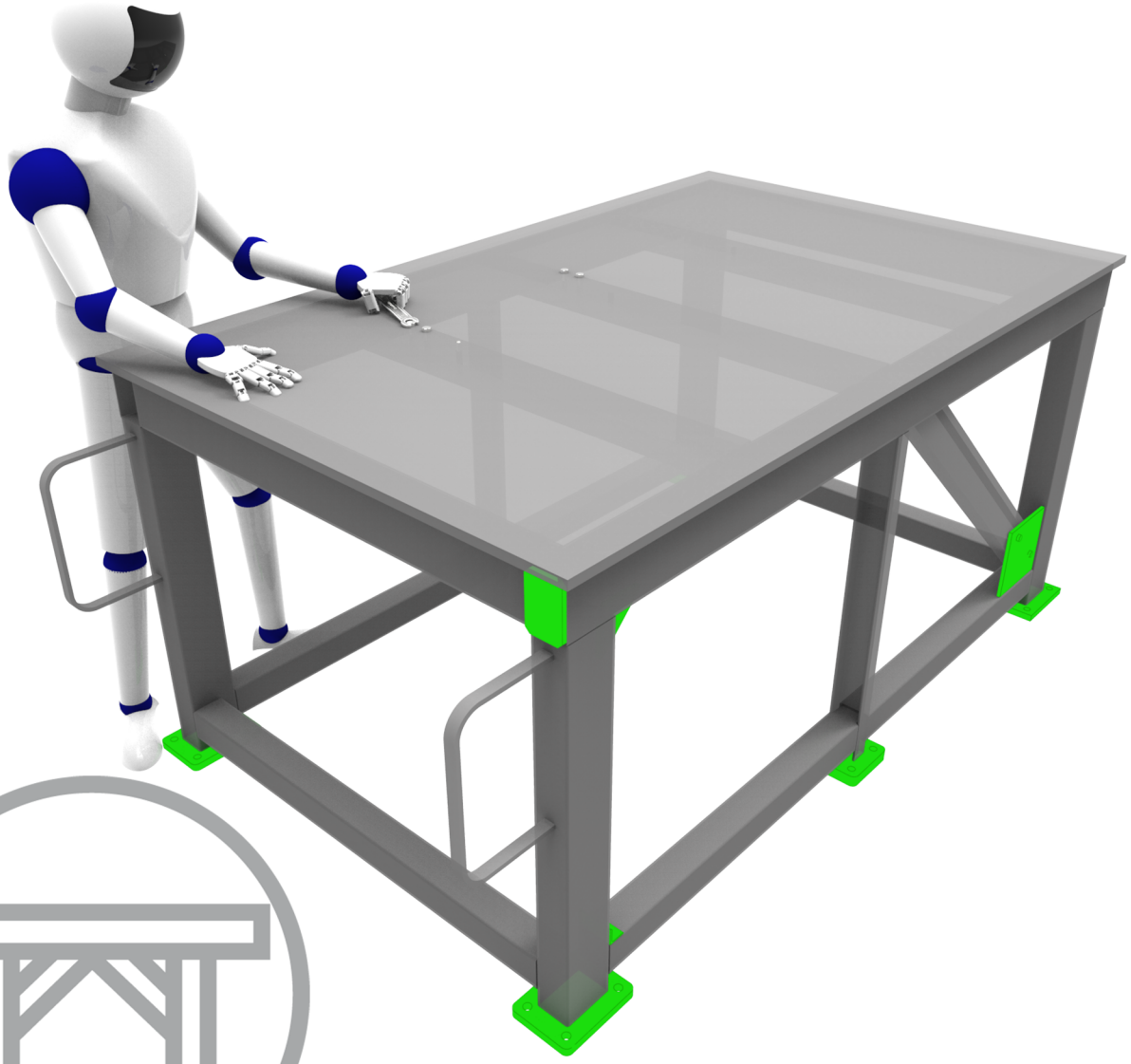
Advanced Framework Extension 5.0

Training Guide: Basic Tutorial



Version, 09.05.2018

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Introduction

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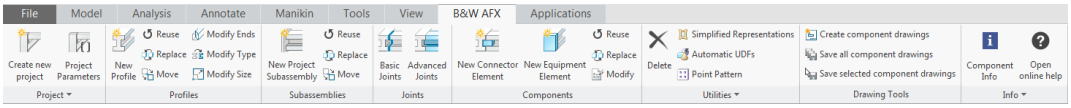
1.1 Objective of this document

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

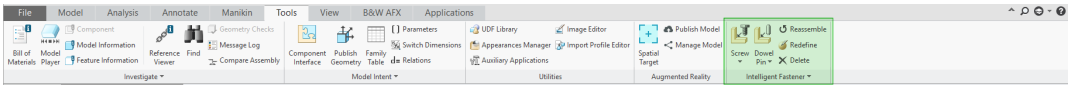
It is assumed that you are familiar with the basic functions of **Creo Parametric**.

1.2 Overview

The **Advanced Framework Extension 5.0** (**AFX**) is an add on software to **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 5.0** 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 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
	**Reuse:**

	**Replace**
	**Move**
	**Modify Ends**
	**Modify Type**
	**Modify Size**
	**New Subassemblies**: Contains functions for working with project subassemblies
	**Reuse:** Reuse an existing subassembly
	**Replace:** Replace an existing subassembly
	**Move:** Move an existing subassembly
	**Basic Joints**: Basic profile joint commands
	**Advanced Joints**: Advanced profile joint commands
	**New Connector Elements**: Create new connector element
	**New Equipment Elements**: Create new equipment element
	**Reuse:** Reuse an existing connector or equipment
	**Replace:** Replace an existing connector or equipment
	**Modify:** Modify an existing connector or equipment
	**Delete**: Delete AFX element
	**Simplified Representations**: Contains functions for defining weldment groups to structure smaller structural assemblies into different weldments
	**Automatic UDFs**: Automatic-UDFs commands
	Point Pattern:
	**Weld Groups**: Weld group 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
	**Online Help:**
	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)

2

Design Frames with Steel Profiles

Overview
Design Process using AFX
Define a Project
Assemble Steel Profiles
Define Joints between Profiles
Move Profiles
Modify and Replace Profiles

2.1 Overview

This chapter explains the basic functions for designing framework with AFX. It is done by using a simple steel frame, but the used functions also apply for designing frames with aluminum profiles.

Objectives:

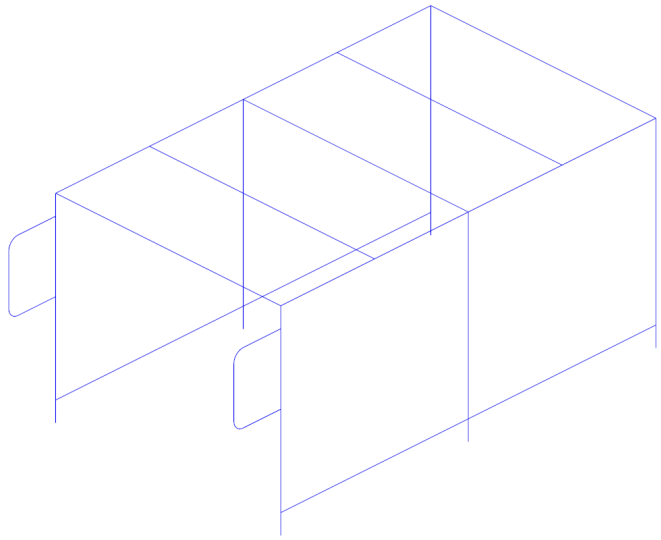
- Understand the design process of **AFX**
- Define a project
- Assemble straight profiles
- Assemble bent profiles
- Define joints between profiles
- Move profiles
- Replace and modify profiles

2.2 Design Process using AFX

AFX supports a top-down design process of framework assemblies. The following steps are recommended when designing a framework assembly with **AFX**:

- Create a datum curve skeleton for the main profiles of the assembly
- Assemble profiles on these datum curves or between points
- Define joints between profiles
- Assemble connector elements between profiles
- Assemble other equipment elements
- Create screw connections

Open the assembly `steel_frame.asm` in folder `steel_frame_start` which already contains a curve skeleton.



2.3 Define a Project

At the beginning of an **AFX** design a project needs to be defined, which means you have to enter a project shortcut (usually the drawing number of the framework assembly). This shortcut is used by **AFX** to automatically generate part names.

Hint 1 — Part Names. Without special configuration **AFX** composes automatically generated part names as follows:

`<PROJECT_SHORT>_<PROJECT_SHORT ELEMENT_NAME>_<ELEMENT_NUMBER>`

- `PROJECT_SHORT`: User defined assembly parameter
- `ELEMENT_NAME`: The element name depends on the element type (e.g. ENDPLATE)
- `ELEMENT_NUMBER`: The element number is a counter that increases by 1 every time you assemble a new element

NOTE:

The project shortcut is stored in an assembly parameter `PROJECT_SHORT`. If you copy and rename framework assemblies using Windchill you need to change parameter `PROJECT_SHORT` manually to the new drawing number.

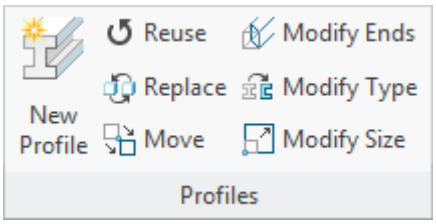
More information about configuring naming scheme can be found in **AFX** online help chapter Configuring **AFX** > To Define Component Naming Scheme **OR** in chapter 8.3.

Define a project for the assembly `steel_frame.asm`.

Select **Create new project** to create a new **AFX** project and enter **[sf]** as project shortcut.

2.4 Assemble Steel Profiles

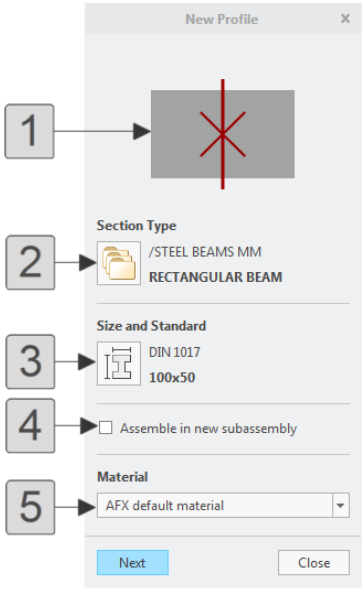
Profiles action is subdivided into seven instances:



- **New Profile** instance to create new parts.
- **Reuse**. An existing instance will be assembled again.
- **Replace**. An existing instance will be replaced.
- **Move**. An existing instance can be regulated by moving position relative to curve or rotation relative to selected orientation plane.
- **Modify Ends**. The profile end of an existing instance can be modified.
- **Modify Type**. The type of an existing instance can be changed.
- **Modify Size**. The size of an existing instance can be modified.

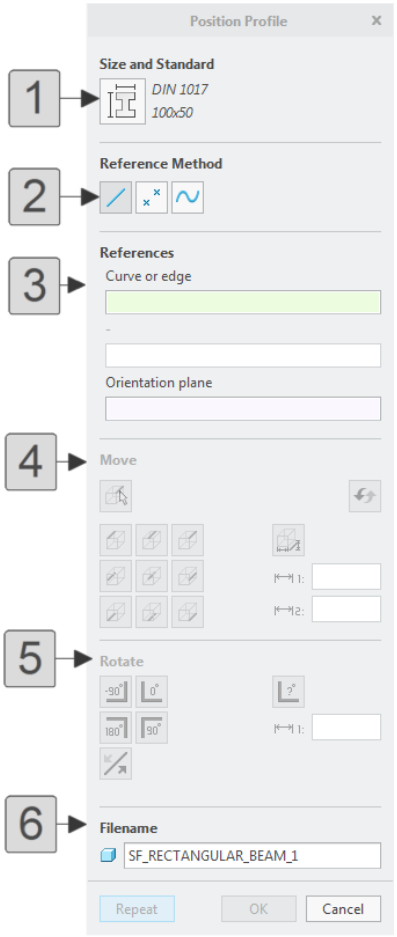
Assembling profiles with **AFX** is done with a dialog box which can be opened by selecting **New Profiles**.

The **New Profile** Dialog:



1. **Preview image**:
   - Shows image of selected profile
   - Red cross represents profile reference curve
   - Red line indicates the profile orientation plane
2. **Profile type**.
3. **Profile size**.
4. **Assemble in new subassembly**.
5. **Material**.

Click **Next** to open the **Position Profile** dialog box.



1. **Profile size**.
2. **Reference Method**:
   - On straight curve
   - Between two points or curve/edge end points
   - On bend datum curves
3. **References**.
4. **Position relative to curve**:

The assembly position is relative to the reference curve, which is left-top, center-top ... right-bottom.

The definition of left, right, top and bottom depends on the current view direction on the assembly.

5. **Rotation relative to selected orientation plane:**

With the rotation buttons you can set the rotation angles (0° , +90°, -90° or 180°) of the profile relative to the profile orientation plane.

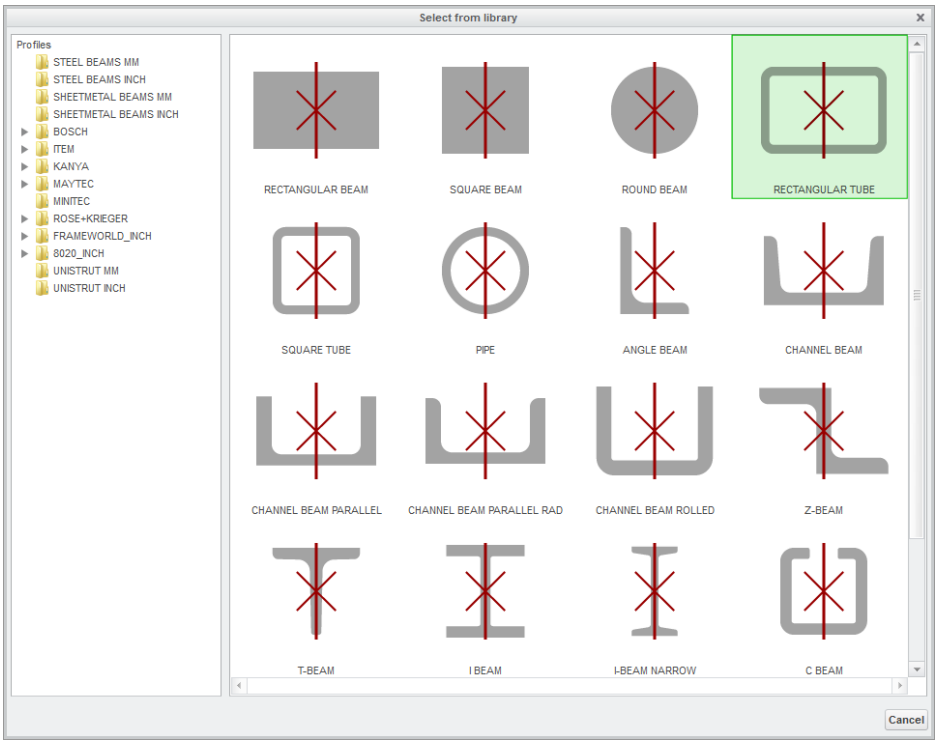
6. **Filename**.

Assemble a New Instance of a Profile on a Straight Datum Curve

To assemble a new instance of a profile (e.g. rectangular tube DIN 2395 120x80x4) on a datum curve of the assembly `steel_frame.asm` perform the following steps.

1. Click **New Profile** to open the New Profile Dialog box. The first time you open the dialog you will be forwarded to **Select from library** dialog automatically.
2. Click to open the **Select from library** dialog box. Here you can select the profile group (e.g. steel profiles in mm, Bosch, etc.) and the profile section type.
3. In this case select **[STEEL BEAMS MM]** and the **[RECTANGULAR TUBE]**

   As the rectangular tube is available in different standards and sizes the **Element definition** dialog box for selecting a standard and size opens.



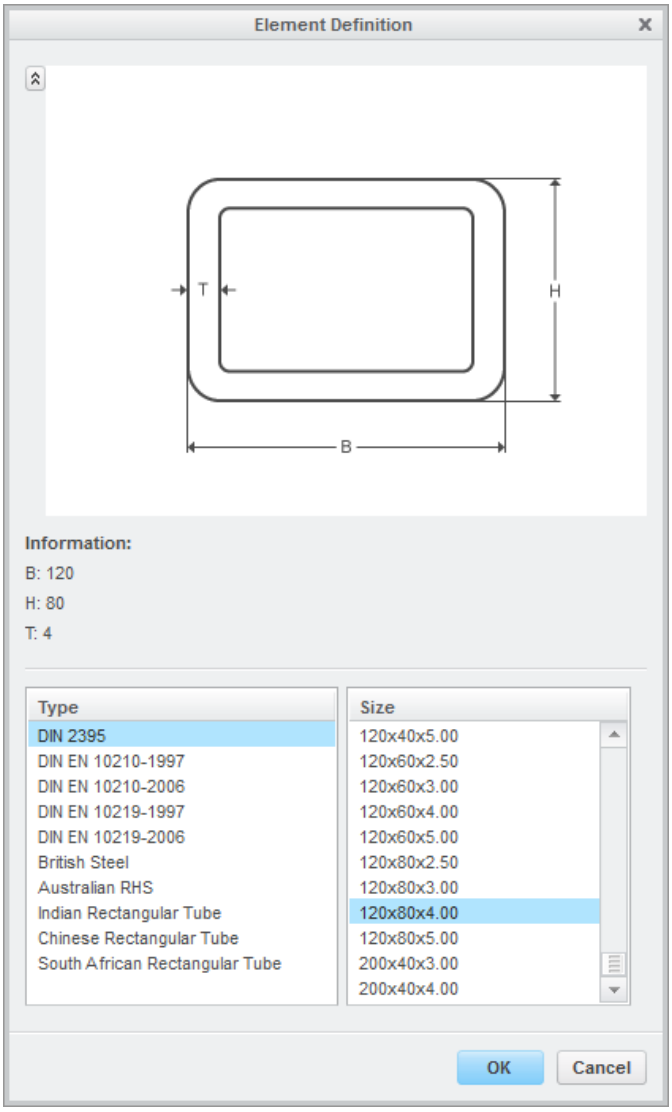
After selecting the **RECTANGULAR TUBE** the **Element definition** dialog box will automatically open.

You can also open this dialog within the **Profiles** dialog by clicking  .

In this example select:

[TYPE: DIN 2395]

[SIZE: 120x80x4.00



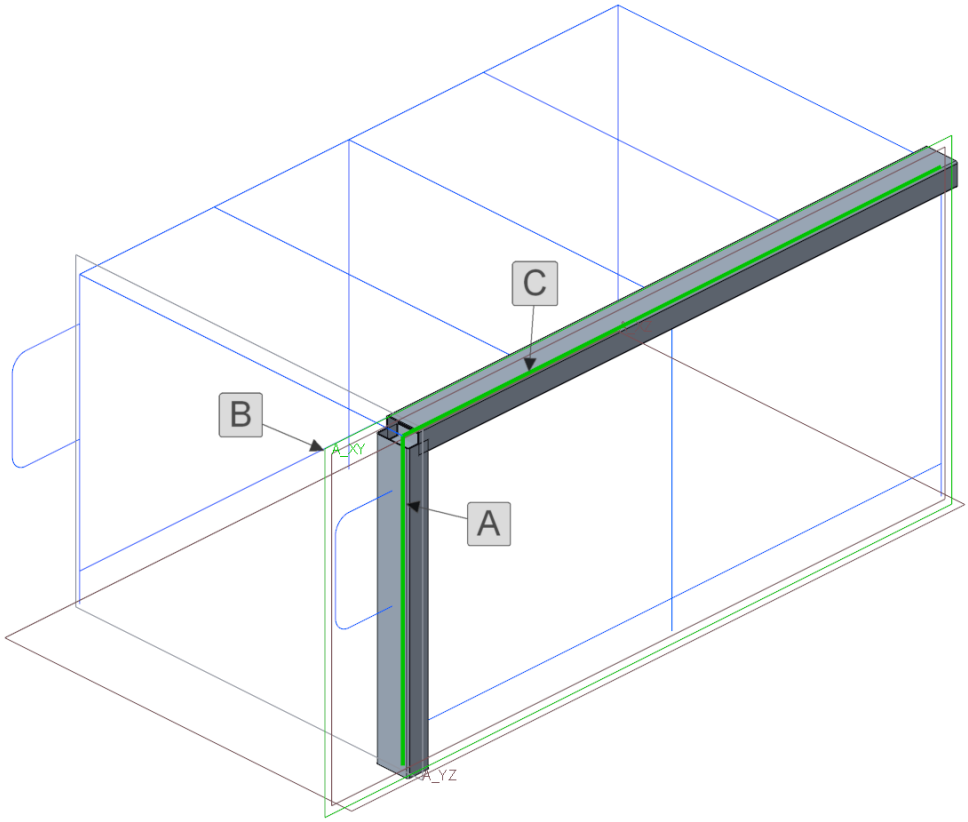
4. Click **[OK]** to close the **Element definition** dialog box.
5. Click **[Next]** to open the **Position Profile** dialog box.

6. Select the horizontal front curve **[A]** like shown in the picture.
   - A new profile part will be assembled on the curve as a preview with an automatically selected orientation plane.
7. To define the orientation of the profile around the reference curve you have two options:
   - To rotate the profile click one of the buttons 

or click ?° and enter the rotation value into the input panel. In this example click 90° so the profile will be oriented like shown in picture.

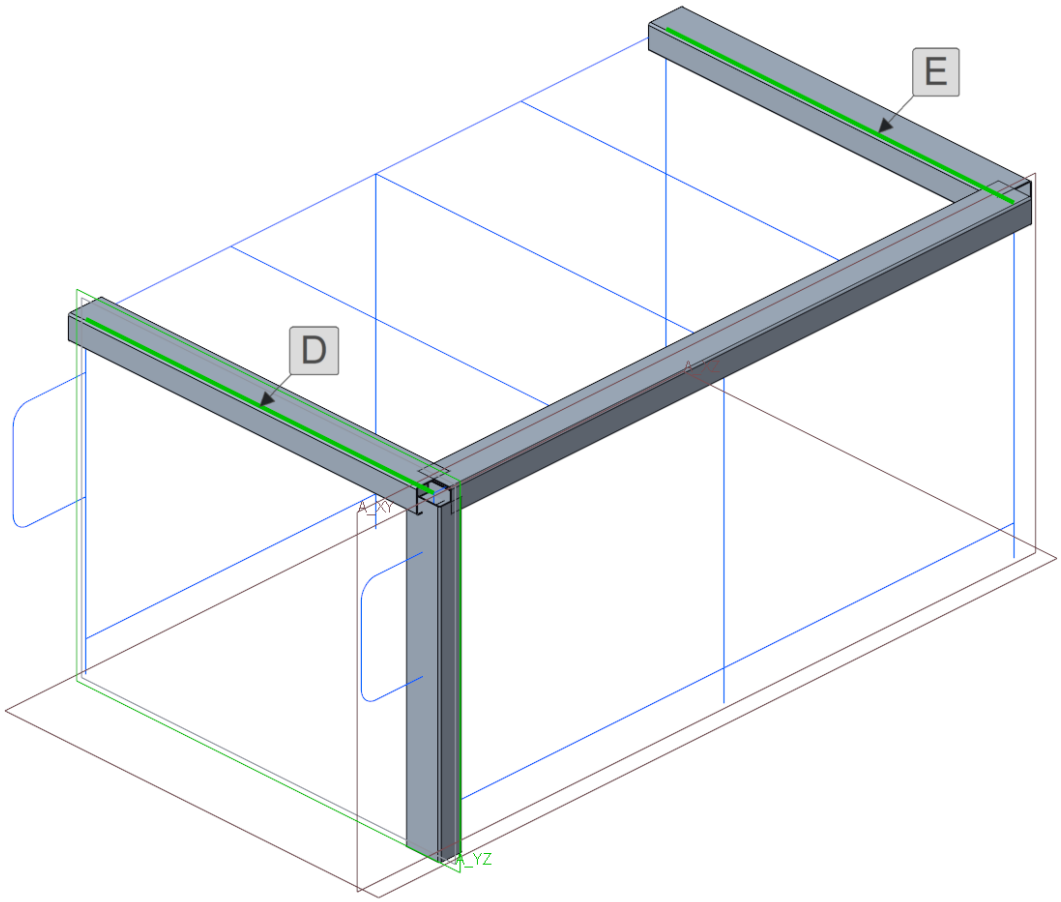
   or

   - Make sure that the **Orientation plane** reference collector is active and then select another planar surface or datum plane as orientation plane (parallel to the long red line in the section preview picture). Take care that the plane is parallel to the curve/edge. I. e. you could select the datum plane A_XY **[B]**.
8. Click **[Repeat]** to permanently assemble the profile and continue with placement of next profile..
9. Select the left vertical curve **[C]**.
   - Another profile with active section will be assembled.

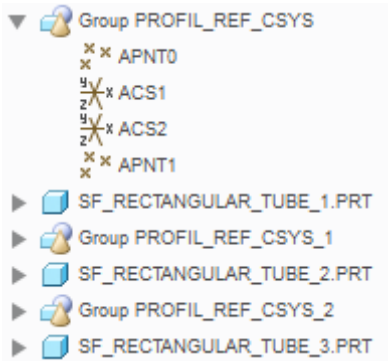


Take care that the active plane (selected last and highlighted in green) is parallel to the curve/edge you want to select. If it is not parallel, select a new plane first, then select the desired curve.

10. Click  to rotate profile by 90 degrees so it is oriented like shown in picture and then **[Repeat]**.
11. Now select the left upper curve **[D]** and then the datum plane A_YZ as new orientation plane **[E]**.
    - A new rectangular tube will be assembled.
12. Click  to rotate profile by 90 degrees so it is oriented like shown in picture and then **[Repeat]**.
13. Now select the upper right curve **[E]**.
    - Another rectangular tube will be assembled as new part.
14. Click  to rotate profile by 90 degrees so it is oriented like shown in picture and then **[OK]** to close the dialog.



In the model tree you see that every profile is a unique part and their names are made as follows.



- project shortcut – **SF**
- the profile type – **RECTANGULAR_TUBE**
- an incremental number – **1**

In front of every profile part is a group PROFIL_REF_CSYS. It contains a coordinate system on which the profile part is assembled. The coordinate system contains offset dimensions to the first coordinate system of the group by which the profile can be moved or rotated relatively to the curve.

The section size of every profile can be shown as 3D-note if you unhide the layer 00_PROFIL_SIZE_NOTE.

