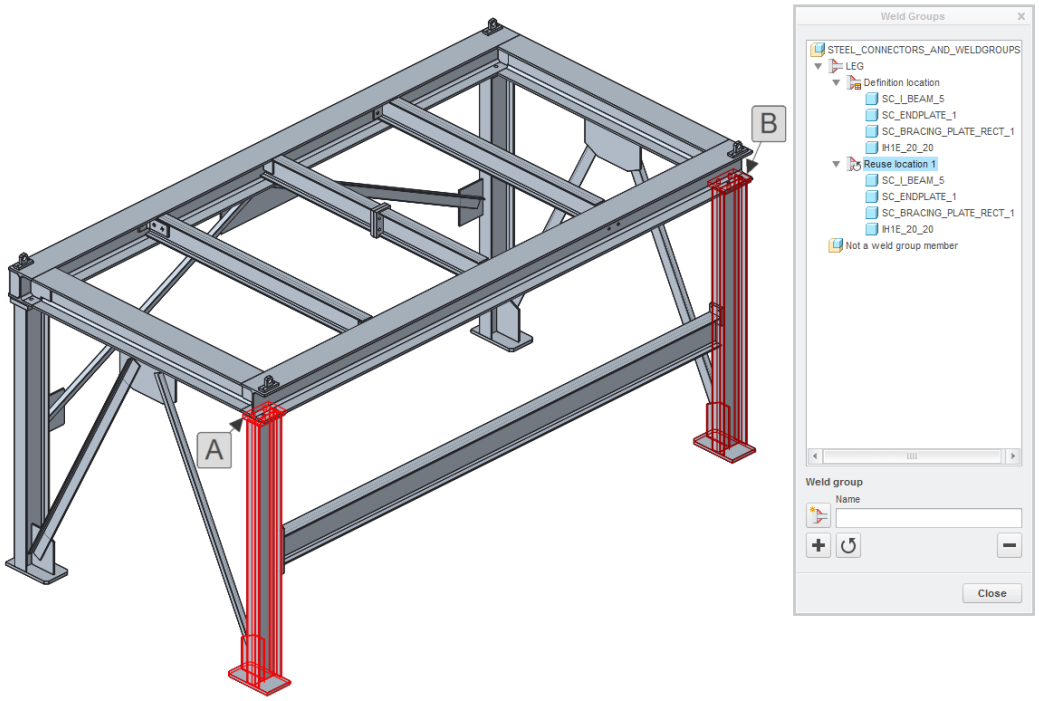


The simplified representations are updated and another bulk item `WG_LEG.prt` is assembled.

3.4 Add components

In this example we forgot to add the **top endplate** of the legs to the weldment group. To add the top endplate to the leg weldment group proceed as follows.

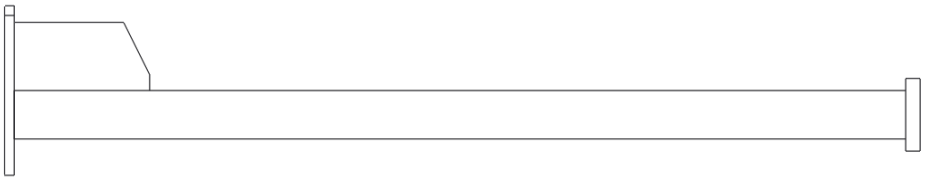
1. Click on one of the leg weldment group items in the weldment group tree and press ✚.
2. At the definition place of the leg weldment group select the upper endplate **[A]**. Confirm your selection via OK or **[MMB]**.
   - As the selected weld group has a reuse location, the endplate must also be selected for the reused place.
3. Select the endplate at the reuse location **[B]** and confirm your selection via OK or **[MMB]**.



The endplate is now added to the weld group **LEG**.

3.5 Drawing creation of Weldment Groups

To create drawings of weldment groups with regular **Creo Parametric** functionality you can create views which just show the simplified representation that belongs to the weldment group and not the complete assembly.



1. Open the drawing `steel_connectors_and_weldgroups.drw`, it shows one view of the complete assembly.
2. Create a new sheet and make it active.

3. Insert a new general view and select orientation **VORNE**.
4. Switch to **View States** and select the Simplified Representation **[WG_LEG]**.
   - The drawing view is created and shows only the members of the weld-group WG_LEG.

3.6 Create BOMs of weld-groups

To create BOMs of weld-groups and assemblies with weldment elements you can use regular **Creo Parametric** repeat region functionality.

1. Add parameters to the bulk part WG_LEG.prt.
   - NAMING = Leg weld group
   - DESIGNATION = 12345
2. Set the simplified representation **WG_LEG** as active model.
3. Add the table afx_bom.tbl from the tutorial folder.

Pos	Qty	Name	Description
1	1	ENDPLATE	IH I E 20 20
2	1	I BEAM	DIN 1025 IPE 200 x 1850
3	1	PLATE	200x350x20
4	1	PLATE	281x188x10

   Only the parts of the simplified representation will be shown in the BOM.
4. Switch back to sheet 1.
5. Set the simplified representation **[NOT_WELDMENT_MEMBERS]**.
6. Add the table **[afx_bom]** from the tutorial folder.

Pos	Qty	Name	Description
1	4	Angle	IW 20 21 180x90x10
2	4	ANGLE BEAM	DIN 1029 L 80x40x6 x 1918
3	2	ANGLE BEAM	DIN 1029 L 80x40x6 x 1976.7
4	2	ENDPLATE	IH I E 16 16
5	2	ENDPLATE	IH I E 20 20
6	2	ENDPLATE	IS 16 4 80
7	2	I BEAM	DIN 1025 HEB 200 x 1800
8	2	I BEAM	DIN 1025 HEB 200 x 4200
9	2	I BEAM	DIN 1025 IPE 160 x 1971
10	2	I BEAM	DIN 1025 IPE 160 x 970.5
11	2	I BEAM	DIN 1025 IPE 200 x 1850
12	1	I BEAM	DIN 1025 IPE 200 x 3760
13	2	Leg weld group	12345
14	3	PLATE	
15	1	PLATE	100x120x20
16	1	PLATE	100x320x20
17	4	PLATE	168x169.5x10
18	2	PLATE	168x84.5x10
19	2	PLATE	200x350x20
20	2	PLATE	281x188x10
21	2	PLATE	476x296x10
22	1	PLATE	700x250x10
23	1	PLATE	95.5x107.6x10

As the current simplified representation is `NOT_WELDMENT_MEMBERS`, only components which do not belong to any weldment group are listed, so the parts which belong to the leg weldment are not listed. As the bulk item `WG_LEG.prt` does not belong to `WG_LEG` simplified representation, but to `NOT_WELDMENT_MEMBERS`, the complete leg weldment group is listed as one item with quantity 2.

4

Seperate weldments

Overview
Define new subassemblies
Edit the subassemblies

4.1 Overview

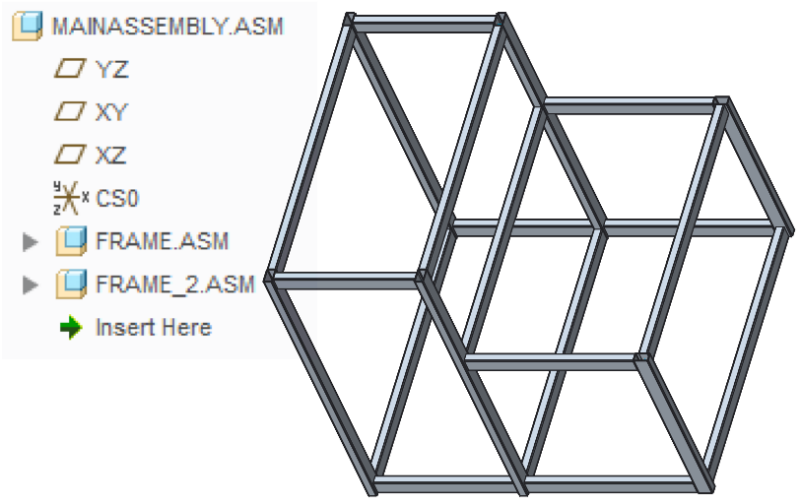
Sometimes it is necessary to divide large weldments into smaller subassemblies, which are assembled in structural steel assemblies later on. This can for example happen, when the weldment has become too large for transportation.

4.2 Define new subassemblies

Open the FRAME.ASM from the separation_of_weldments_start folder to begin with the next exercise.

As you can see the whole frame is in one assembly, which even contains a skeleton part. We now want to split the large assembly into two smaller ones. As the assembly contains a skeleton, this will be a really simple task.

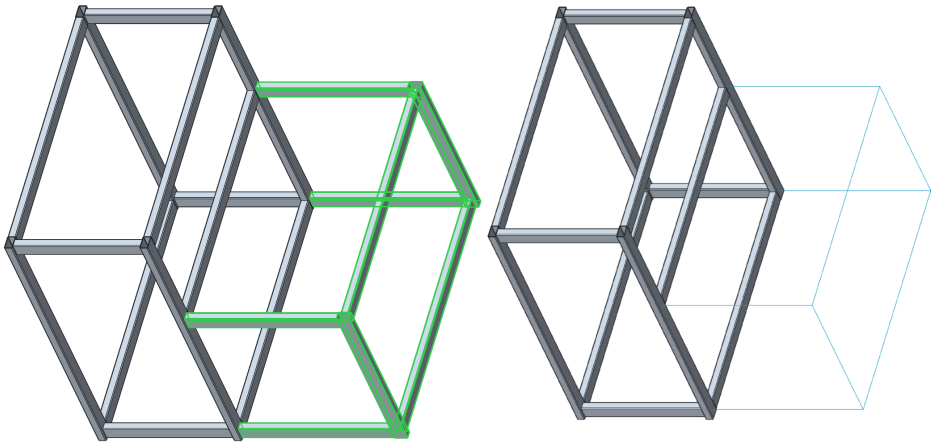
1. Create a new assembly called MAINASSEMBLY.ASM. This will be our main assembly, which will contain the new subassemblies of the split frame.
2. Copy the FRAME.ASM to create two assemblies, which contain the whole content. Name it FRAME_2.ASM.
3. Now assemble the two assemblies FRAME.asm and FRAME_2.ASM into the MAINASSEMBLY.ASM with default constraints.



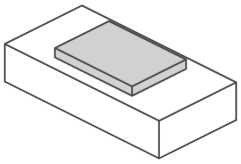
4.3 Edit the subassemblies

Both subassemblies are now on top of each other. In the next steps both sides will be detailed.

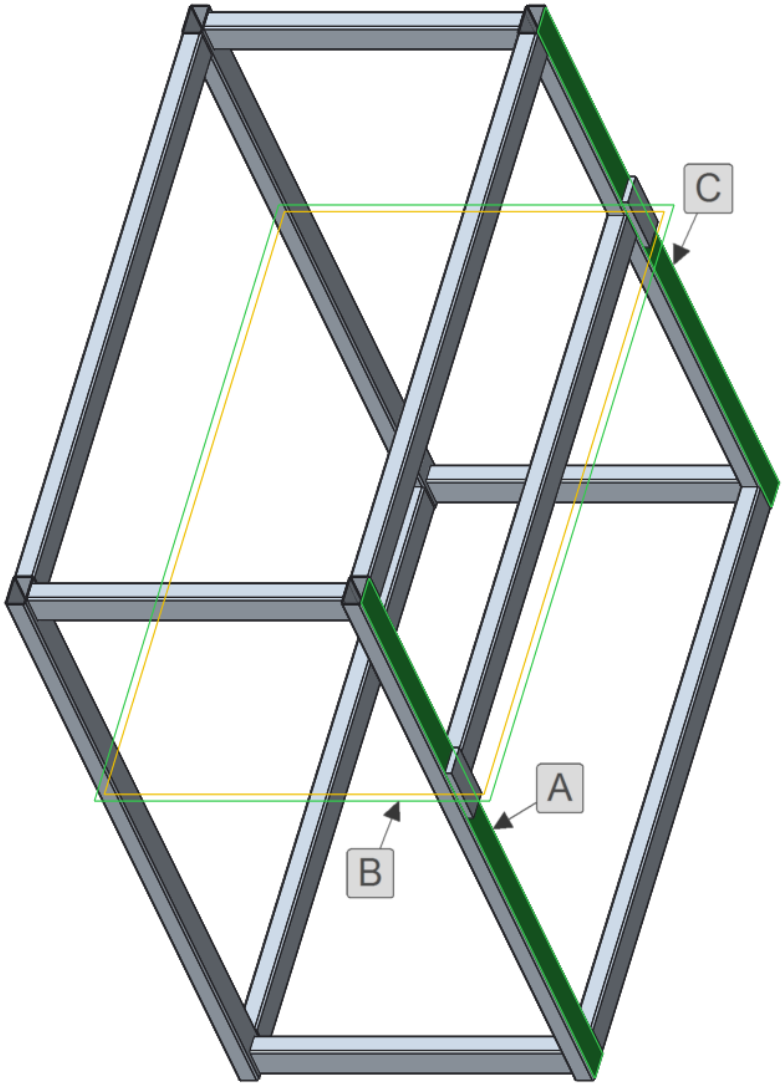
1. Open the FRAME.ASM.
2. Delete all the highlighted Profiles via **Delete** as shown in the picture below.



For the mounting purpose add a **WELD PLATE**.



1. Click **Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [WELD PLATE]** and press **Next**.
   - The Element definition dialog box of the plate opens.

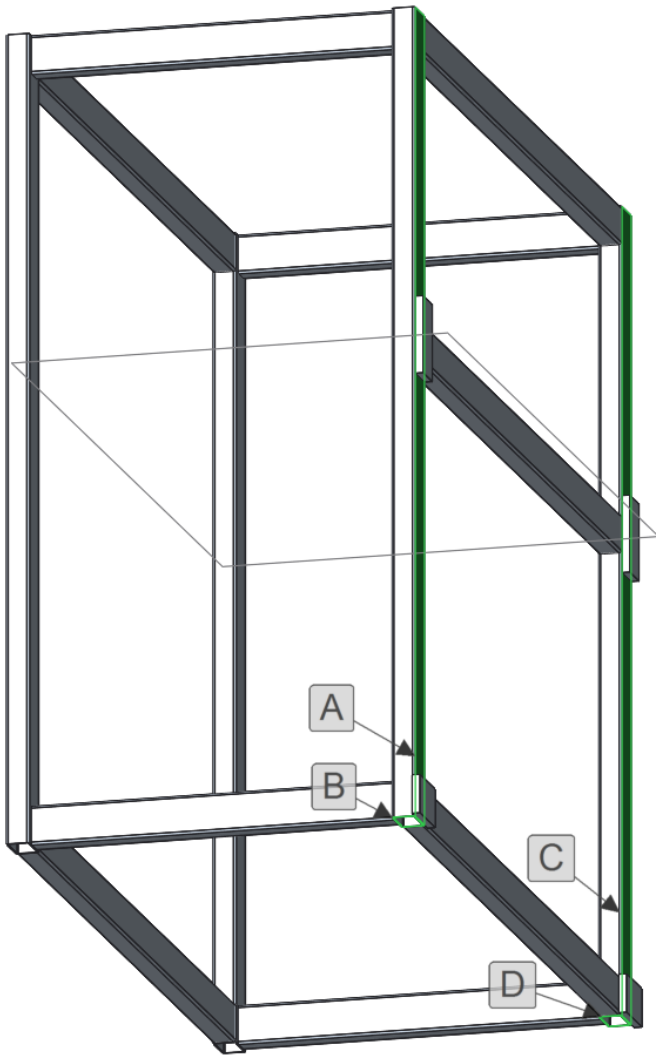


4. Define values: **[W=90]**; **[No. of point columns=1]**; **[OFFSET_1=-100]**.
5. Click **[Attach face (1)]** and select the highlighted surface **[A]**.
6. Click **[Face (2)]** and select `MEDIUM_HEIGHT` of the `SKELETON.prt` **[B]**.
7. Click **[OK]** to complete the definition.
   - The 1st weld plate is assembled.
8. Now click to reassemble the plate to the other side.
9. Select the first plate as element to reassemble.
10. Select the highlighted surface **[C]** as **[Attach face (1)]**.

11. Select the plane MEDIUM_HEIGHT of the SKELETON.prt as **[Face (2)] [B]**.
12. Enter **[-100]** as **OFFSET_1**.
    - The 2nd weld plate is assembled.

The next step is to assemble and reassemble weld plates to the bottom of the frame.

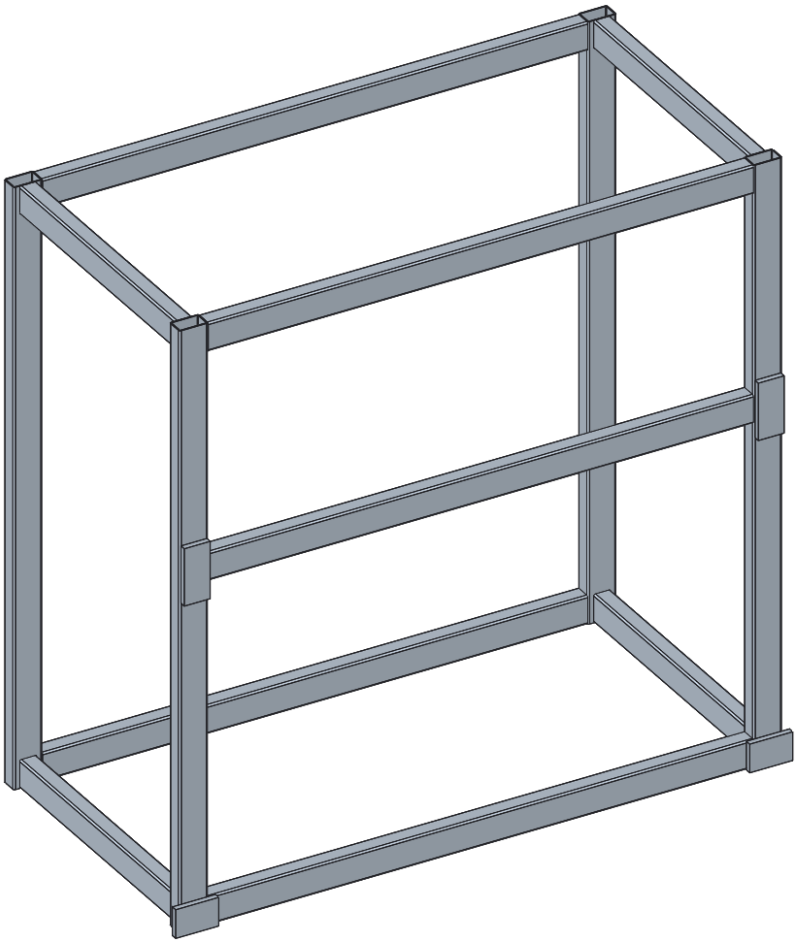
1. Click **Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [WELD PLATE]**.
    - The Element definition dialog box of the plate opens.



4. Define values: **[H=100]**; **[No. of point columns=1]**.
5. Click **[Attach face (1)]** and select the highlighted surface **[A]**.
6. Click **[Face (2)]** and select the highlighted surface **[B]**.

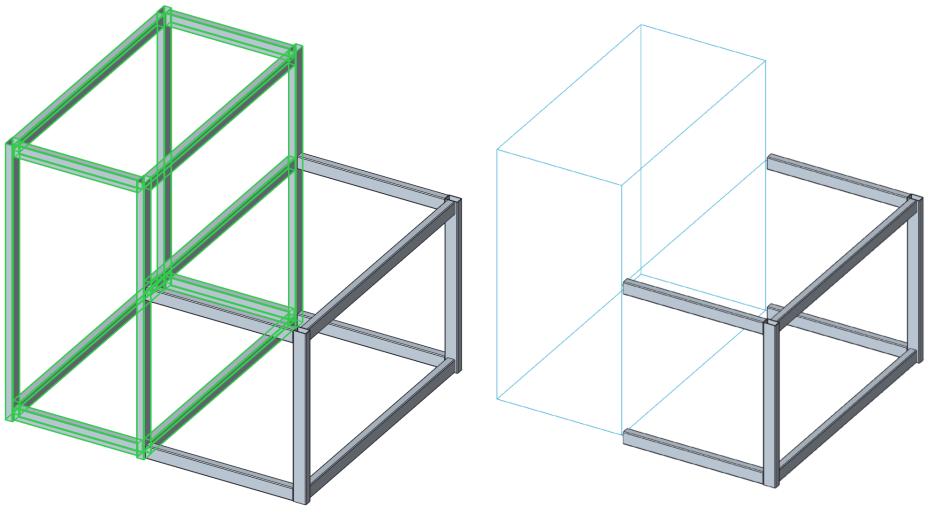
7. Click **[OK]** to complete the definition.
   - The 1st weld plate is assembled.
8. Now click  to reassemble the plate to the other side.
9. Select the first plate as element to reassemble.
10. Select the highlighted surface **[C]** as **[Attach face (1)]**.
11. Select the highlighted surface **[D]** as **[Face (2)]**.
12. Enter **[0]** as **OFFSET_1**.
    - The 2nd weld plate is assembled.

The first half of the seperation process is now complete.

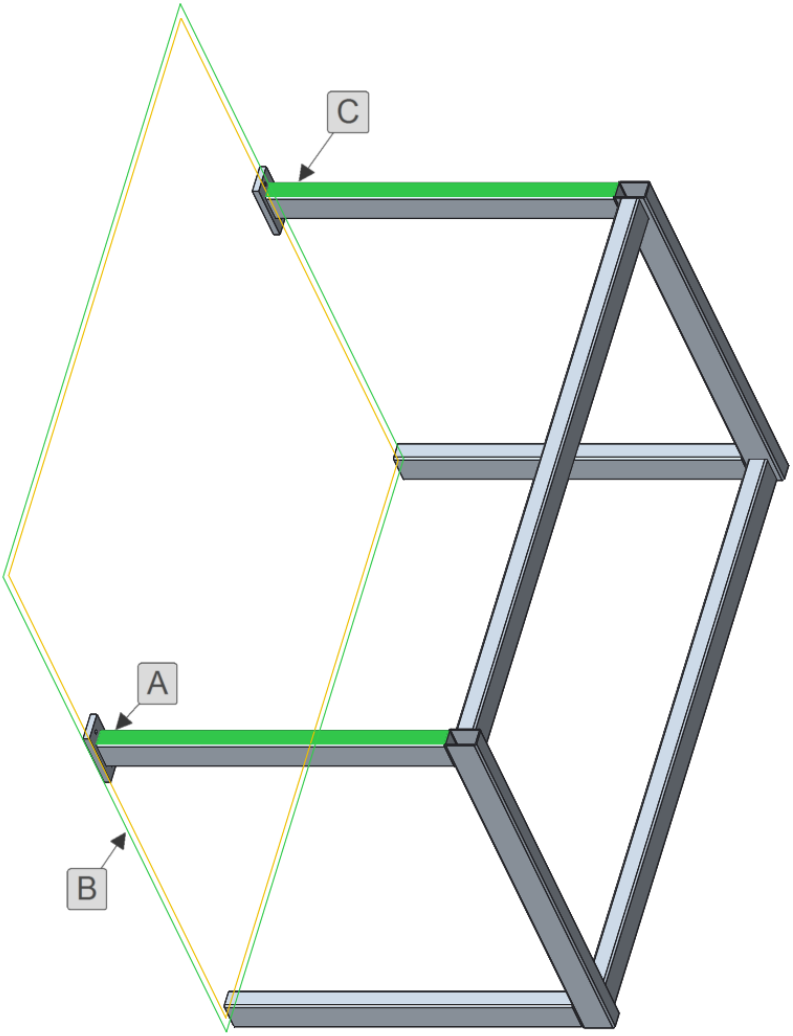


To complete the division of the frame proceed as follows.

1. Open the  `FRAME_2.ASM`.
2. Delete all the highlighted Profiles via  **Delete** as shown in the picture below.



1. Click **Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE]** and press **Next**.
   - The Element definition dialog box of the plate opens.

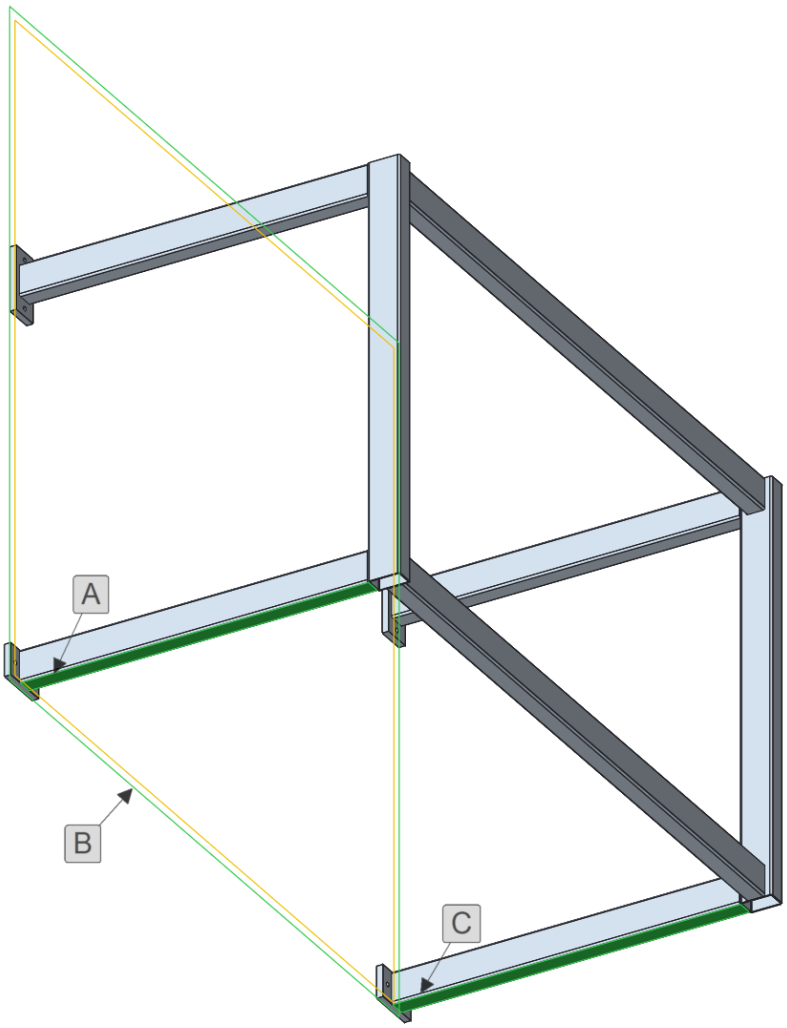


4. Define values: **[W=90]**; **[No. of point columns=1]**; **[HOLE_DM=13]**; **[check option holes (A)]**
5. Click **[Profile top (1)]** and select the highlighted surface **[A]**.
6. Click **[Attach face (2)]** and select the highlighted surface **[B]**.
7. Click **[OK]** to complete the definition.
   - The 1st end plate is assembled.
8. Now click to reassemble the plate to the other side.
9. Select the first plate as element to reassemble.
10. Select the highlighted surface **[C]** as **[Profile top (1)]**.
11. Select the highlighted surface **[B]** as **[Attach face (2)]**.

- The 2nd end plate is assembled.

Now assemble two connectors of type **ENDPLATE TOP**.

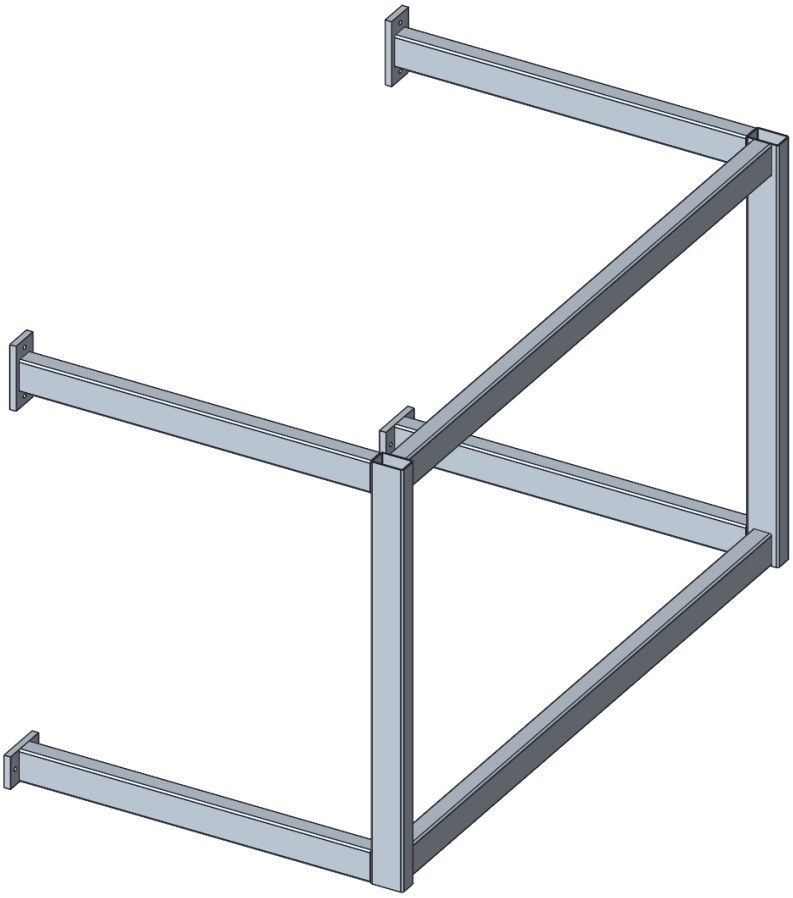
1. Click **Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [ENDPLATE TOP]**.
   - The Element definition dialog box of the plate opens.