Assemble an Existing Instance of a Profile on a Straight Datum Curve

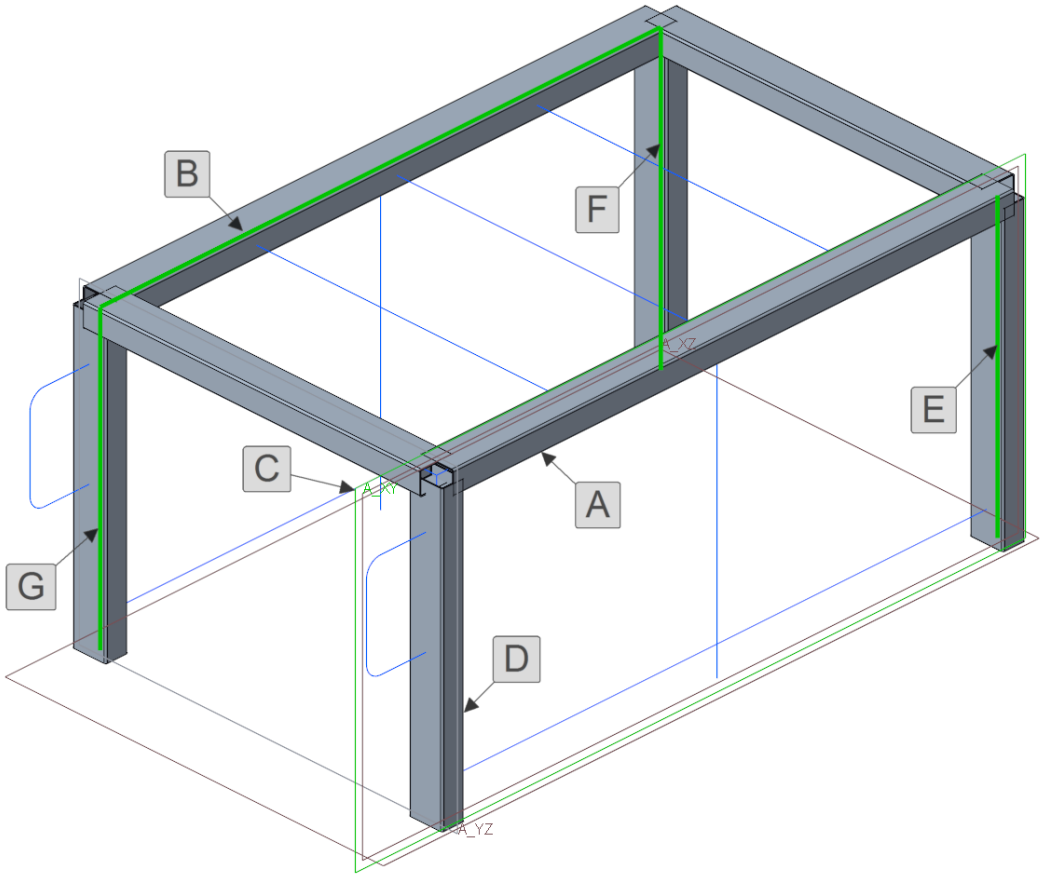
To reassemble an existing profile proceed as follows.

1. Click the **Reuse** button to open the reuse profile dialog box.
2. Select the existing profile part you want to assemble again. In this example select the front horizontal profile SF_RECTANGULAR_TUBE_1.PRT **[A]**.

   OR

1. Select the Profile SF_RECTANGULAR_TUBE_2.PRT.
2. Press the right mouse button and select **Framework** > **Reuse**
3. Click **[Next]** to open the **Position Profile** dialog box.
4. Select the back horizontal curve **[B]** as profile reference curve.
5. Click 90° to rotate profile by 90 degrees so it is oriented like shown in picture and then **[OK]**.

6. The reuse profile dialog box appears again. Make sure that the **Profile to Reuse** reference collector is active and select the left vertical profile SF_RECTANGULAR_TUBE_1.PRT **[D]** to assemble this profile on other curves.
7. Click **[Next]** and select the vertical outside curve **[E]**, rotate profile and click **[Repeat]** or middle mouse button to complete profile placement. Repeat the procedure with outside curves **[F]** and **[G]**.
   - The same **Creo Parametric** part will be assembled on the three curves.



Hint 2—Right mouse button. You might have realized, that many of the **AFX** functions have found their way into the right mouse button menu.

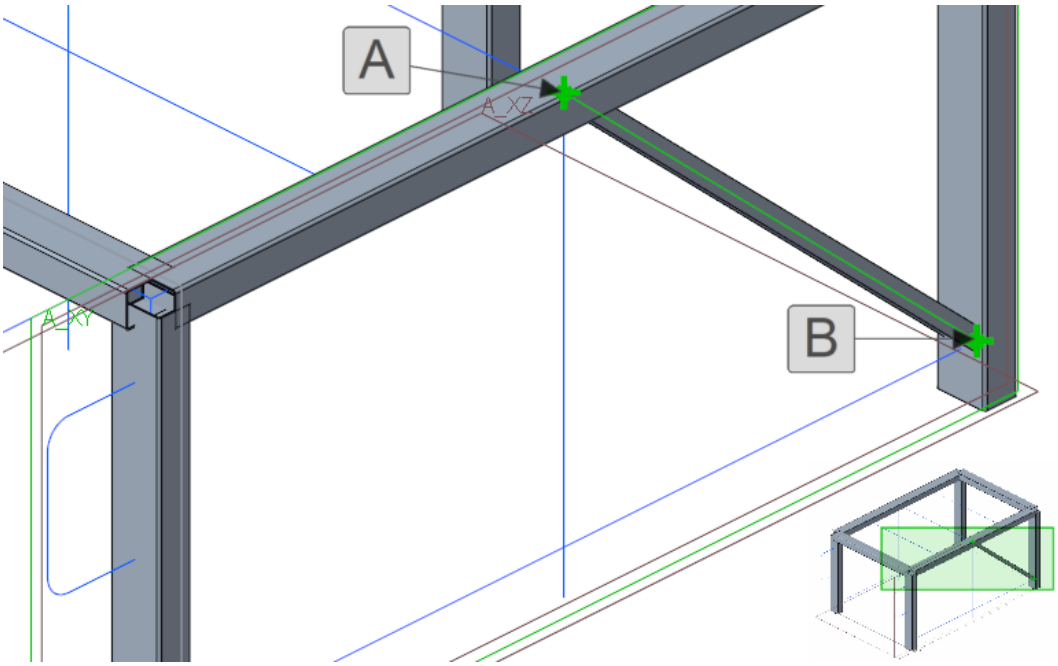
Assemble Profiles between Two Points

Straight profiles can also be assembled between two datum points or curve/edge end points. It is possible to create these points **on the fly** with **Creo Parametric** when **AFX** prompts you to select them.

In this example a **rectangular beam DIN 1017 80x40** shall be assembled between two curve end points.

1. Click **New Profile** to open the **New Profile** dialog box.
2. Click and select **[STEEL BEAMS MM] > [RECTANGULAR BEAM]**.

3. In the **Element Definition** dialog box select:
   - **[TYPE: DIN 1017]**
   - **[SIZE: 80x40]**
4. Click **[Next]**. The **Position Profile** dialog box opens.
5. Click  to activate the **between two points** reference method.
6. Select the two curve end points **[A]** and **[B]** as shown in next picture.
   - The rectangular profile will be assembled.
7. Click **[OK]** to close the dialog box.



Assemble Bent Profiles on Curves

With **AFX** you can assemble profiles on planar bend datum curves. It is possible to assemble profiles on a **from-to curve chain** or a **complete curve chain**. There are some limitations:

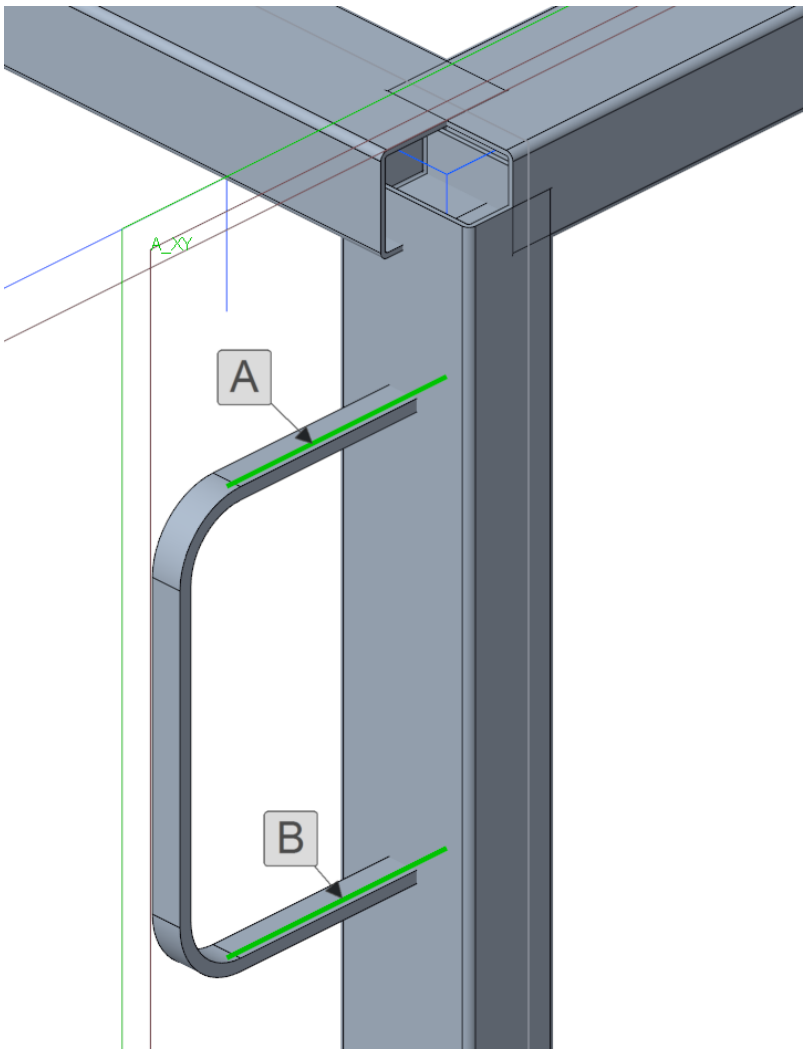
- All segments of the curve must be in one plane.
- The bent profile must be open.
- The start and end point may not align.
- For bent profiles only the following Instance types are supported
  -  Assemble new profile instance
  -  Reuse profile as same instance

In this example a **RECTANGULAR BEAM DIN 1017 25x10** shall be assembled on the left front curve chain.

1. Click **New Profile** and make sure that the **RECTANGULAR BEAM** in the **STEEL BEAMS MM** folder is still active.
2. Click to open the **Element definition** dialog box and select:

   [TYPE: DIN 1017]

   [SIZE: 25x10]
3. Click **[Next]**. The **Position Profile** dialog box opens.
4. Click to activate the Reference method **Assemble profile on a bent curve**

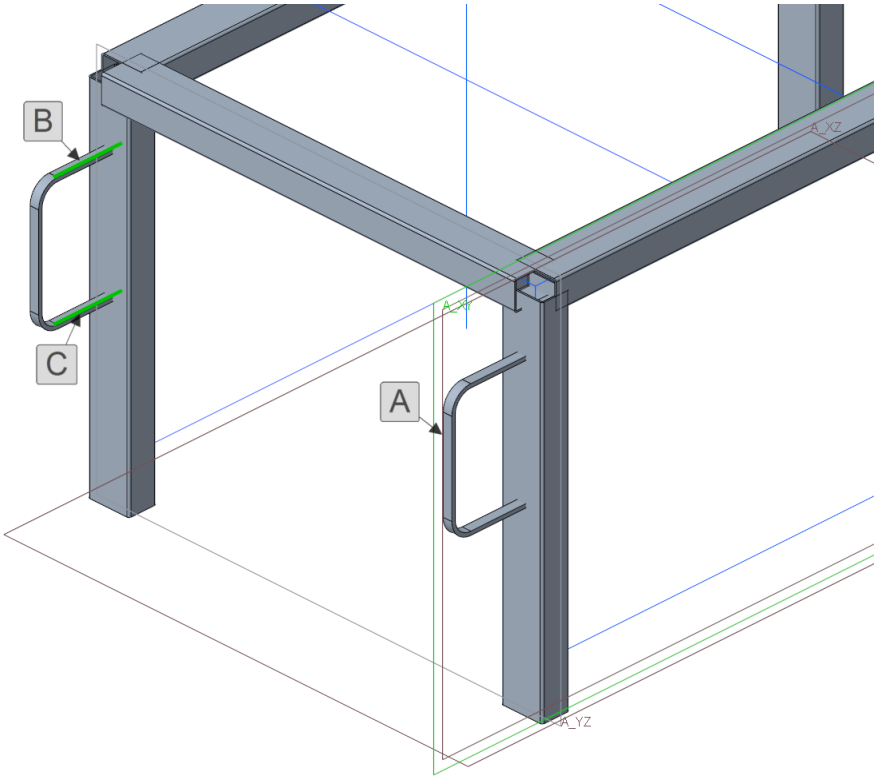


5. Select the start curve segment for the profile on the curve chain **[A]**.
6. Select the end curve segment for the profile on the curve chain **[B]**.
   - The bent profile will be assembled between start and end curve segment on the chain.
   - The orientation plane will be selected automatically

7. Click **[OK]** to close the dialog box.

Reassemble this profile as identical part on the left bend curve chain.

8. Click ↺ to reassemble an existing profile instance.
9. Select the profile assembled previously SF_RECTANGULAR_BEAM_5.PRT **[A]**.
10. Select the upper curve segment of the left bent curve as start curve segment **[B]** and the lower curve as end curve segment **[C]** for assembling the profile again.
    - The bent profile will be assembled as identical part.

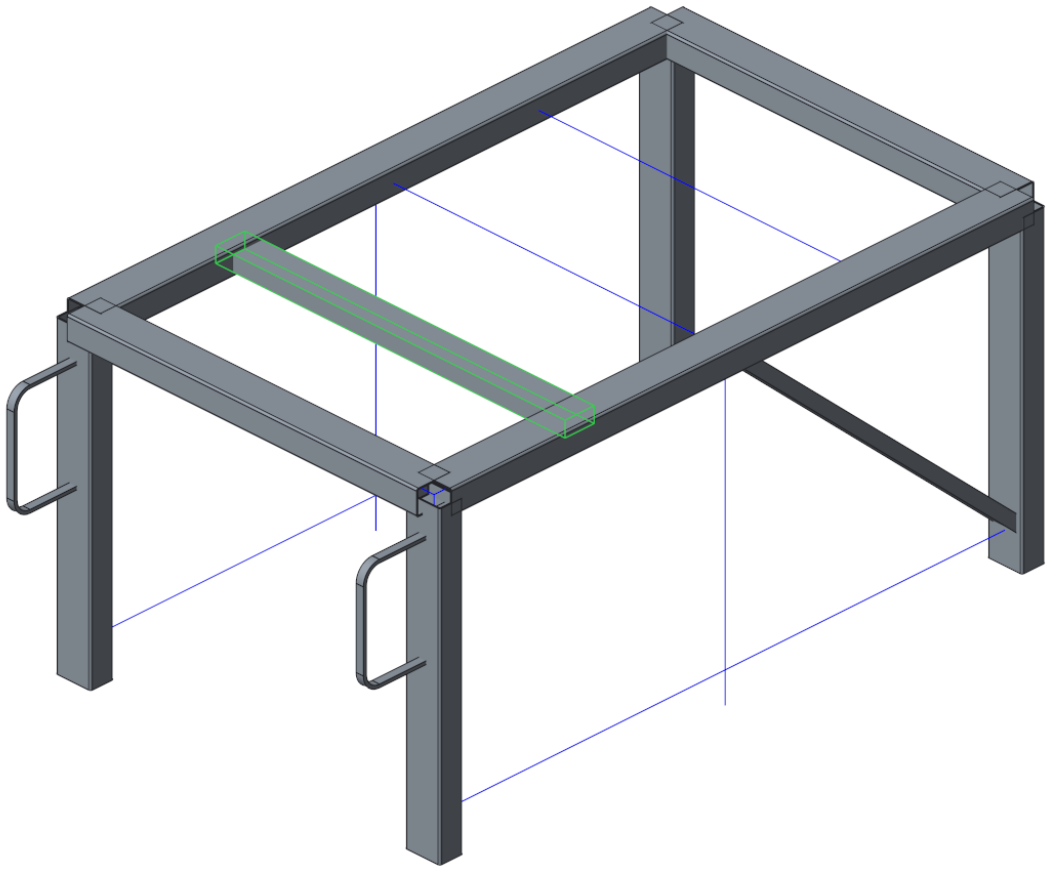


Pattern Profiles

Profiles assembled with **AFX** on patterned reference curves can be reference patterned with standard **Creo Parametric** functionality. However it is necessary to use a special technique. Proceed as follows.

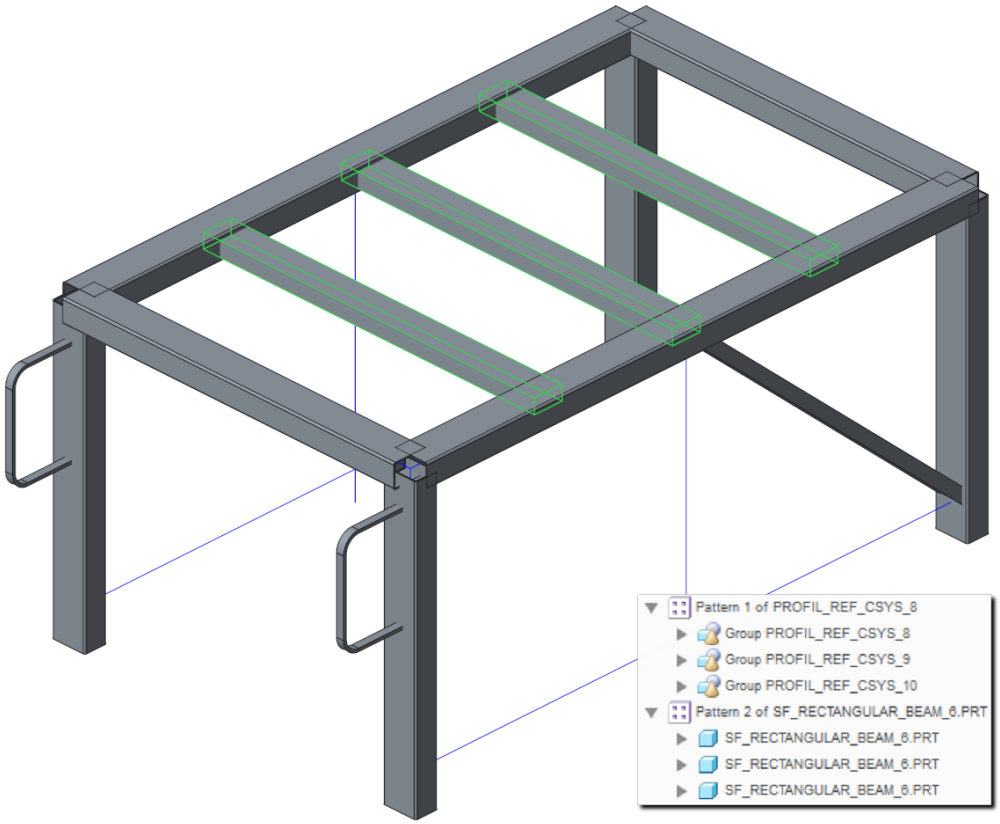
First assemble a **RECTANGULAR BEAM DIN 1017 100x50** like shown in the picture using the functions described in 2.4.1. The datum curve on which the profile is assembled is patterned.

In **Creo Parametric** model tree make the features visible. You see the group PROFILE_REF_CSYS in front of each profile. These groups contain some datum points and coordinate systems. The profile is assembled on the second coordinate system.



As the profile references on the 2nd coordinate system of the group, creating a reference pattern of the profile has to be done in two steps. First you have to create a reference pattern of the PROFILE_REF_CSYS group and then you can create a reference pattern of the profile itself.

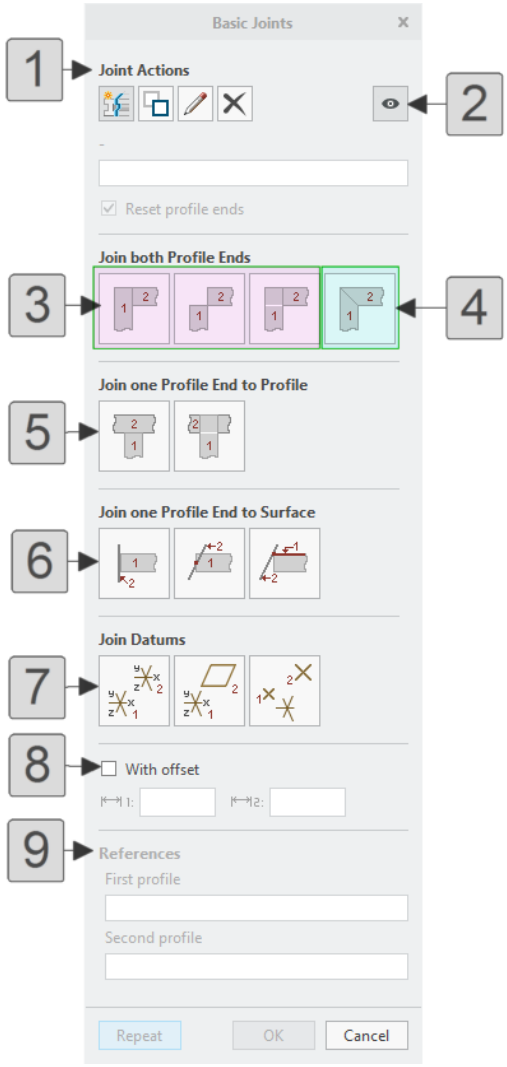
1. Select the group PROFILE_REF_CSYS_8 in front of the profile in model tree with right mouse button and select Pattern.
2. Select pattern type **[Reference]** and click .
   - A reference pattern of the group is created. Now the profile references on a patterned element (the coordinate system of the patterned group) and it is possible to create a reference pattern of the profile.
3. Select the profile part SF_RECTANGULAR_BEAM_6.PRT in the model tree with the right mouse button and select **[Pattern]**.
4. Select pattern type **[Reference]** and click .
   - A reference pattern of the profile is created.



2.5 Define Joints between Profiles

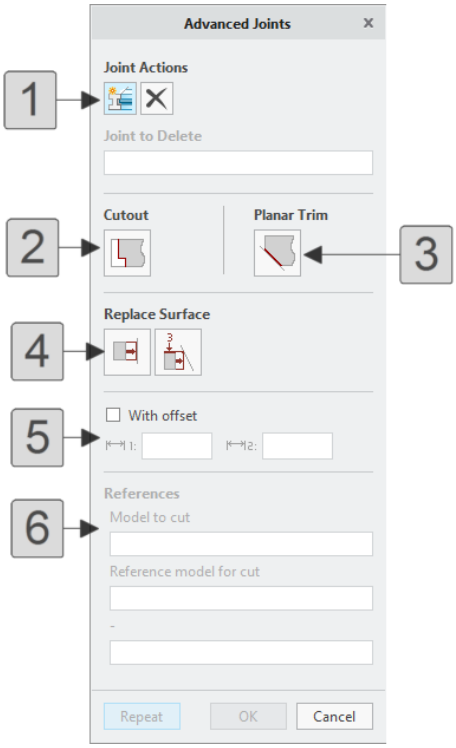
When assembling profiles with **AFX** they are assembled with the length of the reference curve and cut perpendicular to the reference curve. To design the connection points between profiles there are several functions for defining joints. You can define corner joints, mitre joints, T-joints, etc. You also can cut profiles using the geometry of other profiles, create additional planar cuts on profiles and create replace surface joints to join surfaces of non-profile parts to other planar surfaces. The definition of joints with **AFX** is done with two dialog boxes: **Basic Joints** and **Advanced Joints**.

To open **Basic Joint**s press **Basic Joints** in the framework ribbon.



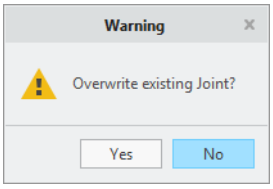
1. **Joint Actions**
2. **Blank/unblank joint symbols**
3. **Corner Joints**
   - both profile ends are modified
4. **Mitre joints**
   - both profile ends are modified
5. **T-joints**
   - Only the 1st profile end is modified
6. **To selected surface**
   - Only the 1st profile end is modified
   - The 2nd part doesn't have to be a profile
7. **Join Datums**.
8. **With or without offset**.
9. **References**.

To open **Advanced Joint**s press **Advanced Joints** in the framework ribbon.



1. **Joint Actions**.
2. **Cutout**.
   - Cut profile or other part with the shape of a profile
   - Uses external references
3. **Planar Trim**.
   - Cut profile or other part with a planar surface
   - Uses external references
4. **Replace Surface**.
   - Cut/extend profiles
   - Uses external references
5. **Input panels for offsets**.
6. **References**.

Hint 3 — Overwrite Joints. If you want to **modify** or **redefine** a regular joint (defined by one of the 12 buttons of basic joints) just create a new joint and accept the query whether you want to overwrite the old joint.



If you want to modify a **profile cutout** or **planar trim** delete it and create it again.

The important advantage of regular joints compared to a profile cutout or planar trim is that **AFX** automatically locates the proper attachment surface in most cases and that external references are avoided. **AFX** just modifies dimensions in the coordinate systems CS_S and CS_E within the profiles.

Always use regular joints as long as profile ends are cut planar.

Create Corner Joints

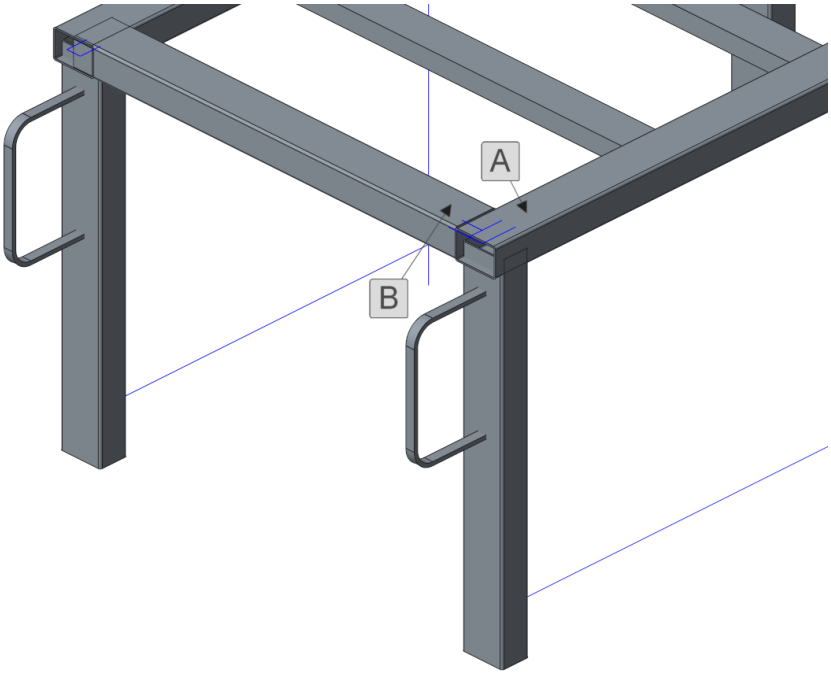
By defining a corner joint with **AFX** you can design a corner so one profile end overlaps and the second attaches or both profile ends overlap or the corner area itself remains free.

1. Click 

to define a corner joint in **Basic Joints** dialog box.
2. Make sure that **First profile** reference collector is active and select the first overlapping profile near its end **[A]**.
3. Make sure that **Second profile** reference collector is active and select the second attaching profile near its end **[B]**.

   The joint is created as a preview.
4. Click **[Repeat]** to apply and reuse the current joint type.

- The joint is created permanently and a joint symbol is displayed which looks like the joint.