4. Define values: **[H=100]**; **[W=150]**, **[No. of Point rows=1]**; **[offset=0]**; **[HOLE_DM=13]**; **[check option holes (A)]**.
5. Click **[Attach face (2)]** and select the highlighted surface **[A]**.
6. Click the **[Profile top (1)]** and select the `DIVIDE_PLANE` of the `SKELETON.prt` **[B]**.
7. Click **[OK]** to complete the definition.
   - The 1st end plate top is assembled.
8. Now click to reassemble the plate to the other side.

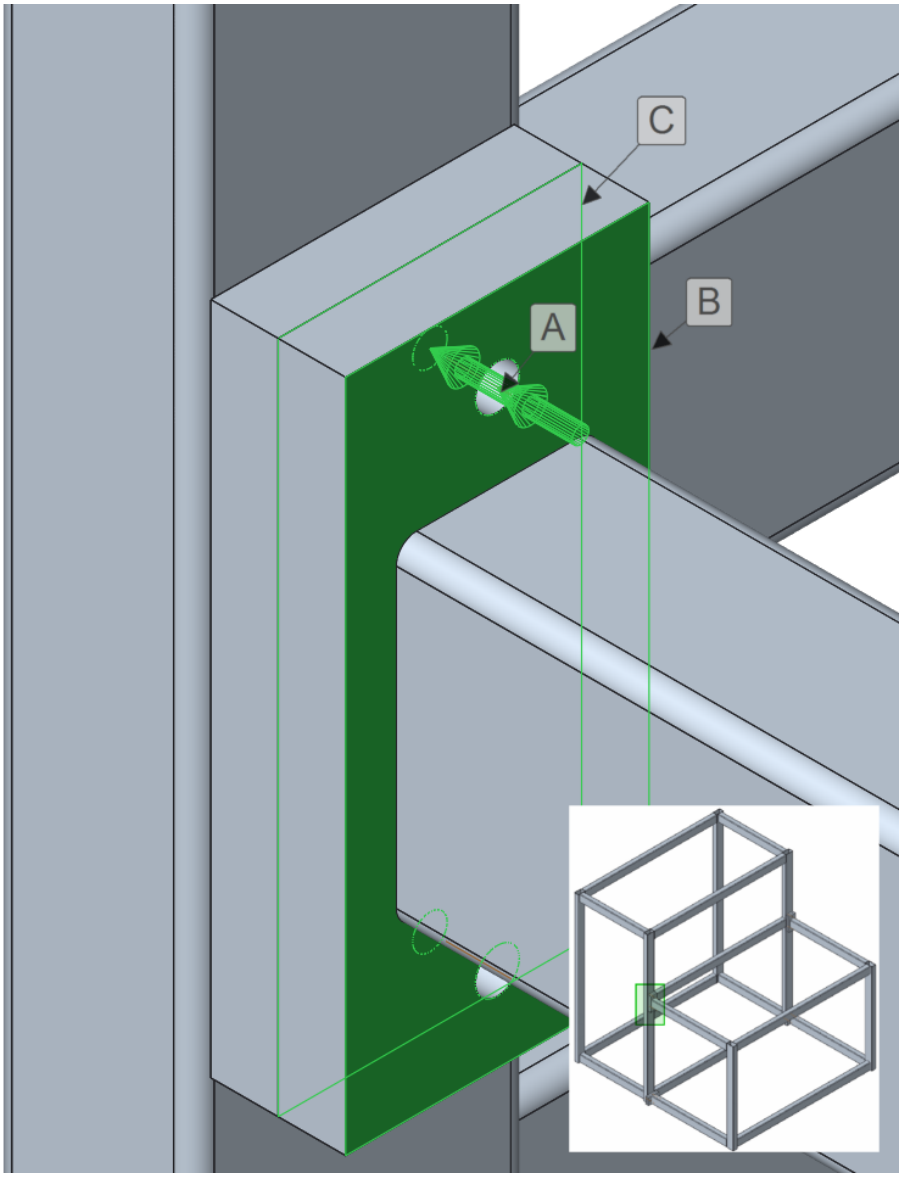
9. Select the previously assembled plate as element to reassemble.
10. Select the highlighted surface **[C]** as **[Profile top (1)]**.
11. Select the highlighted surface **[B]** as **[Attach face (2)]**.
    - The 2nd end plate top is assembled.

The 2nd half is now complete.



Now switch back to the MAINASSEMBLY.ASM.

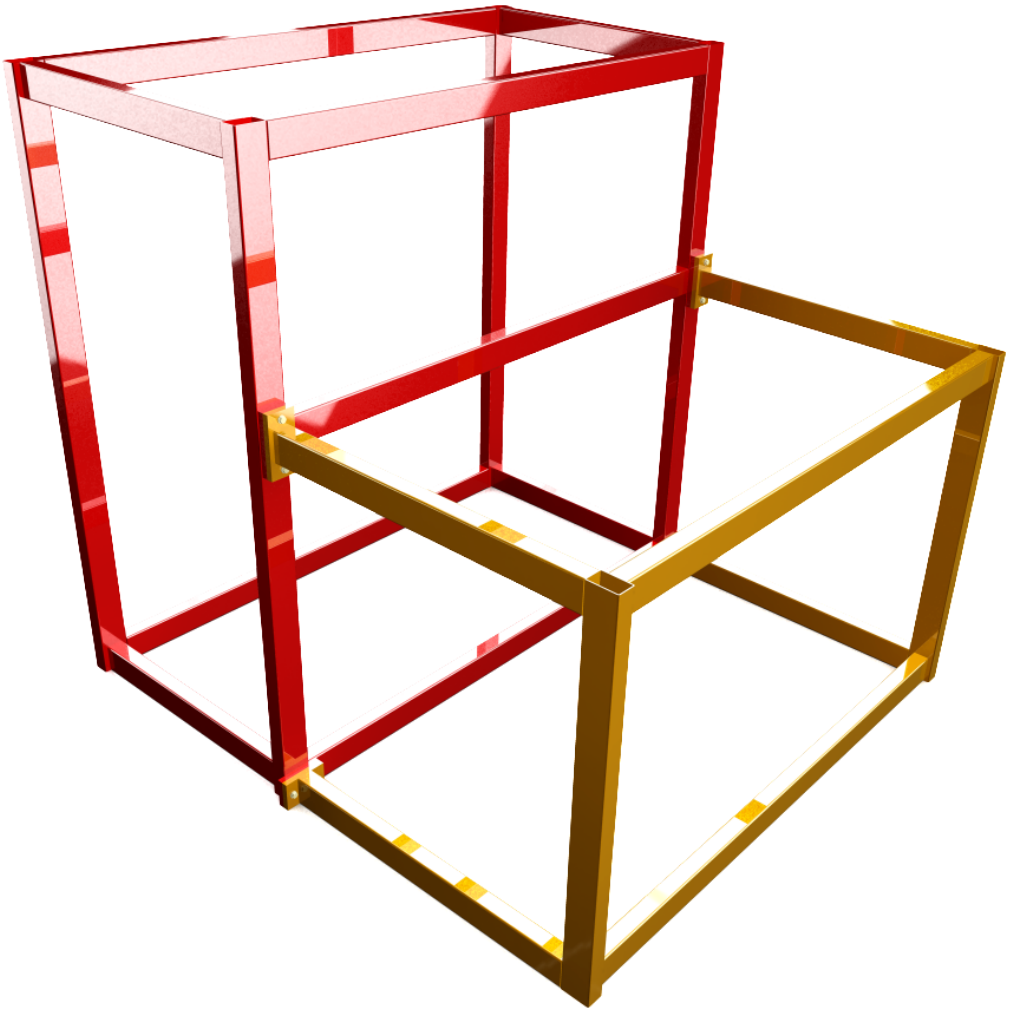
To complete the assembly add screw connections to the just added plates.



1. Click **Assemble on point or axis**.
2. Select the axis **[A]** as **Position Reference**.
3. Select the surface **[B]** of the plate as **screw head** reference.
4. Select the surface **[C]** as **thread start surface**.
5. Click **[OK]** to complete the definition.
   - The screw dialog opens.
6. Define the screw connection:
   - Screw type **ISO 4017 8.8**
   - Washer type **DIN 125-1-A**
7. Press and select the hole surface to automatically receive the correct screw diameter.

8. Press  and the screw length will be changed to achieve a correct depth.
9. Click **[OK]** and select Pattern screw connection to complete the definition.

Optionally you can define screw connections to the remaining plates. The assembly is now complete.



5

Project Subassemblies

Overview
Assemble subassemblies
Reassemble subassemblies
Exercise
Copy subassemblies
Move and rotate
Exercises
Review

5.1 Overview

Besides structuring assemblies with the **AFX** weldment group functionality, **AFX** also offers a different solution to work with project subassemblies. This method will result in real **Creo Parametric** subassemblies.

The main advantage of this technique is that the **Creo Parametric** model tree structure corresponds to the weldment or assembly structure, so it is easy to make drawings, **BOMs** and also to perform **Windchill PDMLink** tasks.

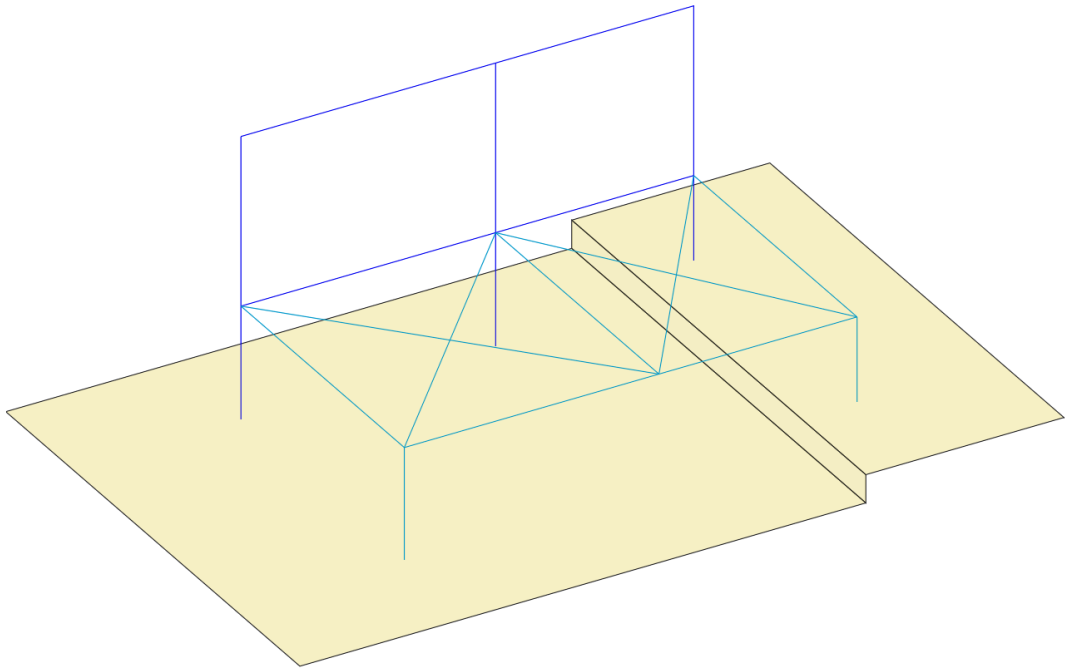
The main disadvantage is that it is necessary to know the weldment structure before starting the design. Restructuring is more difficult than using the **AFX** weldment group functionality. To restructure your design you can use the method described in chapter 4.

There are two main techniques when working with **AFX** project subassemblies:

1. Referencing profiles directly on the curve skeleton
   - This technique generates a lot of **external references** from the subassembly to the top level assembly and makes copying and moving of subassemblies more **difficult** and sometimes confusing.
2. Referencing profiles to curves inside the project subassembly
   - This technique tries to **avoid external references** from the subassembly to the top level assembly to make copying and moving of subassemblies easier and generate **better Creo Parametric** models.

   This technique is described in the following chapter.

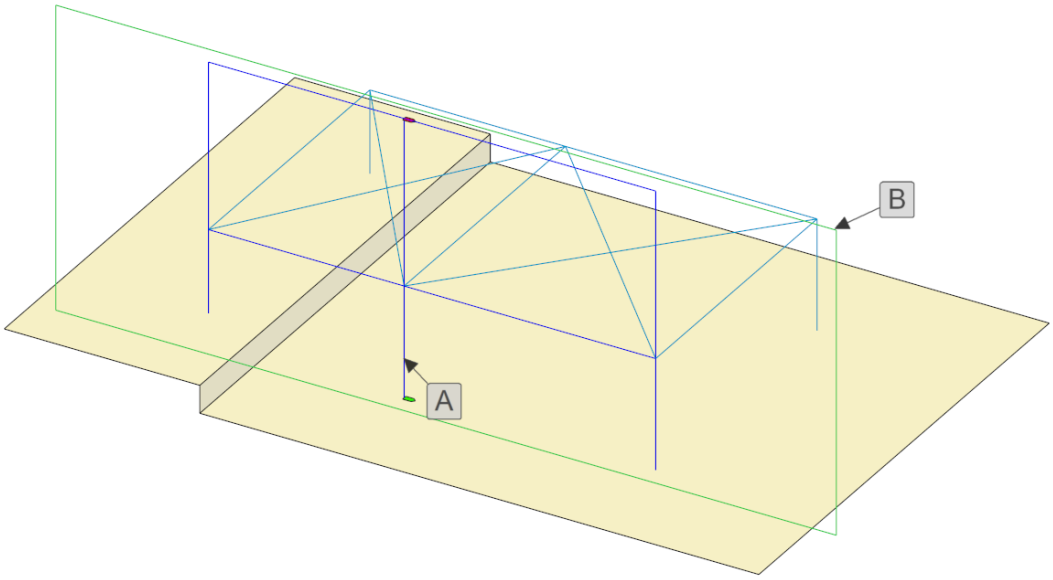
For the examples in this chapter open the assembly `PROJ_SUBASSEMBLIES.ASM`, which already contains a curve skeleton. Also a new **AFX** project called **ps** has already been set.



5.2 Assemble subassemblies

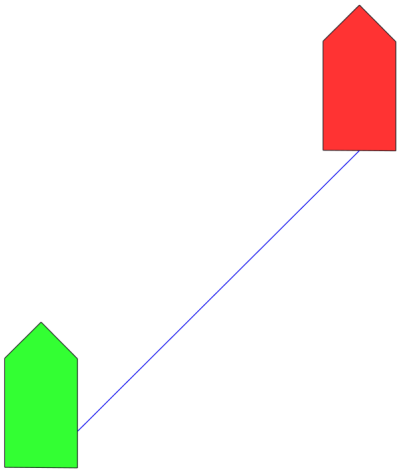
To design a column of the frame, in the project subassemblies dialog box proceed as follows.

1. Click **New Project Subassembly**.
2. Select the center vertical curve **[A]** as placement curve.
3. Select datum plane A_XY **[B]** as orientation plane.
4. Enter name **[ps_column_1]** into the subassembly name input field.
   - A new subassembly named ps_column_1 is assembled.



The subassembly consists of the following elements.

- Three datum planes
- Four coordinate systems
- Two arrow shaped surfaces at the assembly **start** and **end**.
- One datum curve (which corresponds in length with the selected placement curve).

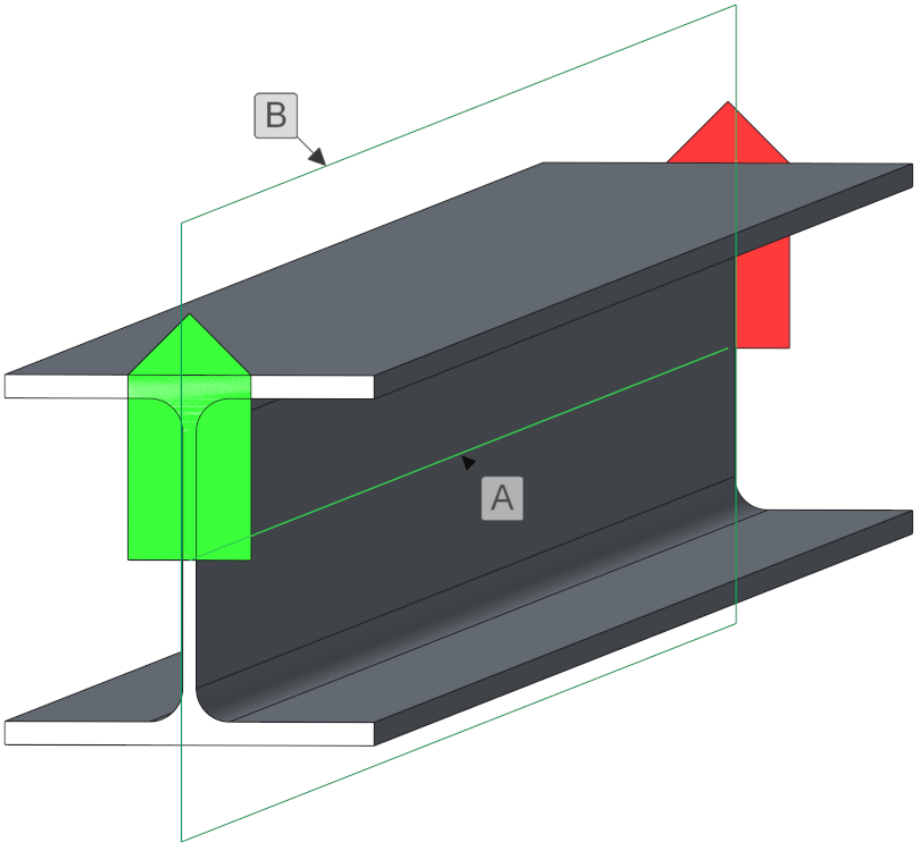


Now you can continue to design the column subassembly either in the main assembly environment or directly in the subassembly ps_column_1 in a separate window.

Due to selecting the correct references in the subassembly it's recommended to **open** the subassembly in a **new window**.

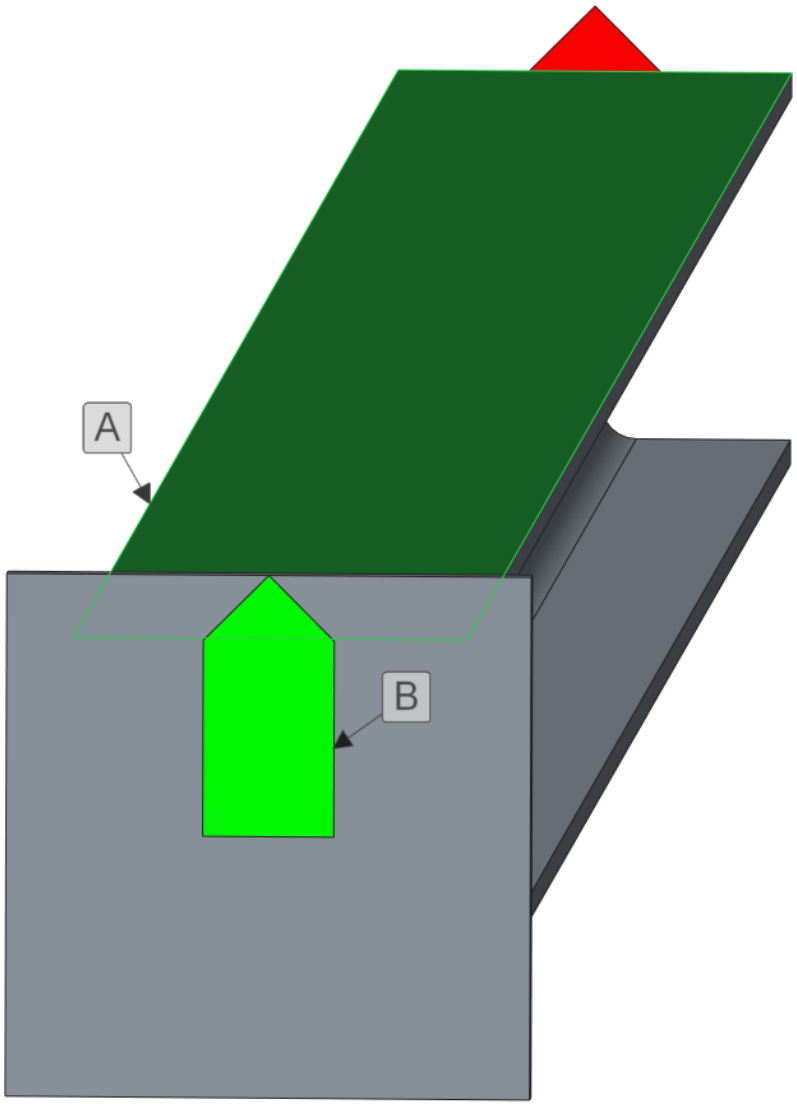
To assemble a new Profile into the subassembly proceed as follows.

1. Click **Profiles** to open the **Profiles** dialog.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL BEAMS MM] > [I-BEAM] > [DIN 1025 IPB/HEB] > [300]**.
4. If you work in the main assembly make sure to activate the subassembly ps_column_1 before you assemble the beam.
5. Select the datum curve **[A]** of the subassembly. If you select the curve from the main assembly you will generate additional external references. The profile is now added into the subassembly.
6. Select the datum plane A_YZ **[B]** of the subassembly as orientation plane.



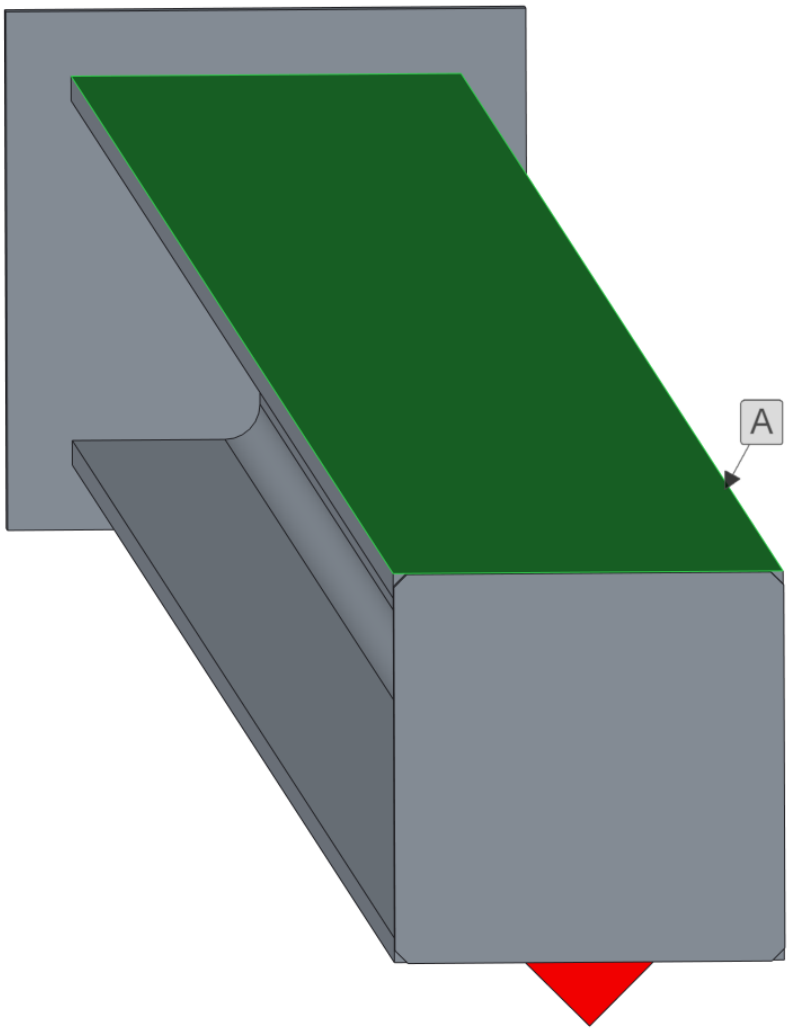
It’s highly recommended that you open the subassemblies in a new window from now on to prevent the selection of wrong references that will lead to external references.

Before you switch into a new window check the orientation of the subassembly. In this case the green arrow touches on the floor.



1. Click **Connector Elements**.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NO STANDARD] > [ENDPLATE]**.
   - The Element definition dialog box of the end plate opens.
4. Enter values: **[H=400]**; **[W=400]**.
5. Click **[Profile end (1)]** and select the highlighted surface **[A]**.
6. Click **[Attach face (2)]** and select green arrow shaped surface **[B]**.
7. Click **[OK]** to complete the definition.
   - The end-plate will be assembled and the profile gets shortened.

On the other side of the profile add an end cap.



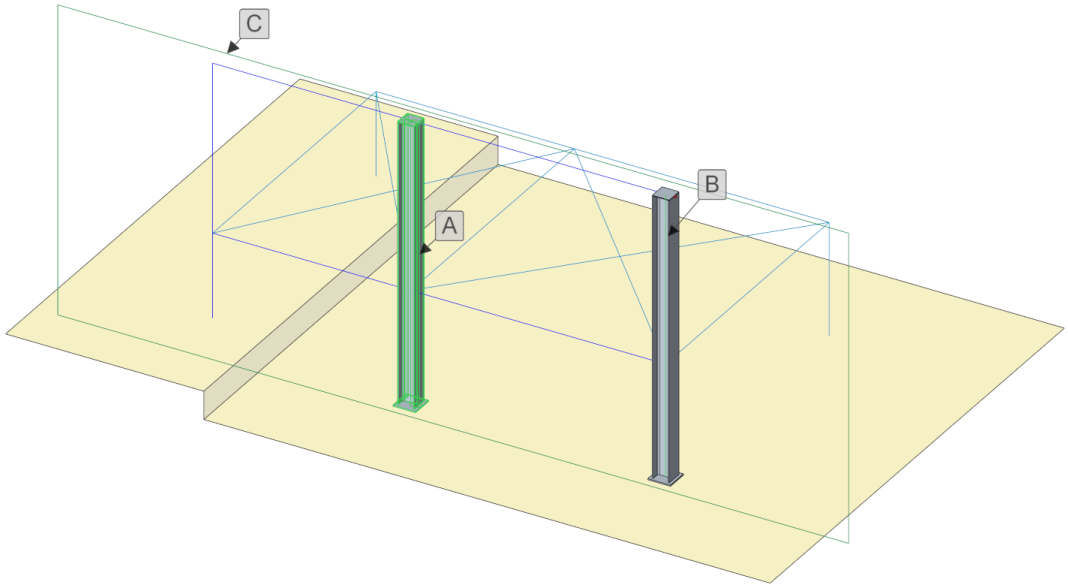
1. Click **Equipment Elements**.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [BEAM END PLATES] > [ENDCAP RECT]**.
4. Click **[Profile end (1)]** and select the highlighted surface **[A]** of the profile.
5. Click **[OK]** to complete the definition.
   - The end cap will be assembled.

Now save and close the assembly and switch back to the main-assembly `PROJ_SUBASSEMBLIES.ASM`.

5.3 Reassemble subassemblies

To assemble the just designed column subassembly again on the right vertical curve proceed as follows.

1. Click **Reuse** in the **Subassemblies** ribbon group.
2. Select the subassembly PS_COLUMN_1.ASM **[A]** as element to reassemble, leave the instance type same instance and press **Next**.
3. Select the right vertical curve **[B]**.
4. Select the orientation plane A_XY **[C]** of the main assembly.
   - The subassembly PS_COLUMN_1.ASM will be reassembled on the right curve.



5.4 Exercise

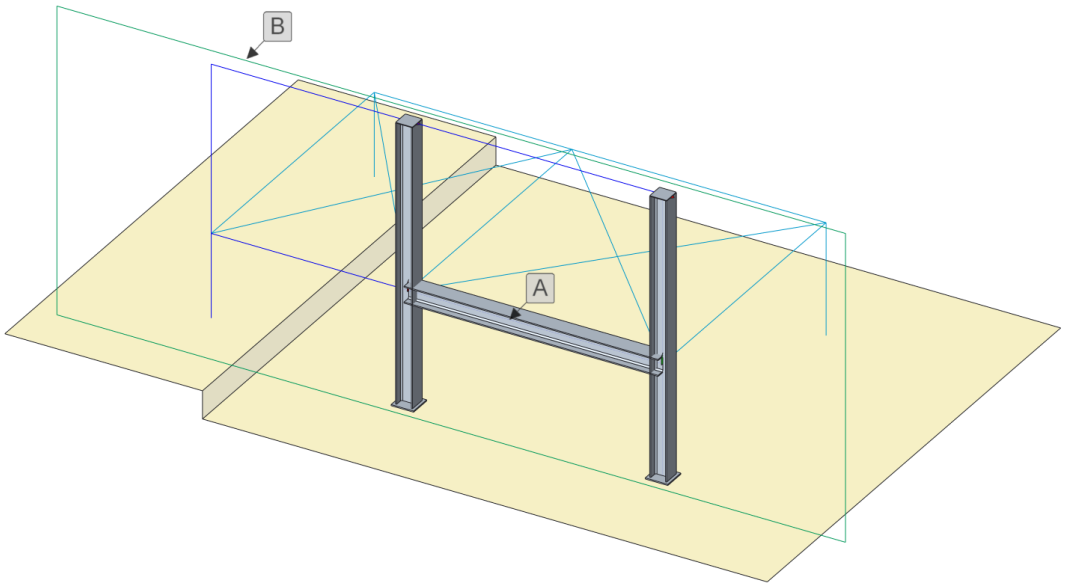
In this section we will create a new subassembly with various additional elements.

Assemble subassemblies with Profiles dialog

Now design a beam subassembly on the lower right horizontal curve. This time we will create the subassembly with a special functionality of the **New Profile** dialog. If you create a subassembly with this method a profile will also be assembled.

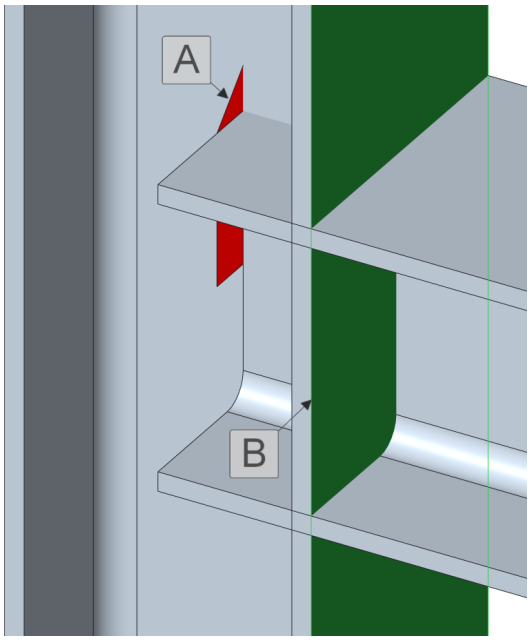
1. Click **New Profile**.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL MM] > [I BEAM] > [DIN 1025 IPB/HEB] > [300]**.
4. Check the option **Assemble in new subassembly**.
5. Press **Next**.

6. Enter **[ps_beam_1]** into the input-panel as name for the new assembly.
7. Select the horizontal datum curve **[A]**.
8. Select A_XY as orientation plane **[B]** and press **OK**.
   - The new subassembly is assembled.



Create Joints to Subassemblies

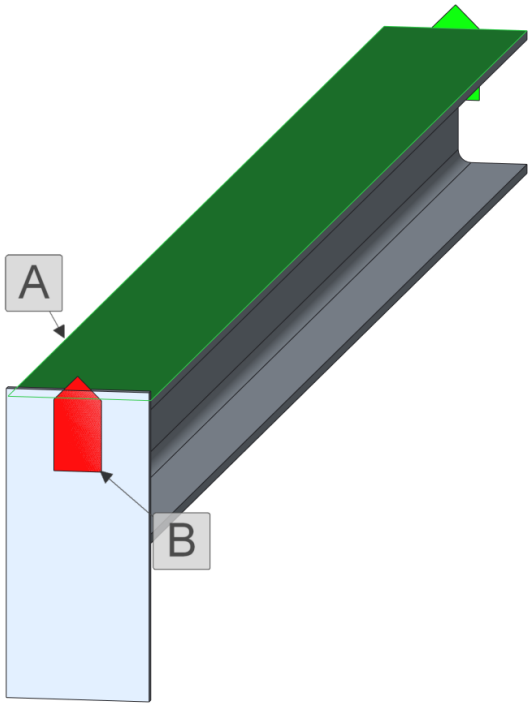
It is also possible to apply joints to the subassemblies by using the arrow shaped surfaces as connection interface.



1. Click **Basic Joints** to open the dialog box.
2. Select joint type **To selected surface**.
3. Select the red arrow shaped surface **[A]**.
4. Select the Profile side face **[B]** and press **Repeat**.
5. Now repeat that step for the green arrow surface and press **OK**
   - The horizontal subassembly is now properly attached to the two columns

Add connector and equipment elements

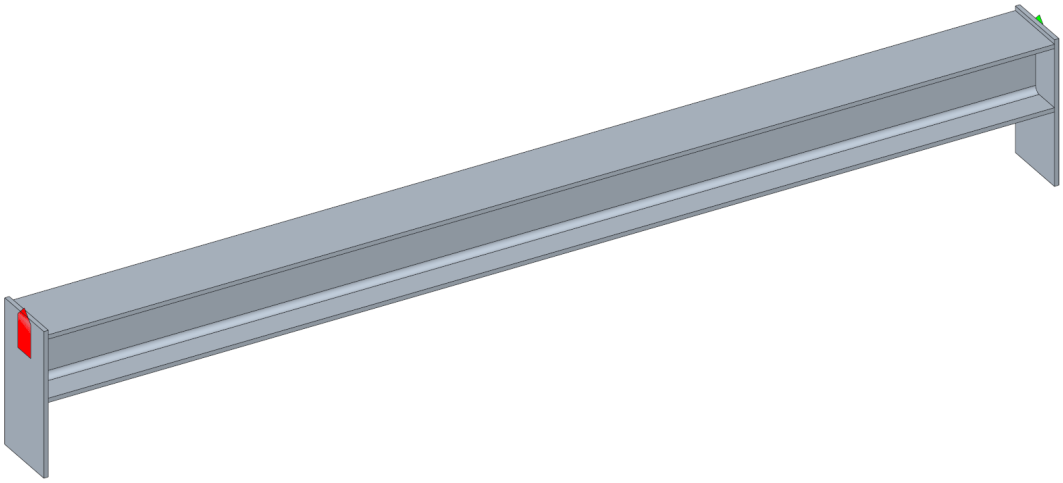
In the next few steps we will add additional equipment and connector elements to the horizontal beam, therefore open the assembly `PS_BEAM_1.ASM` in a new window.



1. Click Connector Elements.
2. Click to open the «Select from library» dialog box.
3. Select **[STEEL CONSTR. MM] > [NO STANDARD] > [ENDPLATE TOP]**.
   - The Element definition dialog box of the plate opens.
4. Enter values: **[H=650]**; **[W=300]**.
5. Click **[Profile top (1)]** and select the highlighted surface **[A]**.

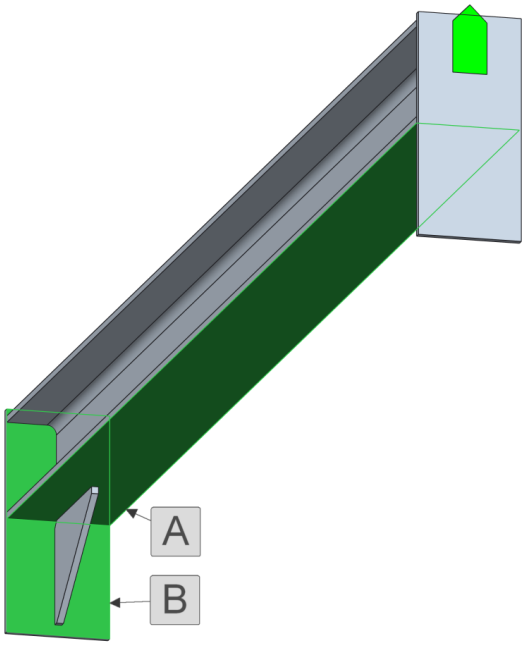
6. Click **[Attach face (2)]** and select red arrow shaped surface **[B]**.
7. Click **[OK]** to complete the definition.
   - The end-plate is assembled and the profile gets shortened.

Now reassemble the end-plate on the other side of the profile via . This time select the green arrow shaped surface as **Attach face**.



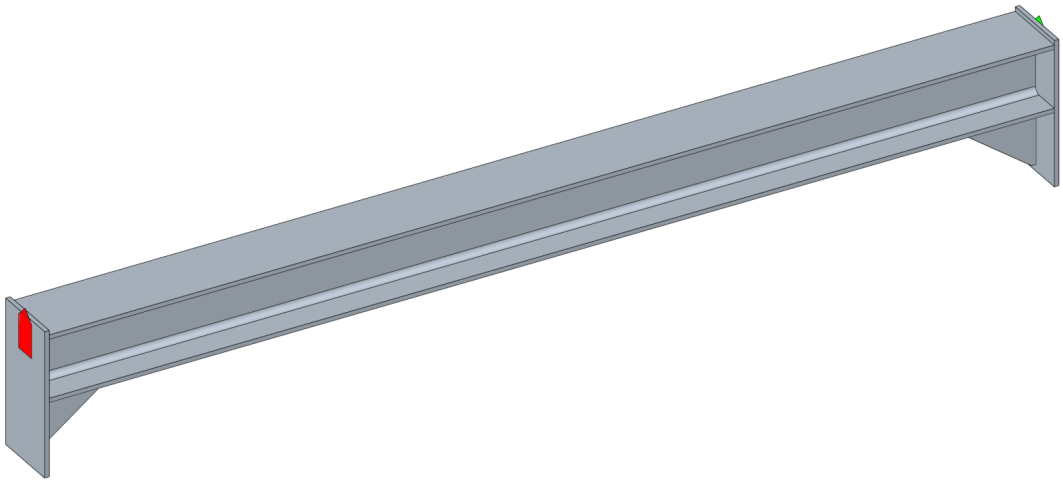
Next assemble a triangular corner plate. Proceed as follows.

1. Click **Connector Elements** to open the dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NO STANDARD] > [TRIANG CORNER PLATE]**.
   - The Element definition dialog box of the plate opens.



4. Enter values: **[L=400]**; **[H=300]**; **[S=20]**; **[L1=20]**; **[L2=20]**; **[H1=20]**.
5. Click **[Profile end (1)]** and select the highlighted surface **[A]**.
6. Click **[Attach face (2)]** and select the highlighted surface **[B]**.
7. Click **[OK]** to complete the definition.
   - The corner plate is assembled.

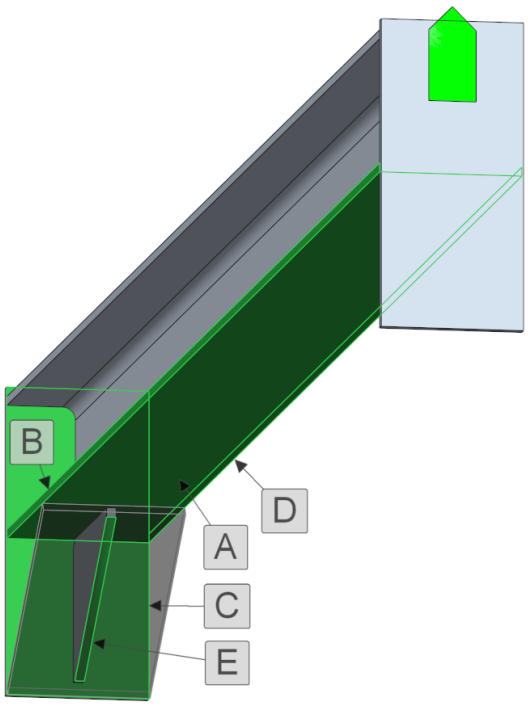
Reassemble the corner plate on the other side via ↺.



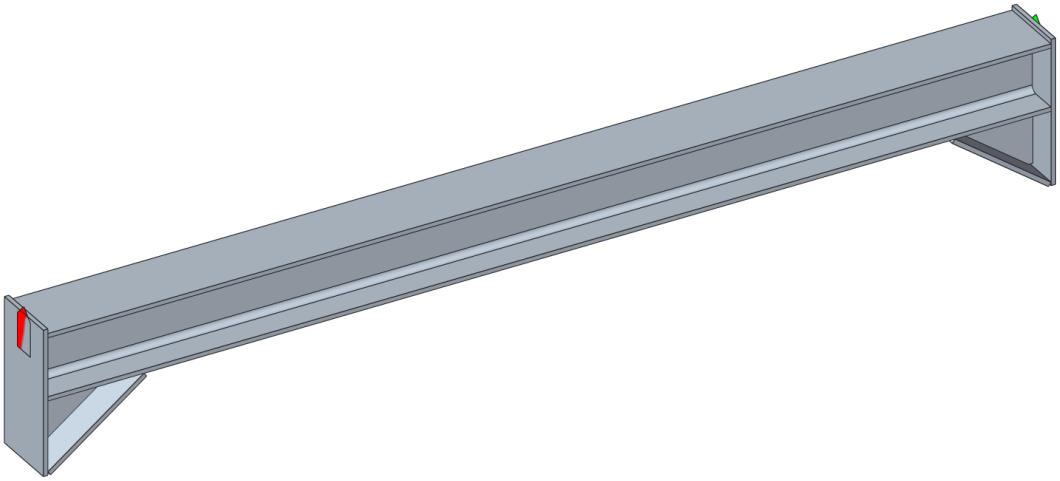
For additional reinforcement we will also add a four sided plate to the triangular corner plate.

1. Click **Equipment Elements** to open the dialog box.
2. Click to open the **Select from library** dialog box.

3. Select **[PLATES MM] > [PLATE 4 PLANES]**.
   - The Element definition dialog box of the plate opens.
4. Enter values: **[T=20]**.
5. Click **[Plane (1)]** and select the highlighted surface **[A]**.
6. Click **[Plane (2)]** and select the highlighted surface **[B]**.
7. Click **[Plane (3)]** and select the highlighted surface **[C]**.
8. Click **[Plane (4)]** and select the highlighted surface **[D]**.
9. Click **[Attachment plane (5)]** and select the highlighted surface **[E]**.
10. Click **[OK]** to complete the definition.
    - The plate is assembled.

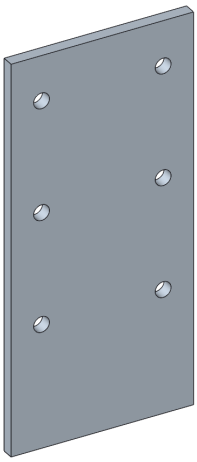


Now reassemble the plate as on the other side via ↺. Take care to select the references **counterclockwise**. After all references are selected you will be prompted to enter the attach option. Select **[1]** to assemble the plate mated of the selected attach face.



Create AutoUDF's and Screw connections

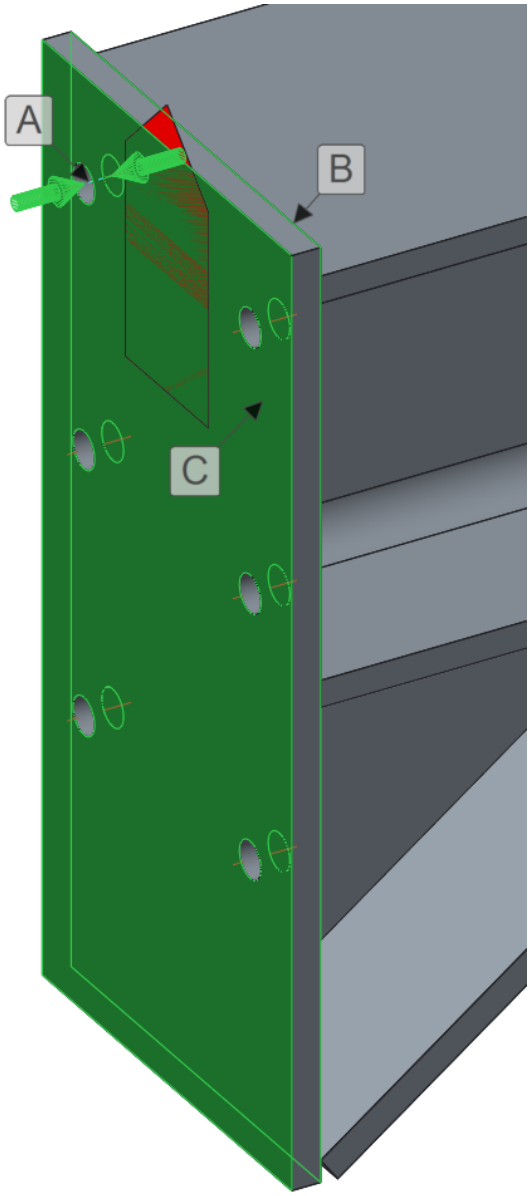
First of all redefine the previously assembled **ENDPLATE TOP**.



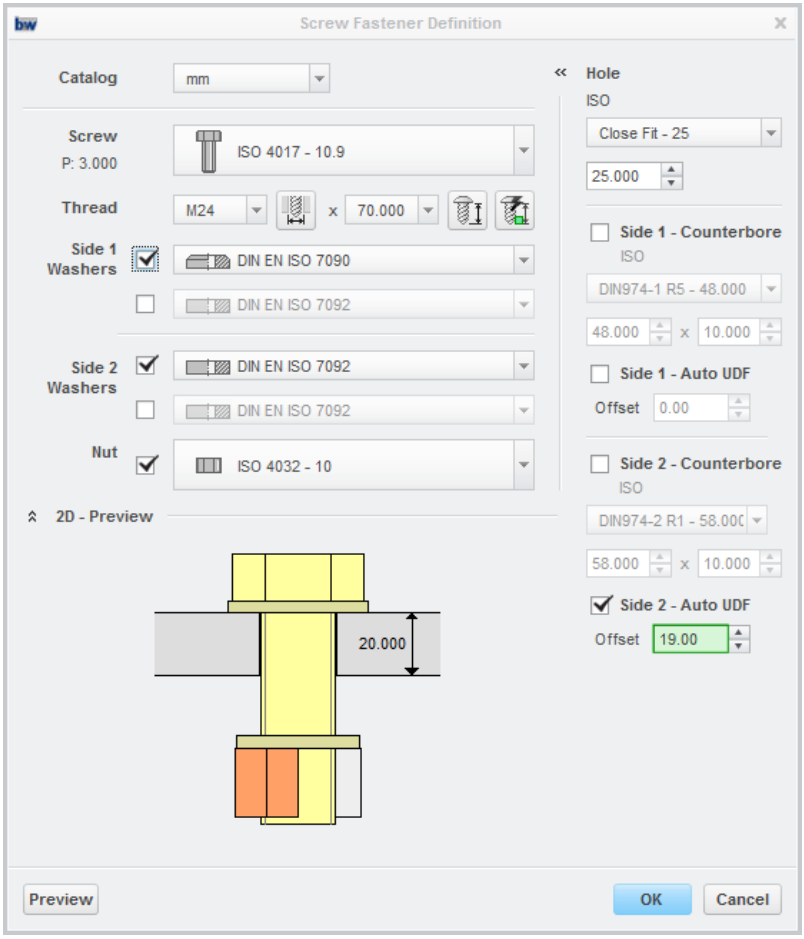
1. Select PS_ENDPLATE_TOP_1.PRT.
2. Press **RMB** and select **[Framework] >** **[modify connector/equipment element].**
   - Since the part was reassembled you will be asked if you want to modify all occurrences.
3. Accept the prompt with **[y]**.
4. Modify the values: **[H1=185]**; **[W1=200]**; **[H2=75]**; **[No. of point rows=3]**; **[HOLE_DM=26]**.
5. Check both Options: **[Holes (A)]**; **[Auto UDF holes (B)]**.
6. Click **[OK]** to complete the definition.
   - The plate is successfully modified on both sides.

Hint 9 — Assembly structure. Most constructing engineers prefer a small main assembly structure. Therefore it's a common practice to put as many parts (especially standard items like screws) into subassemblies. This creates a well organized BOM structure later on.

The next step is the assembly of screw connections for the subassembly. Although we don't have a part to attach the nut or screw head it is still possible to create the connection. Before getting started make sure the **IFX** configuration option `CREATE_CSYS_FOR_AUTO_UDF_CREATION` is set to `YES`. Define the screw connection as follows.



1. Click **Assemble on point or axis** to define a connection.
2. As **Position Reference** select the axis `A_3` **[A]**.
3. Select the highlighted surface **[B]** as **Screw head placement surface**.
4. Select the highlighted surface **[C]** as **Nut placement surface**.
   - The **Screw Fastener Definition** dialog opens.
5. Define the screw connection as shown in the picture.



6. Click **[OK]** to complete the definition.
7. Select **[Pattern Fastener?]** in the upcoming dialog and press click **[OK]**.
   - The screw connection is assembled and patterned.

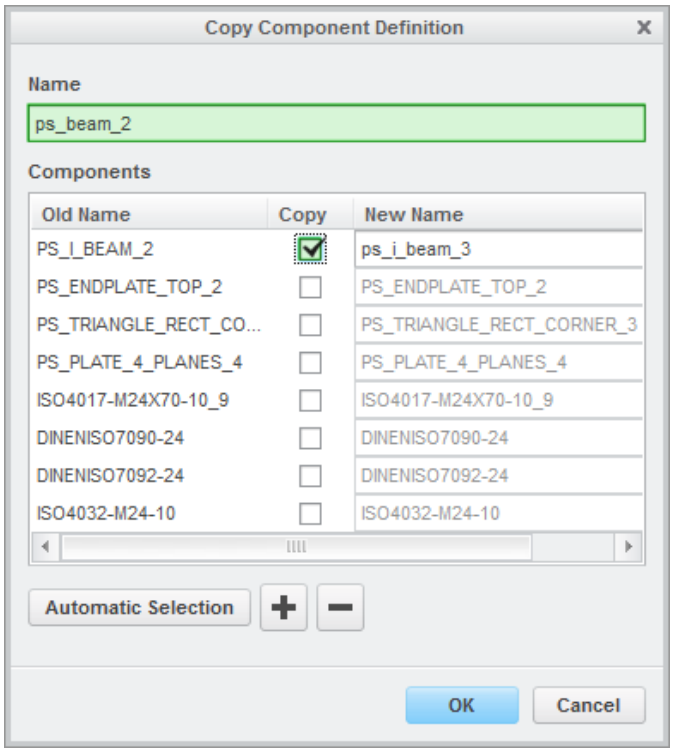
To complete the assembly click **Reassemble** and assemble the connection on the other side.

The subassembly is now complete. Save the model and switch back to the main-assembly `PROJ_SUBASSEMBLIES.ASM`.

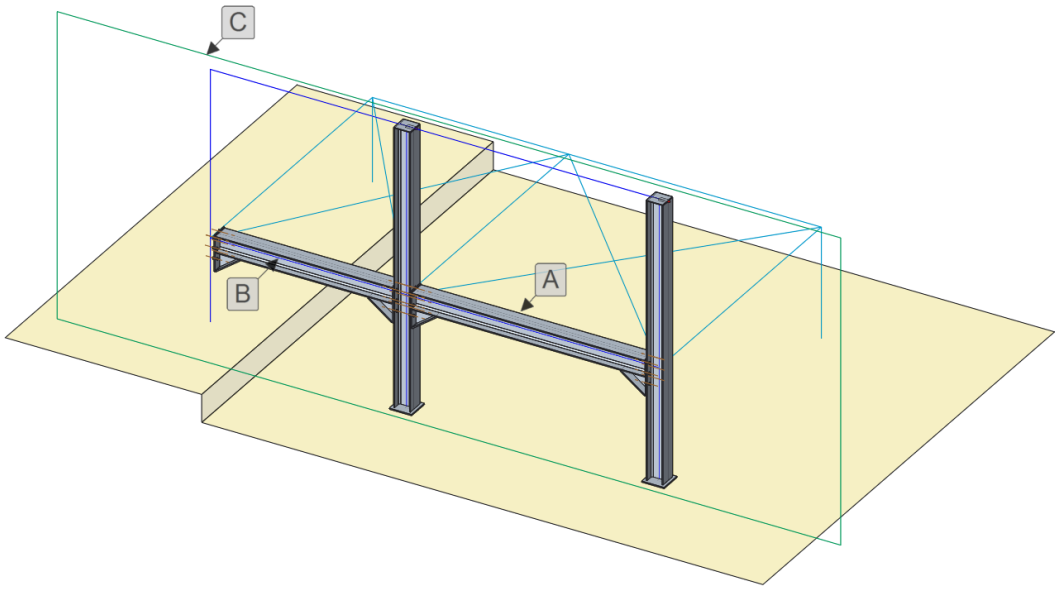
5.5 Copy subassemblies

The PS_BEAM_1.ASM assembly will now be reused at different locations of this assembly. In most cases the used profile of the subassembly differs in length, but is also possible to copy all parts of a subassembly. As the subassembly is designed without external references it is very easy to copy, reuse and modify it on other locations.

1. Click **Project Subassemblies**.
2. Click and select the subassembly PS_BEAM_1.ASM **[A]** as element to copy.
   - The **Copy Component Definition** dialog opens. Enter **[ps_beam_2]** as new name and check the copy box to copy the beam PS_I_BEAM_2.

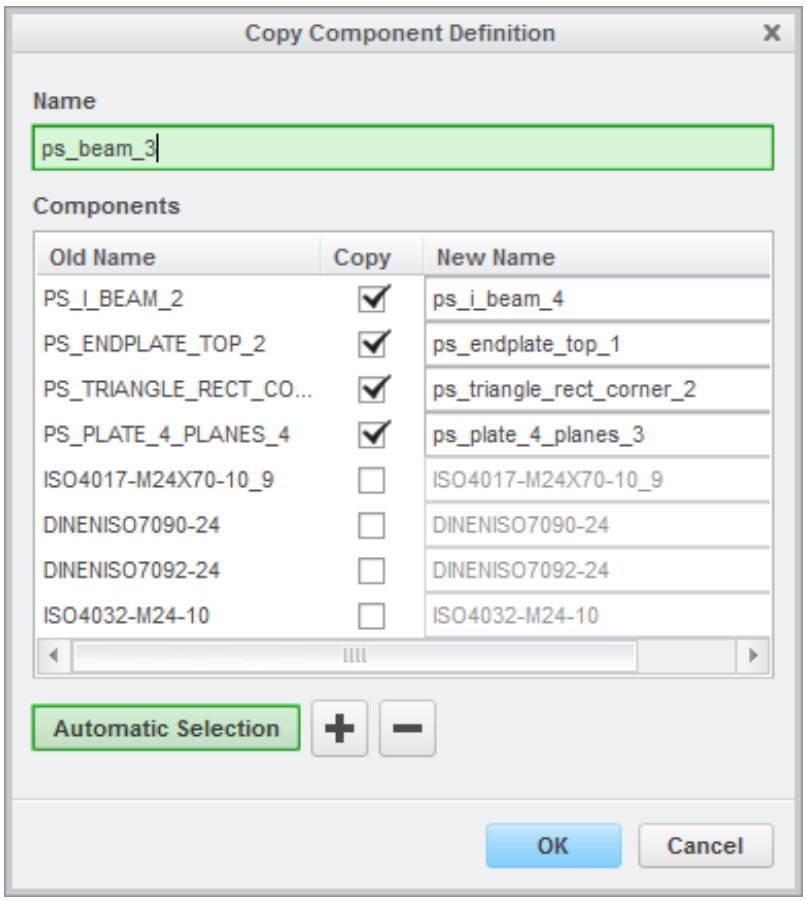


3. Select the right curve **[B]**.
   - The new subassembly PS_BEAM_2.ASM will be assembled on the left curve.
4. Select the orientation plane A_XY **[C]** of the main assembly.
5. Click OK to complete the process.