Now create another **corner joint** at the rear left corner. As the rear horizontal profile is the same part than the front horizontal profile you see its profile end already is modified. You also can see this at the joint symbol which looks like half of a joint symbol which is already

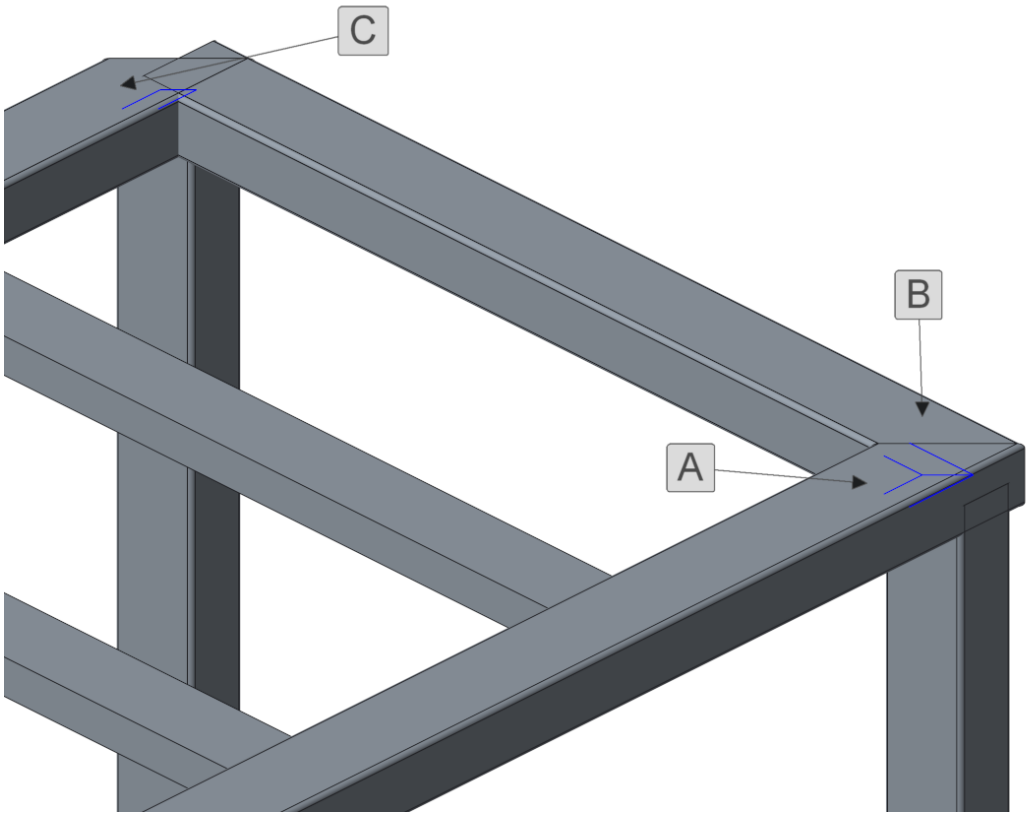
controlling this profile end. Anyhow you can create the joint, but be aware that both joint definitions should result in same shape of the profile end (i.e. do not make a corner joint at one place and a mitre joint at the other place).

Create Mitre Joints

The creation of mitre joints is done similar to corner joints.

1. Click 

in the **Basic Joints** dialog box.
2. Select the first profile near its end **[A]**.
3. Select the second profile near its end **[B]**.
4. Click **[OK]**
   - The mitre joint is created and a joint symbol is displayed which looks like the joint.

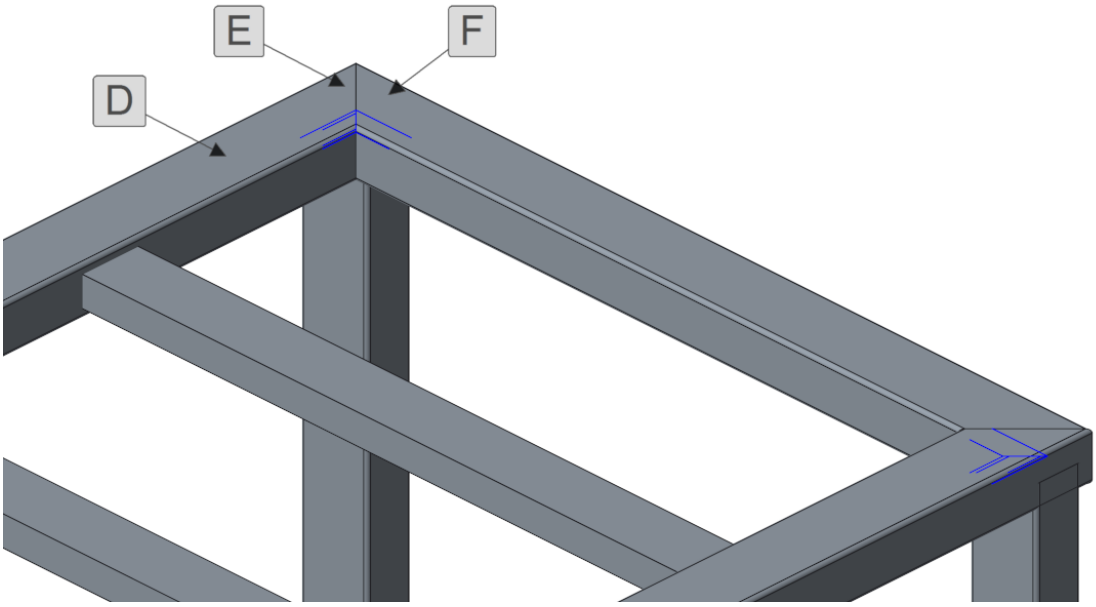


As the rear horizontal profile is the same part than the front horizontal one, the mitre cut also appears on the rear profile **[C]**. Unfortunately the cut has same orientation as on the front profile, which means on this place it has wrong orientation. The profile needs to be rotated by 180 degrees. Therefore **AFX** has functions to move or rotate profiles (see also chapter 2.6).

To rotate the rear profile and then create the mitre joint proceed as follows.

5. Press Move in the **Profiles** group.

6. Click  to rotate profiles by 180 degrees.
7. Select the rear horizontal profile **[D]**.
8. Click **[OK]** in selection dialog. The profile is rotated by 180 degrees.
9. Now open **Basic Joints** dialog box again.
10. Click  in **Basic Joints** dialog box.
11. Select the first profile end **[E]**.
12. Select the second profile end **[F]**.
13. Click [**OK**]
    - The mitre joint is created.



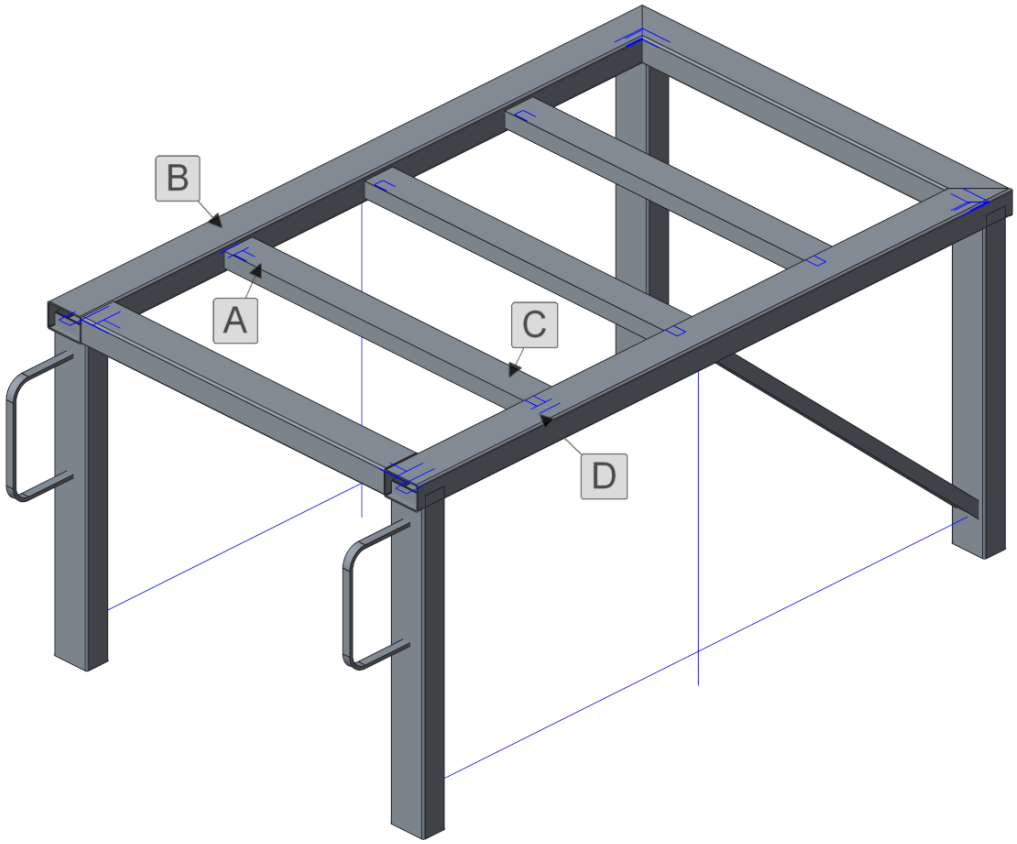
Create T-Joints

While corner joints and mitre joints modify both profile ends, a joint from type T-joint only modifies one profile end. You can use them if

- a profile end needs to be attached to a profile side surface
- more that two profiles meet at one point and between two of them a corner or mitre joint is already defined.

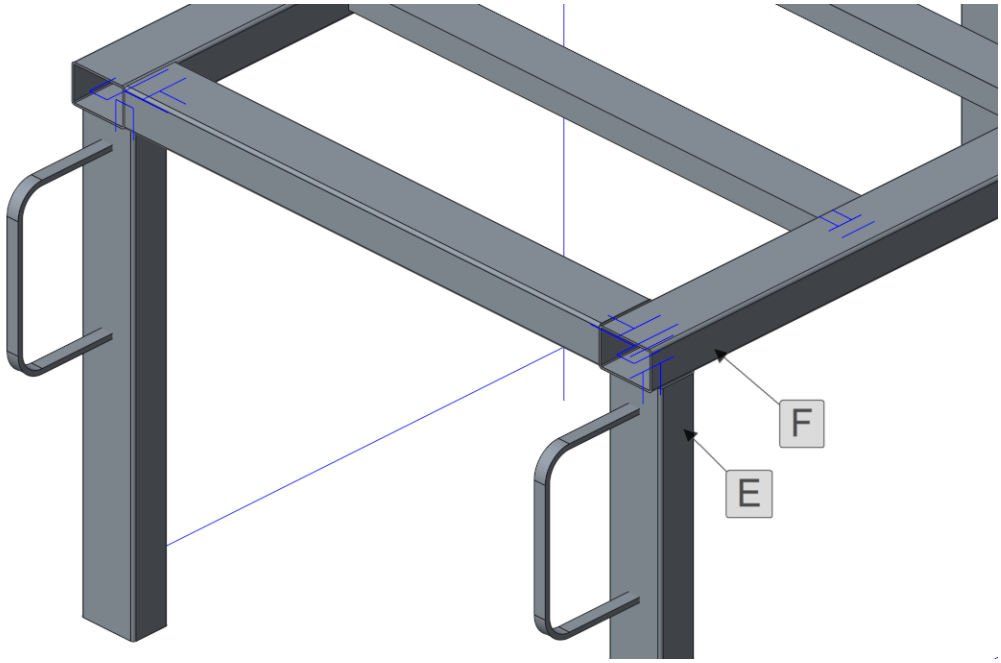
1. Click  in **Joints** dialog box.
2. Select the rear profile end of the rectangular beam **[A]**.

3. Select the rear horizontal rectangular tube **[B]**.
   - The T-joint is created as a preview.
4. Click **[Repeat]** to apply and reuse the current joint type.
5. Select the front profile end of the rectangular beam **[C]** and the front rectangular tube **[D]** to attach the other side as well.
   - The T-joint is created as a preview. As the profile is patterned the pattern instances are modified too

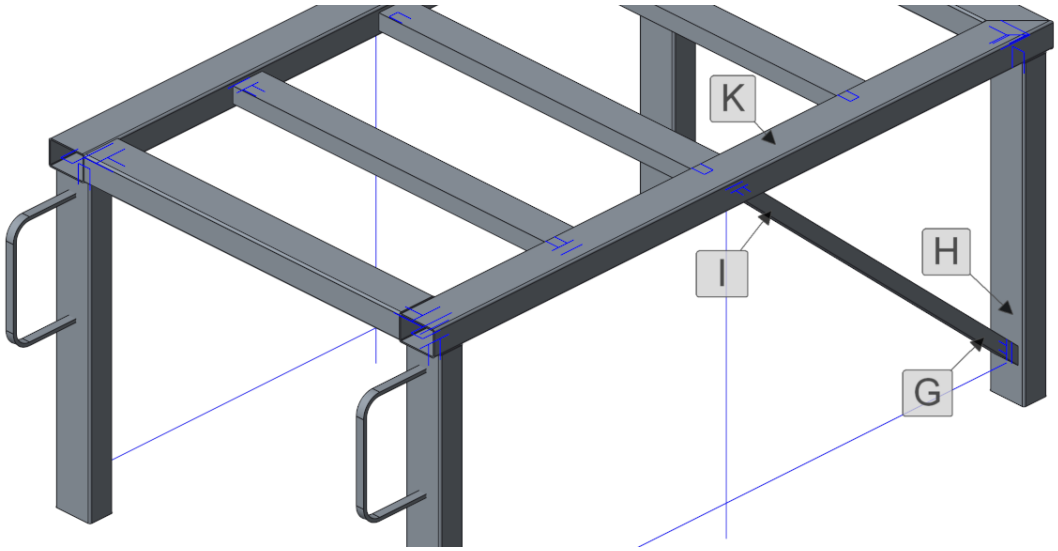


6. Click **[Repeat]** to apply and reuse the current joint type.
7. Select the front vertical profile **[E]** and the front horizontal profile **[F]** to fit the vertical profile as third profile into the existing corner.

- As this profile is assembled four times the other three instances are shortened, too.

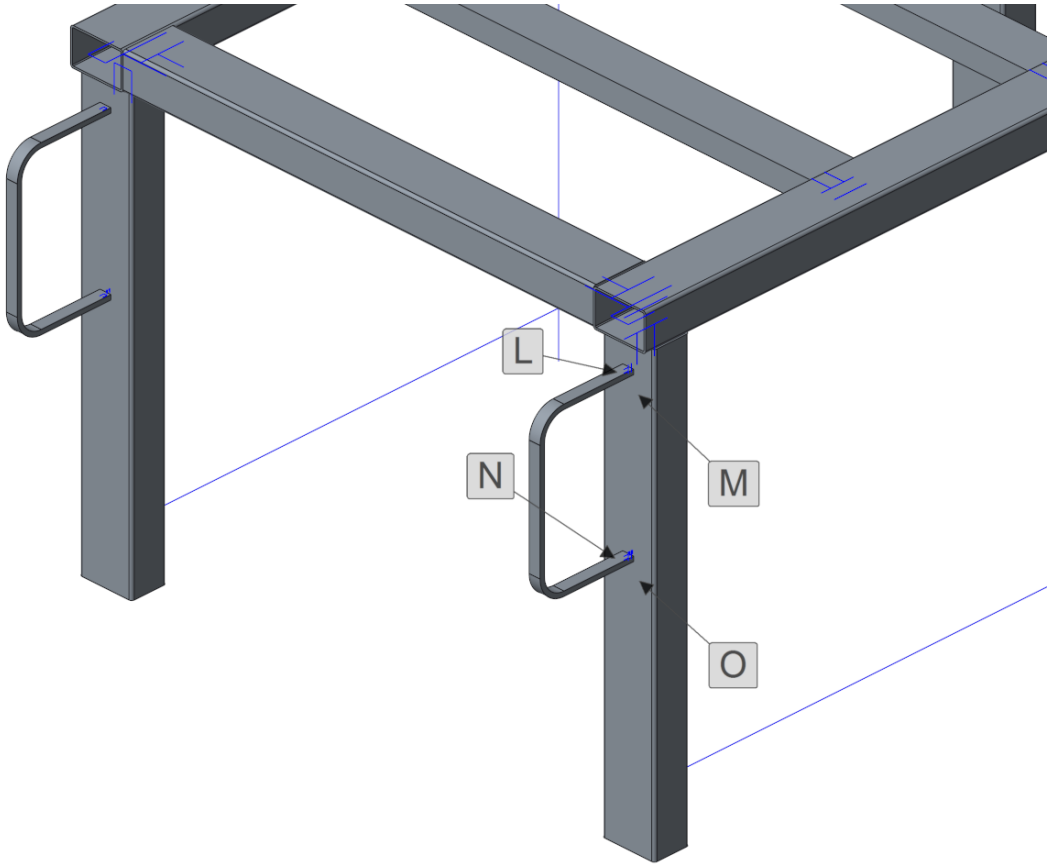


8. Click **[Repeat]** to apply and reuse the current joint type.
9. Select the lower end of the diagonal profile **[G]** and the right vertical profile **[H]** to attach the diagonal profile on its lower end and click **[Repeat]**.
10. Select the upper end of the bent profile **[I]** and the front horizontal profile **[K]** to attach the bent profile on its upper end and click **[Repeat]**.



11. Select the upper end of the bent profile **[L]** and the front vertical profile **[M]** to attach the bent profile on its upper end and click **[Repeat]**.
12. Repeat this step with the lower end of the bent profile **[N]** **[O]**.

- As the bent profile is assembled as existing instance at the left rear side, the profile ends are modified on this instance as well.



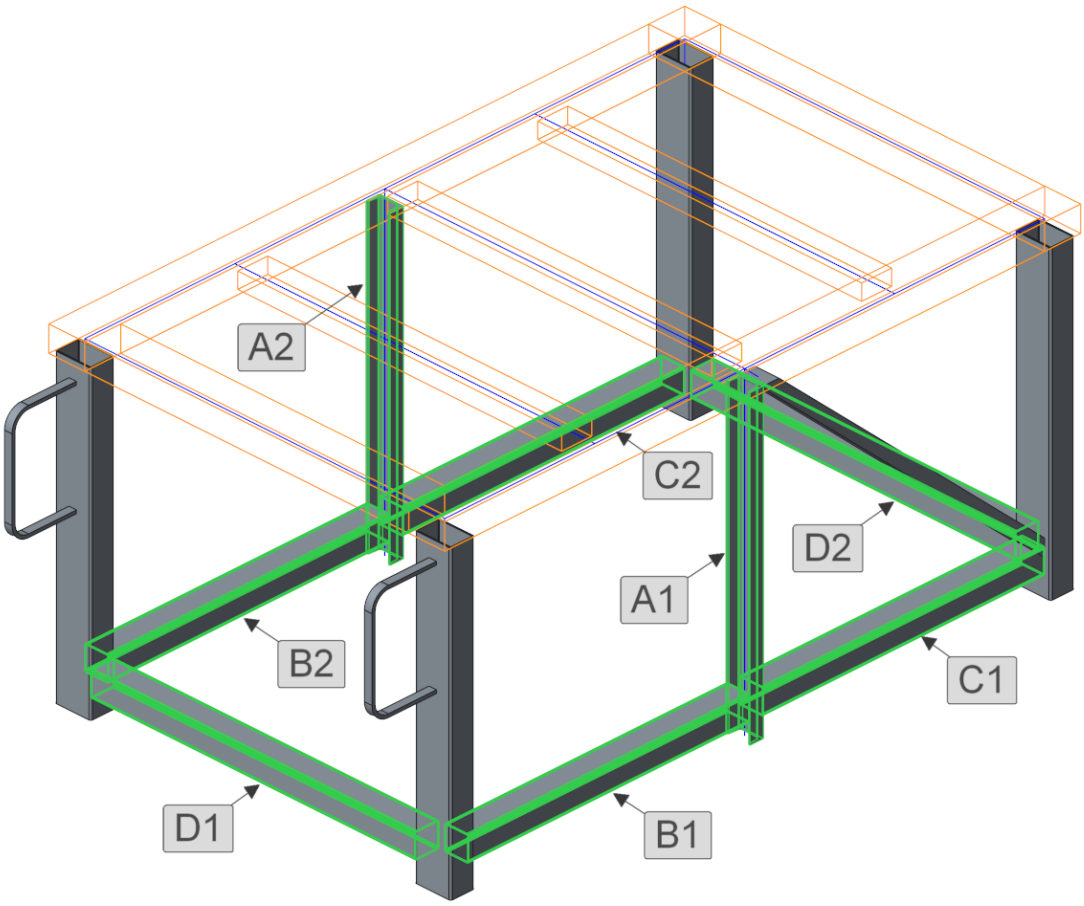
Hint 4 — Joints at reuse places. If you have assembled profiles as **existing instance** multiple times try to define the joints at the first assembly place. Define joints at reuse places only if necessary.

Exercise

Complete the frame with **SQUARE TUBES DIN2395 80x80x4.0** and **U-BEAMS DIN1026 U80** like shown in the next picture. Create the necessary joints. The upper horizontal profiles are hidden in the picture.

1. Assemble one **[CHANNEL BEAM DIN 1026 U80]** as new profile instance on the vertical middle front curve **[A1]**, and then this profile as **reused instance** on the vertical middle rear curve **[A2]**.
2. Assemble one **[SQUARE TUBE DIN 2395 80x80x4]** as new profile on the lower left front curve **[B1]** and then this profile as **reused instance** on lower left rear curve **[B2]**.
3. Assemble one **[SQUARE TUBE DIN 2395 80x80x4]** as new profile on the lower right front curve **[C1]** and then this profile as **reused instance** on lower right rear curve **[C2]**.

4. Assemble one **[SQUARE TUBE DIN 2395 80x80x4]** as new profile between the two left end points of the lower horizontal curves **[D1]** and then this profile as **reused instance** between the two right end points of the lower horizontal curves **[D2]**
   - **HINT:** Switch to wire frame display and select by query which makes it easier to select the curve end points).
5. Create the necessary T-Joints at the first assembly positions of each profile.



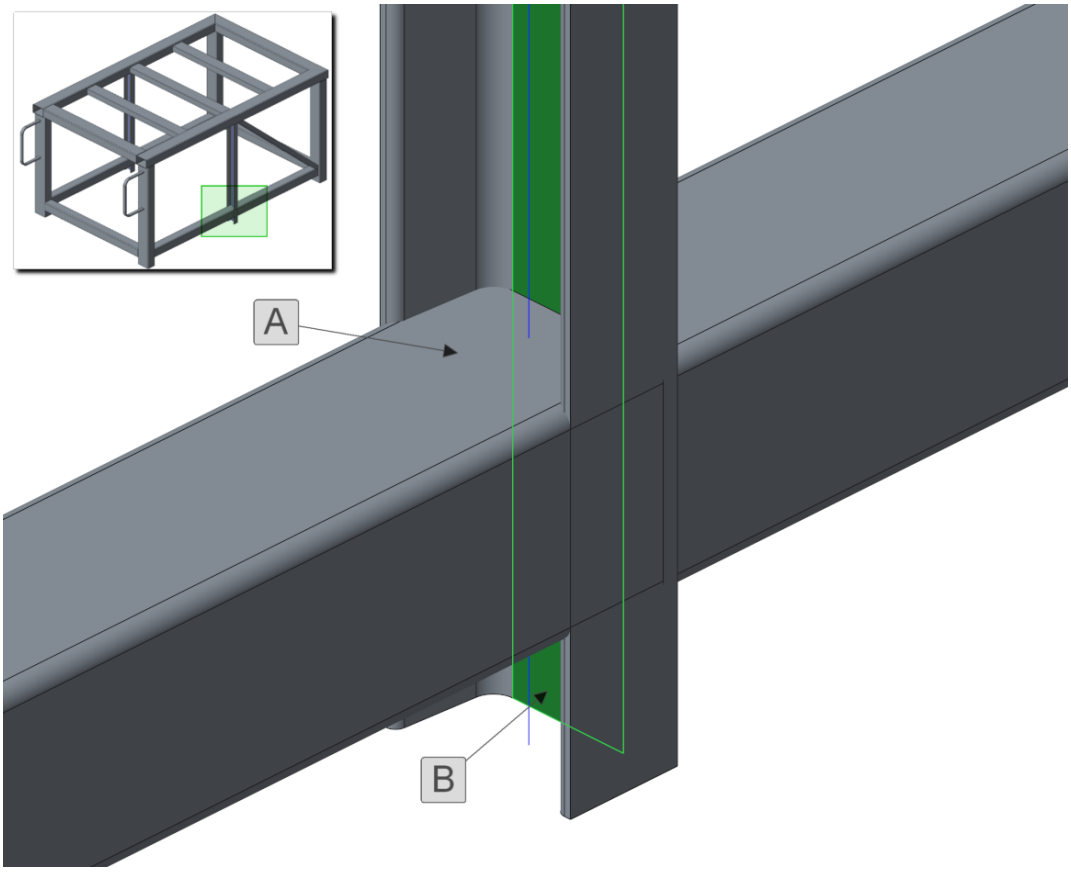
Creating Joints to Selected Surfaces

Joints from type **corner joint**, **mitre joint** or **T-joint** usually find the proper attachment surfaces automatically. If this is not the case or you want to create a joint to a surface of a non profile part you can use the joint type **selected surface**. This joint allows you to select the desired attachment surface for the profile end. Zoom to the place where the lower left square tube and the U-beam meet. You see that the square tube does not extend to the inner surface of the U-shape, but just to the bounding box of the U-beam.

Now extend the square tube to the inner surface of the U-beam.

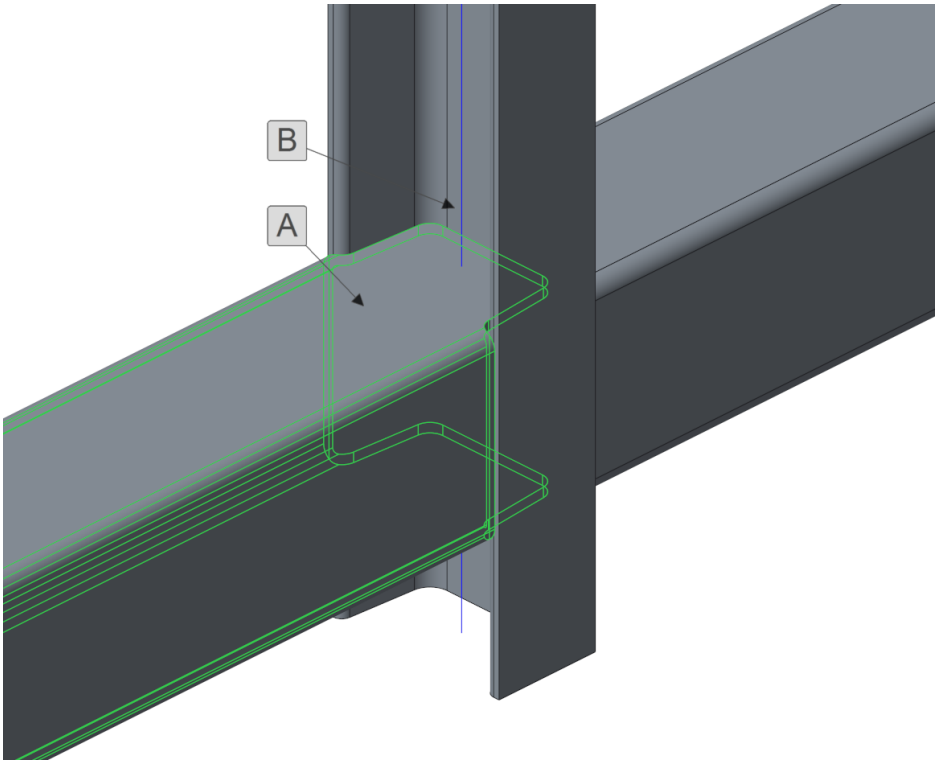
1. Click  in **Basic Joints** dialog box.
2. Select the lower left square tube near its right end **[A]**.
3. Accept the prompt **overwrite the existing joint**.

4. Select the inner surface of the U-beam **[B]**.
   - The square tube is attached to the selected surface but not cut with the U-shape.
5. Click **[OK]** to apply and close dialog box.



Creating Profile Cutouts

1. Check **[With offset]** in **Advanced Joints** dialog box for a profile cutout with offset.
2. Enter the offset value in the input field right to 1: **[2.0]**.
3. Click  for a profile cutout with offset.
4. Select the square tube to cut out **[A]**.
5. Select the reference profile (U-beam) for cutout **[B]**.
   - The profile cutout is created.
6. Click **[OK]** to apply and close dialog box.

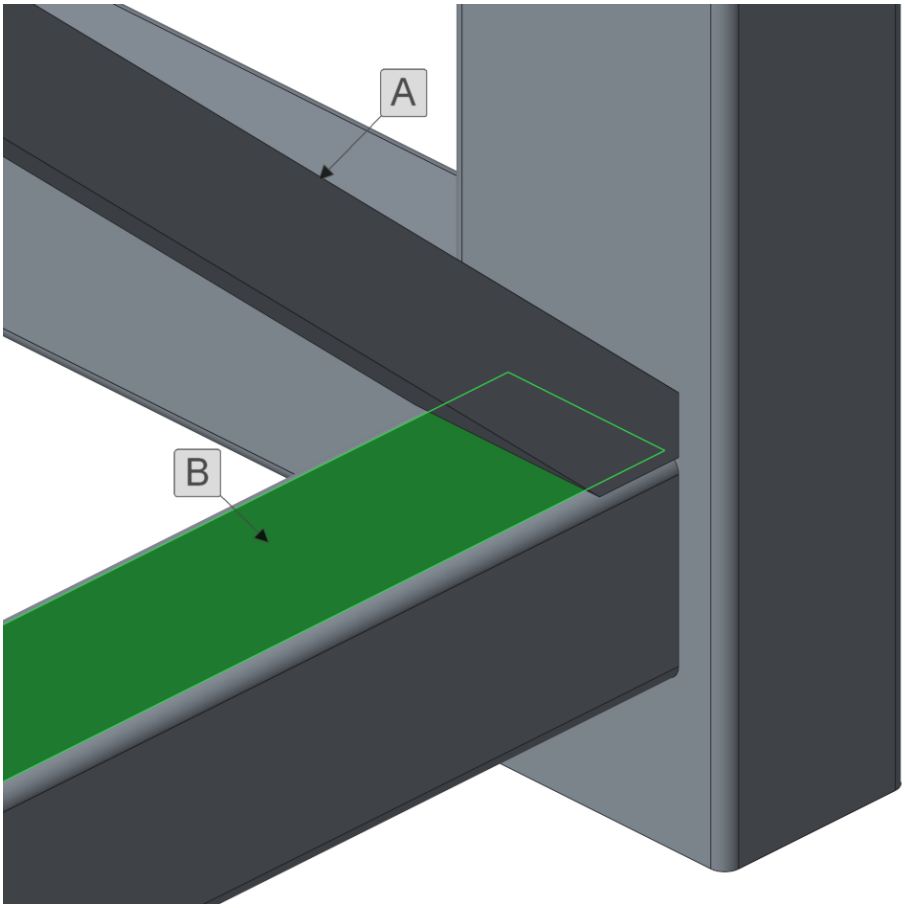


Hint 5 — Joint references. Unlike regular joints (corner joint, mitre joint etc.) a profile cutout creates an external reference to the other profile.

Create a Planar Trim

In some cases (e.g. the lower right end of the diagonal profile) a profile is cut with two or more planar cuts. In this case first create a regular joint (T-joint, Selected Surface joint) then create a **Planar Trim** for the second cut.

1. Click  in **Advanced Joints** dialog box for a planar trim.
2. Select the rectangular beam to cut **[A]** at a position which shall be kept after cutting.
3. Select the plane which shall be used for cutting **[B]**.
   - The profile will be cut at this plane.
4. Click **[OK]** to apply and close dialog box.

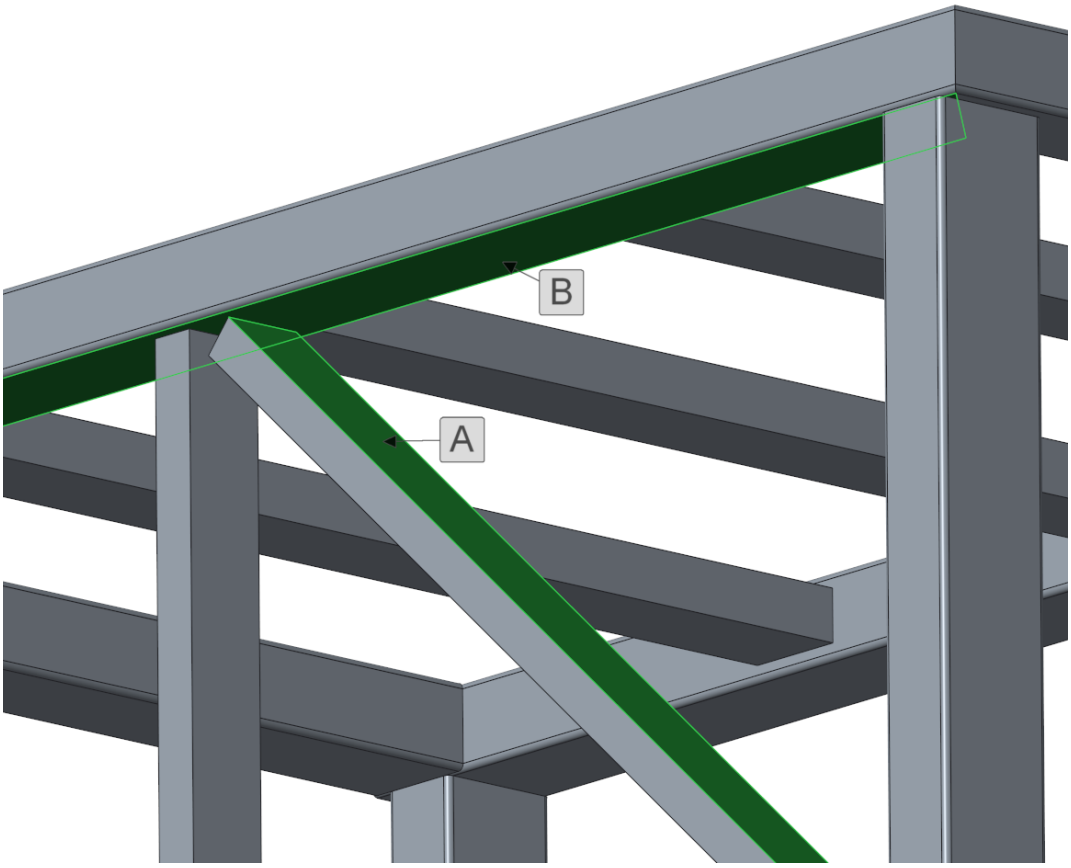


Hint 6 — Delete special Joints. Unlike regular joints (corner joint, mitre joint etc.) a planar trim creates an external reference to the other profile. A profile cutout can be deleted with ✕. Be sure to select the surface that has been modified by the joint.

Create Joints to Selected Surfaces with Normal Cut

If a profile end shall be attached to a surface at an angle, in some cases the profile end surface is not attached directly to this surface, but is cut normal to profile length axis and only the profile end surfaces edge touches the attachment surface.

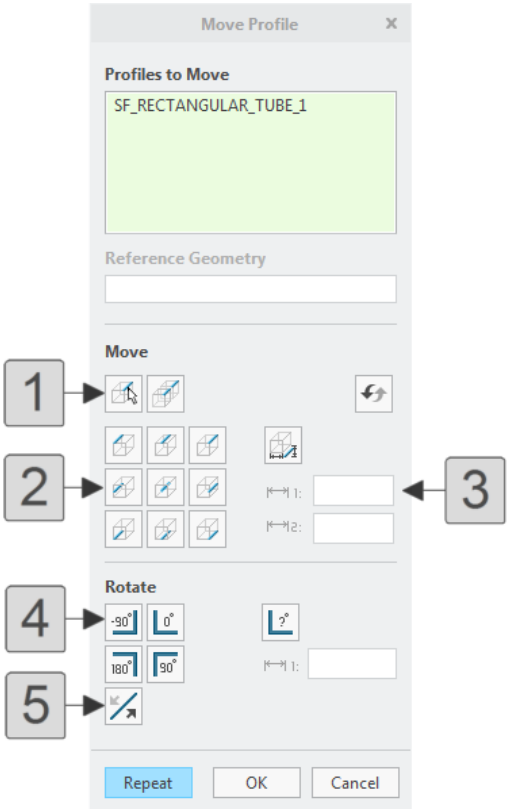
1. Click  in the **Basic Joints** dialog box, to create a joint of type **selected surface with normal cut** between the upper left end of the diagonal profile and the upper horizontal rectangular tube.
2. Select the diagonal profile at the upper side surface near the upper left end **[A]**. Accept the prompt **overwriting the existing joint**.
3. Select the upper horizontal rectangular beam at its bottom surface **[B]**.
   - The diagonal profile is cut normal to its length axis along the intersection line between the two selected surfaces.
4. Click **[OK]** to apply and close dialog box.



For further exercises in this training guide overwrite the just defined joint by a regular T-joint as described in chapter 2.5.3.

2.6 Move Profiles

Usually profiles are assembled with their center aligned to the reference curve and oriented to the orientation plane. Later on they can be moved or rotated. This can be done in the **Profiles** dialog in the tab **Move**.



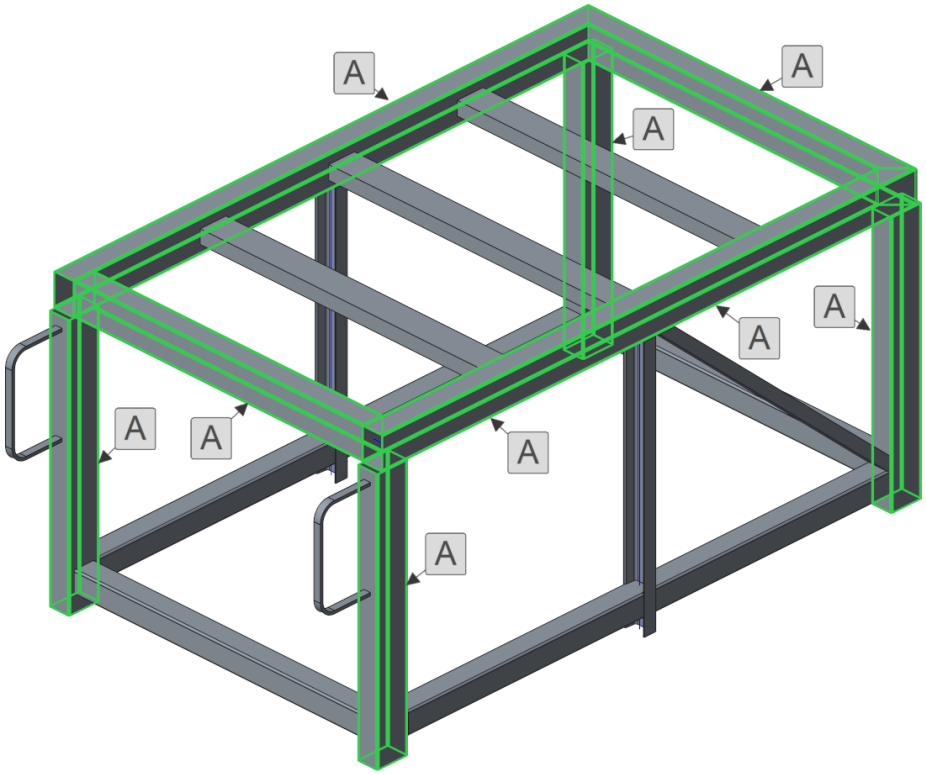
1. **Move commands**
   - Move by mouse pick
   - Align with other profile entity
   - Reset profile movement
2. **Move to predefined positions**
3. **Move by values**
4. **Rotate profiles**
5. **Switch start and end**

Rotate around Profile Axis

Besides rotating profiles when assembling them, they can ne rotated later on when joints already are defined.

1. Select the highlighted profiles **[A]**.
2. Click 90° in **Move Profile** dialog box.

3. Select the highlighted profiles **[A]**.
4. Click **[Repeat]** or middle mouse button.
   - The profiles are rotated and adjusted in length according to their joint definitions.



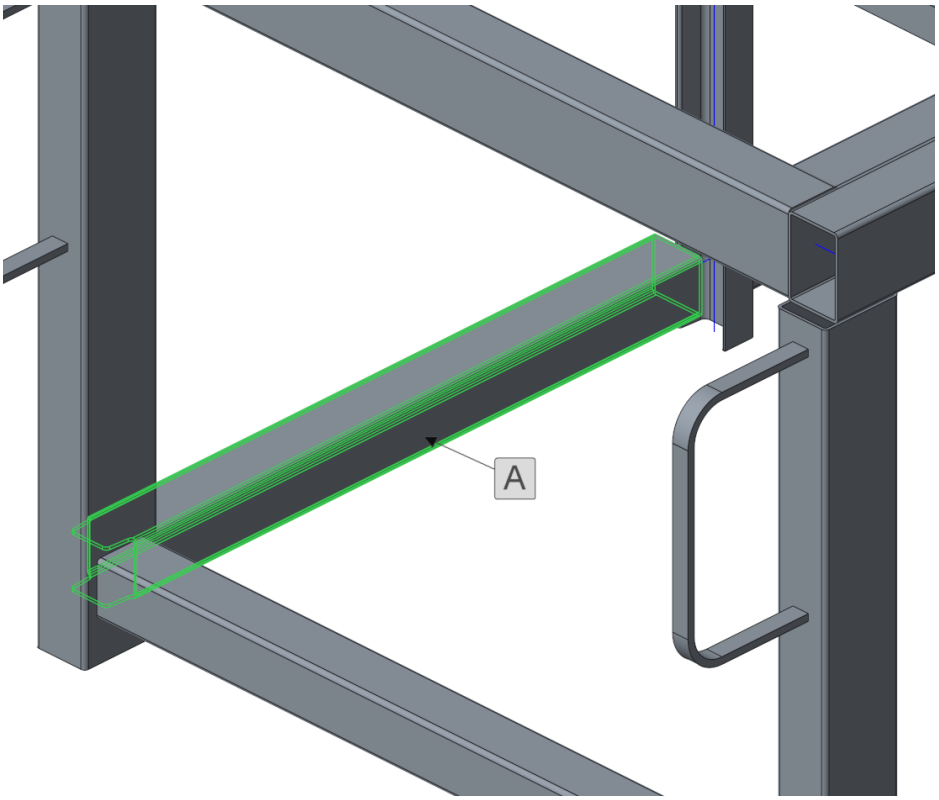
Rotate Profiles to Switch Profile Start and End on the Curve

Two cases make it necessary to switch profile start and end on the reference curve:

- You assemble a non symmetric profile (e.g. unequal angle) and the desired position can not be set just by rotation around the curve. It requires a mirroring of section geometry which is switching profile start and end on the reference curve.
- You have assembled an existing profile instance again and on the reuse place profile cuts (i.e. holes, cutouts etc) appear on wrong profile end, so you need to switch start and end.

1. Select the left rear square tube **[A]**. This is assembled as existing instance of the left front square tube. As the front square tube got a profile cutout using the U-profile, the cutout also appears at the rear reuse place.
2. Click  .
   - The profile start and end is switched on the curve, the cutout appears on the other side.
3. Click **[Repeat]** or middle mouse button.
4. Select the left rear square tube **[A]** again.

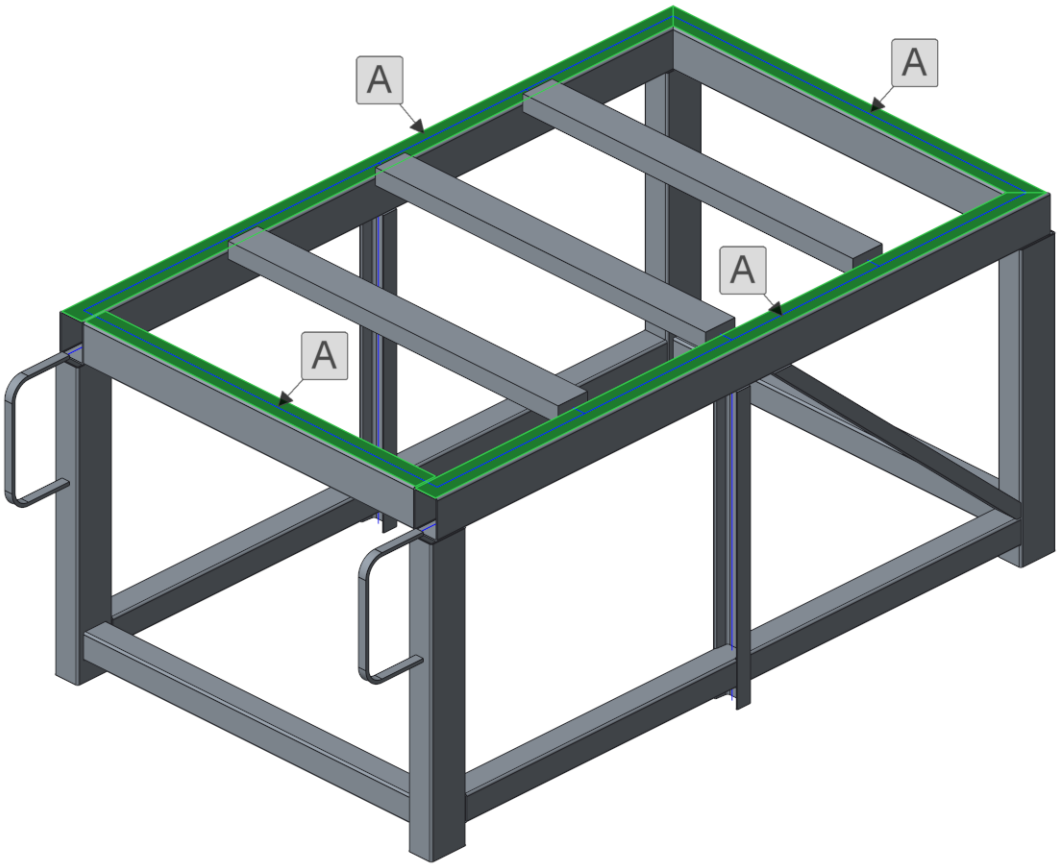
5. Click  of **Move** dialog box.
   - The profile is switched back to its original position.
6. Click **[Repeat]** or middle mouse button.



Move Profiles by Mouse Pick

With **AFX** you can move profiles by selecting a surface or edge so that the selected surface or edge is aligned with the profile reference curve.

1. Select the four upper rectangular tubes on their upper surface **[A]**.
2. Click  in **Move** group of **Move Profiles** dialog.
   - The four profiles are moved perpendicular to the selected surfaces and the reference curve is aligned with the surfaces.
3. Click **[Repeat]** or middle mouse button.

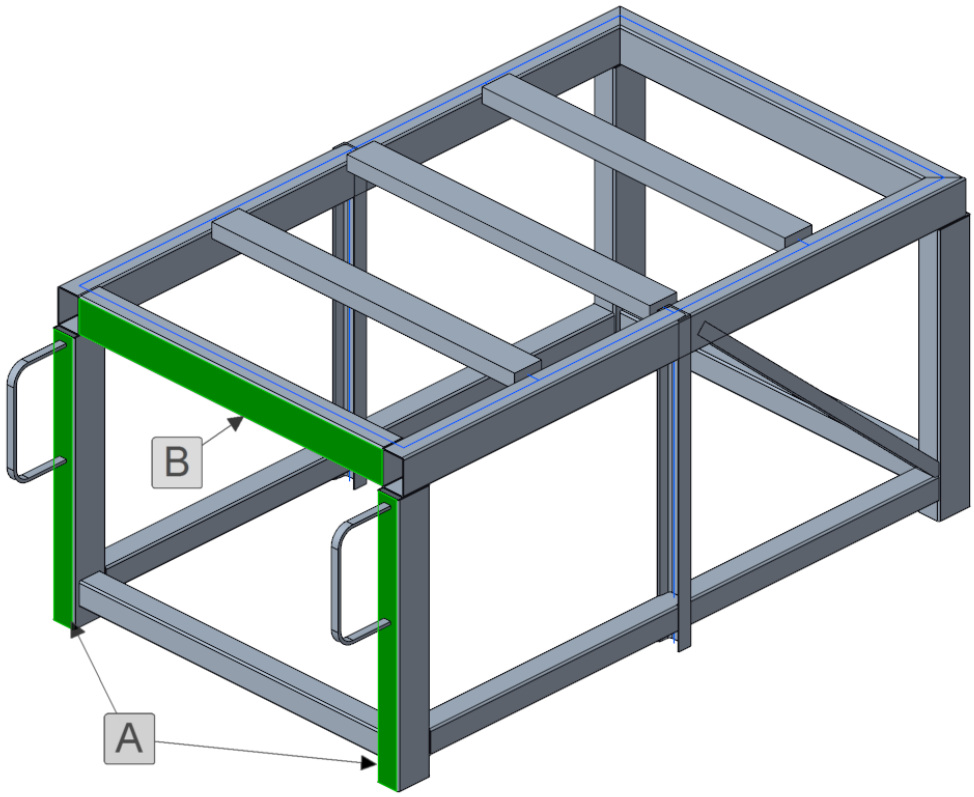


Hint 7 — Move profiles multiple times. With this method you can also move profiles multiple times with different surface selections. Use to undo the profile movement.

Move Profile to other entity

Using **AFX** you can move profiles to surfaces, edges or points of other profiles or parts.

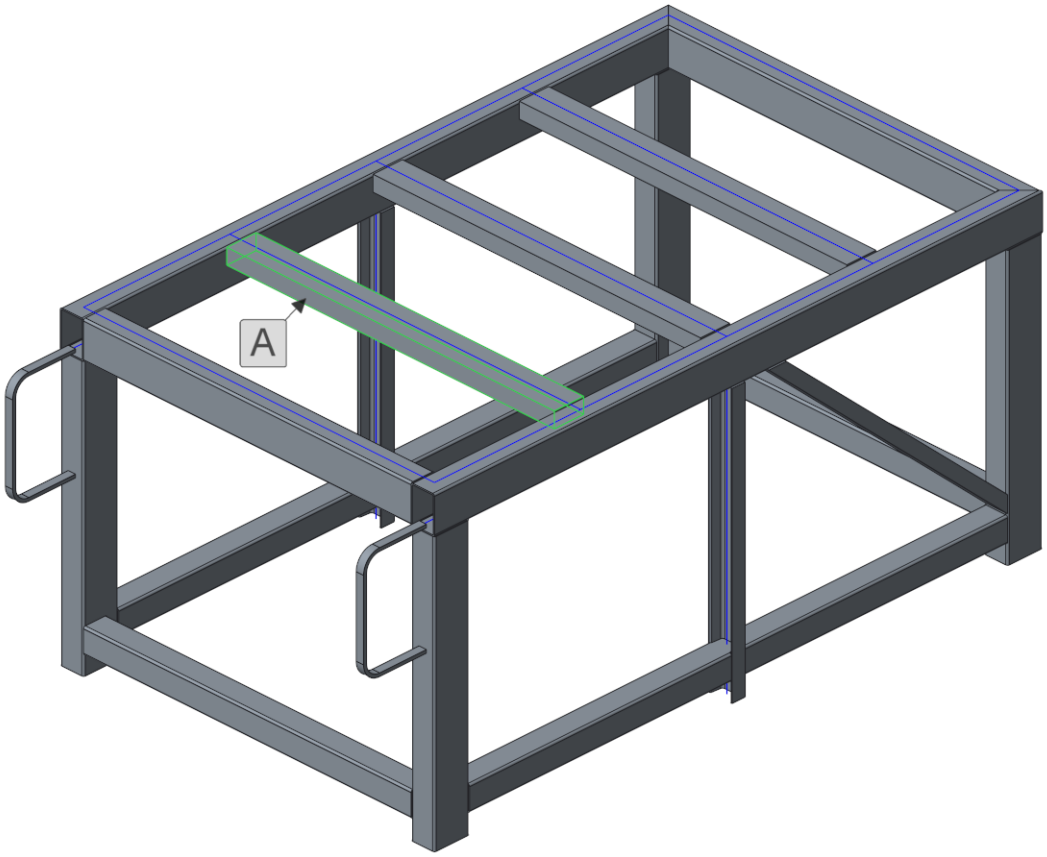
1. Select the two left vertical rectangular tubes on their outward surfaces **[A]**.
2. Click of **Move Profile** dialog box.
3. Select the outward surface of the left upper rectangular tube **[B]** as reference surface, to which the other profiles shall be moved.
   - The two profiles are moved perpendicular to the surfaces so that the selected surfaces of the profiles are aligned with the selected surface.
4. Click **[Repeat]** or middle mouse button
5. Repeat the procedure on the right side of the frame.



Move Profile to Predefined Position

Using **AFX** you can move profiles to predefined positions (e.g. left-top, right-top, left-bottom) relative to the reference curve. The definition of left, right, top and bottom depends on the current view direction you look at the assembly. This means that using the same button you get different results depending from which direction you look at the assembly. If a profile runs nearly parallel to the screen the definition of left, right, top and bottom may be not clear, therefore this function sometimes moves the profiles not as desired. In this case rotate the assembly (different view direction) or use a different function for moving.

1. Select the rectangular profile **[A]**.
2. Click  .
   - The profile is moved so its upper left edge aligns the reference curve.
3. Click  .
   - The profile is moved so that the reference curve aligns the upper center of the profile. As the profile is patterned, the pattern instances are moved as well.
4. Click **[OK]** to accept and close dialog box.



2.7 Modify and Replace Profiles

With the **Modify** commands you can:

- modify profile sections
- replace profile sections
- move profile ends relative to the reference curve.

Now click to open **Modify Ends** in Framework ribbon.

Modify profile ends

- Modify profile end using dimension
- Make profile end the same as another profile end
- Reset profile end

If you click to open **Modify Type** in Framework ribbon, you will be able to change the type of profile.

If you click to open **Modify Size** in Framework ribbon, you will be able to change the size of profile.

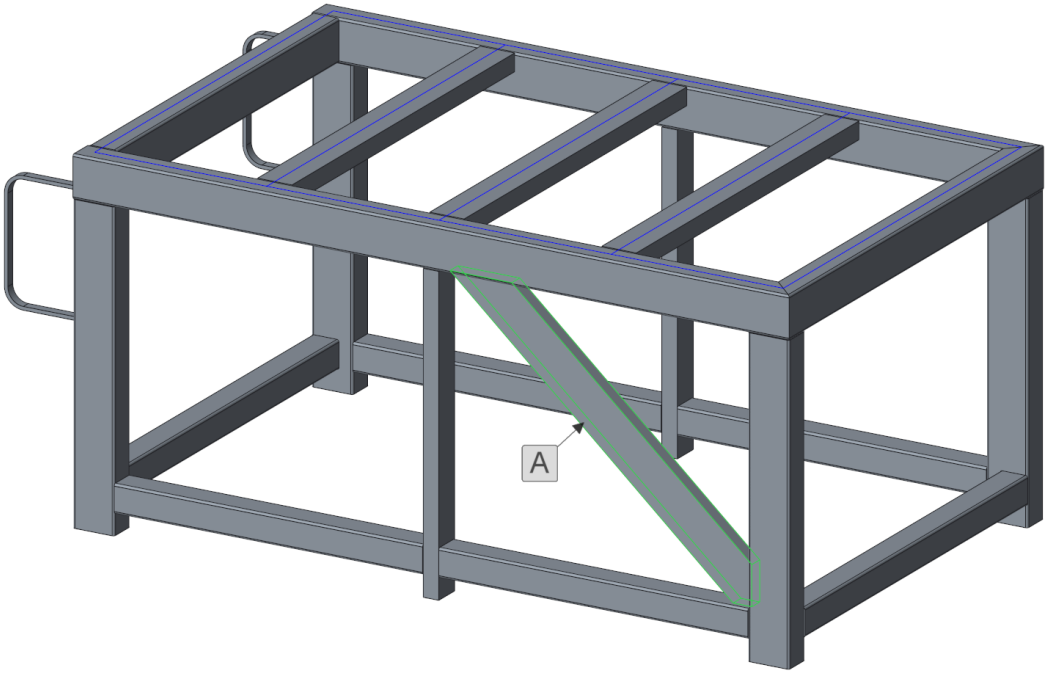
Modify Profile Section

As described in 2.7.0 two cases appear when modifying profile sections:

- **Modify section size** of same section type **AFX** keeps **Creo Parametric** model and just modifies the section dimensions. Holes and cuts made in the profile are kept.
- **Modify to different section type AFX** creates a new **Creo Parametric** model. Holes and cuts made in the profile are lost.