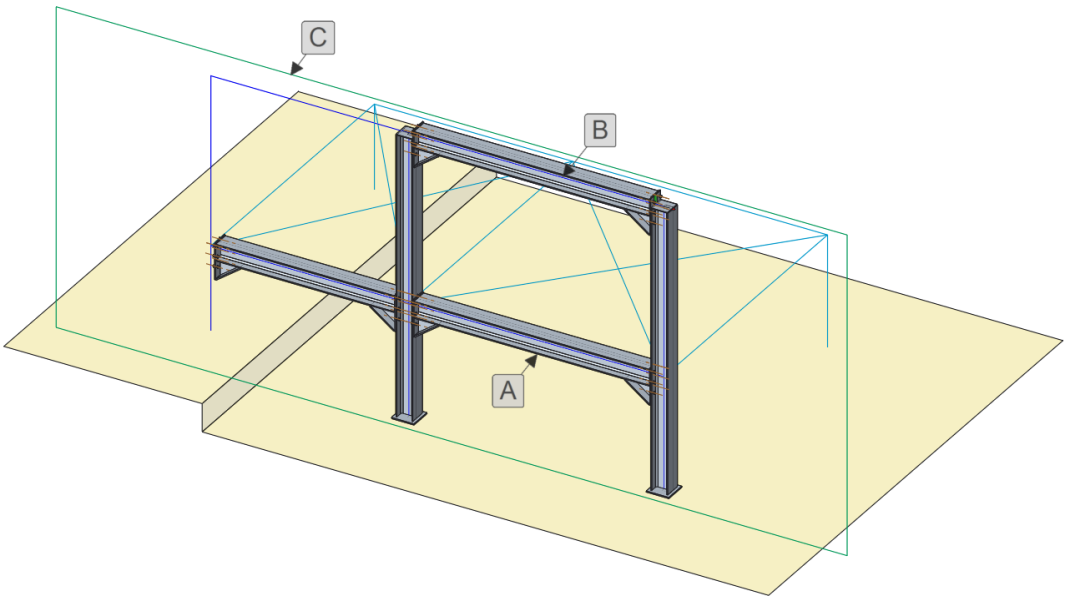


Now we will copy the assembly once more, but this time we will replace all project parts by a copy.

1. Click **Project Subassemblies**.
2. Click and select the subassembly `PS_BEAM_1.ASM` **[A]** as element to copy.
   - The **Copy Component Definition** dialog opens. Enter **[ps_beam_3]** as new name and click Automatic Selection to copy all relevant **project components** as well.



3. Select the upper right curve **[B]**.
   - The new subassembly PS_BEAM_3.ASM will be assembled on the upper right curve.
4. Select the orientation plane A_XY **[C]** of the main assembly and click OK.

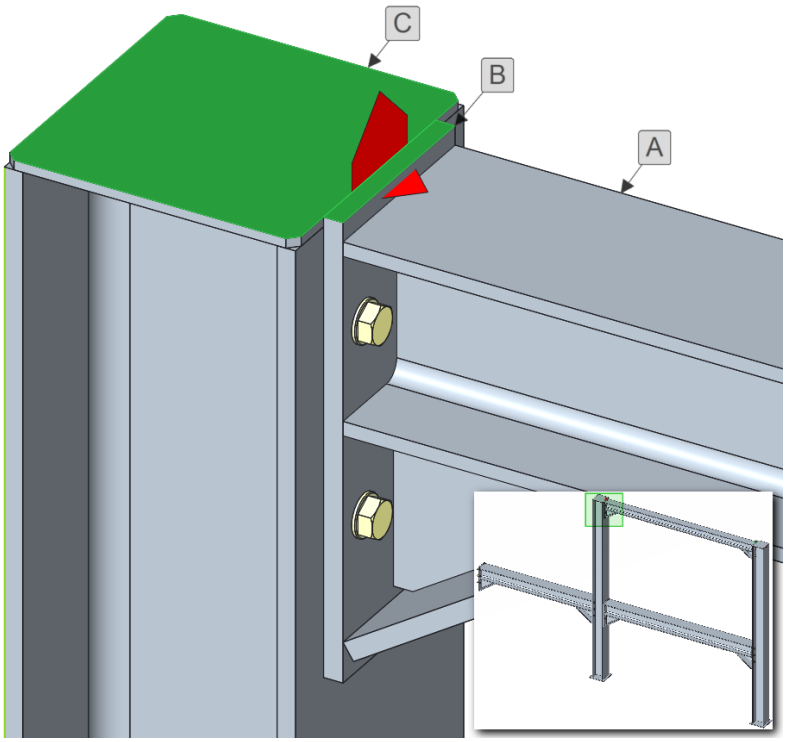


Due to the fact that we copied the relevant components, we are now able to modify the components independently from PS_BEAM_1 and PS_BEAM_2.

1. Select the main **I-beam** and press **[RMB] > [Framework] > [Modify section size]** and choose size: **[DIN 1025 IPB/HEB] > [200]**
2. Select the **ENDPLATE TOP** and press **[RMB] > [Framework] > [Modify connector/ equipment element]**, type "**y**" to change all plates at once and change the following values: **[H=475]**; **[W=200]**; **[H1=200]**; **[W1=125]**; **[H2=120]**; **[No. of point rows=2]**.
3. Select the **triangular rect corner plate** and press **[RMB] > [Framework] > [Modify connector/equipment element]**, type "**y**" to change all plates at once and change the following values: **[L=300]**; **[H=200]**.

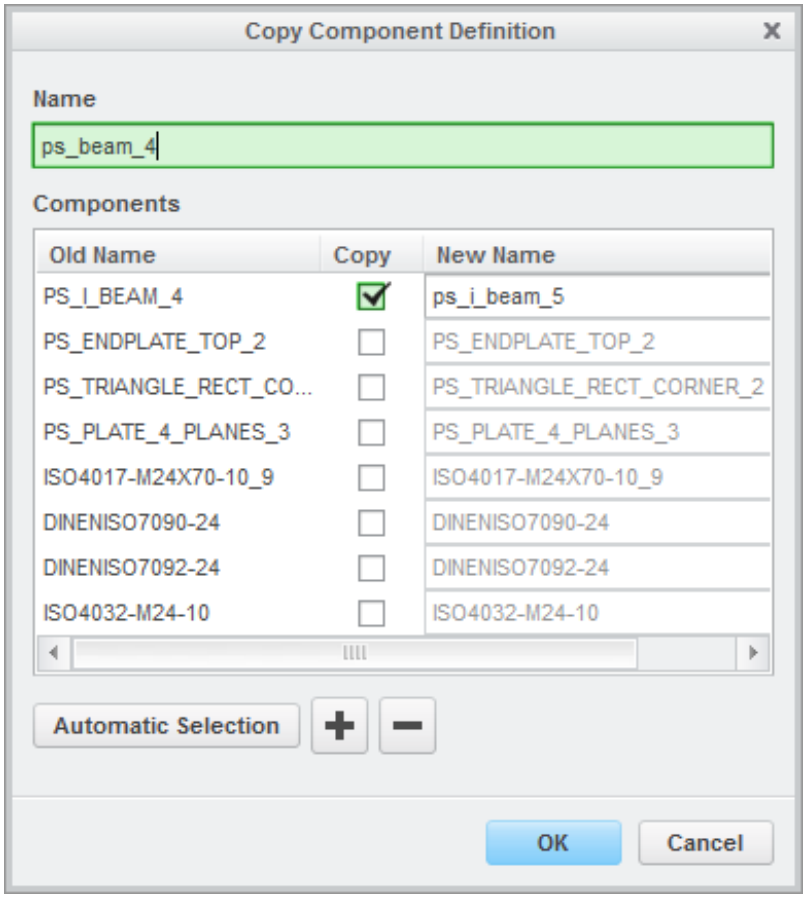
And finally align the subassembly with the **end-cap** of PS_COLUMN_1.ASM.

1. Click **Move** in the **Project Subassemblies** Ribbon group.
2. Select the subassembly PS_COLUMN_3.ASM **[A]**.
3. Click to align a subassembly with a selected entity.
4. Select the highlighted reference surface of the end-cap [**B**].
5. Select the highlighted surface **[C]** of the end-plate top.
   - The completed subassembly is aligned with the end-cap.



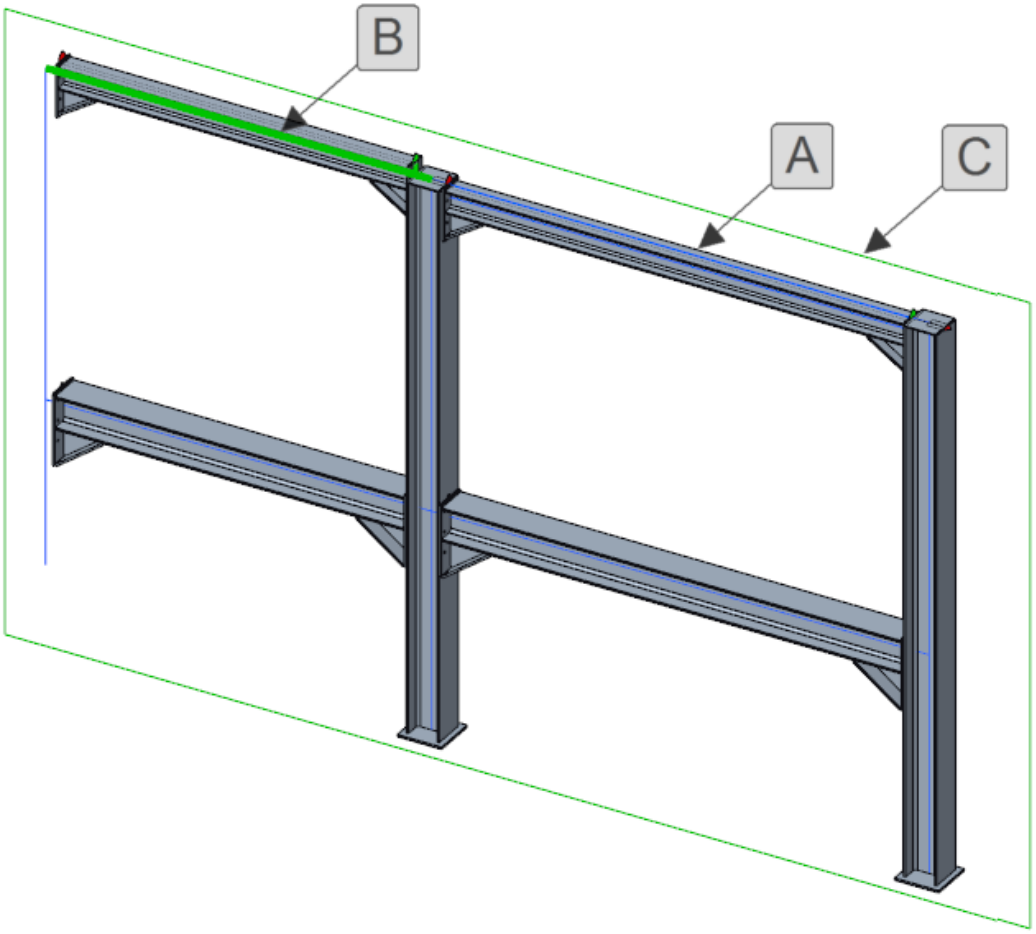
After successfully modifying the upper subassembly it can now be copied to the left.

1. Click **Reuse** in the **Project Subassemblies** Ribbon group.
2. Click and select the subassembly `PS_BEAM_3.ASM` as element to copy.
   - The **Copy Component Definition** dialog opens. Enter **[ps_beam_4]** as new name and check the copy box to copy the beam `PS_I_BEAM_4`.



3. Select the upper left curve **[B]**.
4. Select the orientation plane `A_XY` **[C]** of the main assembly.
   - The new subassembly `PS_BEAM_3.ASM` will be assembled on the upper left curve.

To finalize the definition **move** the subassembly like shown on the **previous page**.



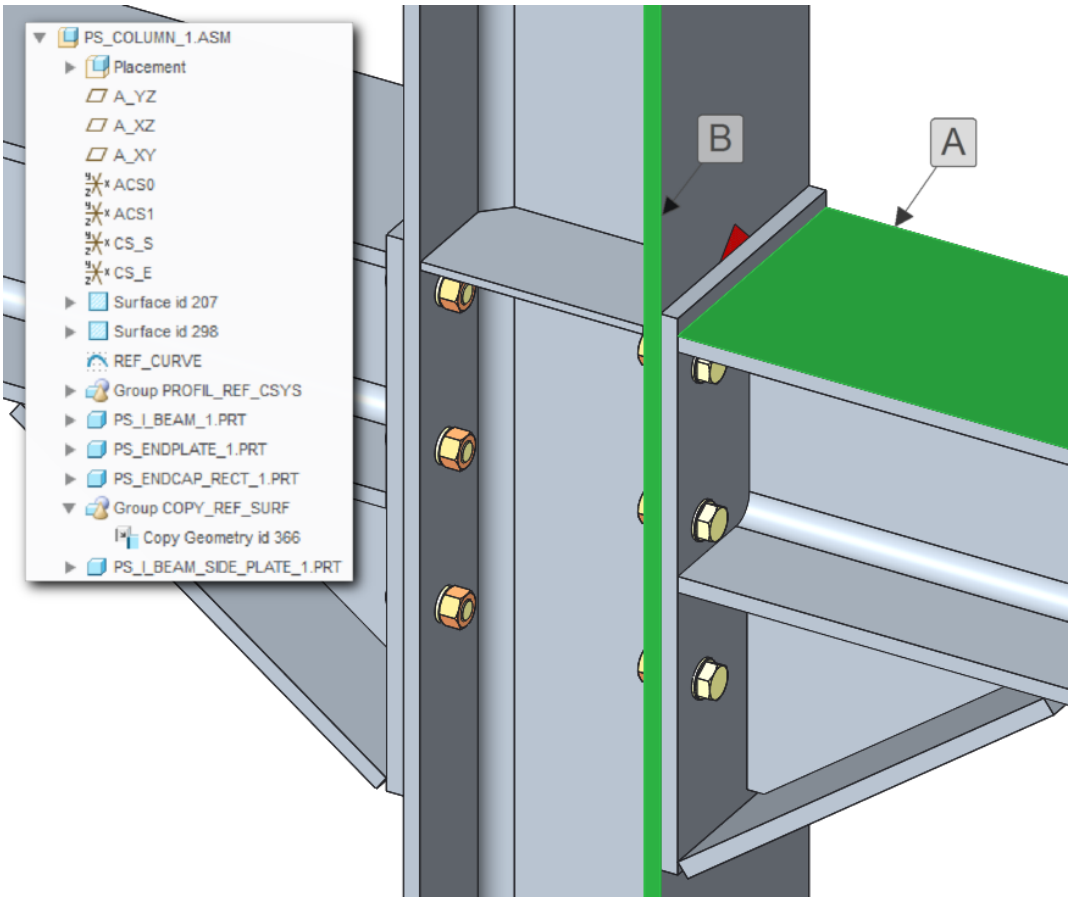
Fix external references

If you assemble elements to project subassemblies using references from other project subassemblies external references will be created. This means that you will have to replace the external reference if you copy those subassemblies.

We will now assemble a connector to PS_COLUMN_1.ASM that will need an external reference.

1. Select PS_COLUMN_1.ASM and click **Activate**.
   - All connectors will now be assembled into PS_COLUMN_1.ASM.
2. Click **Connector Elements** to open the connector dialog.
3. Click to open the **Select from library** dialog box.

4. Select **[STEEL CONSTR. MM] > [NO STANDARD] > [I-PROFIL SIDE PLATE]**.
   - The Element definition dialog box of the plate opens.
5. Click **[Placement plane (1)]** and select the highlighted surface **[A]**.
6. Click **[I-Profile side (2)]** and select the highlighted surface **[B]**.
7. Click **[OK]** to complete the definition.
   - The plate will be assembled into `PS_COLUMN_1`.



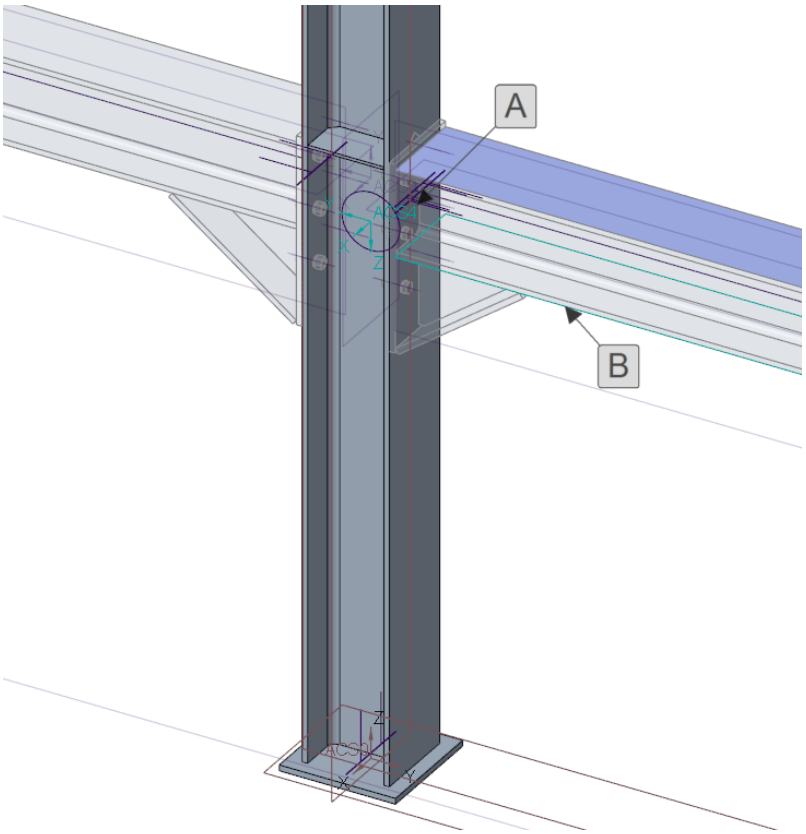
Let us take a closer look what happened:

Inside the `PS_COLUMN_1` a copy geometry feature got created. This is an external reference, because the selected reference **[A]** does not belong to the subassembly. To avoid the external reference you could have also created a datum plane in the subassembly to use it as reference for the plate.

To assemble the plate again without external references on the bottom side proceed as follows.

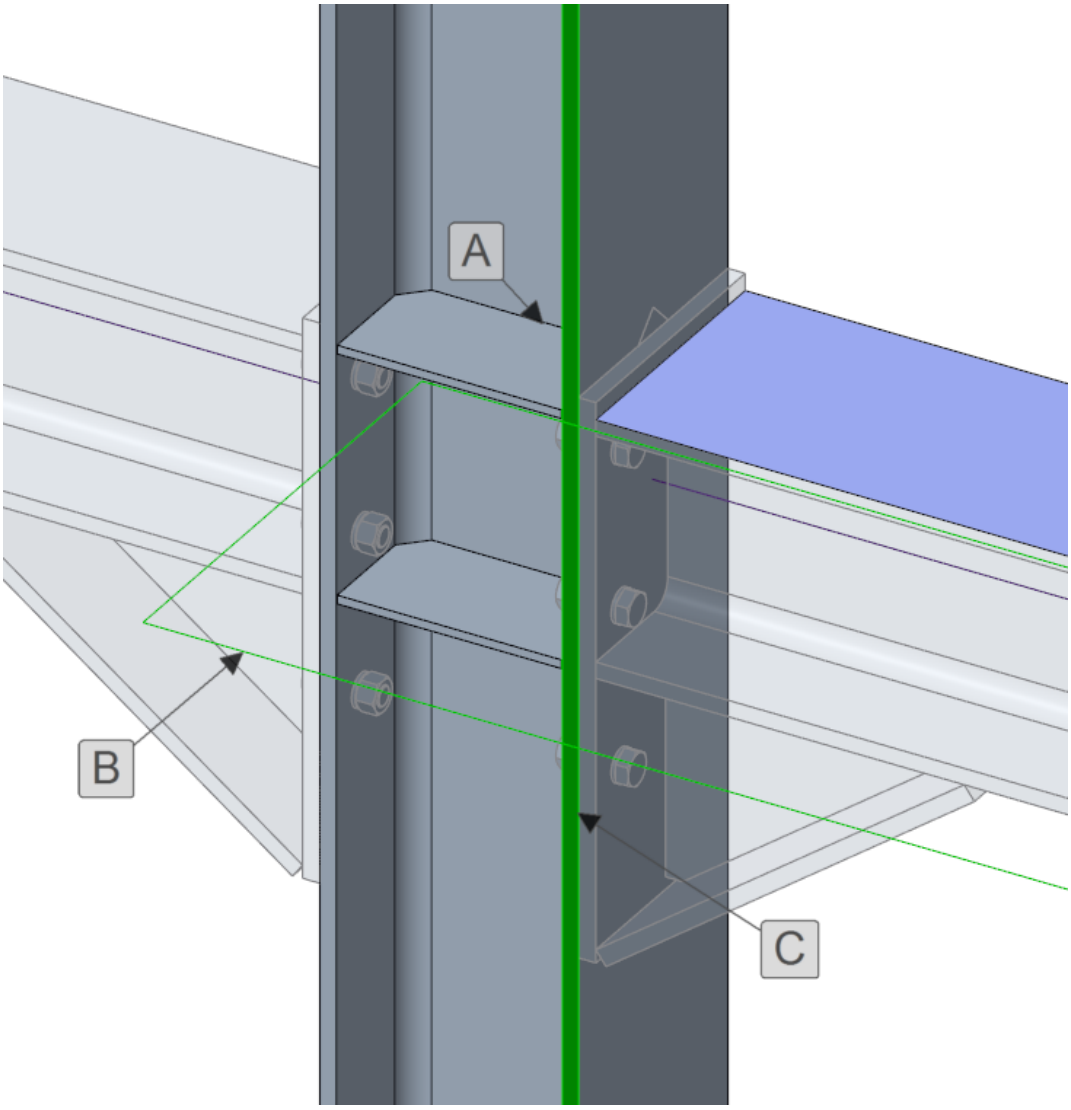
1. Activate the subassembly `PS_COLUMN_1.ASM` again.
2. Create a Csys on the default Csys `ACS0` of the assembly.
3. Click Basic Joints and select joint type **offset csys to plane**.

4. Select the just created Csys ACS4 **[A]** and the lower surface of the I-Beam **[B]**

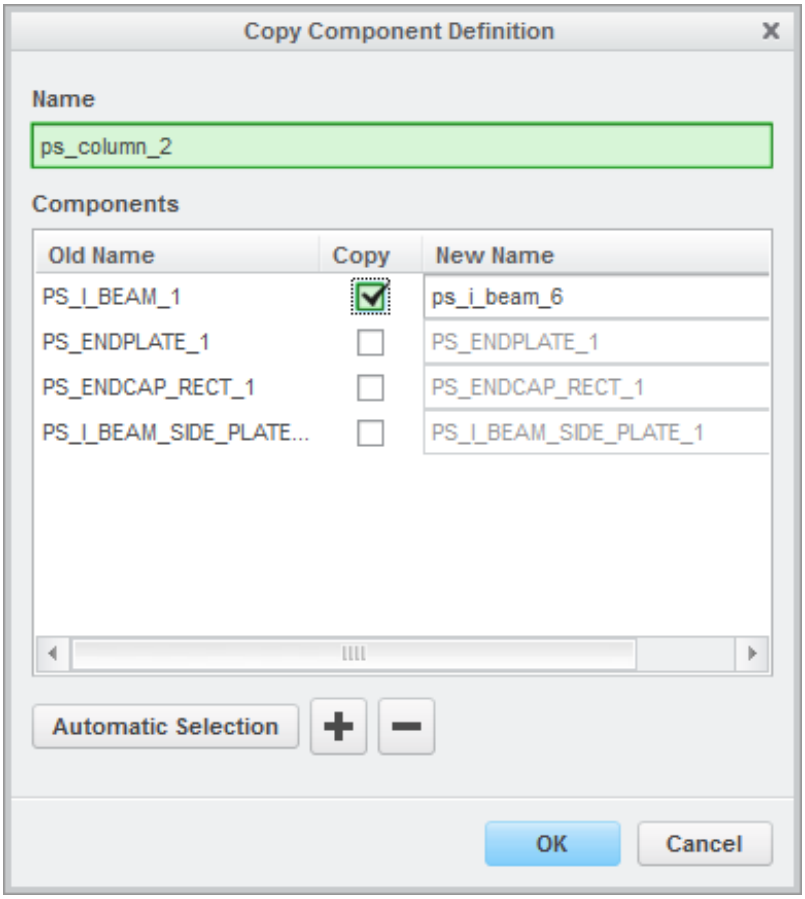


5. To finish the reference create a datum plane through the Csys Plane XY
6. Click **Connector Elements** to open the connector dialog.
7. Click and select the previously assembled plate **[A]**.

8. Select the newly created datum plane ADTM5 **[B]**.
9. Select the highlighted surface **[C]**.
    - The plate will be reassembled into PS_COLUMN_1.ASM.



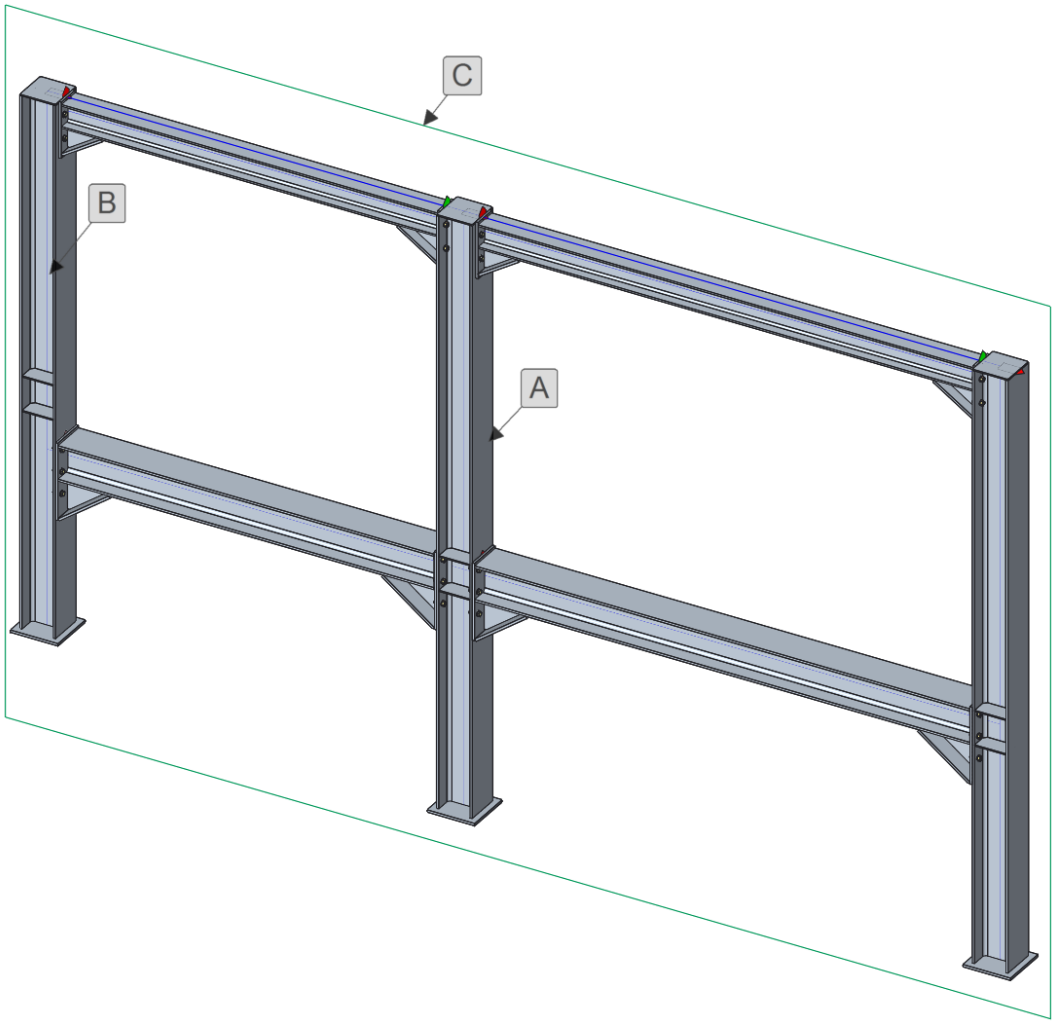
The next step is to copy assembly PS_COLUMN_1.ASM on the remaining vertical datum curve.



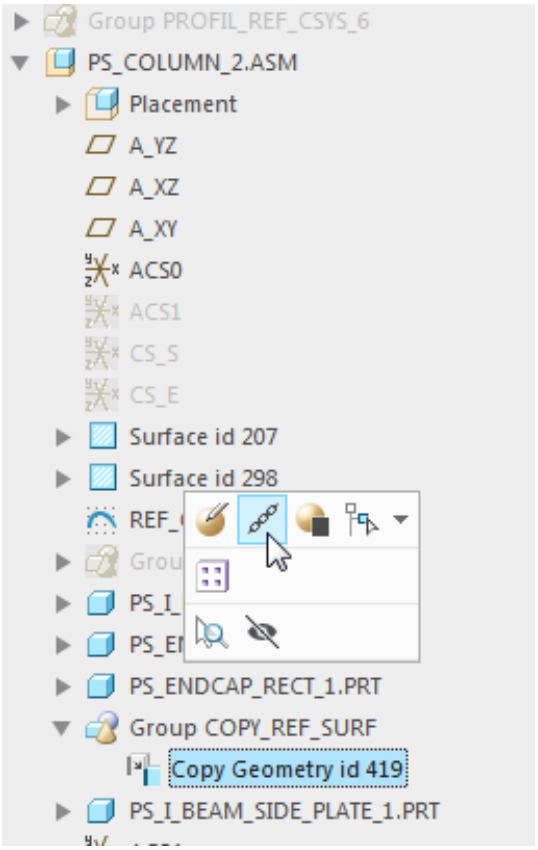
1. Click **Reuse** int the **Project Subassemblies** Ribbon group.
2. Click and select the subassembly `PS_COLUMN_1.ASM` **[A]** as element to copy.
   - The **Copy Component Definition** dialog opens. Enter **[ps_column_2]** as new name and check the copy box to copy the beam `PS_I_BEAM_1`.
3. Select the left curve **[B]**.
4. Select the orientation plane `A_XY` **[C]** of the main assembly.

- The assembly PS_COLUMN_2.ASM is assembled on the left curve. Unfortunately it is upside down.

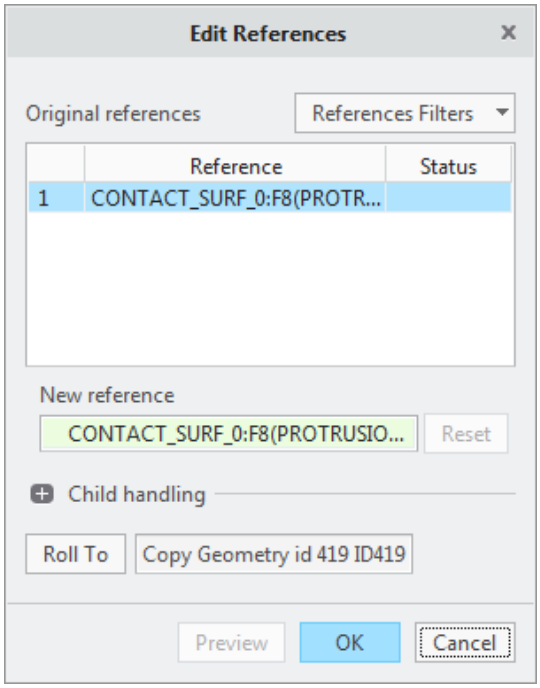
5. Click to switch its start and end position and press OK to finish the definition.
   - The subassembly is now orientated correctly.



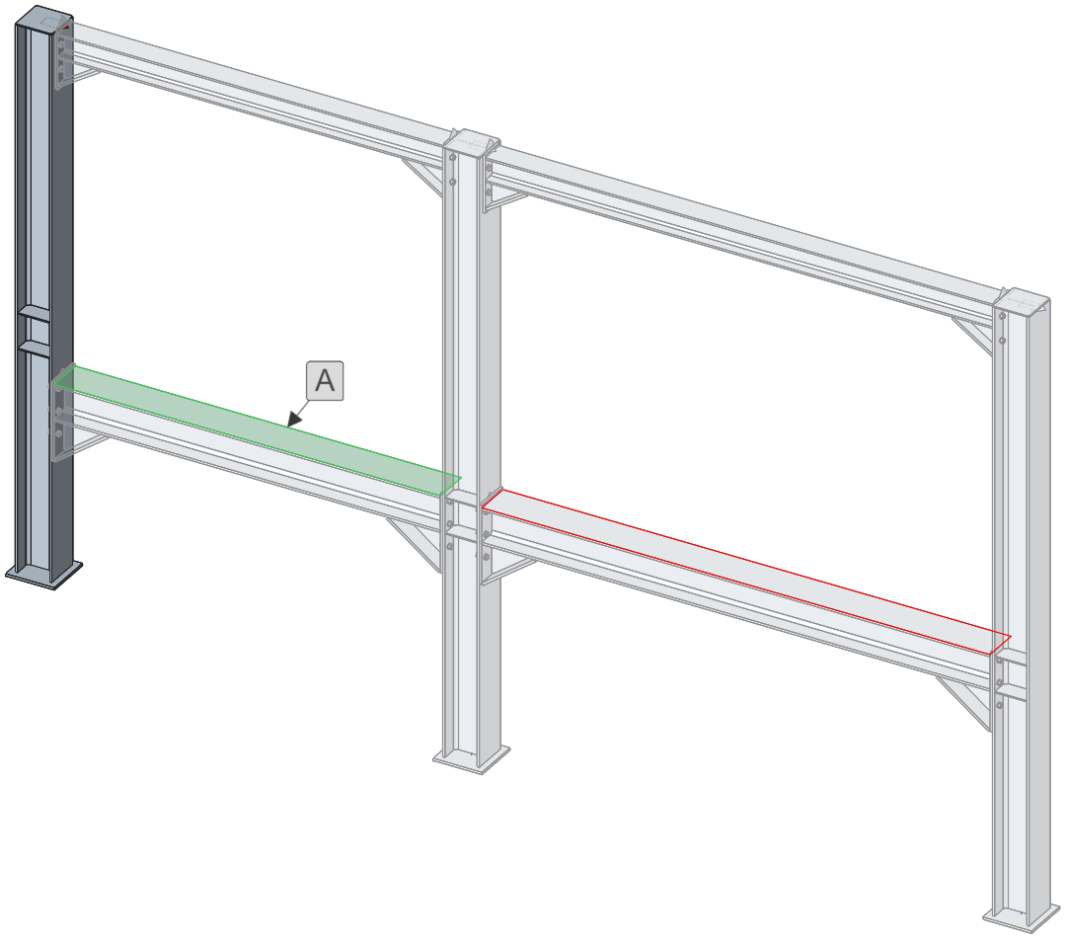
Due to the fact, that the I-profile side plates have external references, we need to manually update these references with standard **Creo Parametric** functionality.



1. Right click on the copy geom feature and click **Edit references**.
   - The **Edit References** dialog shows up.



2. Select the top surface of the left I-beam **[A]** as new reference.
3. Click **[OK]** to complete the definition.
   - The reference is modified and the plate is now aligned to the left I-beam



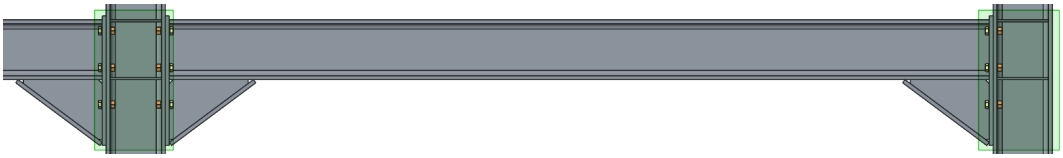
For the second plate open the Basic joints dialog and recreate a **offset csys to plane** Joint with the Csys ACS4 of PS_COLUMN_2 and the lower surface of the I-Beam.

Replace subassemblies

With the **Replace** subassembly functionality, you can replace a project subassembly in two ways.

- By another already assembled project subassembly via . This works similar to replacing a profile by another existing profile.
- By a copy of itself via . This can be used if you have assembled a project subassembly several times with the **Reuse** functionality and some instances need to be different.

In this exercise it is required to replace the right subassembly by a copy of itself, because the right I-beam doesn't need holes for the fasteners on both sides.



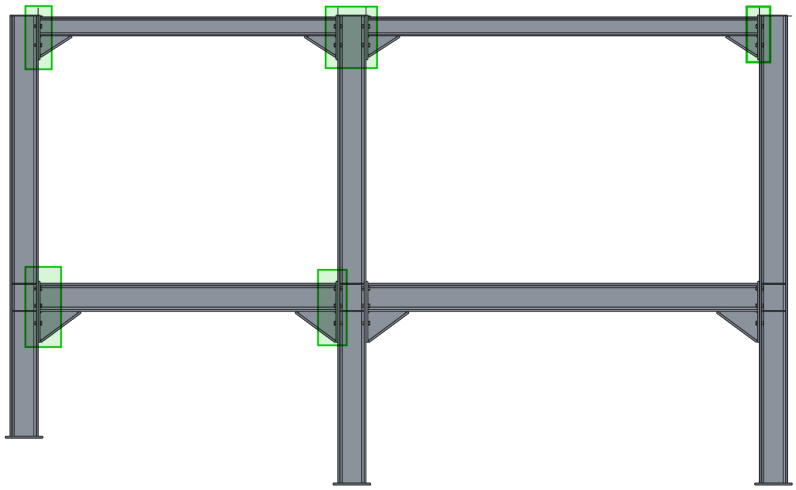
1. Click **Replace** in the **Subassemblies** ribbon group.
2. Click to replace by a copy of itself.
3. Select the `ps_column_1` as subassembly to replace.
4. In the **Copy Component Definition** dialog enter the new name **[PS_COLUMN_3]** and mark the I-beam to copy.
5. Click **[OK]** to complete the definition.
   - The subassembly will be copied and the I-beam profile becomes a new part, which is independent from the `PS_COLUMN_1.ASM`.

Although it may seem as if the definition is now complete, there are still some problems left:

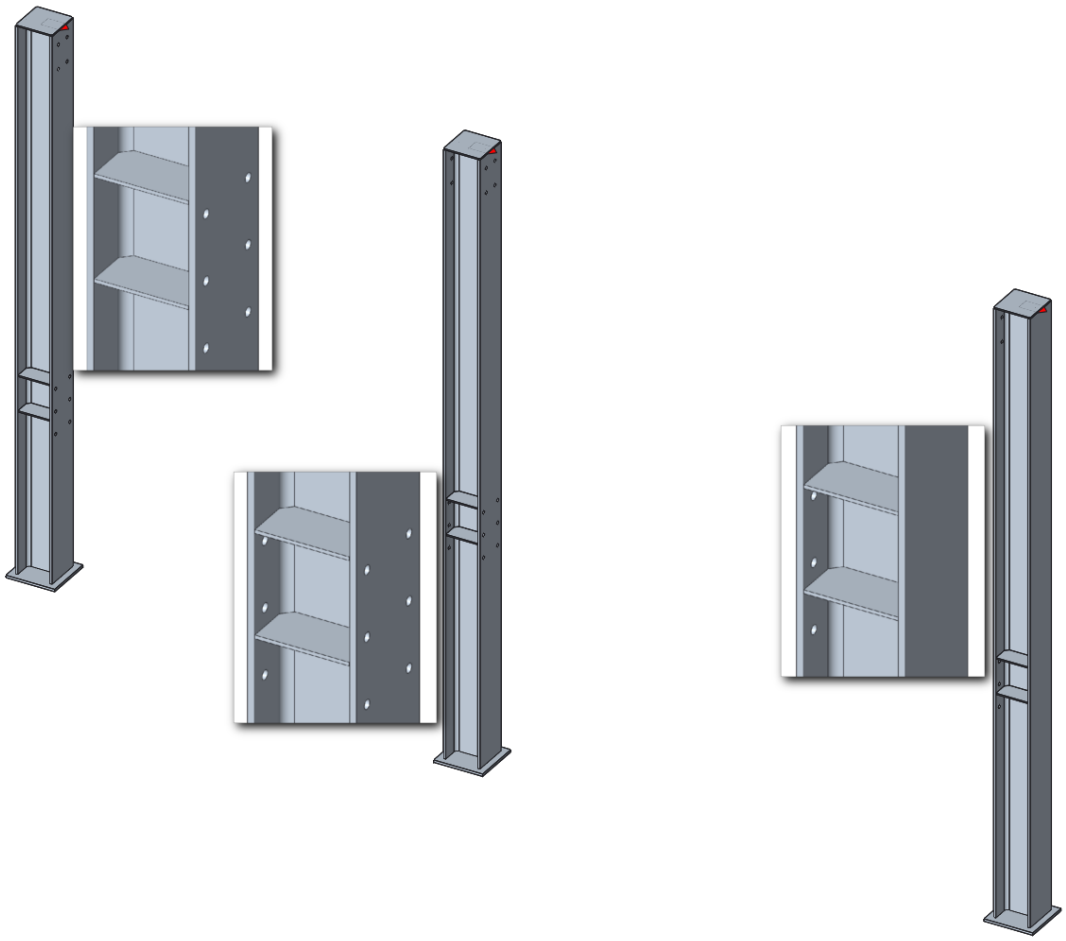
- The copied subassemblies do **not** have joint definitions
- The I-profile side plates of the right subassembly need a new reference for the copy geom feature
- The **AutoUDF** holes must be added to the I-beams

To fix this problems proceed as follows.

1. Define joints to the copied subassemblies.
   - Use the **To selected surface joint**.
   - Use the arrow shaped surfaces to create the joints.



2. Redefine the copy geom to fix the external reference:
   - Select the copy geom features of `PS_COLUMN_3.ASM` and edit the references via .
   - Select the top surface of the I-beam in `PS_BEAM_1.ASM` as new reference.
3. Add the AutoUDF holes to the I-beams.
   - Open the **Automatic UDFs** dialog.
   - Press the update button.
   - The holes are created. Their definition is controlled through the **ENDPLATE TOP** connector element.



5.6 Move and rotate

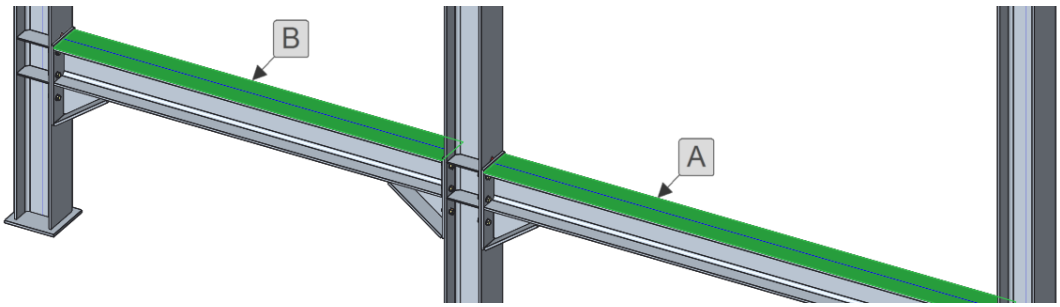
Moving and rotating project subassemblies works similar to moving and rotating profiles.

To move the two beam subassemblies down so that the top of the I-Beams are aligned to the skeleton curves.

1. Click **Move** in the **Subassemblies** Ribbon.
2. Select the 1st assembly `PS_BEAM_1.ASM` that should be moved
3. Click to move a subassembly by mouse pick and select the highlighted surface **[A]**.

- The assembly will be moved and the selected plane is **aligned with the curve** of the skeleton.

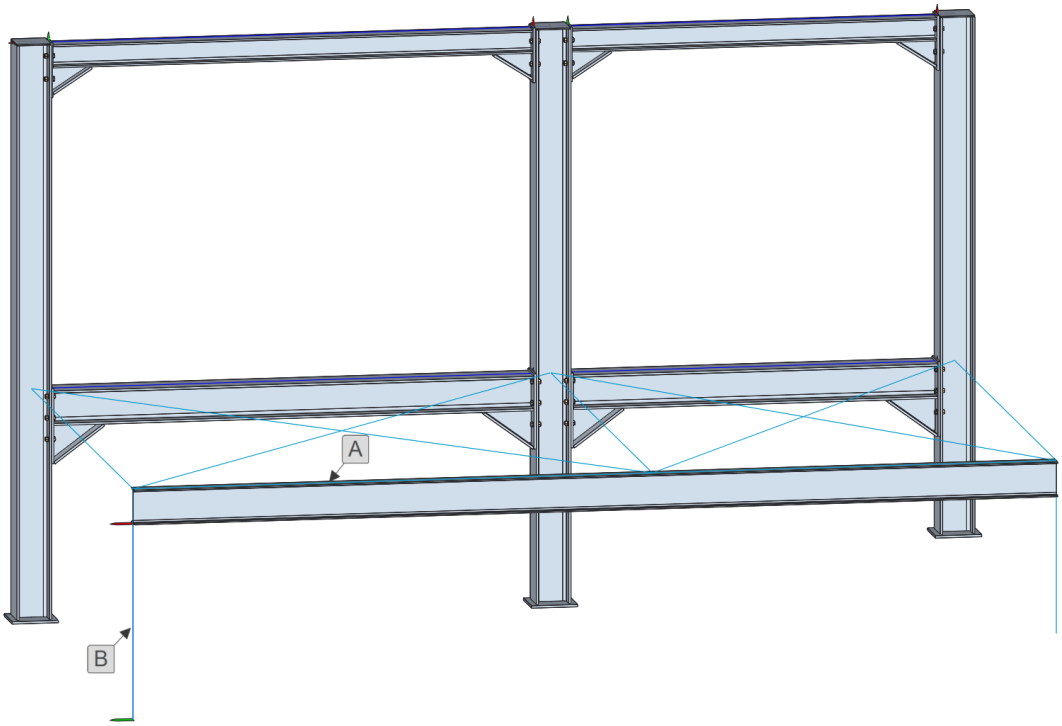
4. Select the 2nd assembly `PS_BEAM_3.ASM` that should be moved.
5. Click to move a subassembly by mouse pick and select the highlighted surface **[B]**.
   - The assembly will be moved and the selected plane is **aligned with the curve** of the skeleton.



5.7 Exercises

In this section some more exercises to project subassemblies will be described. If you have problems with these exercises please repeat the proper chapters.

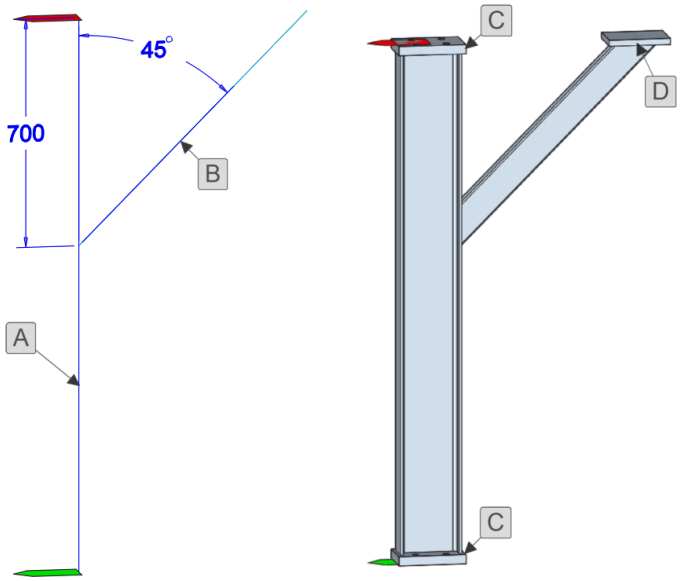
1. Assemble an **[I-Beam > DIN 1025 IPE > 300]** on the rear horizontal curve **[A]** as member of the **top level assembly**.
2. Use the move commands, so that its top surface is aligned with the curve.
3. Assemble a new project subassembly named **[ps_column_4]** on the vertical curve **[B]** like shown in the picture.
4. Create a joint **To selected surface** between the arrow shaped surface and the lower surface of the just assembled I-beam.



Open the new subassembly and proceed as follows.

1. Create a new sketch on A_YZ with the red arrow shaped surface as reference. Define the sketch as shown in the picture.
2. Assemble an **[I-Beam > DIN 1025 IPE > 200]** on the vertical curve **[A]**.
3. Assemble a **[Rectangular tube > DIN 2395 > 100x50x3]** on the just sketched curve **[B]**.
4. Assemble a **[DAST standard > end plate IH1 > IH1 E > 20 20]** on the top and the bottom of the I-beam **[C]**.

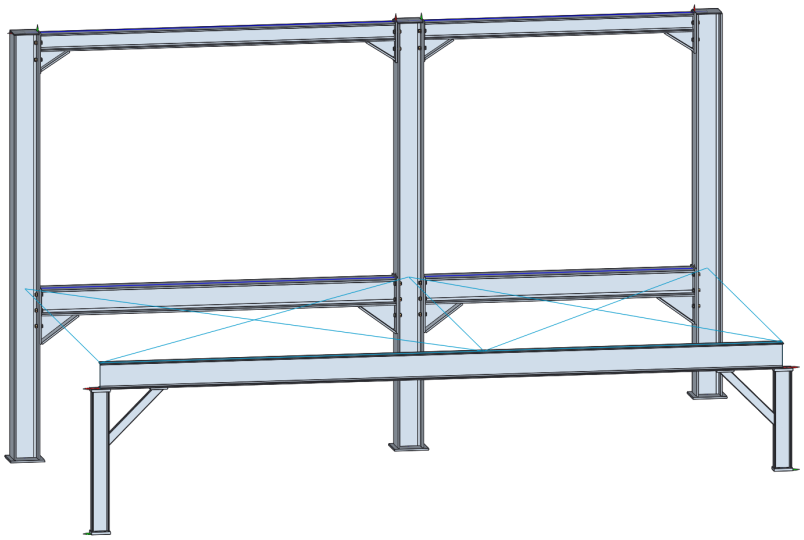
5. Assemble a **[Nonstandard > end plate (default values)]** on top of the rectangular tube **[D]**. Use the arrow shaped surface as attach face!
6. Apply a T-joint between the two profiles.
   - The definition of the subassembly is now complete. Switch back to the main assembly.



Now we will copy the subassembly to the right.

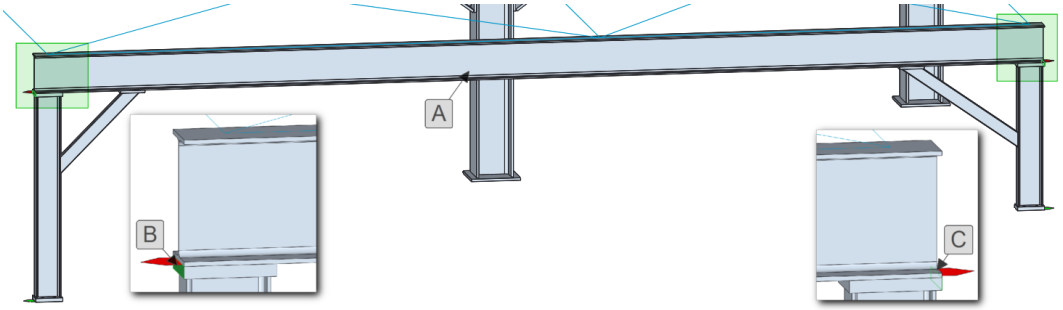
1. Click **Reuse** in the **Subassembly** Ribbon group.
2. Click and select `PS_COLUMN_4` as element to copy.
3. In the **Copy Component Definition** enter the new name **[ps_column_5]** and mark the vertical beam for copy.
4. It is necessary to switch the start and end of the subassembly. Therefore press the button.

5. Also use the rotate commands until the desired rotation is achieved .
6. Create a joint  **Tto selected surface** between the arrow shaped surface of the copied subassembly and the lower surface of the beam, that is part of the main assembly.
   - Both subassemblies are now fully defined.

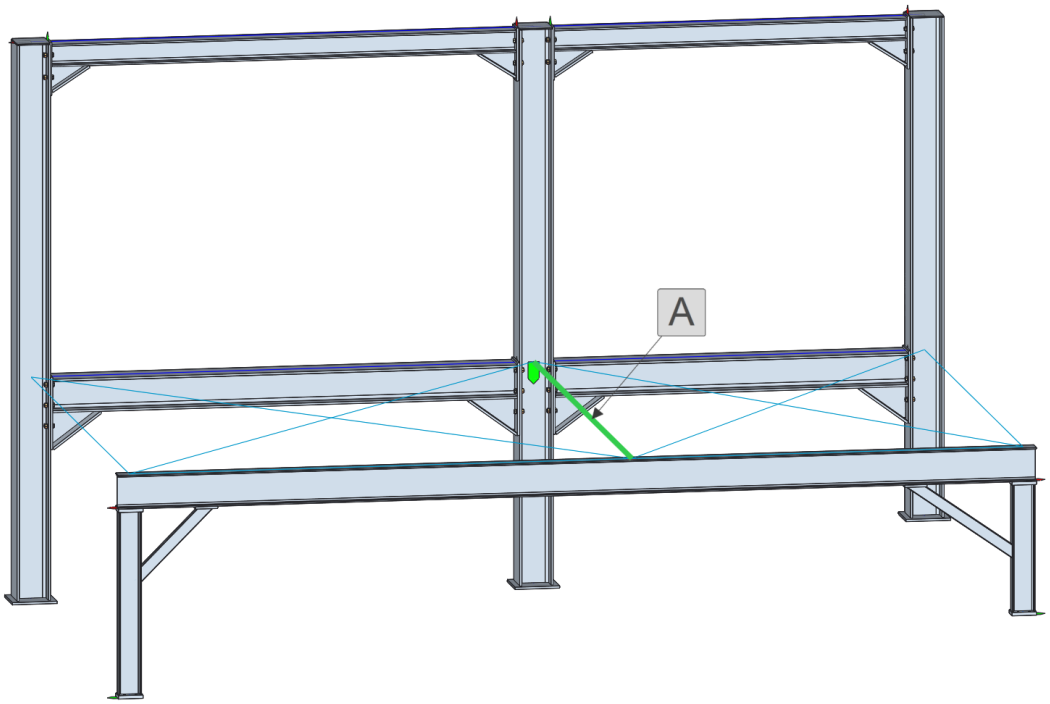


It could be necessary, that the angle of one of the two subassemblies has to be different due to stability optimization. To do so the rectangular tube must be copied first, because at the moment both subassemblies share the same tube model.

1. Click **Replace** in the **Profiles** Ribbon group.
2. Click **Replace by copy of itself** and select the rectangular tube of `PS_COLUMN_5.ASM` and confirm with **OK**.
   - Now that the two profiles are separate models they can be changed independently.
3. Edit the sketch of `PS_COLUMN_5.ASM` and change the angle to **[55°]**.
   - After regeneration all parts will align in their new position and the tube will be extended to fit to the changed curve.
4. Extend the I-beam **[A]** on both sides via joint type  **To selected surface** and reference it to the outer surface of the DAST plate of the two subassemblies **[B]** and **[C]**.



The backside of the frame is now complete. The next step will be to create the connecting profiles between the front and the backside of the frame.

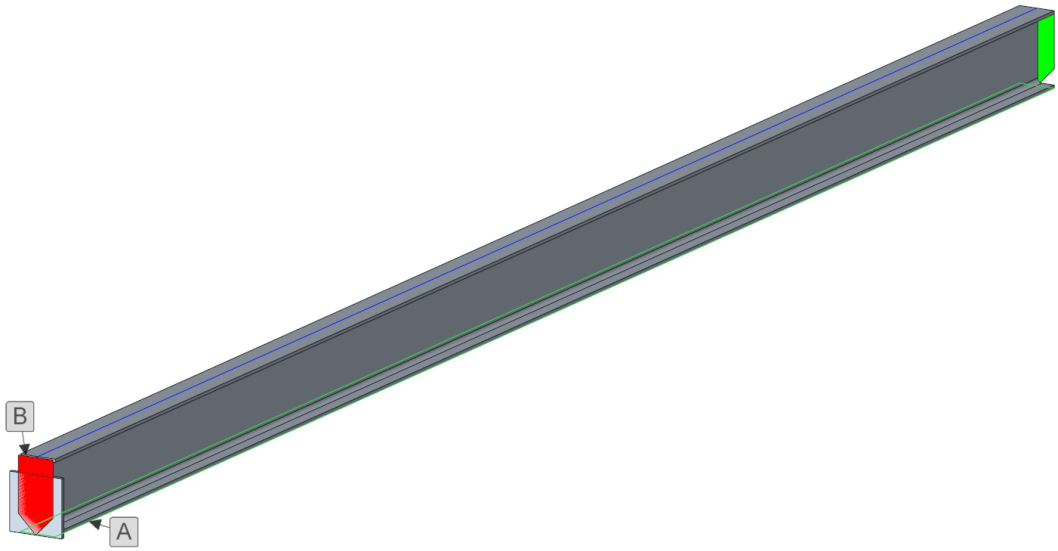


1. Assemble a new project subassembly named **[PS_BEAM_6]** on the middle curve **[A]** use A_YZ as orientation plane.
2. Create a joint type  **To selected surface** with the two arrow shaped surfaces and the I-beam web side surfaces.