To Modify section size of the diagonal profile from **80x40** to **120x40** proceed as follows:

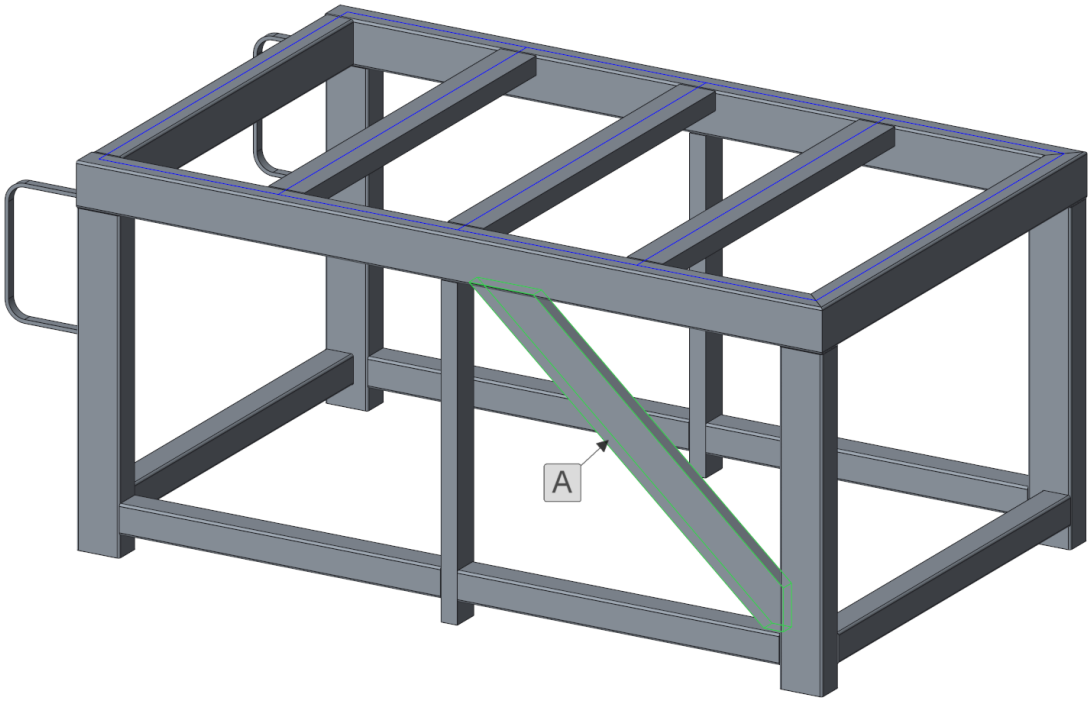
1. Click  to open the **Modify Size** command in **Profiles** group.
2. Select the diagonal rectangular beam **[A]** as reference.
3. Click  .
4. Select **[DIN 1017 120x40]**.
   - The dimensions of the section and BOM parameters are modified because the profile has the same section type. The modification is shown as a preview.
5. Click **[OK]** to accept modification and close dialog box.



To modify section of the diagonal **RECTANGULAR BEAM DIN 1017 120x40** to a **UBEAM DIN 1026 U100** proceed as follows:

1. Click  to open the **Modify Type** command in **Profiles** group.
2. Select the diagonal rectangular beam **[A]**.

3. Select a **[CHANNEL BEAM] [DIN 1026 U 100]**.
   - The rectangular beam is replaced by an U-beam. The U-beam is a new part, additional holes or cuts created in the rectangular beam (i.e. the planar trim) are lost. The resulr is shown as a preview.
4. Click **[OK]** to accept modification and close dialog box.



Hint 8 — Replace profiles. If the replaced profile sections fails, check whether the profile has children in the current assembly. If yes, suppress the children, replace the profile section and then resume the children. In some cases you need to redefine the placement of the children.

Replace Profiles

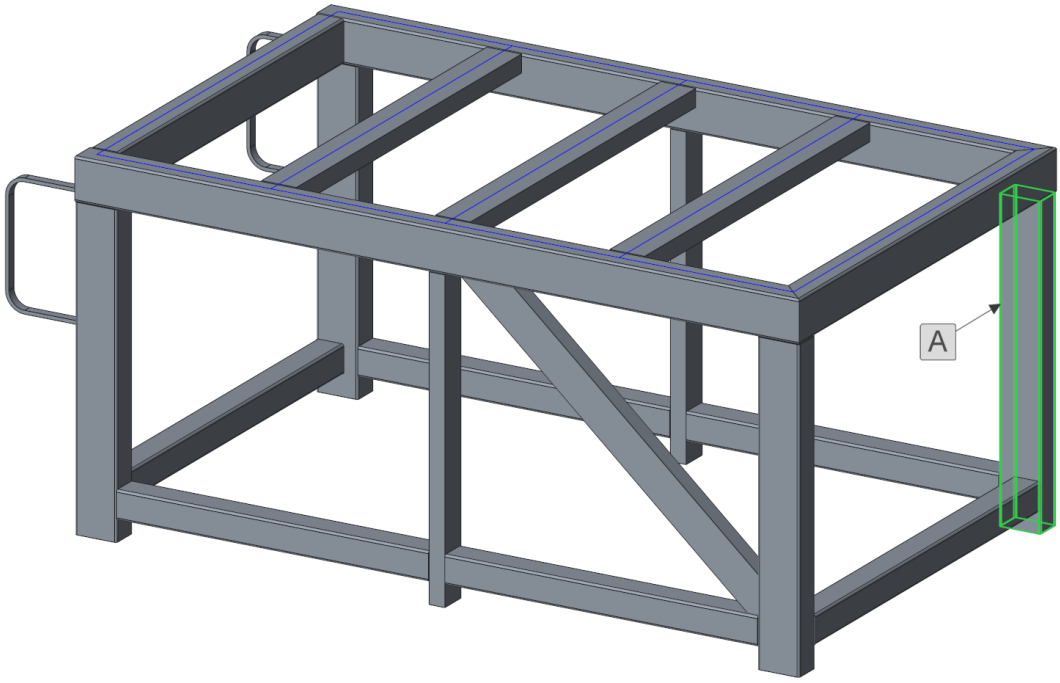
Click on **Profiles** group to open **Replace Profile** dialog box.

The following functions allow the replacement of profiles:

- Replace profile by an already assembled profile
- Replace profile by a copy of an already assembled profile
- Replace profile by a copy of the section of an already assembled profile
- Replace profile by a copy of itself

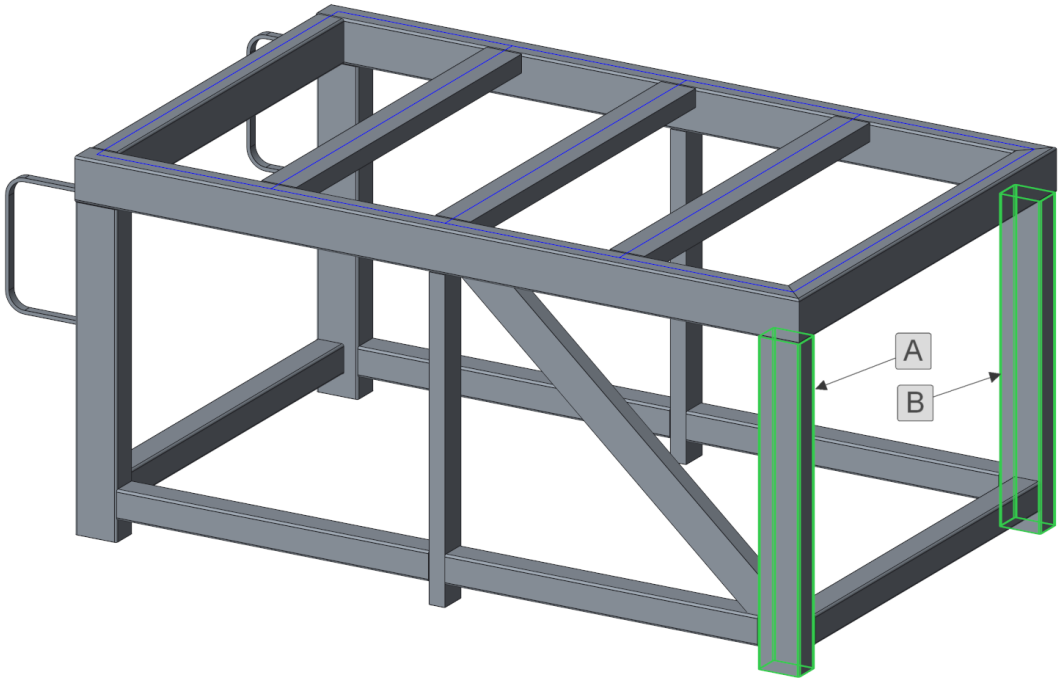
For example if you want to create holes in just one of the four identical vertical rectangular tubes, you have to replace this profile by a copy of itself (which creates a new part as copy of the existing). Afterwards you can create the holes in this part.

1. Click  in **Replace Type** area of dialog box.
2. Select the rear right rectangular tube **[A]**.
   - A copy of the selected profile is assembled at the selected location (a new part with new name appears in model tree) as a preview..
3. Click **[Repeat]** or middle mouse button accept modification.



If you copied a profile by mistake you can undo the procedure above (as the different profiles need to be the same parts again). Proceed as follows.

1. Click ↺ in **Replace Type** area of dialog box.
2. Select the front right rectangular tube **[A]** as existing profile by which the others shall be replaced.
3. Select the rear right rectangular tube **[B]**.
   - The rear right rectangular tube will be replaced by the selected profile, the vertical profiles on all four sides of the frame are identical parts again.
4. Click **[OK]** to accept modification and close dialog box.



Modify Profile Ends

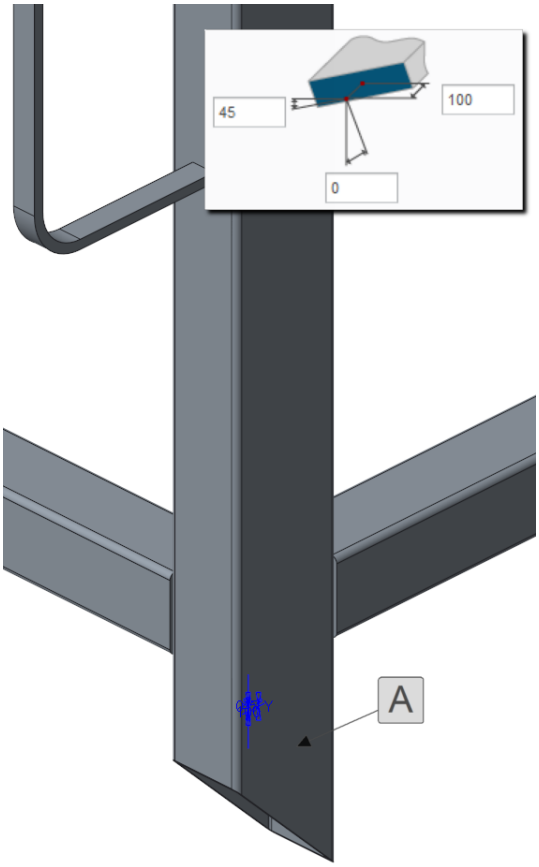
Click on **Profiles** group to open **Modify Ends** dialog box.

The following functions allow modifying the profile end surface relative to the end of the reference curve.

- Modifying profile ends by entering offset dimension, X- and Y-rotation angle in the input boxes or by modifying the dimensions on the model.
- Copying the modification of a profile end to another profile end.
- Reset profile end.

To extend the left front rectangular tube **100mm** and cut it in **45°** proceed as follows.

1. Click on **Modify** tab of **Profiles** dialog box.
2. Select the lower end of the rectangular tube **[A]**.
   - The current profile end dimensions appear on the model and in the input boxes of the dialog box.
3. Enter the desired values in the input boxes **[100]** for the offset and **[45]** for the left angle like shown in the picture.
4. Press **[enter]** or the middle mouse button.
   - The profile end will be modified. Now reset the modification again.
5. Click in the dialog to reset the changes.
6. Select the profile end again **[A]**.
7. The profile end dimensions will be reset to **0**.



Hint 9 — Modify profile ends. Take care that the profile end you want to modify is not controlled by a joint otherwise the joint will overwrite your settings after regenerating the assembly.

3

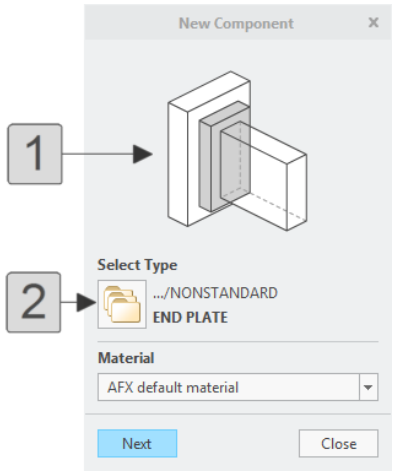
Connector and Equipment Elements

Overview
Assemble new Connectors
Reassemble Connectors
Reassemble Connectors as Copy
Modify Connectors
Replace Connectors by Copy of Itself
Assemble typical Connector Elements
Assemble typical Equipment Elements

3.1 Overview

The library of **AFX** contains a lot of connector and equipment elements for structural steel and aluminum profiles. When assembling these elements with **AFX**, not only the components are assembled, but also the necessary modifications (cutting the profile, creating holes, . . .) are made. These elements can be accessed by the **Connector elements** dialog box or by the **Equipment elements** dialog box. Both dialog boxes are similar they just access different areas of the library.

To open the **Connector elements** dialog box click **New Connector Elements**.

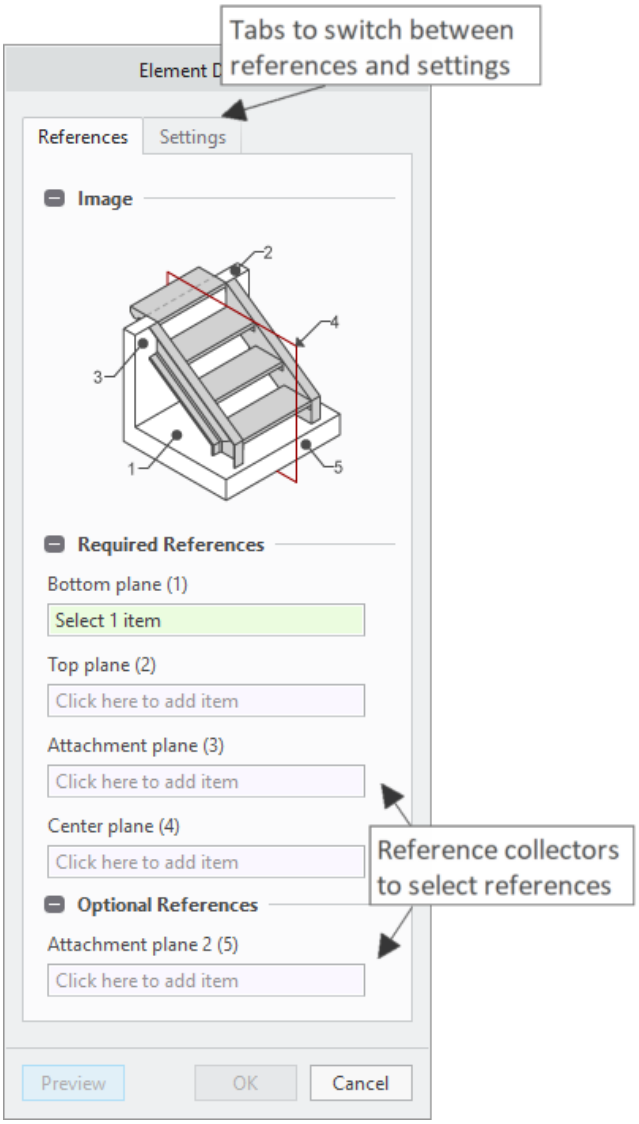


1. **Preview image**
2. **Choose connector**

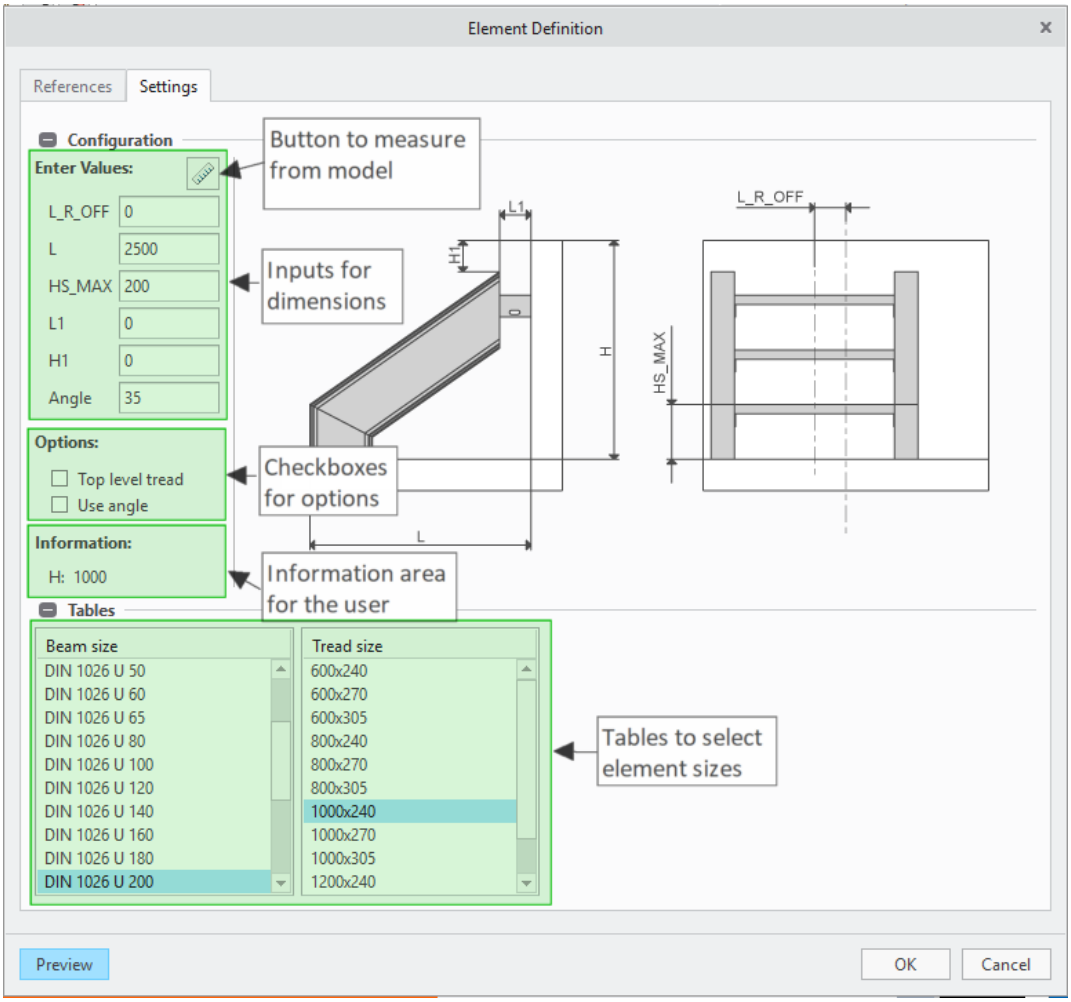
Connector and Equipment Elements

All connector and equipment elements of **AFX** library (especially for structural steel) have an element specific configuration dialog box, in which you can make the desired configuration settings and which assists you in selecting the necessary references. As the different elements need different settings and references, all configuration dialog boxes are different. However they have a common look and feel and consist of the same dialog elements. You should always use them in the same way. The structure of a element configuration dialog box is shown in the following pictures.

Tab 1: References selection



Tab 2: Definition of settings, dimensions and options



Other functions of the Element Definition dialog are:

- Measure in **Creo Parametric** (diameter, distance, length)

Use these dialog boxes with following procedure:

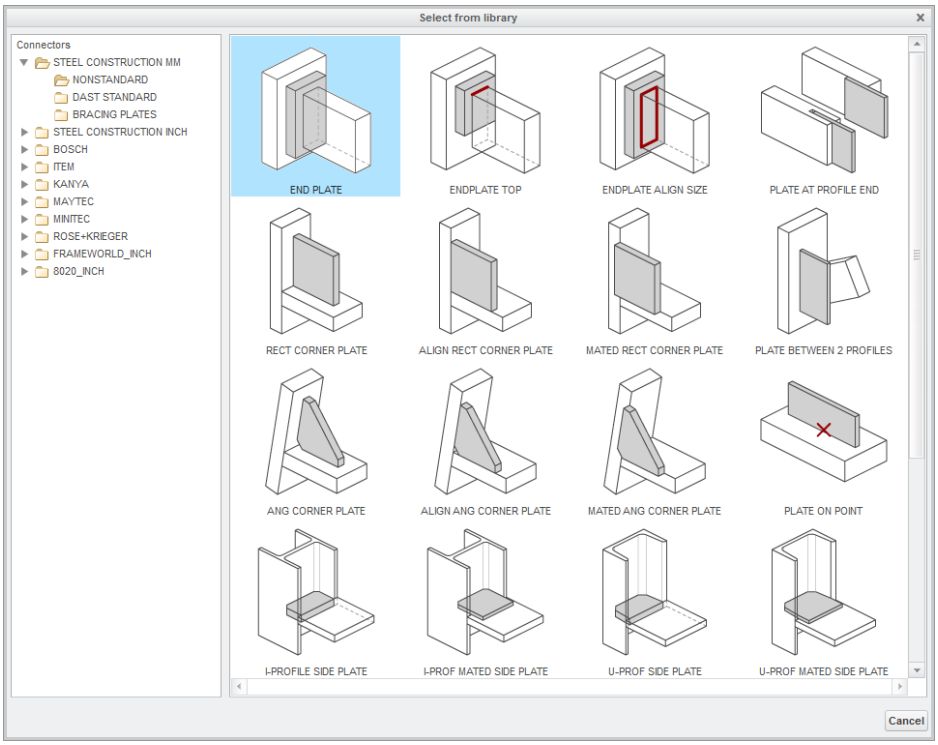
1. When the dialog box opens for creating an element the **References** tab is active. Look at the picture to understand necessary placement references (marked by numbers).
2. Define **Required references** and **Optional references**: Look in the picture for the number in brackets which can be found in the **references area** and think about which reference of your model corresponds with the picture.
3. Make sure that the desired reference collector is active and select the reference. The selected reference also is highlighted in the model.
4. Repeat step 3 for all required references.
5. **Optional references**: If required also repeat step 3 for optional references.
6. Toggle to the **Settings** tab. Look at the picture to understand dimensions and options (marked by letters).
7. If tables are available in the lower area of the dialog box, perform the table selections. The table headline shows you what you can select. Perform all table selections - no table should remain unselected.

8. **Options**: If options are available check the required options. The meaning of options is described by the text and picture.
9. **Enter values**: If input boxes are available enter the desired dimensions or parameters. The meaning of dimensions are described in the picture. If you see a «?» in the box no value is specified. You can overwrite default values. If a **?** in a box changes after performing a table selection this box is controlled by table values. You can overwrite these values after performing all table selections. If you select a table after overwriting such a value it will be set to the default from the table.
10. Click **[Preview]** to create the element as a preview.
11. Click **[OK]** after completing the configuration.
    - The element will be assembled.

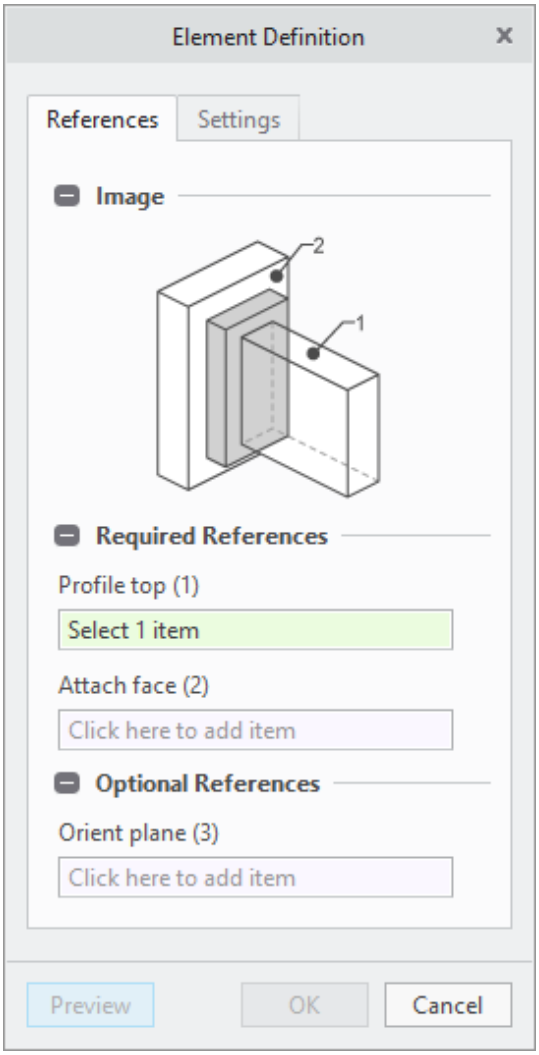
3.2 Assemble new Connectors

When assembling a new instance of a connector element, a new part is copied from library and assembled. In this example, an **end plate** shall be assembled.

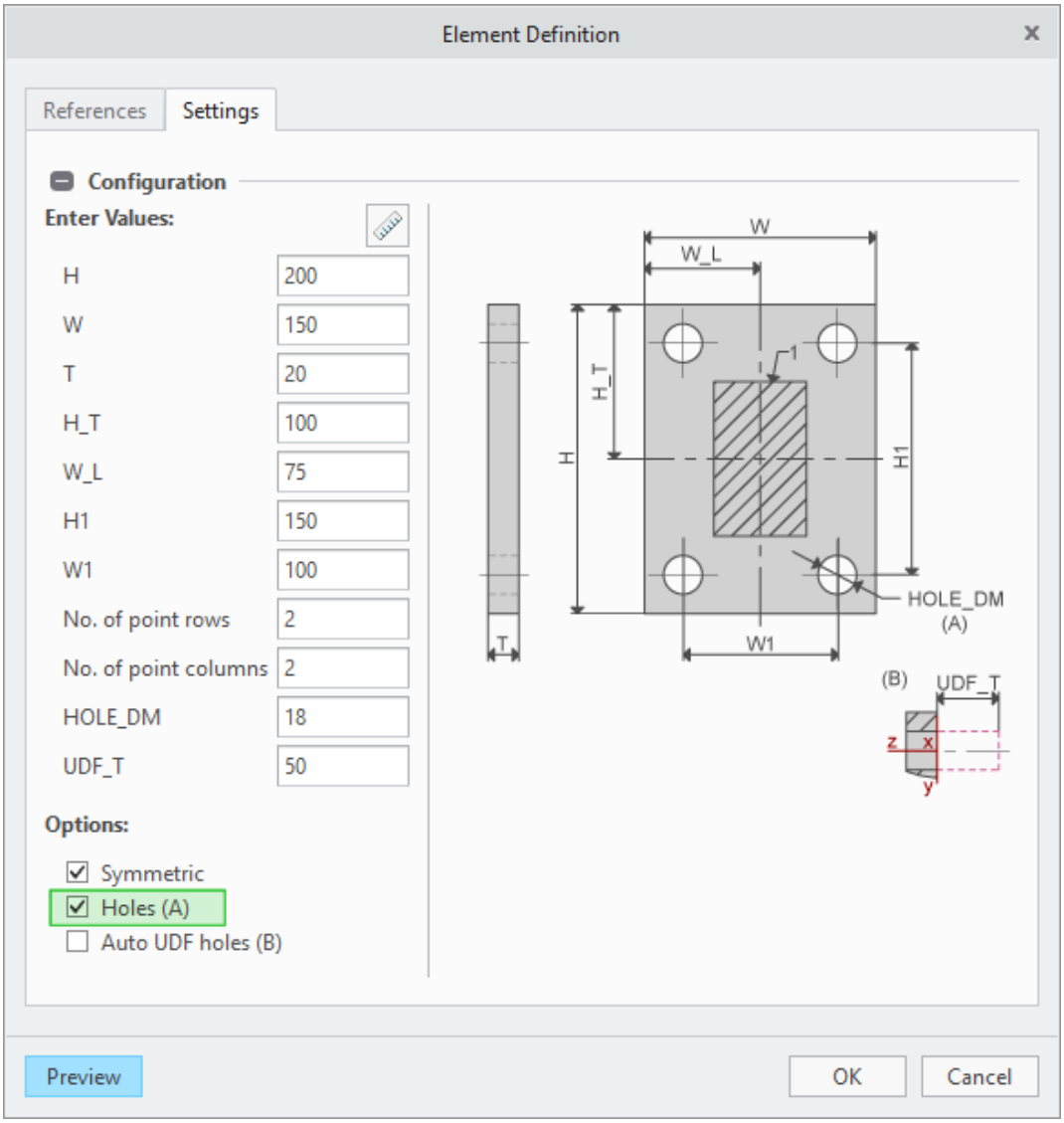
1. Click  to open Select from library dialog box for selecting a connector element.
2. Select **[STEEL CONSTRUCTION MM] > [NONSTANDARD]** and then the element **[END PLATE]**.