After the subassembly is added and the joints are defined you should open the subassembly in a separate window to avoid external references.

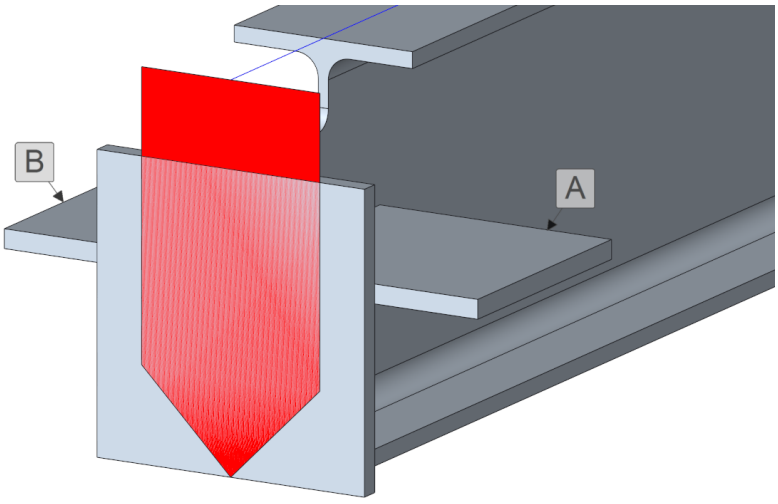
Follow these steps to complete the just created subassembly.

1. Assemble an **[I-Beam > DIN 1025 IPE > 200]** on the curve and use A_YZ as orientation plane.
2. Move the profile via , so that it is aligned with the datum curve.
3. Assemble a non-standard **[ENDPLATE TOP]**.
4. Enter values: **[H=155]**; **[T=10]**; **[OFFSET=0]**.
5. Use the bottom side of the I-beam as **Profile top (1) [A]**.
6. Use the red arrow shaped surface as **Attach face (2) [B]**.



7. Create a **[PROFILE END CUT OUT]** from the **[connector > STEEL CONSTRUCTION MM > NONSTANDARD]** folder.
8. Enter values: **[L=80]**; **[H=45]**; **[R=10]**.
9. Select the I-beam on the top surface as **Cut surface**.

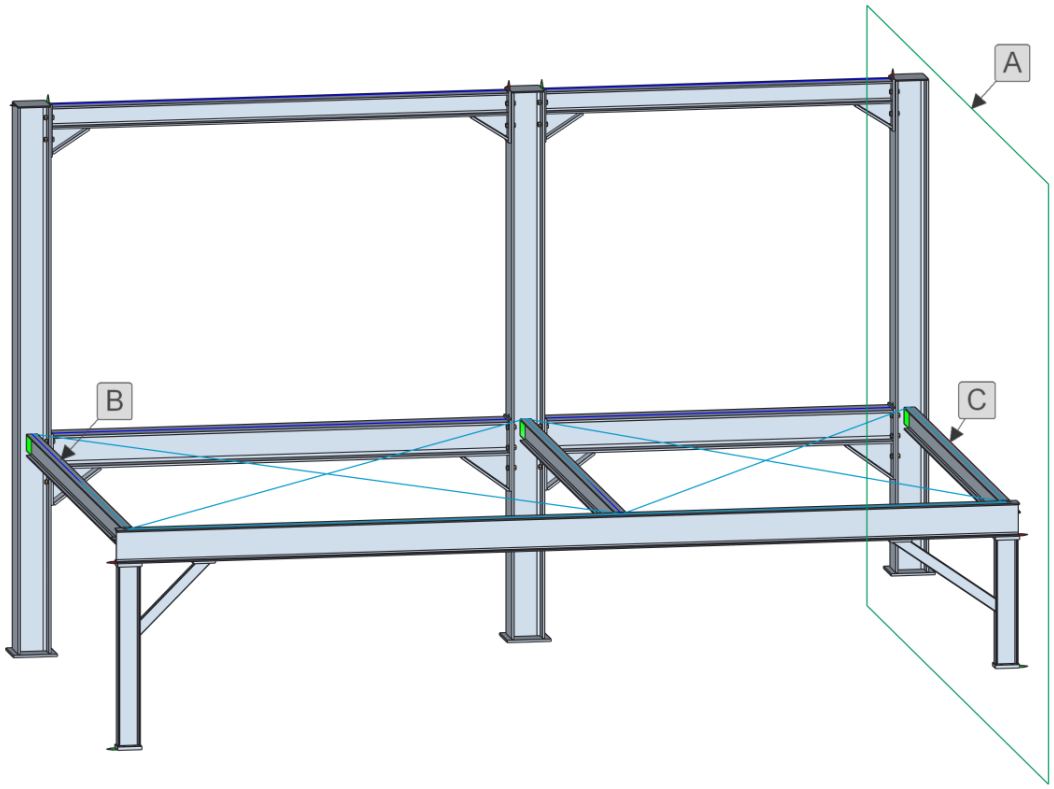
   NOTE: The **PROFILE END CUT OUT** feature does not have references to any other I-beam shape. It can be redefined with **AFX**, but also directly in **Creo Parametric**.
10. Assemble **[Connectors > STEEL CONSTRUCTION MM > NONSTANDARD > RECT CORNER PLATE]** as shown in the picture **[A]**.
11. Enter values: **[L=130]**; **[H=130]**.
12. Reassemble the plate **[A]** with **Reuse** to the other side **[B]**.



After the assembly of the corner plates the subassembly is complete and can now be copied. Switch back to the main assembly `PROJ_SUBASSEMBLIES.ASM`.

Now copy the subassembly.

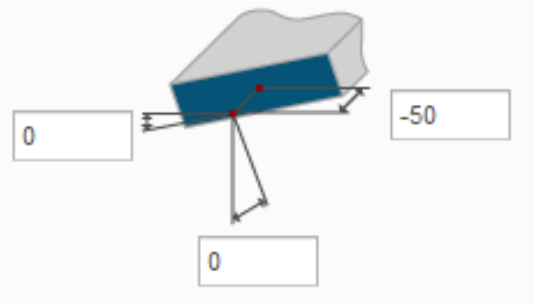
1. Click **Reuse** in the **Subassemblies** ribbon group..
2. Click and select PS_BEAM_6 as element to copy.
3. In the **Copy Component Definition** enter the new name **[ps_beam_7]** and click **OK**.
4. Select the left curve **[B]**.
5. Select the orientation plane A_YZ **[A]** and click **OK**.
   - The subassembly is copied.
6. Now repeat the above steps and copy the subassembly once more. The new name is **[ps_beam_8]**.
7. Select the right curve **[C]** as placement curve.



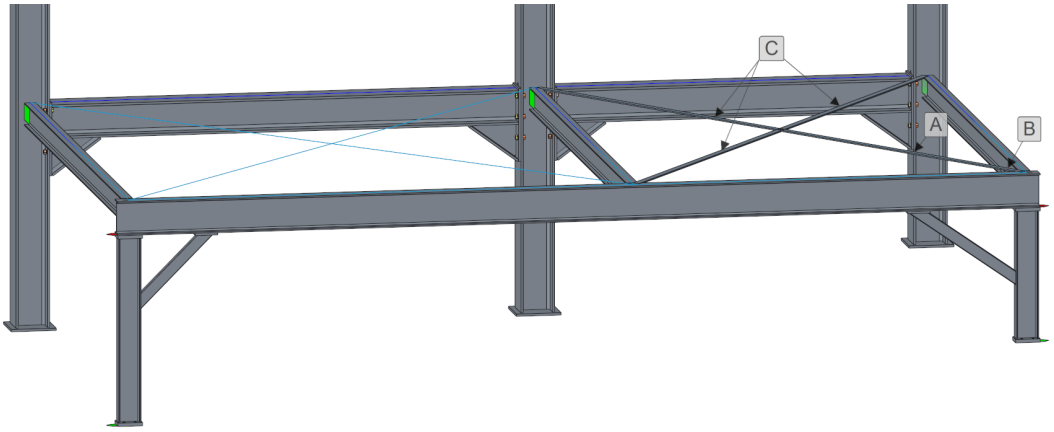
As we don't need the **RECT CORNER PLATES** facing outwards of the PS_BEAM_7 and PS_BEAM_8 simply delete them with the **AFX** delete functionality, but be careful not to delete the complete subassembly.

The next step is to assemble the center profiles.

1. Assemble a **[Rectangular beam > DIN 1017 > 60x10]** on one the curve **[A]** and use A_XZ as orientation plane.
2. Open the **Modify Ends** dialog and modify the profile end **[B]**. Enter offset value: **[-50]**.



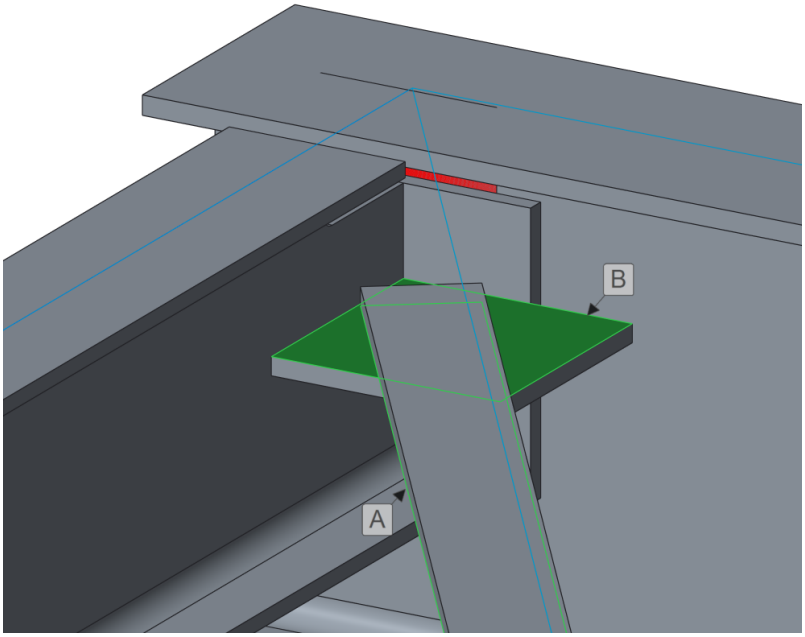
3. Reassemble the just assembled profile on the remaining three curves **[C]**. Rotate or switch start and ends to align the profiles correctly.



Now align the new profiles with the corner plates.

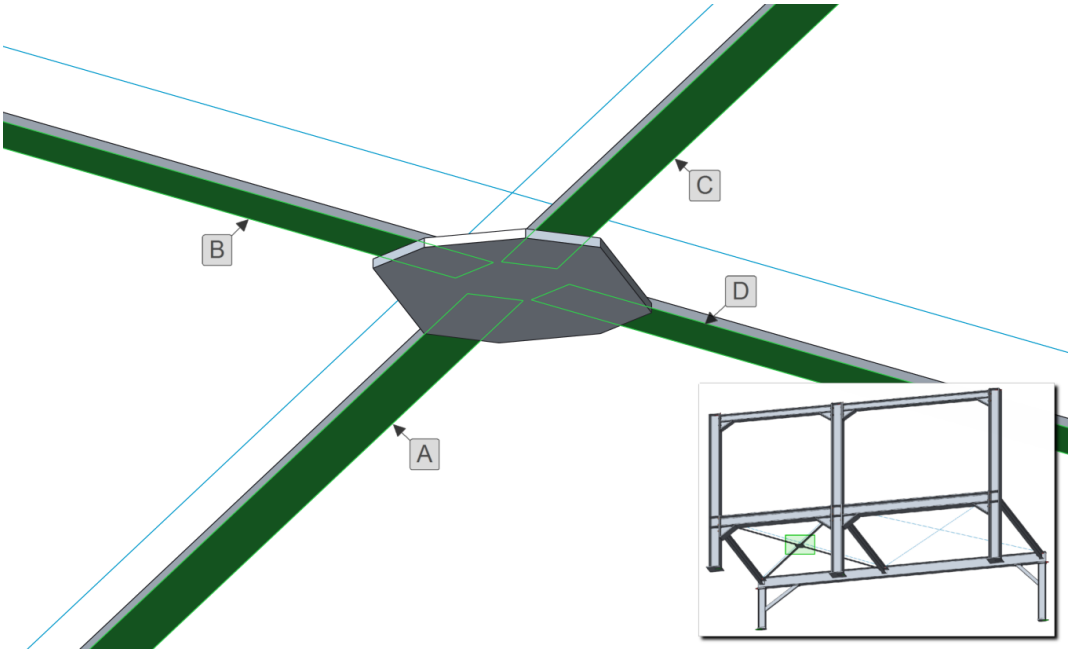
1. Click **Move** in the **Profiles** Ribbon group.
2. Select the surface **[A]** of **ALL** four profiles.
3. Select the highlighted surface of the **RECT CORNER PLATE [A]** as reference.

- The profiles are now aligned with the **RECT CORNER PLATE**.



To assemble the **Bracing plate cross** in the center of the profiles proceed as follows.

1. Select **[connectors > Steel construction mm > bracing plates > bracing plate cross]**.
2. Enter valus: **S=10**; **L1=50**; **L2=100**; **L3=80**; **L4=25**; **L5=50**.
3. Click and select the references **[A]**, **[B]**, **[C]** and **[D]**.
   - The plate is assembled.



Optionally you can copy the four profiles and the bracing plate cross to the other side. If you do so copy the first profile and use the reassemble command afterwards.

This exercise is now complete.

5.8 Review

In this chapter you learned:

1. How to work with subassemblies correctly.
   - Avoid external references by opening the assembly in a new window.
   - Try to work with AutoUDFs if you copy assemblies.
2. How to keep references healthy.
   - Redefine external references.
   - Joint definition.

Notes:

6

Stairs and Handrails

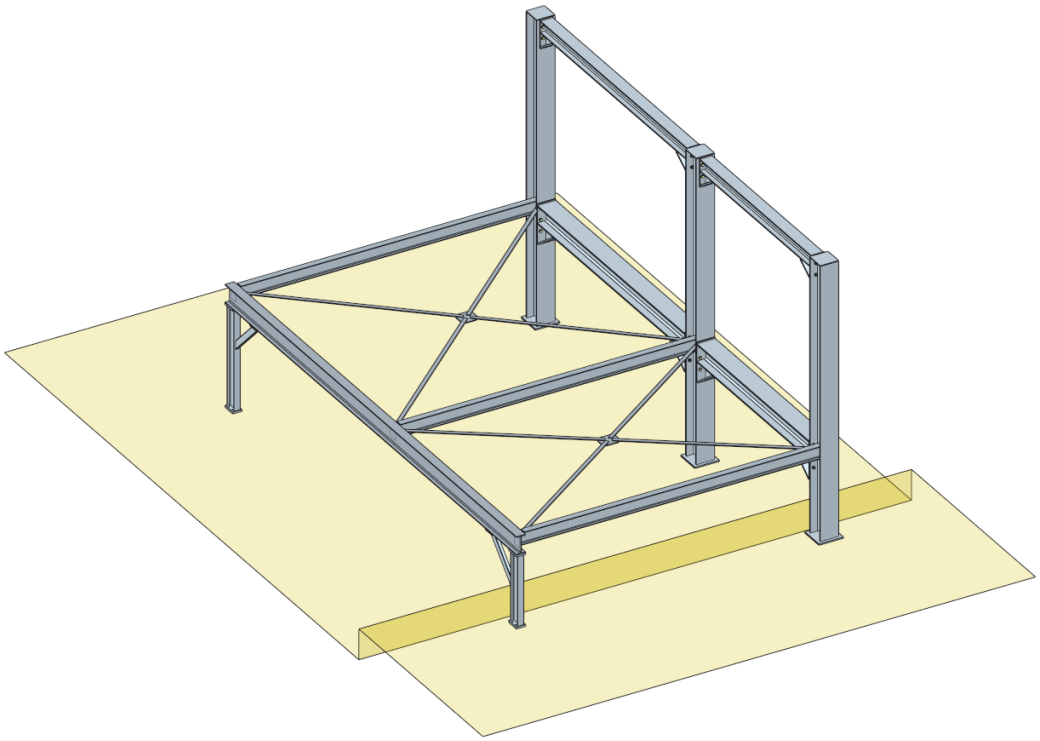
Overview
Assemble and modify stairs
Copy modified stairs
Assemble and modify handrails
Custom handrails

6.1 Overview

The **AFX** equipment library contains some types of stairs and handrails. If these types do not fit your requirements, you can assemble and modify them with **AFX** or standard **Creo Parametric** functions to adjust them to your needs. You can also use these modified stairs and handrails and copy them to other locations with **AFX** functionality.

If you did not finish the previous chapter you can switch to the folder `subassemblies_stairs_and_rails_done_p1`

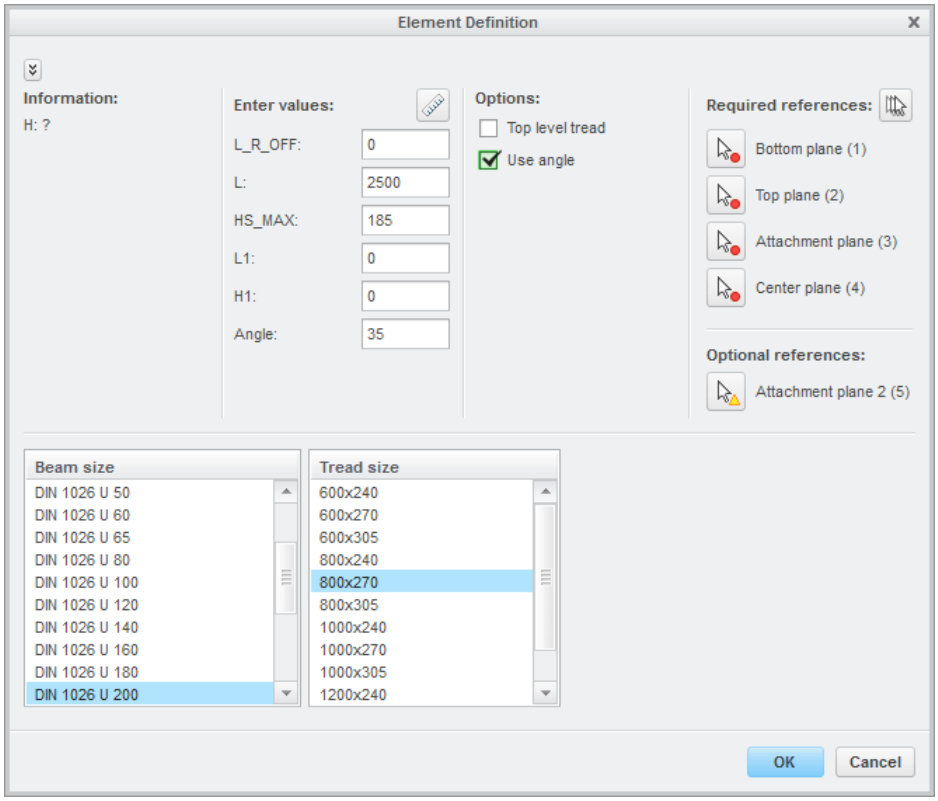
and reopen the `PROJ_SUBASSEMBLIES.ASM`.



6.2 Assemble and modify stairs

The **AFX** library contains two types of stairs. While type **stairs_1** is mainly used when the stairs are placed on the floor, type **stairs_2** is used when the stairs run from one platform level to another platform level.

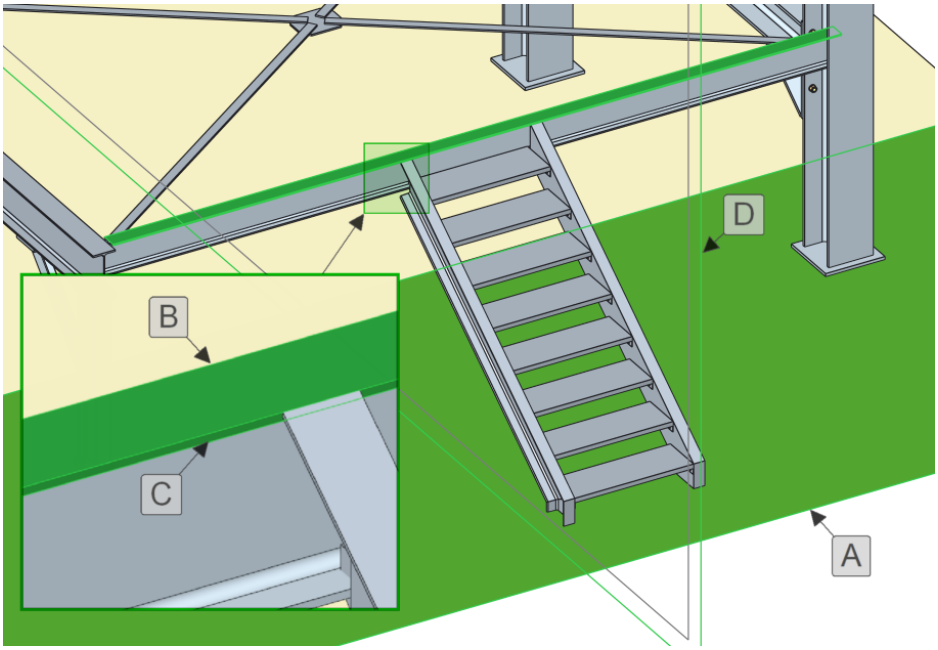
1. Click **Equipment Elements** to open the **Equipment Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTRUCTION MM] > [STAIRS] > [STAIRS_1]**.
   - The Dialog opens. Define the element as shown below in the picture.



Hint 10 — Stairs definition. The value in input field **Angle** will be used for stairs steepness. If you do not check this option, then the value in input field **L** will be used. Together with the measured height this will result in a certain steepness. If you want to measure the length distance between attachment plane and another plane you can use the optional selection **Attachment plane 2**.

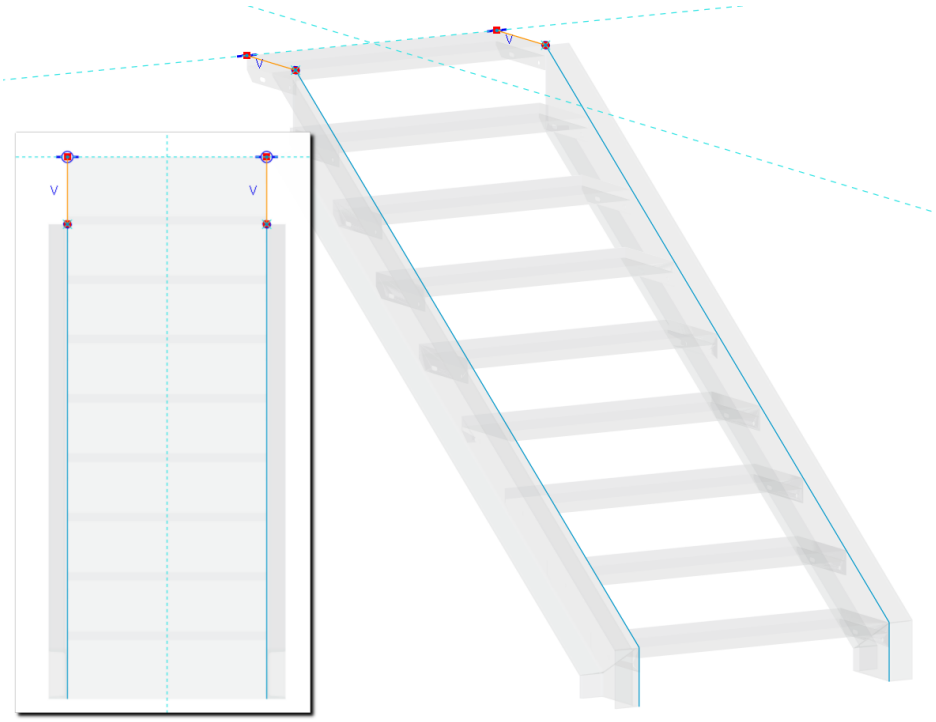
4. Click **[Bottom plane (1)]** and select the highlighted surface **[A]**.
5. Click **[Top plane (2)]** and select the highlighted surface **[B]**.
6. Click **[Attachment plane (3)]** and select the highlighted surface **[C]**.

7. Click **[Center plane (4)]** and select ADTM3 **[D]**.
8. Click **[OK]** to complete the definition.
   - The stairs equipment element is assembled.

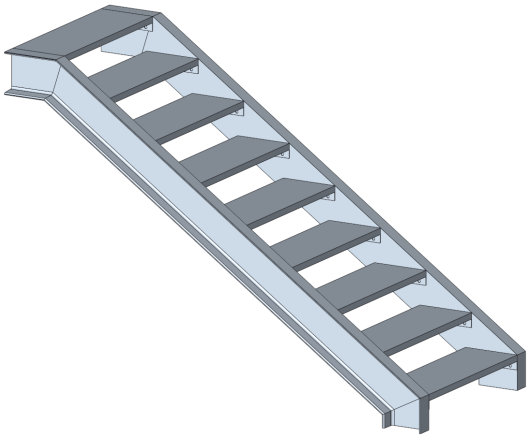


Sometimes stairs also have a tread and two additional U-beams at the top level. To mount the stairs, end-plates are very common. The following steps describe how to modify the stairs like this.

1. Open the equipment dialog and use the command to redefine the stairs.
2. Check box **[Top level tread]**.
   - Using the input fields **L1** and **H1**, you can move the upper end of the U-beams relative to the **Attachment plane**.
3. Enter Value: **[L1=270]** (tread depth).
4. Click **[OK]** to modify.
5. Open the stairs assembly in a new window.
6. Create a new sketch on ADTM1. Use A_XY and the ends of the CONTROL_CURVE as sketch references.



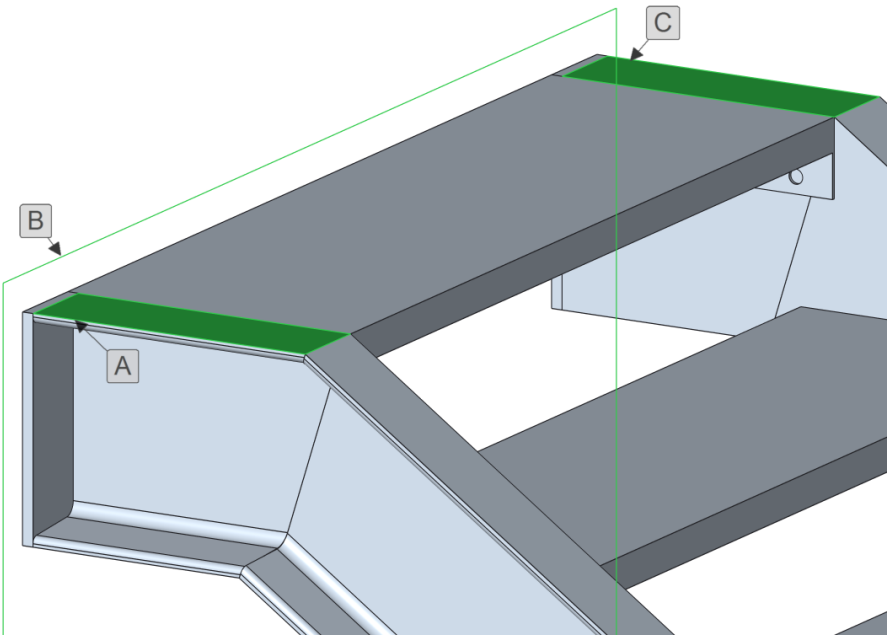
7. Add two beams of type **[channel beam] > [DIN 1026] > [U200]** using the just sketched curves as references. Rotate and move the beams and create two mitre joints between the existing and the new profiles.