3. The element definition dialog box opens with References tab active.

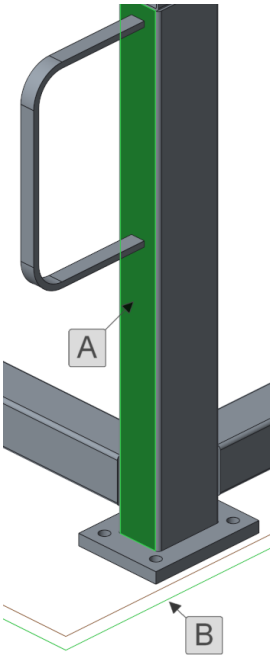


4. Make sure that the **[Profile top (1)]** reference collector is active and select the side surface of the right front rectangular tube near the profile end **[A]**
5. Make sure that the **[Attach face (2)]** reference collector is active and select datum plane A_XZ **[B]**.
6. Toggle to **Settings** tab.



7. Check option **[Holes (A)]** to create the end plate with holes.
8. Keep the default values for plate size.

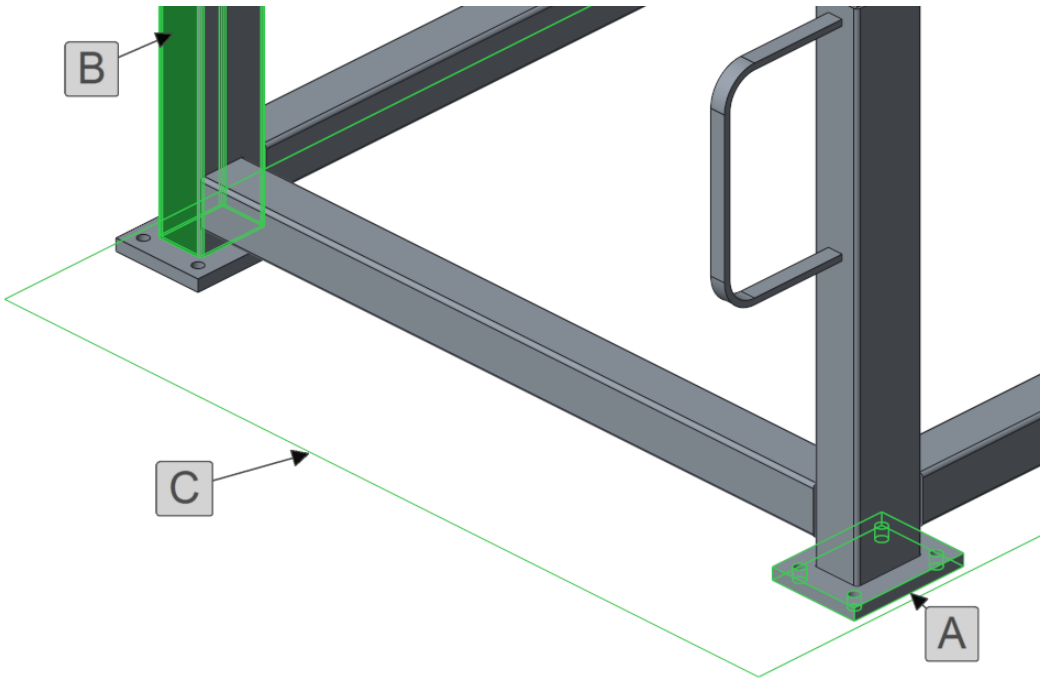
9. Click **[Preview]** or middle mouse button to get a preview..
10. Click **[OK]** or middle mouse button.
    - The end plate will be assembled as new part and the profile is cut.



3.3 Reassemble Connectors

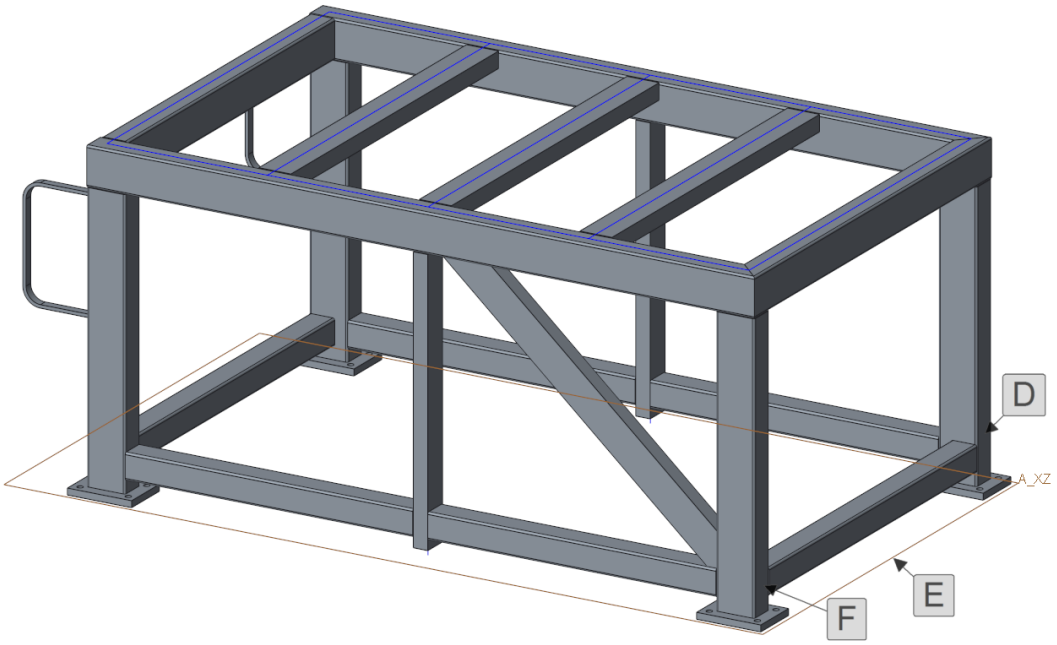
An already assembled connector element can be assembled again as identical part. In this example the previously assembled end plate shall be assembled at the end of the three other vertical rectangular tubes.

1. Click ↺ in **Components** group.
2. Select the end plate assembled in previous chapter **[A]**.
3. Press **[Next]** or middle mouse button. The element definition dialog box opens with References tab active.
4. Make sure that the **[Profile top (1)]** reference collector is active and select the side surface of the left front rectangular tube near the profile end **[B]**
5. Make sure that the **[Attach face (2)]** reference collector is active and select datum plane ▱ A_XZ **[C]**.



6. Click **[Preview]** or middle mouse button to get a preview.
7. Click **[OK]** or middle mouse button.
   - The end plate will be reassembled and the profile is cut.

   Now assemble the end plate on the other two vertical rectangular tubes.
8. Click **[Next]** or middle mouse button in the **Reuse Component** dialog box.
9. Repeat step 4 to 7 and select side surface of rear right rectangular tube near the profile end **[D]** as **Profile top** and datum plane A_XZ **[E]** as **Attach face**.
   - The end plate is assembled at the rear right rectangular tube.
10. Click **[Next]** or middle mouse button in the **Reuse Component** dialog box.
11. Repeat step 4 to 7 and select side surface of front right rectangular tube near the profile end **[F]** as **Profile top** and datum plane A_XZ **[E]** as **Attach face**.
    - The end plate is assembled at the front right rectangular tube.
12. Click **[Close]** in the **Reuse Component** dialog box to quit the process.



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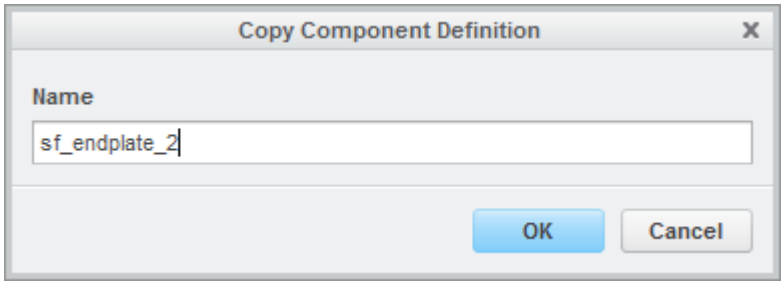
3.4 Reassemble Connectors as Copy

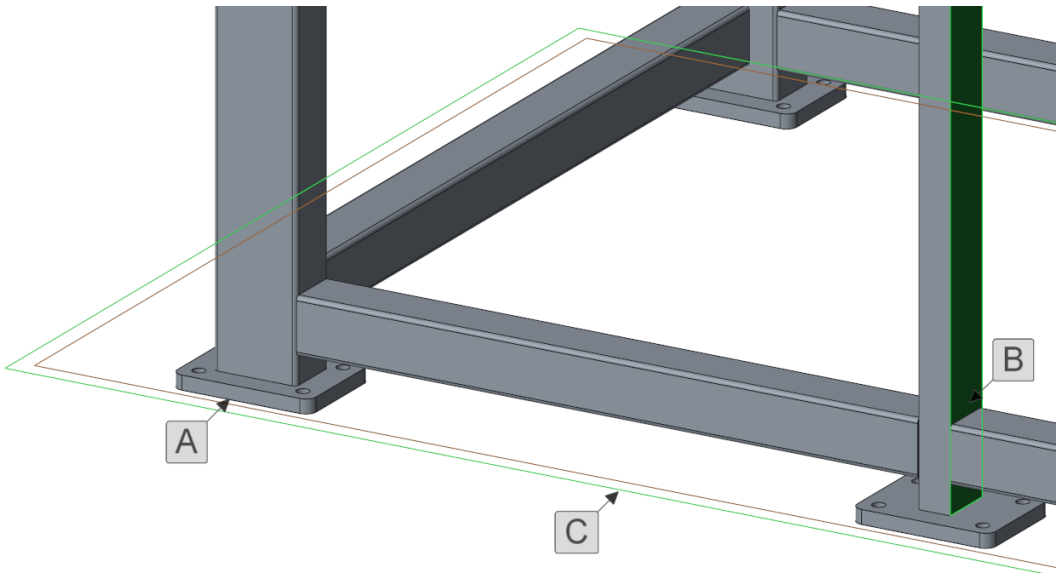
If you have made modifications at an assembled connector element (e.g. created additional features) and want to assemble the connector element at another location including these modifications but different dimensions, it is common practice to assemble the connector element as copy of an existing connector element.

1. Using standard **Creo Parametric** functionality, create 4 rounds at the 4 edges of the previously assembled end plate **[A]**.
2. Open the **Reuse Component** dialog box with **Reuse**.
3. Click to assemble a copy of an existing instance of a connector element.
4. Select the end plate which you want to assemble as copy **[A]**.
   - The **Copy Component Definition** dialog box opens. It allows you to define a name for the copied component. In this case leave the default value.



5. Click **[OK]** to close the dialog and to continue the process. The element definition dialog box opens with References tab active

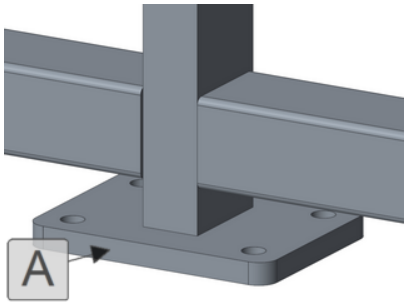
6. Select the side surface of the middle front U-beam near its end **[B]** as **Profile top** and datum plane A_XZ **[C]** as **Attach face**.
7. Click **[Preview]** or middle mouse button to get a preview.
8. Click **[OK]** or middle mouse button.
   - A copy of the end plate will be assembled. Now it can be modified.



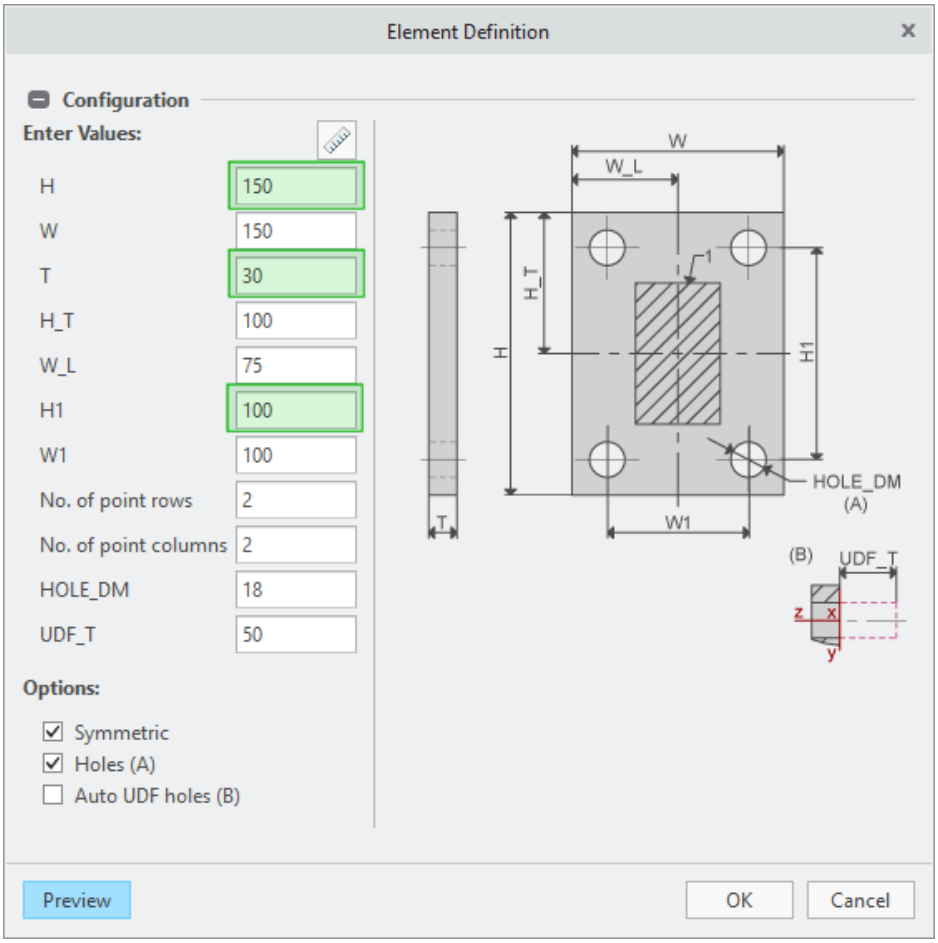
3.5 Modify Connectors

With this function you can modify the size of a connector element including the dimensions of cuts, holes etc. which are created together with the element. However redefining placement references is not possible.

1. Click in **Components** group.
2. Select the end plate which was assembled as a copy in previous chapter **[A]**.
   - The Element definition dialog box for the end plate opens with the **Settings** tab active. The **References** tab is not visible.



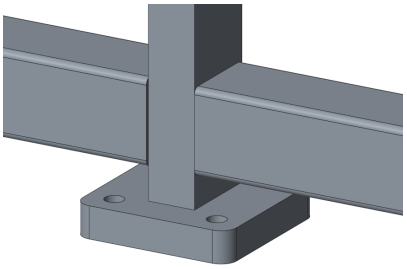
3. Modify the values for **H** to **[150]**, **T** to **[30]** and **H1** to **[100]** like shown in next picture.



The values **H1** and **W1** control a datum point feature with four points (on blanked layer 00_SCREW_POINTS). These points can be used for the placement of fasteners later on.

4. Click **[Preview]** or middle mouse button to get a preview.
5. Click **[OK]** to close the **Element definition** dialog box.

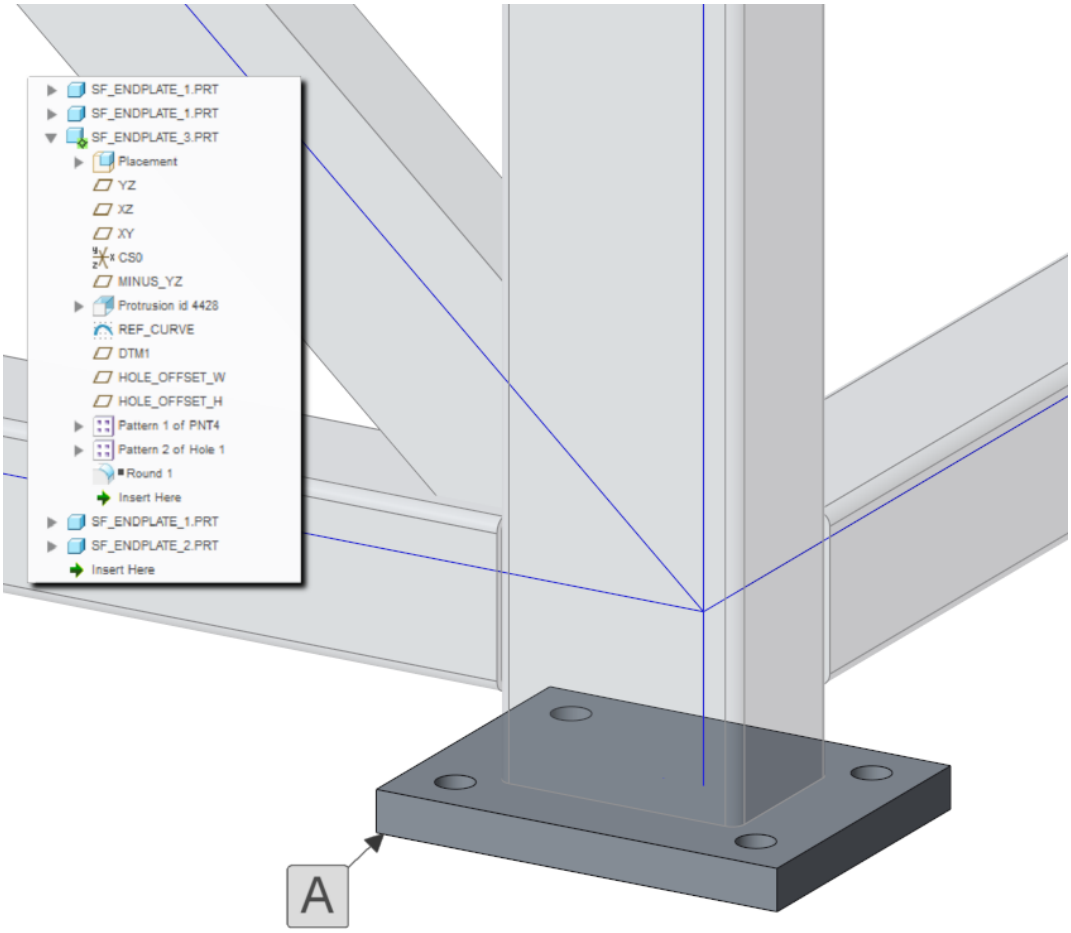
The end plate is modified and the U-beam is adjusted to the new plate thickness.



3.6 Replace Connectors by Copy of Itself

With this function you can replace a connector element by a copy of itself. This is required if a connector element, which previously was assembled as identical model, needs to be different now.

1. Click in **Components** group.
2. Select the end plate at the front right vertical profile, which was assembled as a copy in previous chapter **[A]**.
3. The **Copy Component Definition** dialog box opens. Accept the default name of the copied plate part by pressing **[OK]**.
   - The end plate is replaced by a new model which is a copy of the previous model. Now it can be modified, e.g. delete the rounds at the corner or modify size.

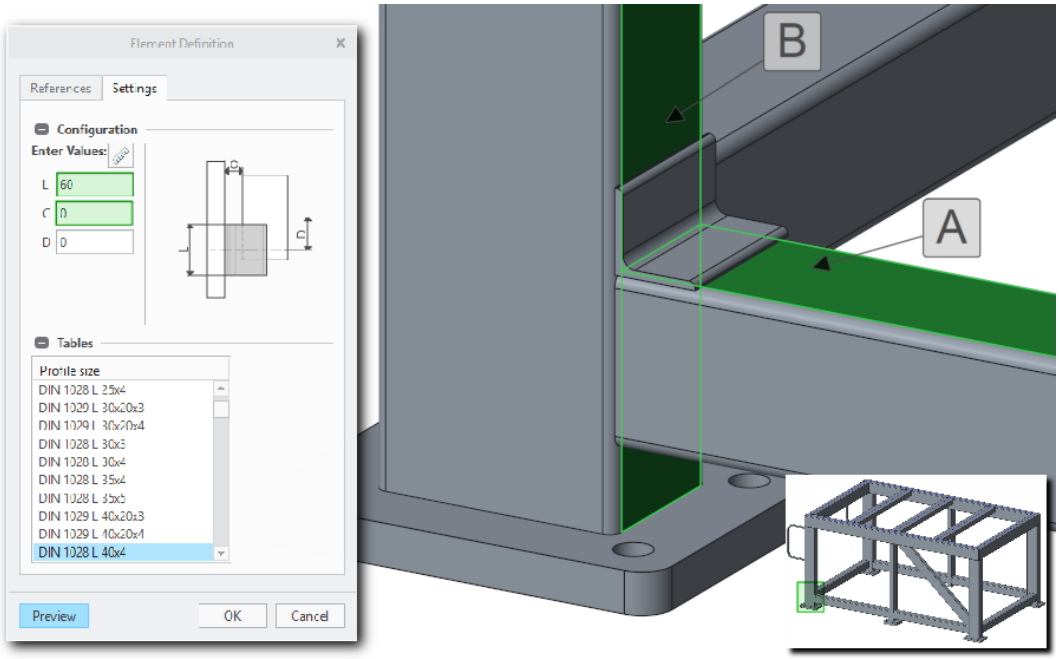


3.7 Assemble typical Connector Elements

This chapter describes some other typical connector elements for steel profiles.

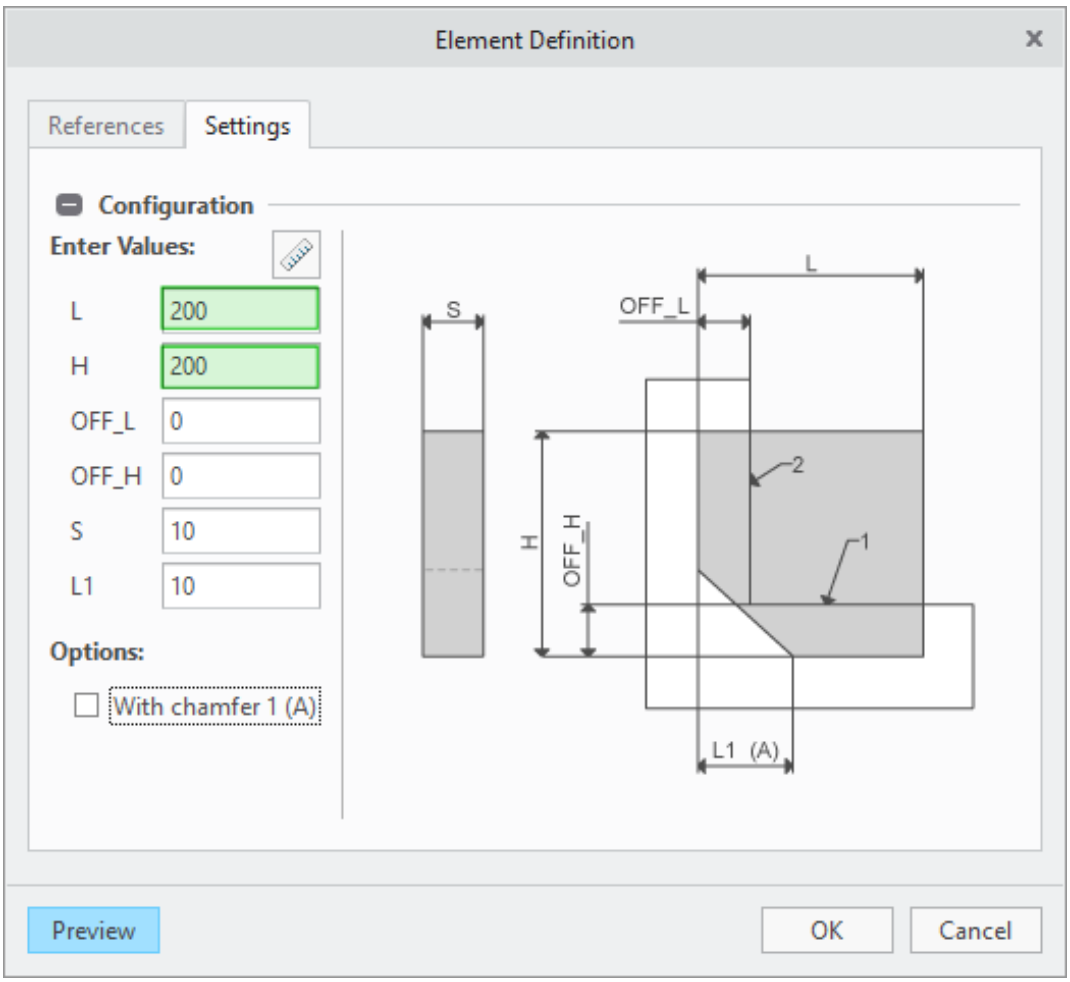
Assemble a Single Angle Connection

1. Click  to open **Select from library** dialog box for selecting a connector element.
2. Select **[STEEL CONSTRUCTION MM] > [NONSTANDARD] > [1 ANGLE]**.
   - The Element definition dialog box of the single angle opens on **References** tab.
3. Make sure that the **Profile side face** reference collector is active and select the profile side surface of the lower left square tube **[A]**.
4. Make sure that the **Attach face** reference collector is active and select the side surface of the left rectangular tube **[B]**.
5. Toggle to the **Settings** tab.
6. Select angle size **[DIN 1028 L 40x4]** from table.
7. Enter the desired angle length **[80]** in input field **L**.
8. Enter the desired offset value between profile end and attachment surface in the input field **C**. In this example enter **[0]**.
9. Click **[Preview]** or middle mouse button to get a preview.
10. Click **[OK]** or middle mouse button to close the Element definition dialog box.
    - The angle is assembled and the profile shortened, if necessary.