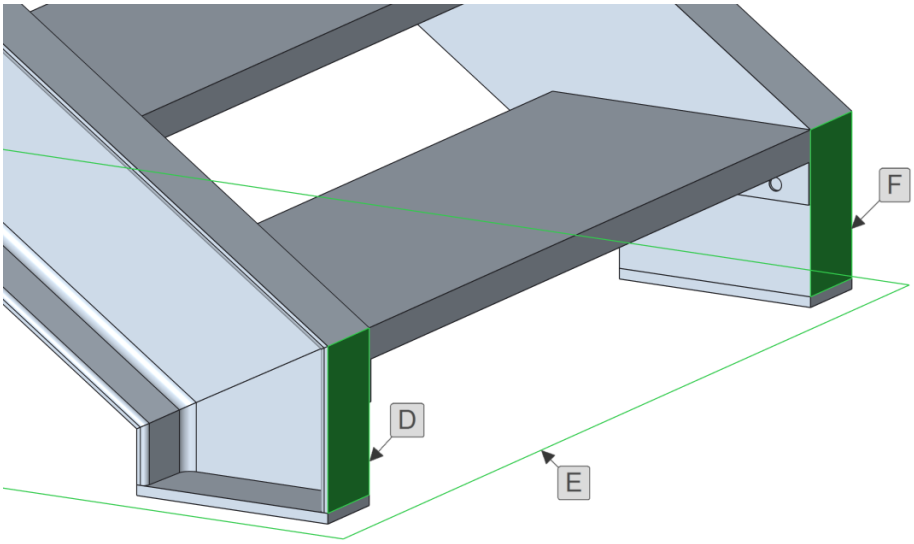
To complete the stairs definition add end-plates to the four channel beams.

1. Click **Connector Elements** to open the **Connector Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE]**.
   - The Element definition dialog box of the end plate opens.
4. Enter Values: **W=75**, **T=10**.
5. Click **[Profile top (1)]** and select the highlighted surface **[A]**.
6. Click **[Attach face (2)]** and select A_XY **[B]**.

7. Click **[OK]** to complete the definition.
8. Select the plate and reassemble it via **[RMB] > [Framework] > [Reassemble]**.
9. Select the highlighted surface **[C]** and the same datum plane as the existing element **[B].**



10. Now reassemble again on the bottom side with the references **[D]**, **[E]** and **[F]**, **[E]**.

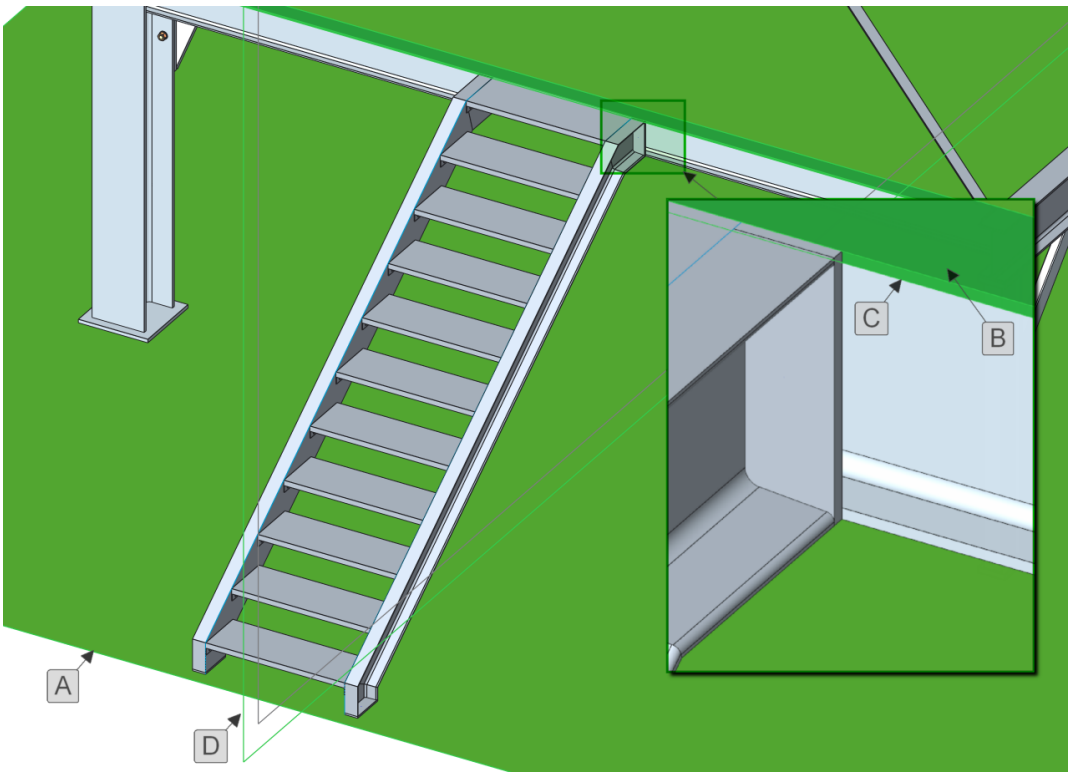


Close the window of the stairs subassembly and switch back to the top level assembly window.

6.3 Copy modified stairs

If you want to use the stairs just designed on other locations of the assembly with different height proceed as follows.

1. Click **Reuse** in the **Equipment Elements** Ribbon group to open the Reuse component dialog.
2. Click , select the previously assembled stairs as element to copy and press **Next**.
   - The **Copy Component Definition** dialog opens.
3. Click Automatic Selection to copy all required project parts.
4. Select the required references in the following order **[A]**, **[B]**, **[C]** and **[D]**.
   - The new stairs assembly is assembled.



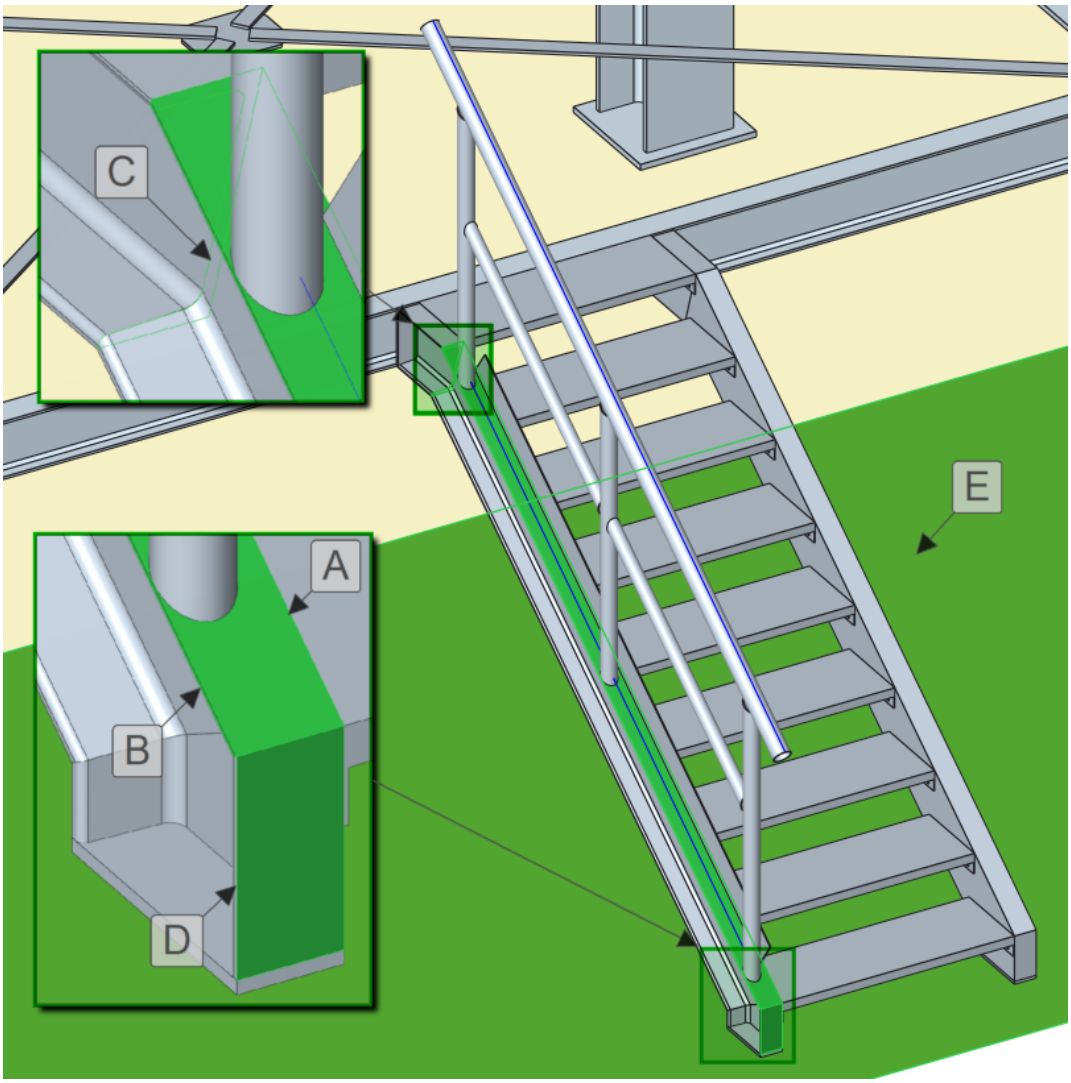
6.4 Assemble and modify handrails

The **AFX** library contains two types of handrails, one with angle beams and one with pipes. To assemble a rail of type rails_2 (with pipes) proceed as follows.

1. Click **New Equipment Elements** to open the **Equipment Elements** dialog box.
2. Click to open the **Select from library** dialog box.
3. Select **[STEEL CONSTRUCTION MM] > [RAILS] > [RAILS_2]** and press **Next**.
4. The Dialog opens. Choose the following sizes:

• Hand rail size: 60.3 x 2.3	• Foot rail size: 100 x 3
• Knee rail size: 44.5 x 2.3	• Post size: 54 x 2.3

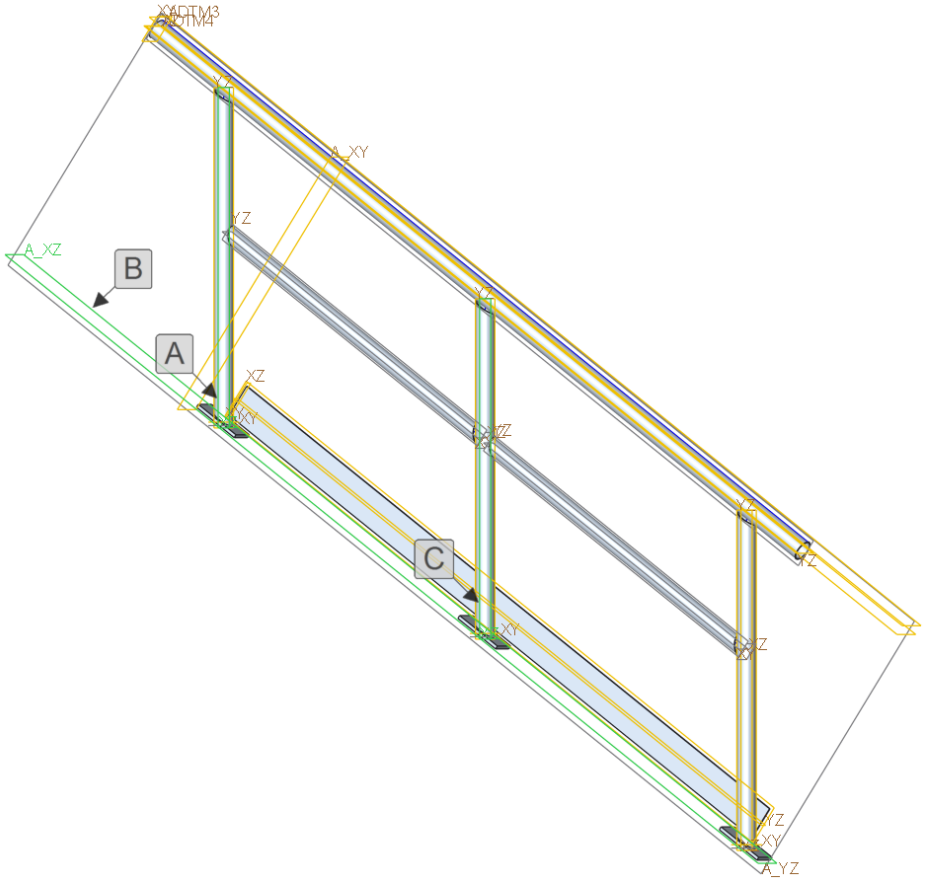
5. Click and select the required references in the following order **[A]**, **[B]**, **[C]**, **[D]** and **[E]**.
6. Click **[OK]** to complete the definition.



To complete the rail design open the assembly of the rail in a new window.

1. Unblank the layer `00_PROFIL_DATUM`.
   - You can see all base datum planes of the profiles and the stairs assembly.
2. Click **Connector Elements** to open the **Connector Elements** dialog box.
3. Click to open the **Select from library** dialog box.
4. Select **[STEEL CONSTR. MM] > [NONSTANDARD] > [END PLATE]**.
5. Enter values: **[H=150]**; **[W=75]**; **[S=10]**.

6. Click left to **[Profile top (1)]**.
7. Select the datum plane XZ of the pipe **[A]**.
8. Click left to **[Attach face (2)]**.
9. Select the datum plane A_XZ as attach plane **[B]**.
10. Click **[OK]** to complete the definition.
    - The first **END PLATE** is assembled.
11. Reassemble the end-plate via at the end of the second pipe (first member of first pattern). Select the datum plane XZ **[C]** of second pipe near the bottom end as **Profile top** and the datum plane A_XZ **[B]** as **Attach plane**.
12. Select the just assembled **end-plate** in the model tree and create a **Creo Parametric** reference pattern.
    - The end-plate will be reference patterned, so when the rails length changes and additional studs are added, the end-plates are added, too.

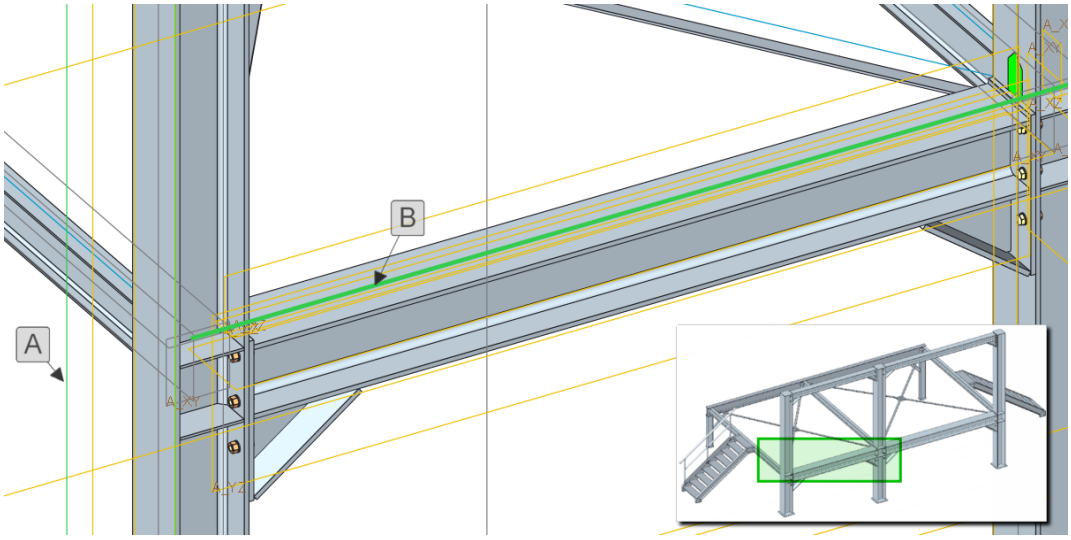


To proceed switch back to the main assembly PROJ_SUBASSEMBLIES.ASM.

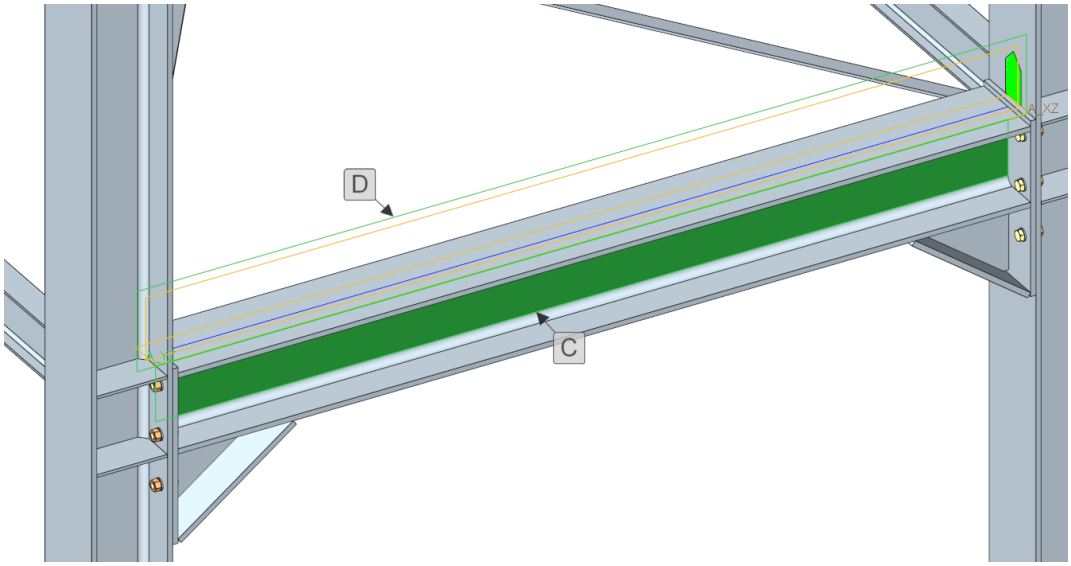
6.5 Custom handrails

In this section we will construct our own handrail. To do so we will use the **AFX** subassembly feature. This will enable us to reuse the design on other locations as well.

1. Click **New Project Subassemblies** to open the **Project subassemblies** dialog.
2. Assemble a new project subassembly on the lower left curve of the frame assembly **[A]**. Use the datum plane A_XY **[B]** of the top level assembly as orientation plane. Name it **[ps_custom_rails_1]**.



3. Align the subassembly via , that the front web surface of the horizontal I-Beam **[C]** is aligned with the datum plane A_YZ **[D]** of the subassembly.

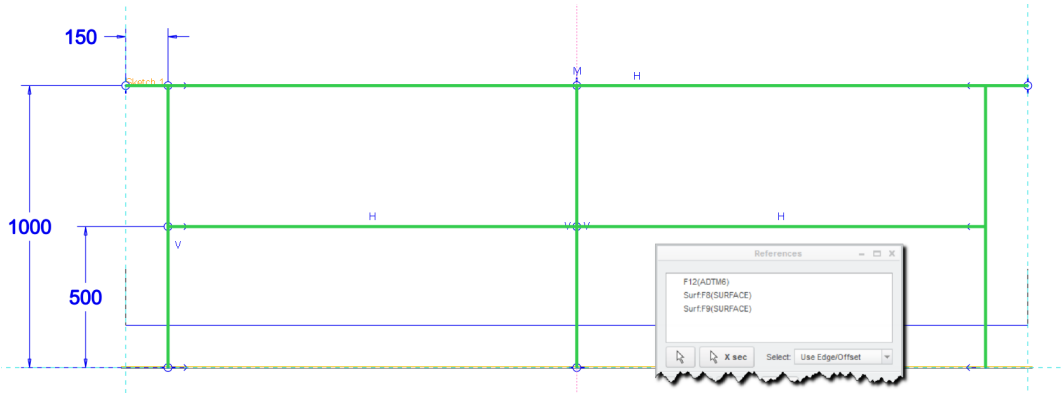


4. Add joints of type **to selected surface** between the arrow shaped surfaces and the vertical I-beams.

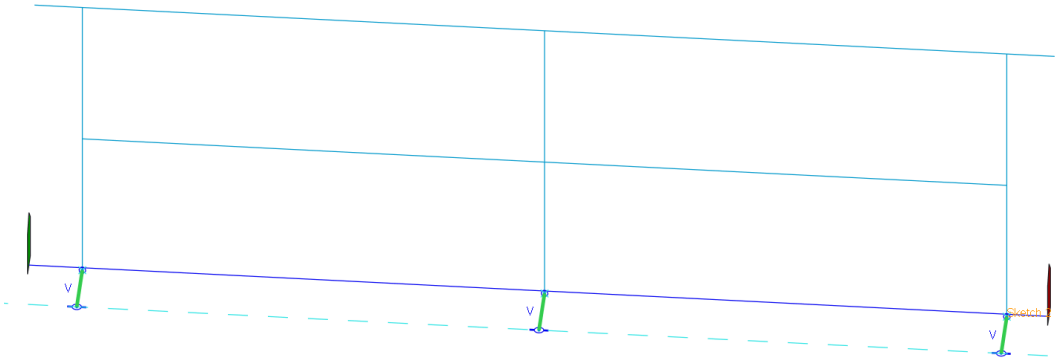
Open the subassembly in a new window and proceed as follows.

1. Create a new datum plane ADTM5 with **[200mm]** offset from A_YZ.
2. Create a datum plane ADMT6 with **[-150mm]** offset from A_XZ.

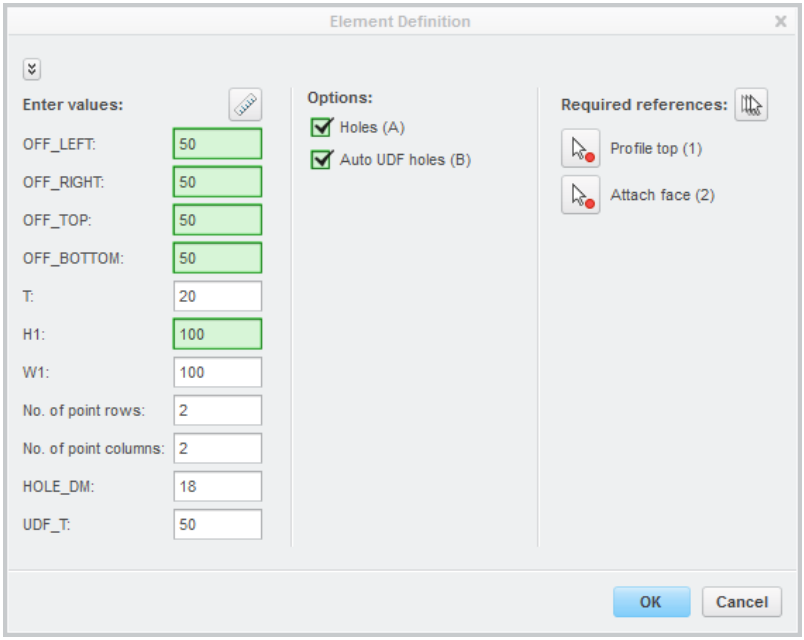
3. Create a sketch on ADTM5. Use two **separate** lines for the lines with **offset 500**.



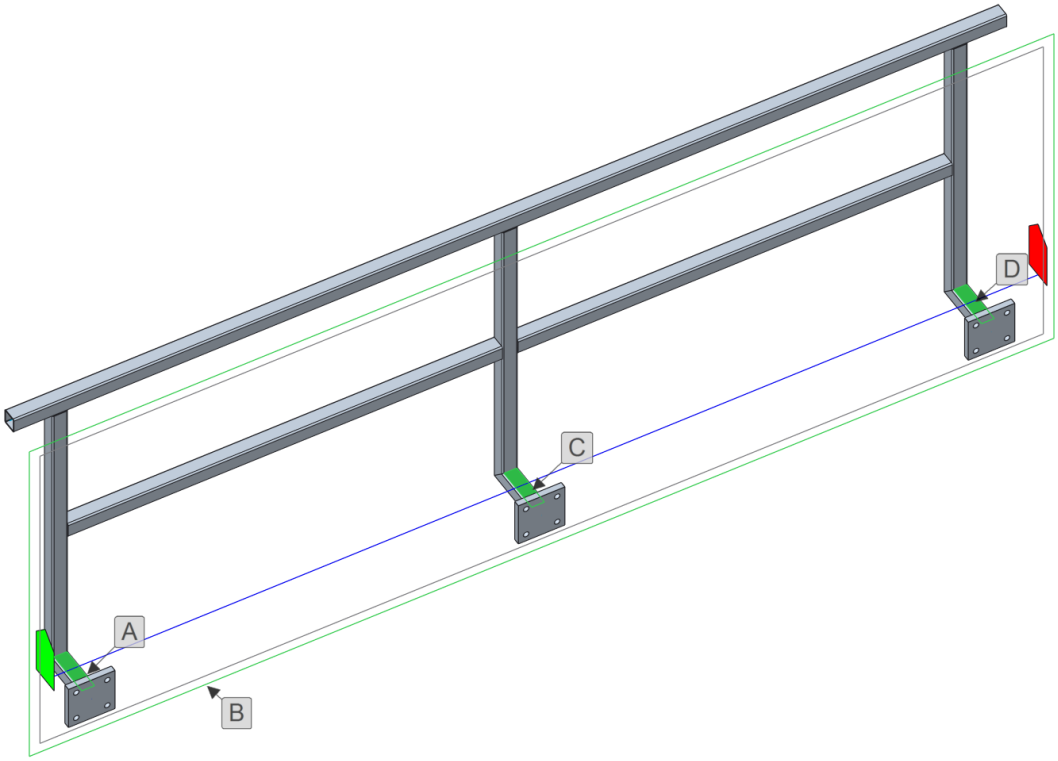
4. Create a sketch on ADTM6 and reference on the **end points** of the just created sketch.



5. Assemble profiles of type **[square tube] > [DIN 2395] > [50x50x3]** on all created sketched lines and create joints of type  and  .
6. The last step is to assemble three plates to connect the rail with the main assembly. Therefore assemble three plates of type **ENDPLATE ALIGN SIZE**.



7. Click **[Profile top (1)]** and select the highligted surface of the fist beam **[A]**.
8. Click **[Attach face (2)]** and select the datum plane A_YZ **[B]**.
9. Use to reassemble the plate on the other two profiles **[C]** and **[D]**.

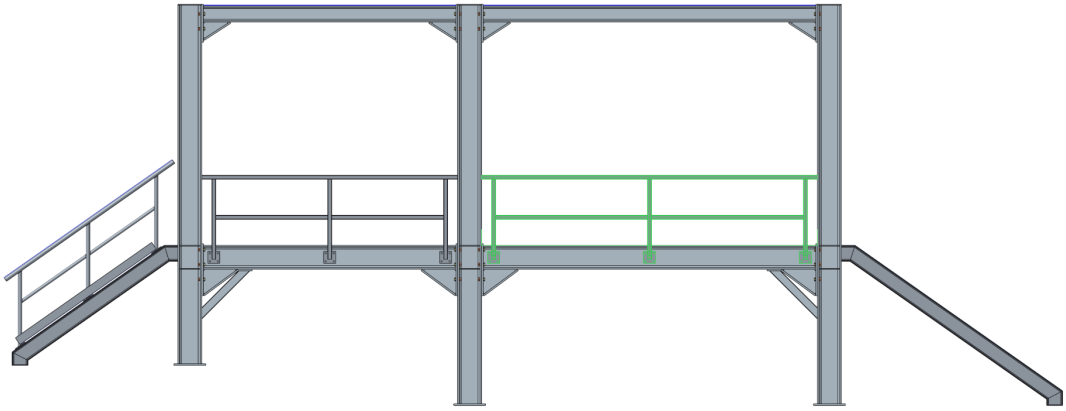


The custom rail subassmbly is now complete and can be reused as copy. Switch back to the main assembly to proceed.

1. Now copy the subassembly to the right with the new name **[PS_CUSTOM_RAILS_2]**. Select the horizontal square tubes to be copied as well, as they differ in length.

2. Add joints of type 

between the arrow shaped surfaces and the vertical I-beams.



The exercise is now complete.

Optionally you can add screw connections using the **Intelligent Fastener** functionality.

Notes:

7

Custom elements

Overview
Create a new Custom template
Define a template subassembly
Define a custom plate
Define a custom equipment assembly
Assemble custom elements

7.1 Overview

To unleash the full potential and productivity when working with **AFX** it is recommended to create your own custom elements. With **AFX** custom parts it is very easy for the administrator to create models according to company standards and forward them to the construction department. If used correctly they can save time and provide a good and unified model quality.