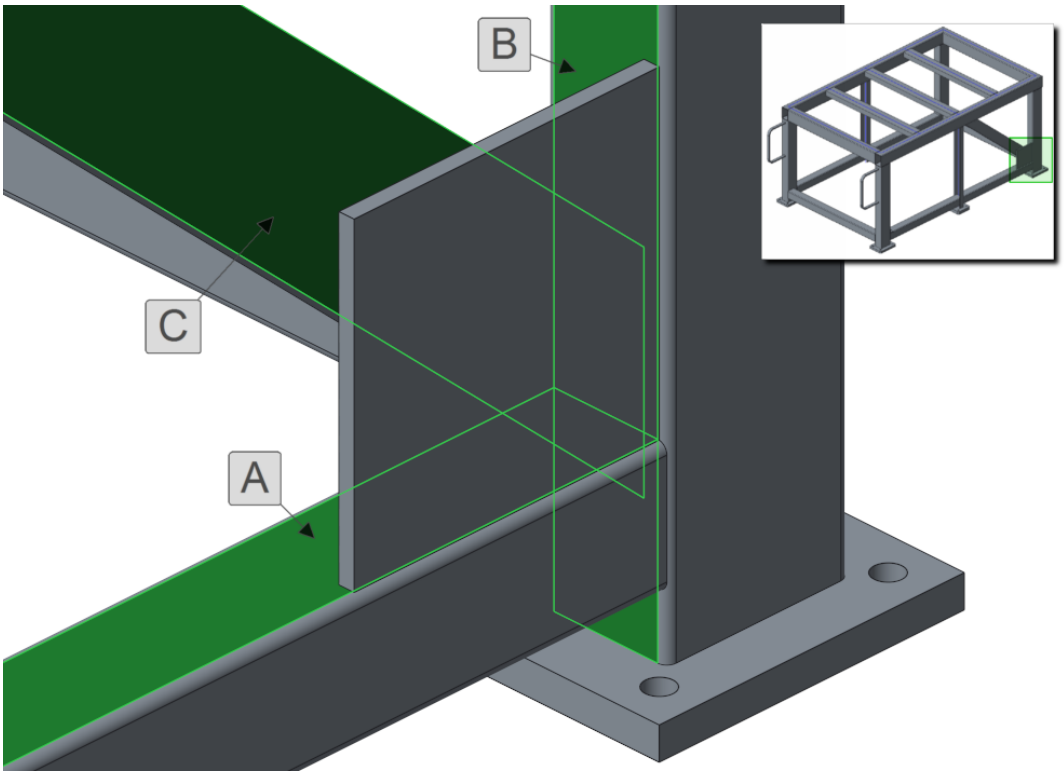
Assemble a Plate in a Rectangular Corner

1. Click  to open **Select from library** dialog box for selecting a connector element.
2. Select **[STEEL CONSTRUCTION MM] > [NONSTANDARD] > [RECT CORNER PLATE]**.
   - The Element definition dialog box of the end plate opens on **References** tab.
3. Make sure that the **Profile end** reference collector is active and select the side surface of the lower right square tube **[A]**.
4. Make sure that the **Attach face** reference collector is active and select the side surface of the right rectangular tube **[B]**.
   - Usually the plate is assembled centered to the profile selected at **Profile end**. Optional you can select a profile side surface to align the bracket to this surface.
5. In this example Make sure that the **Profile side face** reference collector is active and select the side surface of the diagonal rectangular beam **[C]**.
6. Toggle to the **Settings** tab.



7. Enter desired length **[200]** in field **L**.
8. Enter desired height **[200]** in field **H**.

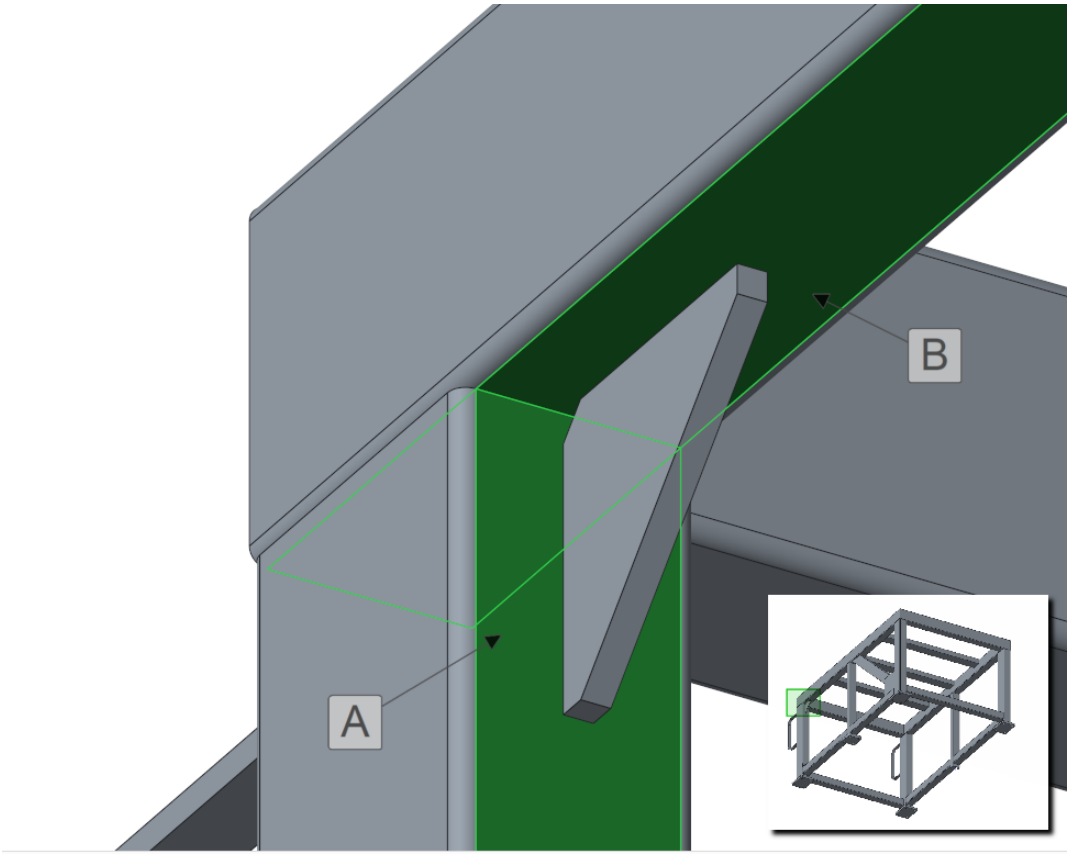
9. Click **[Preview]** or middle mouse button to get a preview.
10. Click **[OK]** or middle mouse button to close the Element definition dialog box
    - The plate is assembled. If necessary you can modify the plate with regular **Creo Parametric** functions and assemble it as existing element or copy of existing element on other locations of the assembly.



Assemble a Triangular Corner Plate in a Rectangular Corner

1. Click  to open **Select from library** dialog box for selecting a connector element.
2. Select **[STEEL CONSTRUCTION MM] > [NONSTANDARD] > [TRIANG CORNER PLATE]**.
    - The Element definition dialog box of the end plate opens on **References** tab.
3. Make sure that the **Profile end** reference collector is active and select the side surface of vertical left rectangular tube **[A]**.
4. Make sure that the **Attach face** reference collector is active and select the side surface of upper horizontal rectangular tube **[B]**.
5. Toggle to the **Settings** tab.
    - Enter desired values for bracket size. In this example keep the default values.
6. Click **[Preview]** or middle mouse button to get a preview.
7. Click **[OK]** or middle mouse button to close the Element definition dialog box.

- The corner plate is assembled.



3.8 Assemble typical Equipment Elements

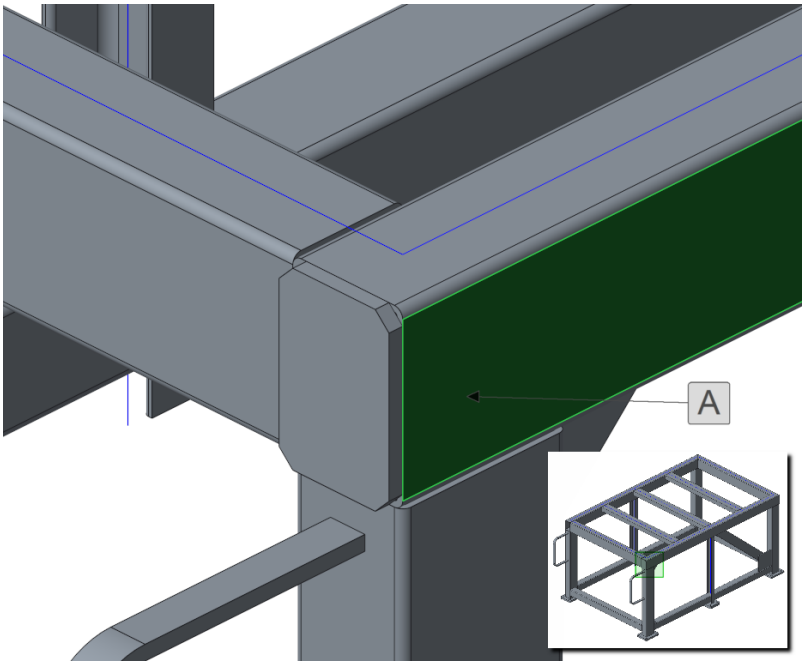
In this chapter some typical equipment elements for steel profiles are explained. They work similar to connector elements, however they are accessible by different menu buttons and a different dialog box.

To open the **Equipment elements** dialog box click **New Equipment Elements**.

Assemble a Profile End Cap

1. Click to open **Select from library** dialog box for selecting a equipment element.
2. Select **[STEEL CONSTRUCTION MM] > [BEAM END PLATES] > [END PLATE RECT]**.
   - The **Element definition** dialog box of the end cap opens on **References** tab.
3. Make sure that the **Profile end** reference collector is active and select the side surface of the rectangular tube **[A]**.

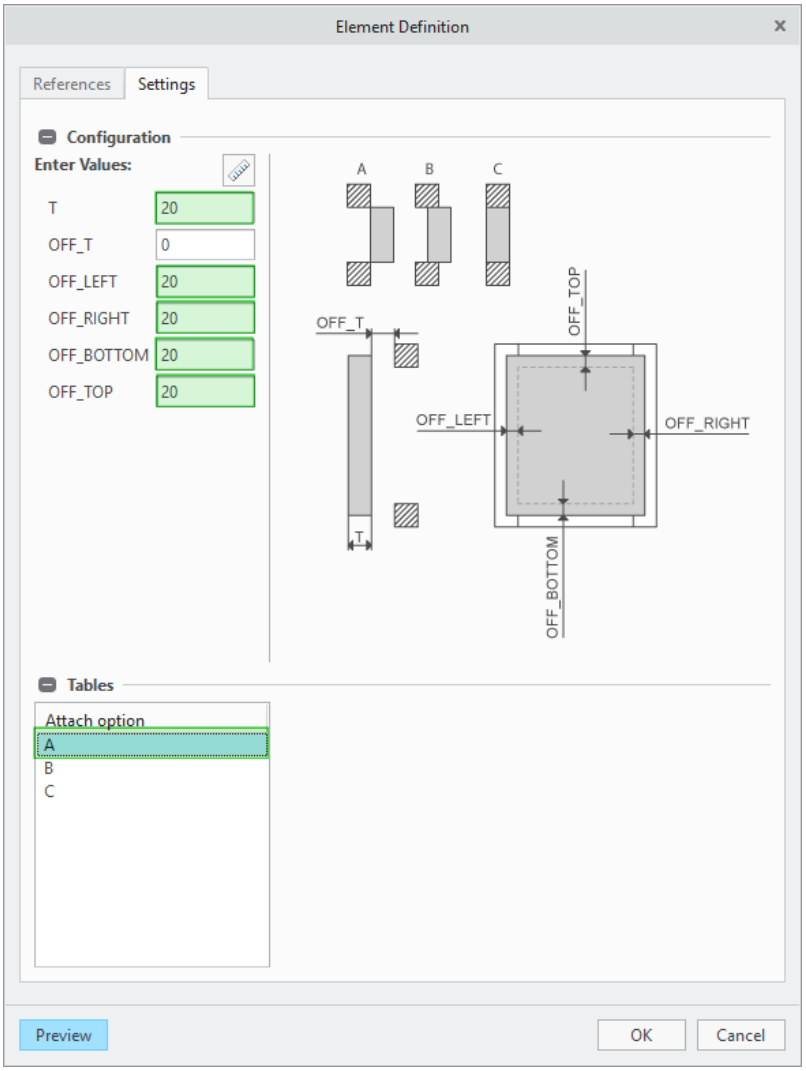
4. Toggle to the **Settings** tab.
   - Enter desired values for cap size. In this example keep the default values.
5. Click **[Preview]** or middle mouse button to get a preview.
6. Click **[OK]** or middle mouse button to close the **Element definition** dialog box.
   - The end cap is assembled.



Assemble a General Rectangular Plate Element

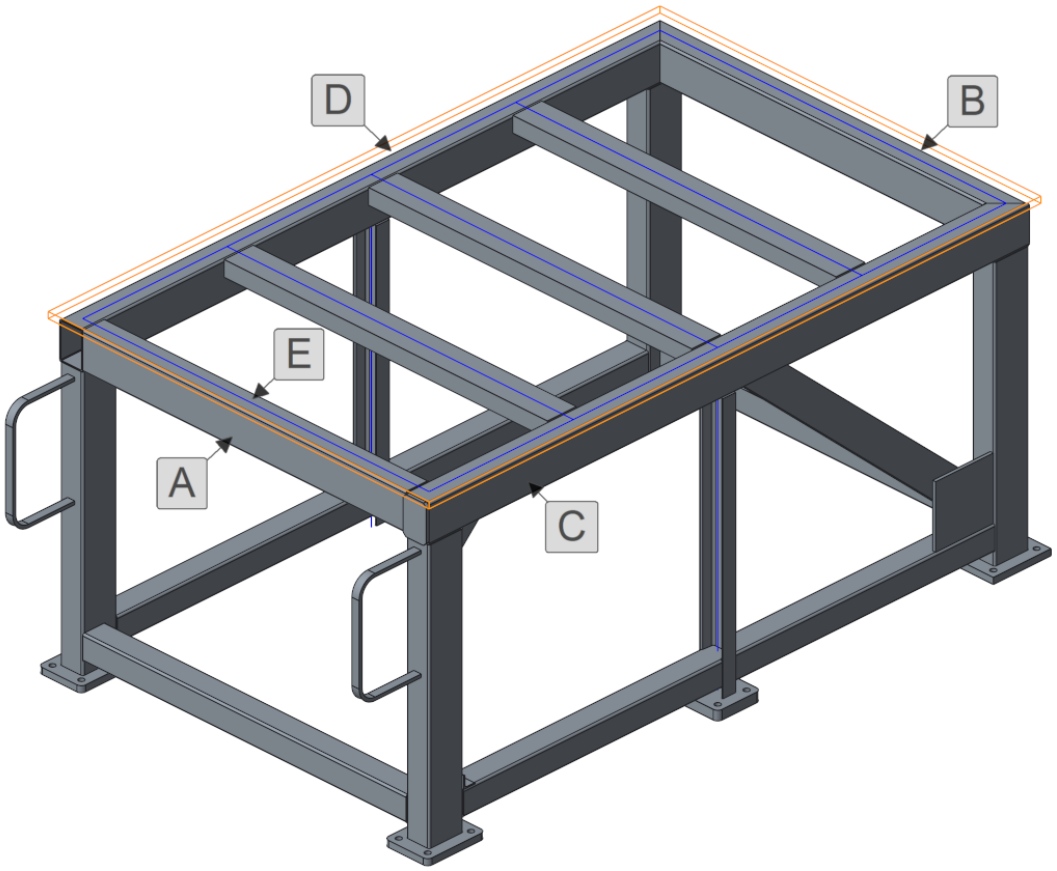
This element can be used as general element for rectangular plate elements (e.g. cover plates, table plates etc.).

1. Click  to open **Select from library** dialog box for selecting a equipment element.
2. Select **[PLATES MM] > [PLATE]**.
   - The Element definition dialog box of the **PLATE** opens on **References** tab.
3. Make sure that the **Left Plane** reference collector is active and select the left profile side surface of upper left rectangular tube **[A]**.
4. Make sure that the **Right Plane** reference collector is active and select the right profile side surface of upper right rectangular tube **[B]**.
5. Make sure that the **Bottom Plane** reference collector is active and select the front profile side surface of front horizontal rectangular tube **[C]**.
6. Make sure that the **Top Plane** reference collector is active and select the rear profile side surface of rear horizontal rectangular tube **[D]**.
7. Make sure that the **Attachment plane** reference collector is active and select the upper profile side surface of front horizontal rectangular tube **[E]**.
8. Toggle to the **Settings** tab.



9. From table select **Attach option [A]** to place the plate on top of the **Attachment plane** (B = centered, C= aligned).
10. Enter desired plate thickness **[20]** in field **T**.
11. Enter desired offset values **[20]** in the four fields **OFF_LEFT**, **OFF_RIGHT** , **OFF_ BOTTOM** and **OFF_TOP**.

12. Click **[Preview]** or middle mouse button to get a preview.
13. Click **[OK]** or middle mouse button to close the **Element definition** dialog box.
    - The plate is assembled.



4

Screw and Dowel Pin Connections

Overview
Assemble fasteners on Points/Axis
Reassemble fasteners
Redefine a Screw Connection
Assemble Fasteners by Mouse Click

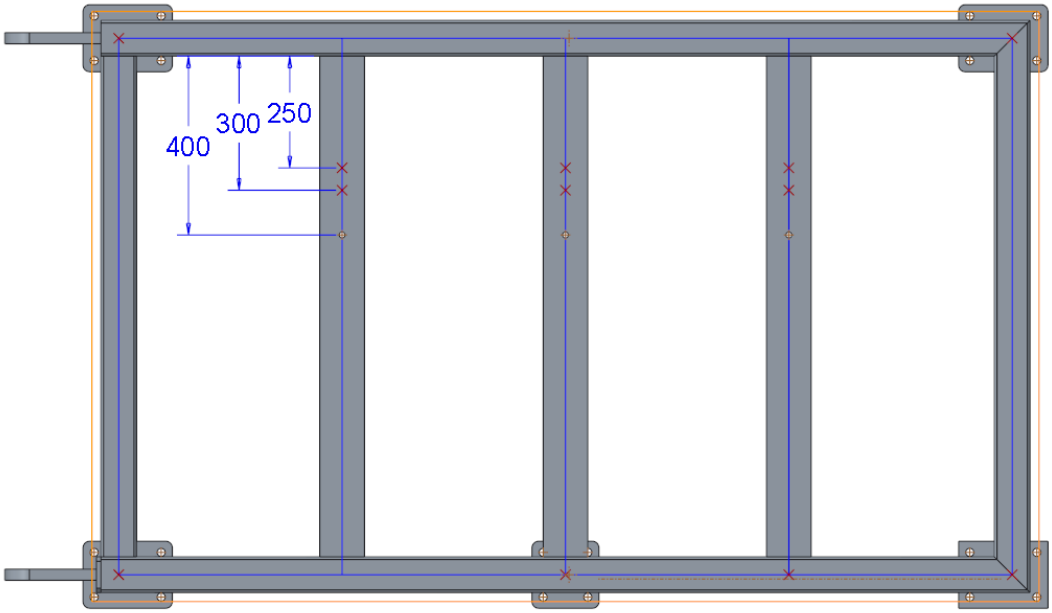
4.1 Overview

AFX also includes a full version of **Intelligent Fastener (IFX).** It is a very powerful tool to define screw and dowel pin connections and is delivered with a big library of standard parts such as screws, nuts, washers and dowel pins. Besides assembling parts **IFX** also takes care of creating holes and counter bores in the parts. As placement reference points, axes or holes can be used. If desired you can configure **IFX** to create the holes without external references.

In this training only screw connections are explained. Dowel pin connections work similar. Read more about dowel pins in the **Creo Parametric** help.

To prepare the next lessons activate the 1st of the three patterned rectangular beams and create:

- a **datum point** feature with **two points**.
- a **through hole** with diameter **[13mm]**.



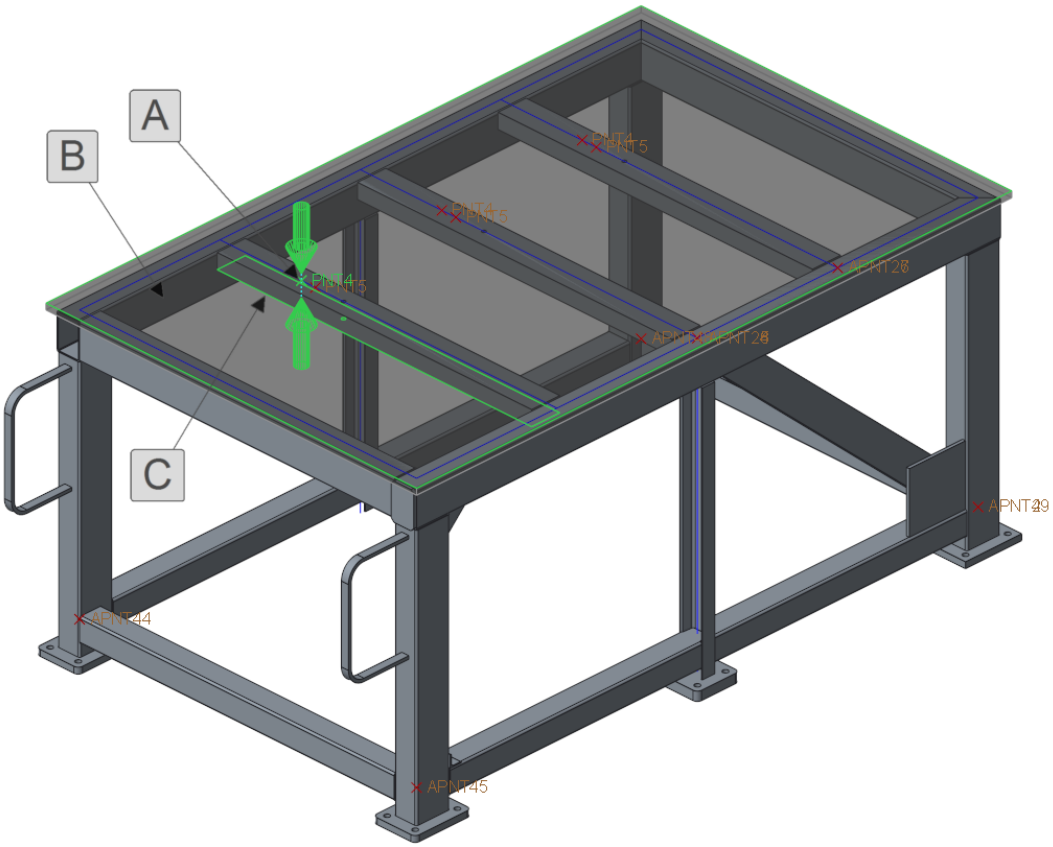
4.2 Assemble fasteners on Points/Axis

To assemble a screw connection with **IFX** you need to select a datum point, axis or hole as placement reference, the surface where the screw head shall be placed and the surface where the nut or thread shall be placed.

As a first example assemble fasteners on datum points.

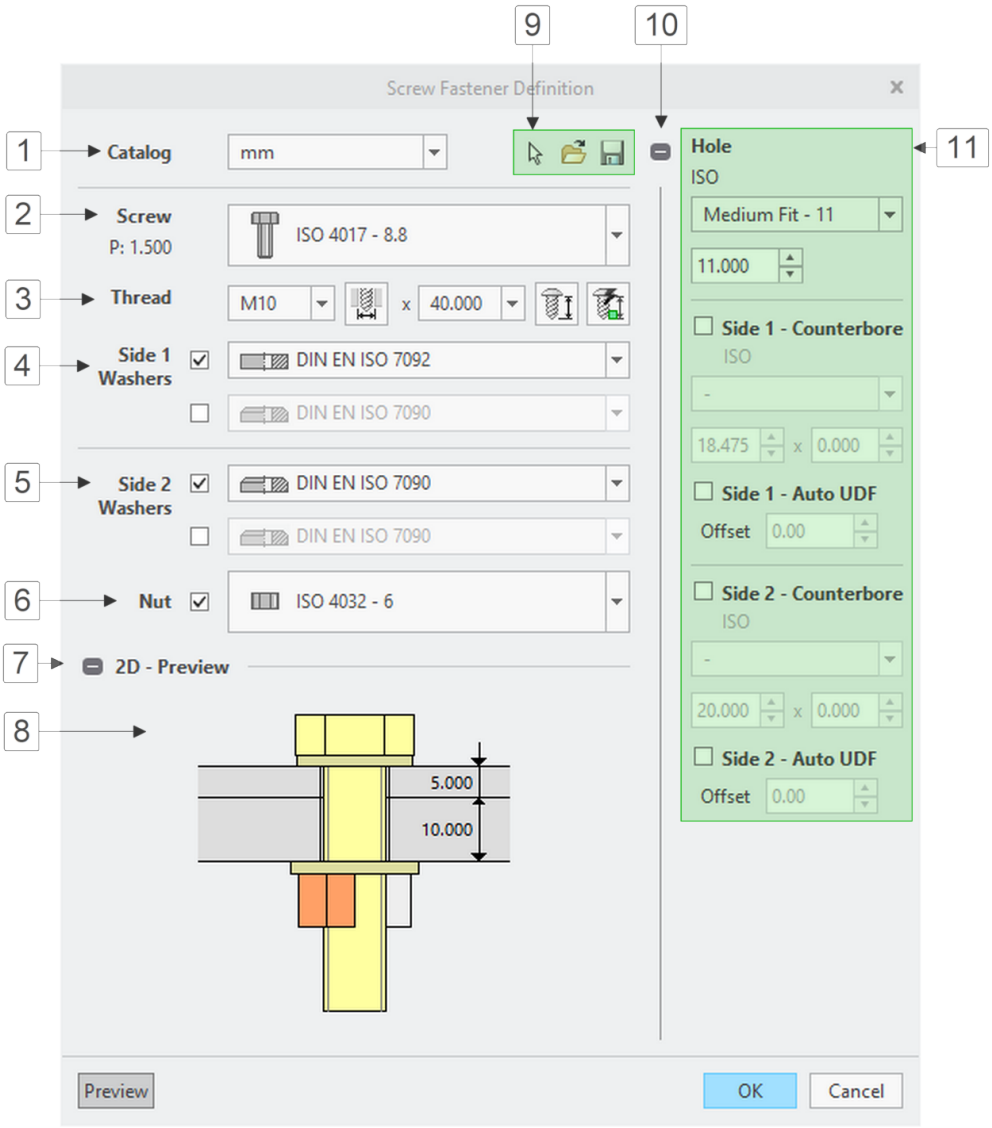
1. Switch to the **Tools Tab** in the ribbon to access the **IFX** functionality.
2. Click **Assemble on point or axis**.
   - The **Select References** dialog box shows up.
3. Select datum point PNT4 as **Position Reference [A]**.
4. Select the upper surface of the plate **[B]** as **Screw Head Placement Surface**.

5. Select the lower surface of the rectangular beam **[C]** as **Nut/Thread Placement surface**.
6. Click **[OK]** or middle mouse button to get continue.



Hint 12 — Connection types. You can also determine the resulting connection type with the help of the indication arrows. Opposing arrows will result in Screw/Nut and aligning arrows in Screw/Thread connection.

The **Screw Fastener Definition** dialog box for configuring fasteners opens. As the screw head and nut/thread surface are opposing, **IFX** assumes that you want to create a fastener connection with nut. If both surfaces are aligned, **IFX** assumes that you want to create a connection with thread. The **Screw Fastener Definition** dialog box looks like this:

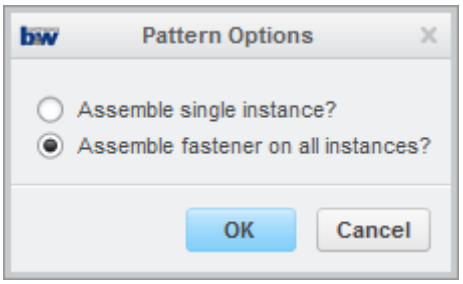


1. **Select catalog**
2. **Screw standards**
3. **Select thread and length**
   -  Measure diameter
   -  Calculate screw length
   -  Toggle automatic screw length calculation
4. **Side 1 washers**
5. **Side 2 washers**
6. **Nut standards**
7. **Toggle 2D-Preview**
8. **2D-Preview**

9. **Load and save definitions**
10. **Toggle hole definition**
11. **Hole definition area**
    - Hole diameter
    - Counterbore options
    - Automatic UDF options

To complete the fastener proceed as follows:

1. Select the screw type **[ISO 4762 – 8.8]**.
2. Select screw diameter **[M10]**.
3. Activate 1st washer on nut side and select type **[DIN 125-1-A]**.
4. Select nut type **[ISO 4032 - 8]**.
5. Click  to perform a screw length calculation or click  to permanently enable automatic length calculation.
6. Click **[OK]** to close the dialog.
    - As the selected datum point belongs to a feature with multiple points, the **Pattern Options** dialog box opens.