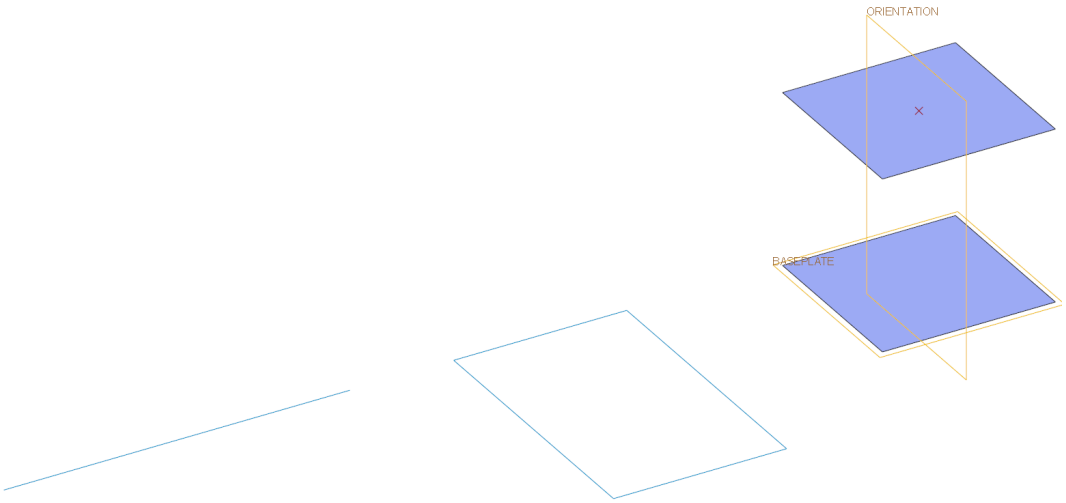
7.2 Create a new Custom template

To start with the customized elements you need to create a new **AFX** template assembly. In this case a template assembly is already created, but you can of course create your own template assembly later on.

To start with the exercise set the folder **CUSTOM_TEMPLATE** as new working directory and open the `CUSTOM_TEMPLATE.ASM`.

The template assembly consists of the following elements.

1. A straight curve (`REF_SUBASM`) – it will be used for a project subassembly.
2. A square curve loop (`REF_PLATE`) – it will be used to assemble a custom plate.
3. A variety of references – they will be used to assemble an equipment subassembly.

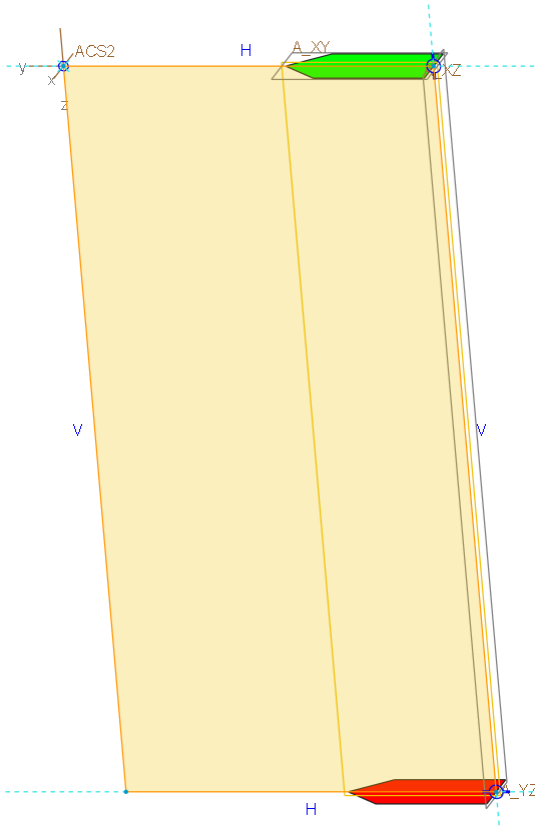


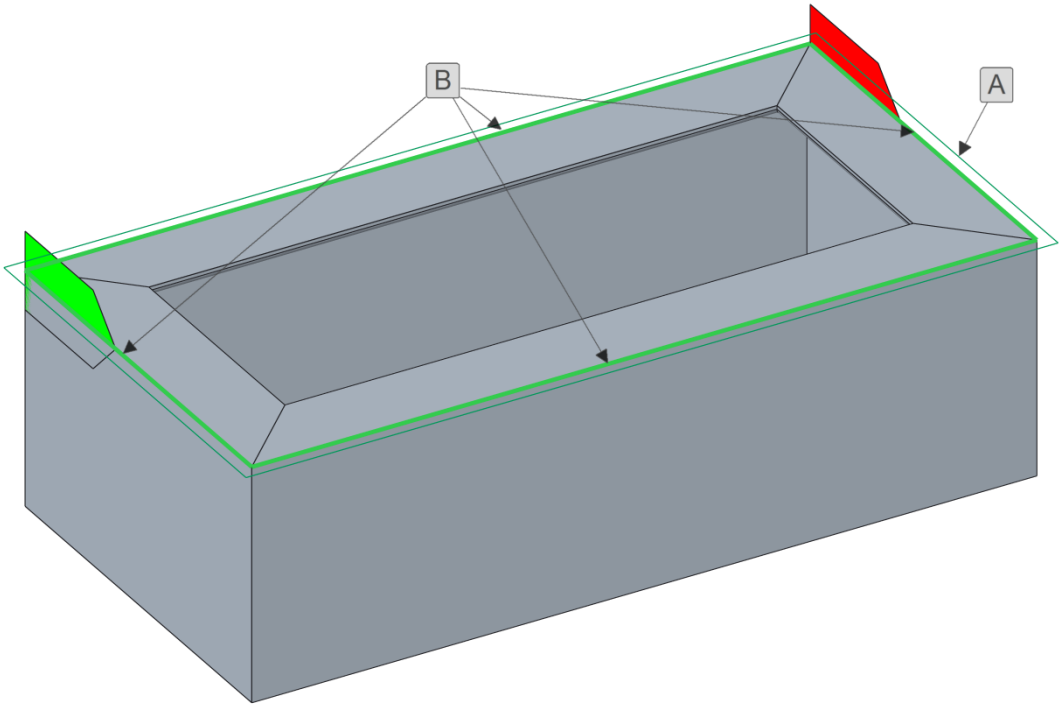
7.3 Define a template subassembly

In the next steps a frame subassembly will be created. Later on it can be placed on a datum curve to control the length and a special joint can be applied to control the width of the frame.

1. Assemble a new project subassembly named **[TEMP_EXTRAFRAME]** on the sketched curve `REF_SUBASM`. Use plane `XY` as orientation plane.
2. Open the assembly `TEMP_EXTRAFRAME` in a new window.

3. Create a new offset coordinate system called **[WIDTH_REF]**. Use ACS0 as reference and enter **[Y=500]**.
4. Create a new sketch on A_YZ. Use ACS0, the red arrow shaped surface and the just created coordinate system WIDTH_REF as reference. Now draw a rectangle and let it snap into the references. No dimensions should be shown.
5. Click **Profiles** and choose **[STEEL BEAMS MM] > [CHANNEL BEAM] > [DIN 1026] > [U300].**
6. Select A_YZ **[A]** of the subassembly as orientation plane.
7. Select the four datum curves **[B]**.
8. Use the rotation and move tools, to align the profiles as shown in the picture.
9. Create mitre joints between all four profiles.

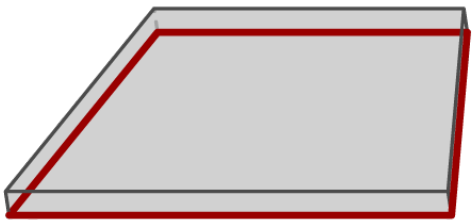




The first custom template is now complete. Switch back to the `custom_templates.asm` to continue.

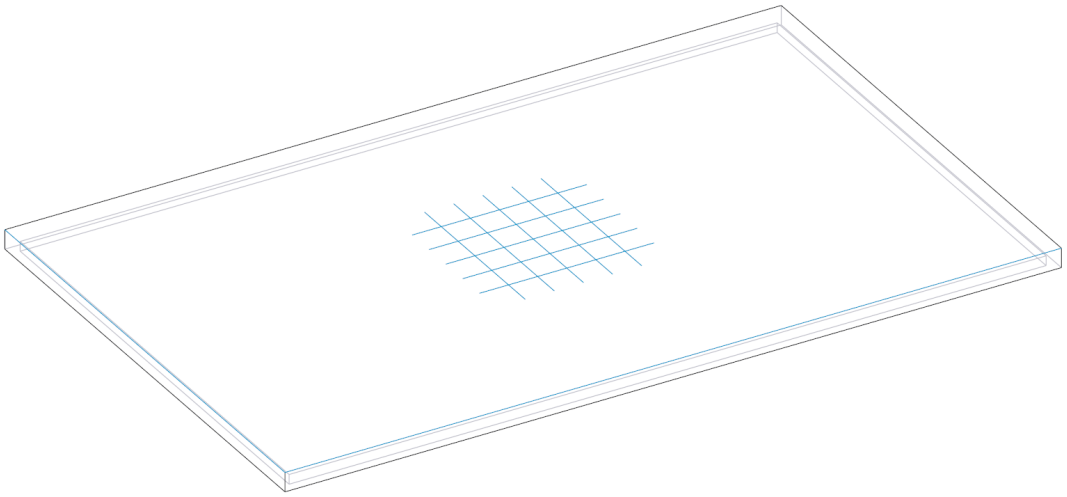
7.4 Define a custom plate

The next step is to create a custom plate. This plate will be a grating with custom parameters and a revised geometry.



1. Click **Equipment Elements**.
2. Click to open **Select from library** dialog box.

3. Select **[PLATES MM] > [PLATE 4 LINES]**.
4. Enter values: **[T=30]**.
5. Place the plate on the four lines of the REF_PLATE sketch.
6. Open the plate in a new window.
7. Create a **Creo Parametric** shell feature **[T=10]** and deselect the bottom surface of the plate. This will decrease the overall mass of the plate. We have to use this trick, because it would be very memory consuming to detail the grating as in reality.
8. To indicate the grating structure of the plate you can define sketch on top of it.
9. Edit the parameter **[BUW_NAME]** and change it to **[GRATING PLATE]** to complete the definition of the plate.
10. To make the part easier to find later on rename it with **Creo Parametric** standard functionality to **[TEMP_GRATING]**.



The custom equipment part is now ready. Switch back to the custom_templates.asm to continue.

7.5 Define a custom equipment assembly

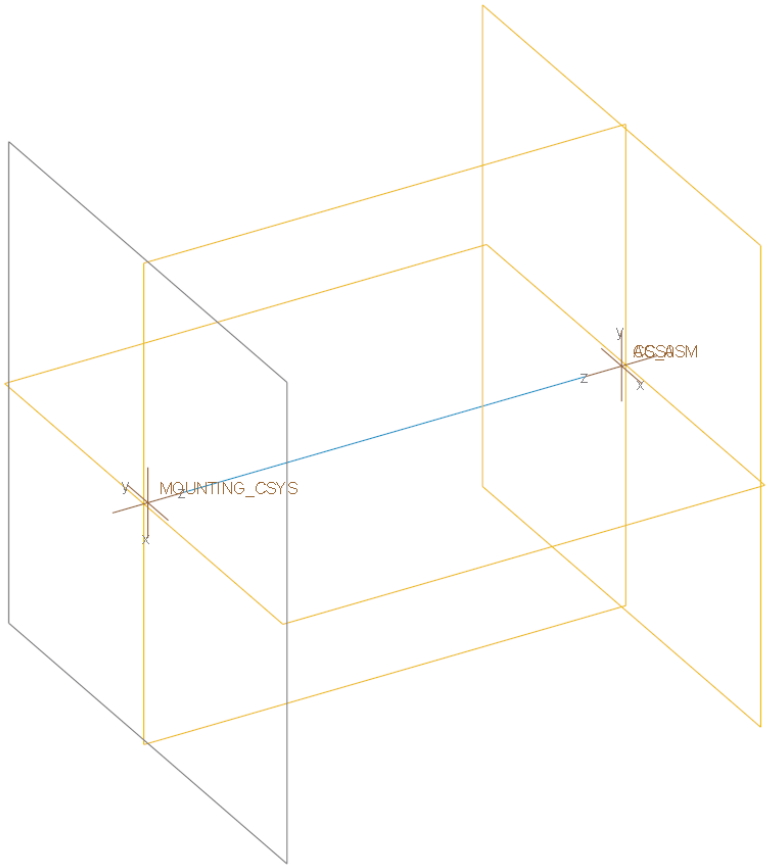
The last custom element will be an equipment subassembly. It will later be used to mount a machine on the frame of the previous chapter.

1. Click **Equipment Elements**.
2. Click to open the **Select from library** dialog box.
3. Select **[TEMPLATE MODELS MM] > [ASSEMBLY PLANE POINT PLANE]**.
4. Select the datum plane BASEPLATE **[A]** as 1st reference.

5. Select the datum point MOUNTINGPNT_MACHINE **[B]** as 2nd reference.
6. Select the datum plane ORIENTATION **[C]** as 3rd reference.
   - The subassembly will be placed. For further detailing open the subassembly in a new window.

Now we complete the equipment assembly. Proceed as follows.

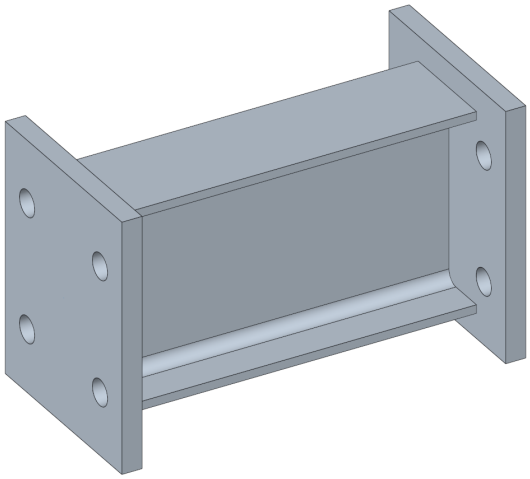
1. Create a new offset coordinate system named MOUNTING_CSYS select ACS0 as Reference and orient it like shown on the next picture. Enter values: **[Z=200]**, **[About X= 180]**, **[About Z=-90]**.
2. Create a new sketch on the datum plane A_XZ and use ACS0 and the MOUNTING_CSYS as references. To complete the sketch simply draw a straight line between the two coordinate systems.
   - As we want to add an I-beam and two end-plates to the assembly, we also need a datum plane as attach face.
3. Create a datum plane through the MOUNTING_CSYS and select XY. Make sure, that the plane has the right orientation.



4. Click **Profiles** and select profile type **[STEEL BEAMS MM] > [I-BEAM] > [DIN 1025 IPE] > [200].**
5. Select A_YZ of the subassembly as orientation plane.

6. Select the just created datum curve.
   - A new profile will be assembled.
7. Click to open the **connector elements** dialog.
8. Click and select **[STEEL CONSTRUCTION MM] > [NONSTANDARD] > [END PLATE]**.
9. Enter values: **[H=250]**; **[W=200]**; **[W_L=100]**; **[H1=125]**; **[W1=125]**; **[HOLE_DM=26]**; **[Holes (A) > ON]**; **[Auto UDF holes (B) > ON]**.
10. Assemble the **END PLATE** on the end of the I-beam.
11. Use reassemble and pace the plate on the other side of the beam.

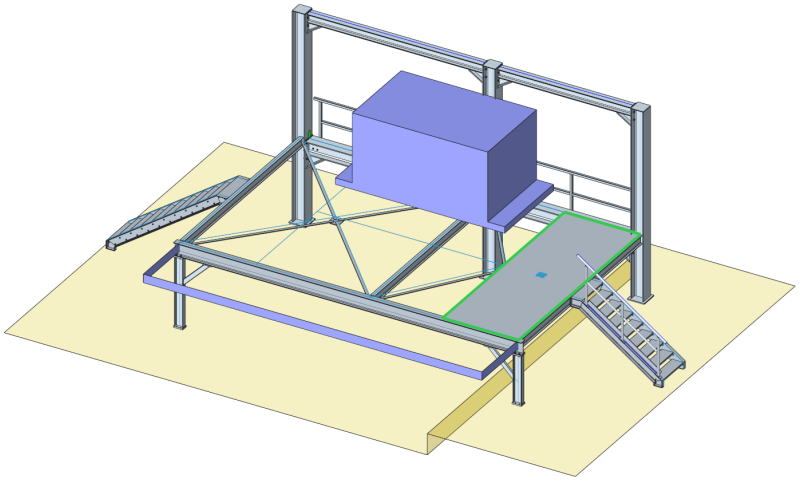
The custom equipment assembly is now ready.



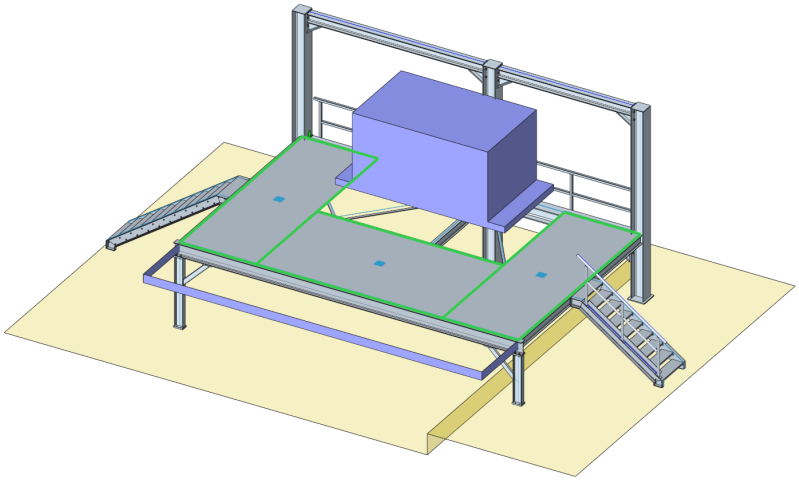
7.6 Assemble custom elements

After the creation of the custom elements, we of course want to assemble them into the main assembly. First of all open the main assembly `PROJ_SUBASSEMBLIES.ASM` from the `SUBASSEMBLIES_STAIRS_AND_RAILS_DONE_P2` folder.

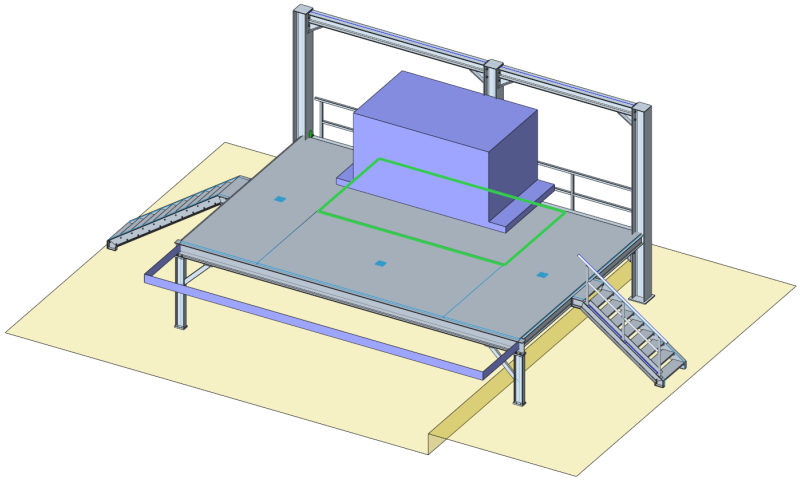
1. Unhide the layer `CUSTOM_ELEMENTS`.
2. Click to open the equipment dialog.
3. Click to assemble a customized equipment part.
4. Select instance type copy .
5. Select `TEMP_GRATING.prt` from the `CUSTOM_TEMPLATES_DONE` folder.
6. Select the curves as shown in the next picture. Be careful to select closed loops from one sketch. As attach option choose 1.



7. Place grating plates on the other two rectangular sketches as shown in the next picture.

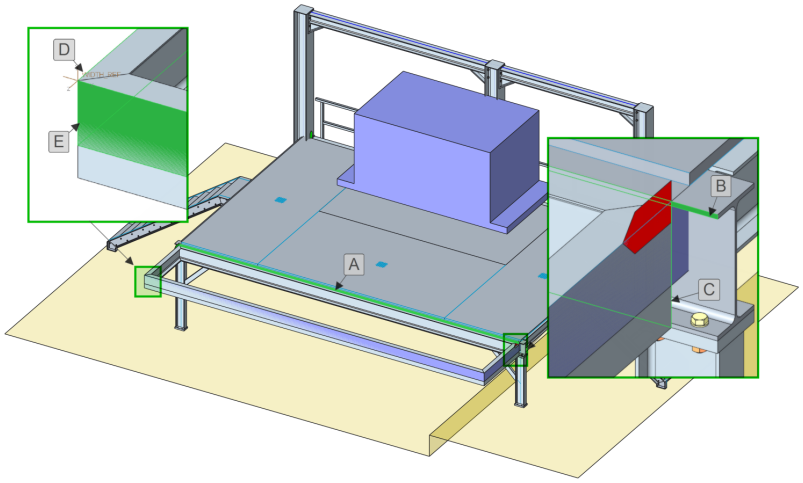


8. Assemble a standard plate (T=30) from the AFX library on the remaining sketch.

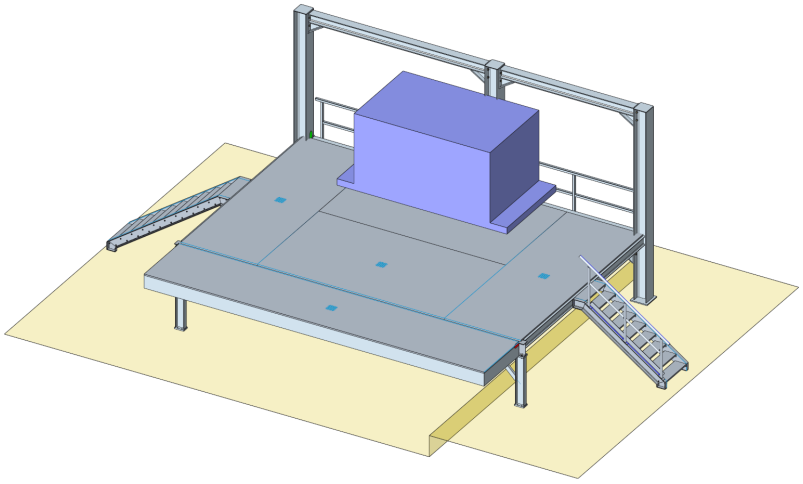


The next step is to assemble the TEMP_EXTRAFRAME.ASM subassembly.

1. Click **Reuse** in the **Subassemblies** Ribbon group to open the **Reuse Subassembly** dialog.
2. Click to assemble an instance from file and open TEMP_EXTRAFRAME.ASM from the CUSTOM_TEMPLATES_DONE folder.
3. Select instance type **Copy existing** and press **Next** to proceed.
4. Enter the new name [PS_SUB_EXTRAFRAME] and mark the two beams for copy.
5. Select the sketched curve **[A]**.
6. Select the datum plane **[A_XZ]** as orientation plane and confirm with **OK**.
7. Open the **Move Subassemblies** dialog and click and select the highlighted surface **[B]** and the highlighted surface **[C]** to align the assembly correctly.
8. Create a joint of type **Csys to plane** and select the coordinate system WIDTH_REF **[D]** and the front face of the surface **[E]** to complete the frame.

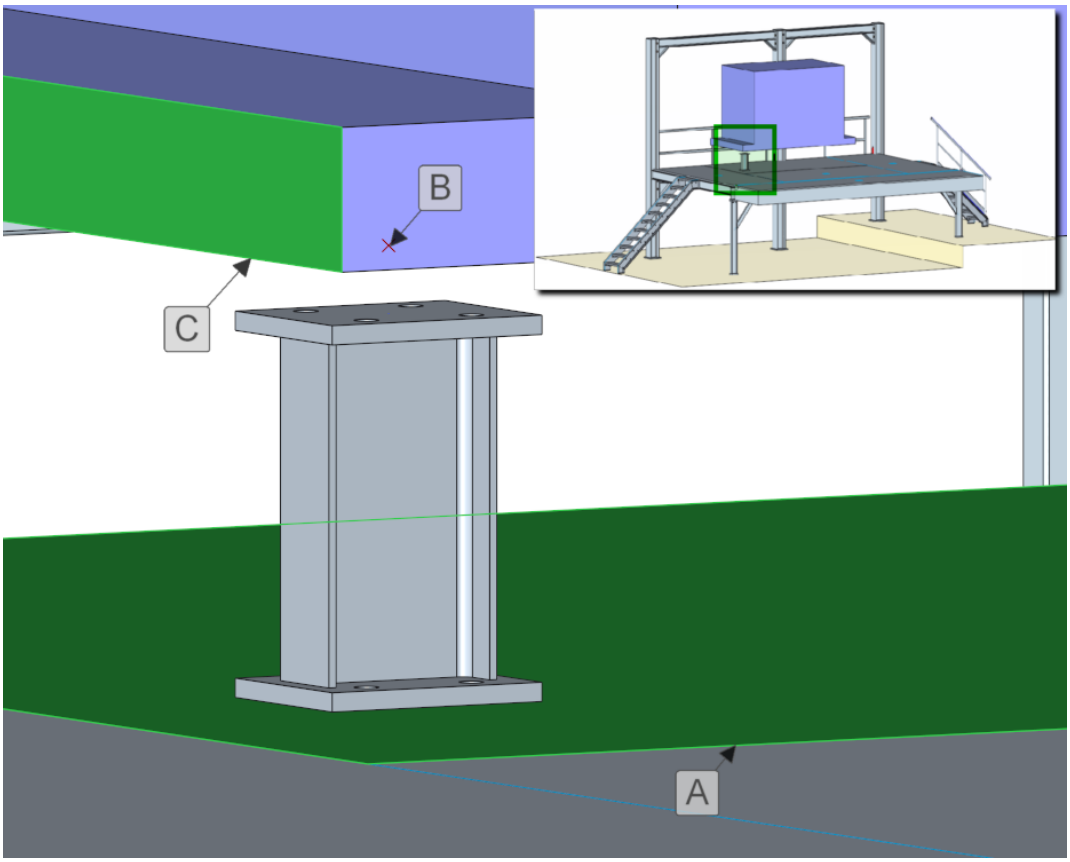


Optionally you can also copy the grating plate on top of the just assembled extra-frame.

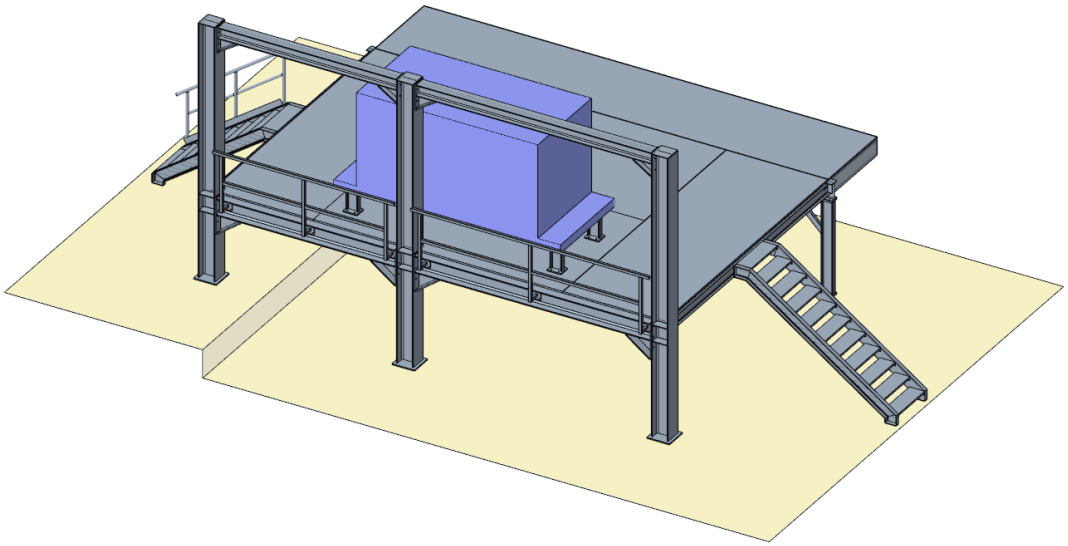


The last step is to assemble the equipment assembly.

1. Click ↺ to open the equipment dialog.
2. Click to assemble a customized equipment part and
3. Select `TEMP_ASSEMBLY_PLANE_POINT_PLA_1.ASM` from the `CUSTOM_TEMPLATES_DONE` folder.
4. Select instance type and press **Next**.
5. Enter new name `ps_foot_1` and select all parts for copy.
6. Select the highlighted references in the order **[A]**, **[B]** and **[C]**.



7. To finish the 1st mounting foot apply a to selected surface joint between the coordinate system `MOUNTING_CSYS` and the bottom surface of the machine skeleton.
8. Afterwards the mounting foot can be reassembled via ↺.



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