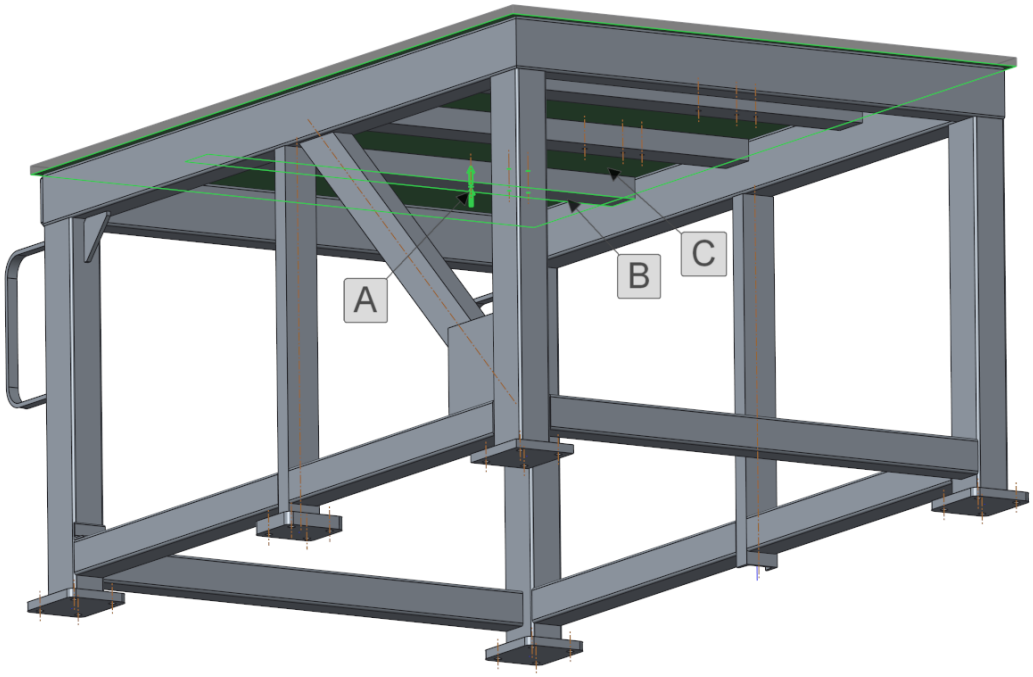


7. Check the option **[Assemble fastener on all instances?]** to assemble the screw connection on all points of the feature.
    - The screw connection is assembled with all components and holes on both points.

Now assemble a screw connection on an **axis**.

1. Click **Assemble on point or axis**.
2. Select the axis of the hole created above as placement reference **[A]**.
3. Select the lower surface of the rectangular beam **[B]** as screw head surface.
4. Select the lower surface of the plate **[C]** as placement surface for nut/thread.
    - As both surfaces have same orientation, **IFX** assumes that you want to have a connection with thread. Therefore the nut area in the dialog box is inactive.
5. Select screw type **[ISO 4014 - 8.8]**.
6. Activate option **[Through Thread]** for the thread hole.

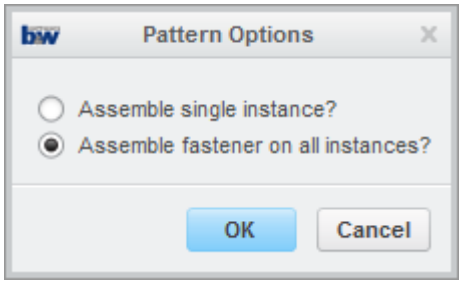
7. Click  to perform a screw length calculation.
8. Click **[OK]** to close the dialog and assemble the screw connection.
   - The screw connection is assembled on the axis with all components and missing holes.



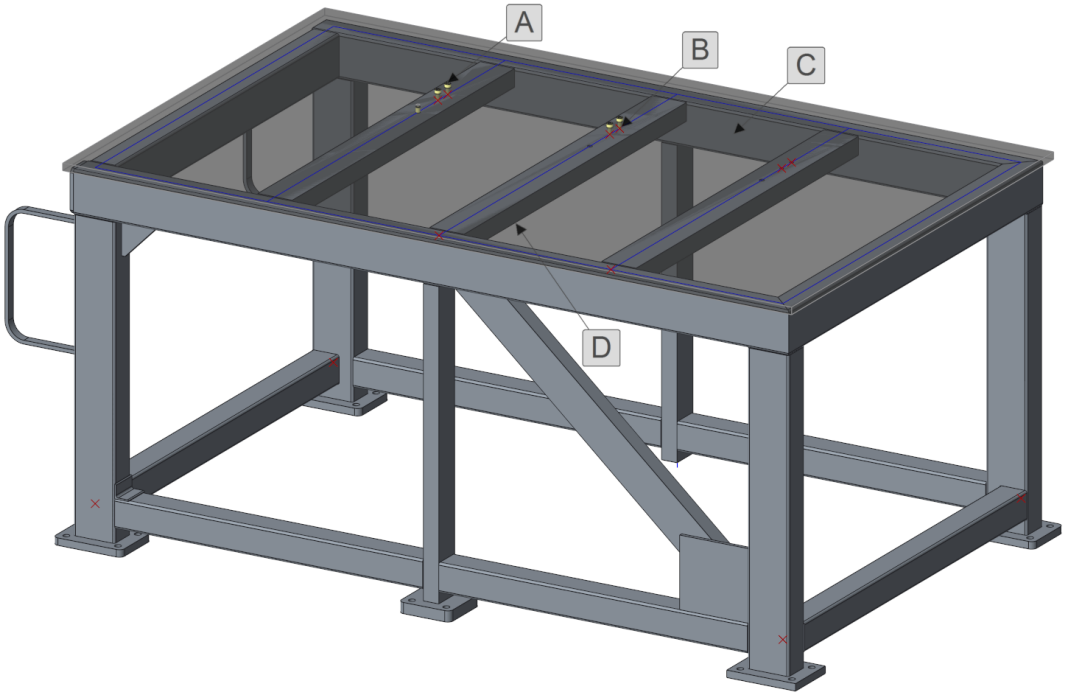
4.3 Reassemble fasteners

Similar to profiles and connector elements existing screw connections can be reassembled.

1. Click **Reassemble**.
2. Select a component of the screw connection assembled as first connection in the previous chapter **[A]**.
   - The **Select References** dialog box shows up.
3. Select the datum point PNT4 **[B]** on the second rectangular beam of the pattern as placement point.
4. If it is not already preselected select the upper surface of the plate **[C]** as screw head surface.
5. If it is not already preselected select the lower surface of the rectangular beam **[D]** as surface for the nut.
6. Click **[Apply]** or middle mouse button to permanently create connection and continue with creating next connection or click [OK] to permanently create connection and then quit the process.
   - As the new datum point belongs to a feature with multiple points, the **Pattern Options** dialog box opens.



7. Check the option **[Assemble fastener on all instances?]** to assemble the screw connection on all points of the feature.
   - The screw connection is assembled again using identical configuration (same parts, same holes).

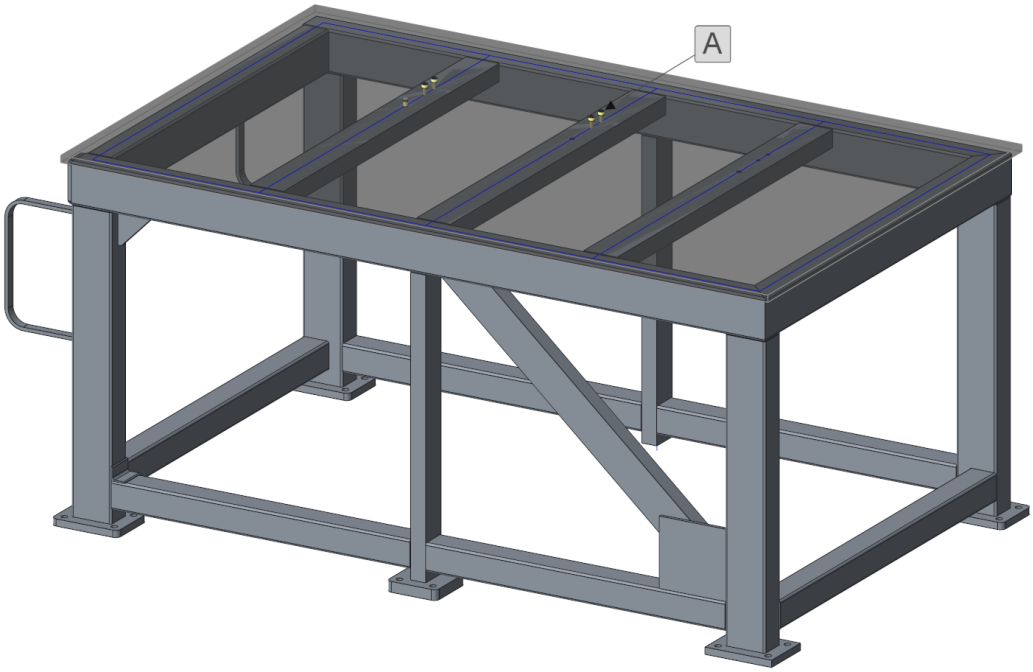


4.4 Redefine a Screw Connection

Screw connections assembled with **IFX** can be redefined (however no new references can be set).

1. Click **Redefine**.
2. Select a component of the screw connection to redefine **[A]**.
   - As the screw connection is reassembled multiple times the **Redefine Options** dialog box appears.
3. Check option **Redefine all elements** and then click **[OK]**.
   - The **Screw Fastener Definition** dialog box for configuring fasteners opens.
4. Select screw type **[ISO 4014 - 8.8]**.

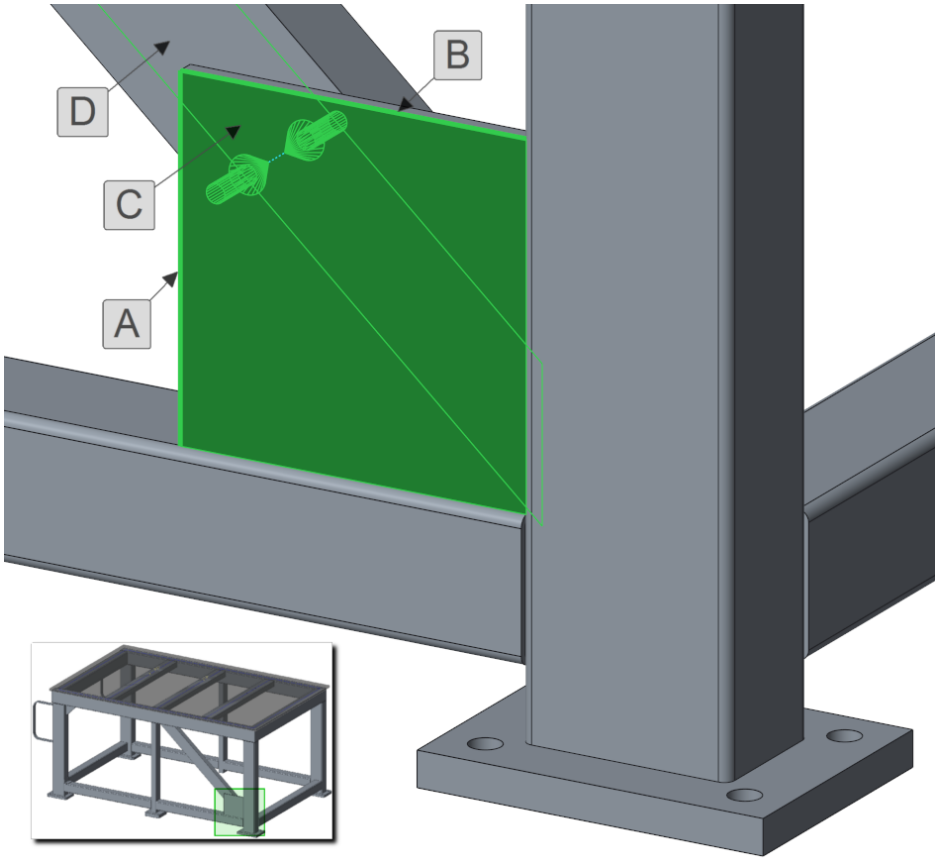
5. Select screw diameter **[M12]**.
6. Activate check box for **[Counterbore]** on screw side.
7. Click  to perform a screw length calculation.
8. Click **[OK]** to close the dialog and to redefine the screw connection.
   - The screw connection is redefined on all locations. All screws, nuts, washers and holes are modified and the hole type is changed to counter bore hole.



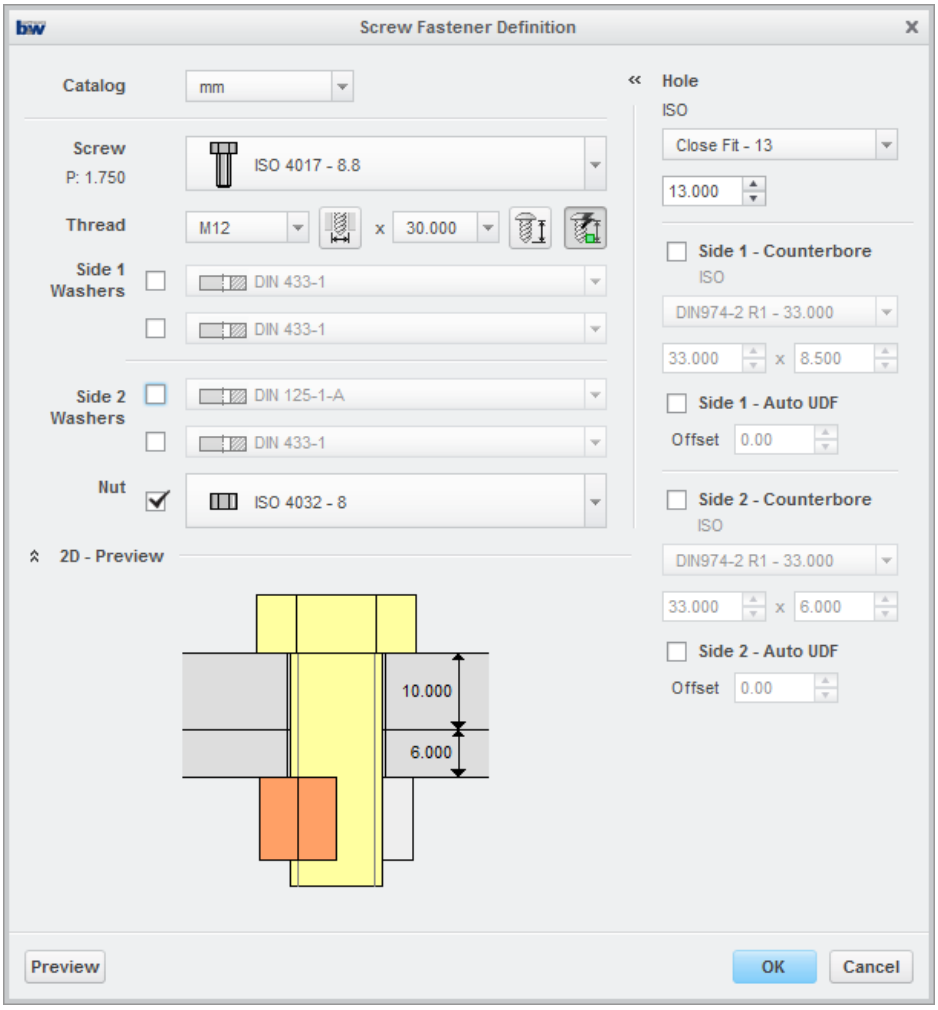
4.5 Assemble Fasteners by Mouse Click

Besides assembling screw connections on points/axes it is possible to assemble screw connections by mouse click.

1. Click **Assemble by mouse click**.
   - The **Select References** dialog box shows up.
2. Select the left side edge of the plate **[A]** as 1st dimension reference for the screw connection.
3. Select the upper edge of the plate **[B]** as 2nd dimension reference for the screw connection.
4. Select the front surface of the rectangular plate **[C]** as screw head surface.
5. Select the rear inside surface of the diagonal U-profile **[D]** as placement surface for the nut.
6. Click **[OK]** or middle mouse button to get continue.
   - The **Screw Fastener Definition** dialog box for configuring fasteners opens.



7. Configure the fastener as shown in the image.



8. Click **[OK]** to close the dialog box. Now you can see a preview of the screw connection on the mouse pointer.
9. Move the mouse to the desired position and click left mouse button.
   - The screw connection is assembled. If the mouse pointer is at a position where the fastener axis does not intersect both surfaces the fastener can not be placed and the preview has a different color.
10. Move the mouse pointer to a different location and click left mouse button.
    - The screw connection is created again.
11. Quit the process with middle mouse button.

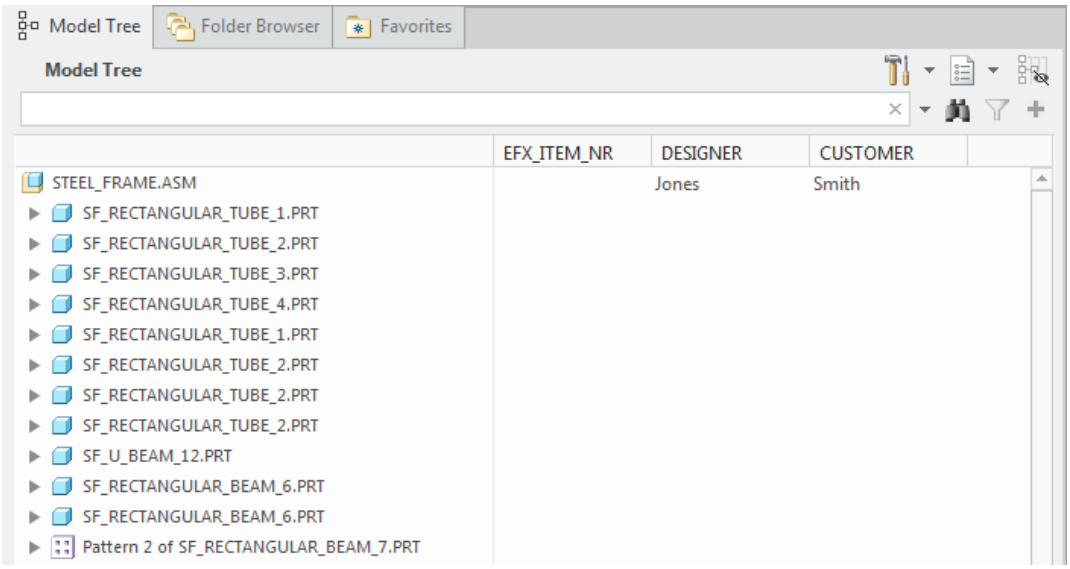
5

Drawing and BOM Creation

Overview
Project Parameters
Creation of BOMs and other reports
Drawing automation tools

5.1 Overview

AFX offers several functions to simplify creation of **drawings** and **BOMs**. This chapter will address these tools. Before performing the next steps of this training guide in **Creo Parametric** model tree load tree configuration file `tree.cfg`. You can see columns `EFX_ITEM_NR`, `DESIGNER` and `CUSTOMER`.

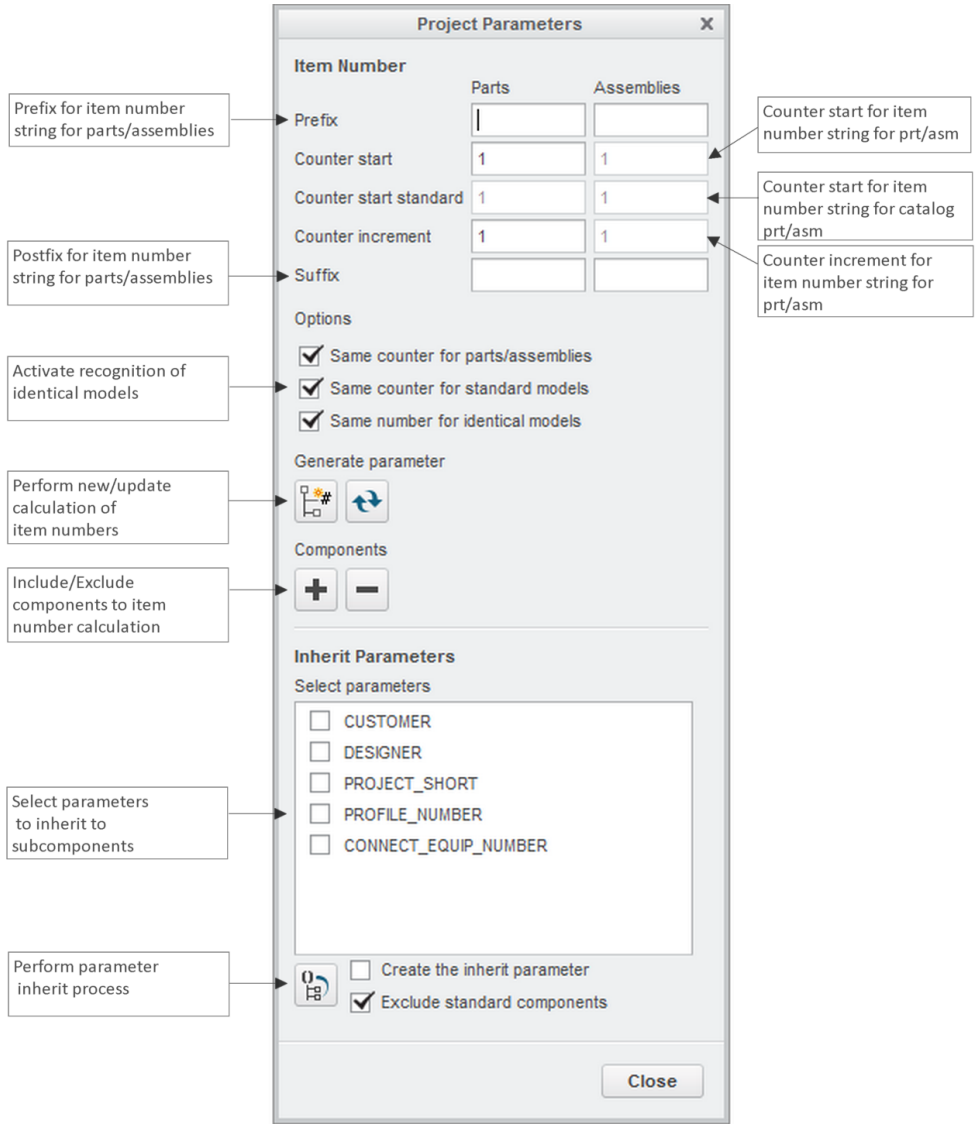


5.2 Project Parameters

AFX has two functionalities for handling project specific parameters for **AFX** parts and subassemblies:

- Inheriting project parameters (like customer, contract number etc.) from top level assembly to parts and subassemblies
- Calculation of Item numbers of parts or subassemblies in a project

These functions can be found in the **Project parameters** dialog box.

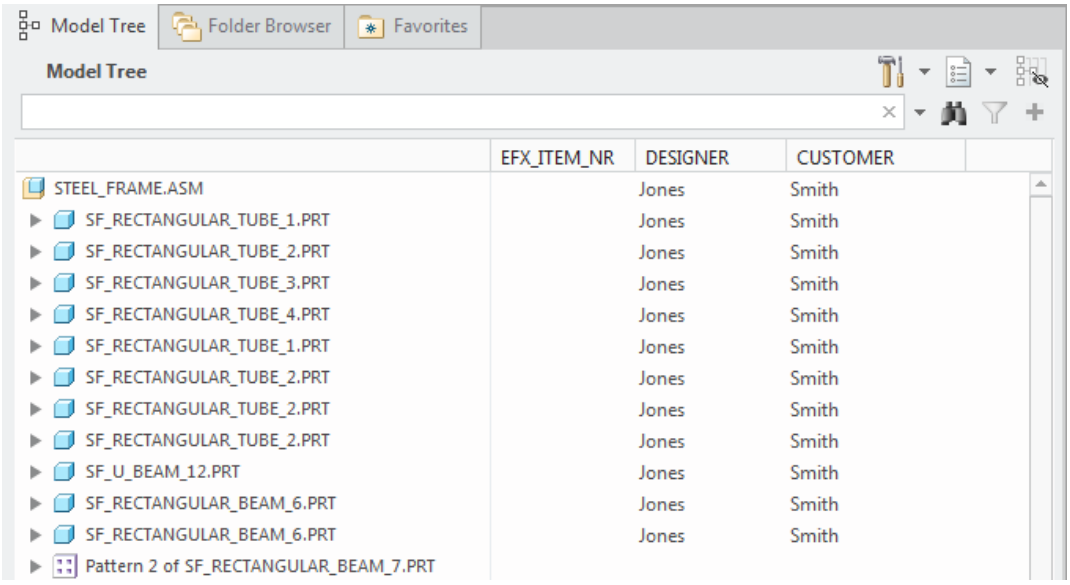


Inherit Parameters

This functionality allows you to inherit project relevant parameters (like customer name, project number, designer etc.) from top level assembly to all project specific sub components. To copy the value of top level assembly parameters DESIGNER and CUSTOMER to all project specific sub components proceed as follows.

1. Click  **Project Parameters**.
   - The **AFX** Project parameters dialog box opens.
2. In Inherit parameter area check the parameters **[CUSTOMER]** and **[DESIGNER]** as parameters to inherit.

3. Check option **[Create the inherit parameter]** to create the parameter in the sub components.
   - If you know that the parameters already exist, it is not required to check this option.
4. Click  to inherit the parameters.
   - In the model tree you can see that the parameters are created in all project specific sub components.

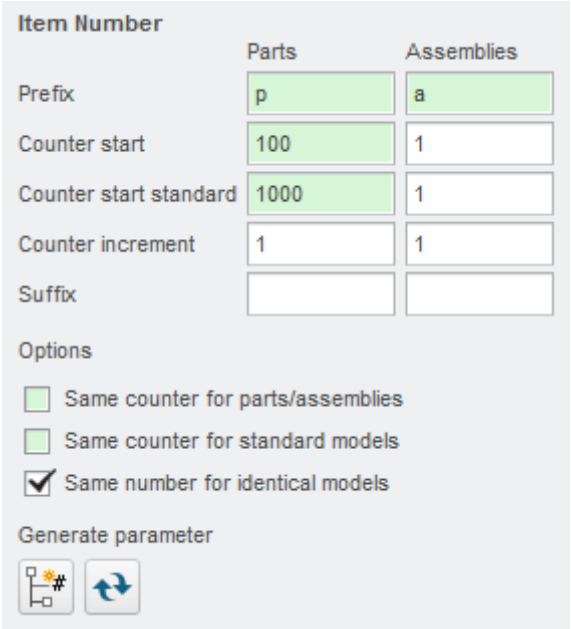


Calculate Item Numbers

This functionality allows you to calculate item number strings for all sub components of top level assembly. Item numbers are similar to the repeat index of a **Creo Parametric** repeat region but have some advantages compared to them:

- Item numbers are available as model parameters for project specific models therefore they can be shown on single part drawings or views.
- Identical models can have the same item number even if the model names are different. **AFX** can perform a recognition of identical models and will assign the same item number if this is the case.
- Item number notes can be shown on multiple views for same model.

To perform an item number calculation for the current assembly proceed as follows.



1. Click  **Project Parameters**.
   - The **AFX** Project parameters dialog box opens.
2. Uncheck option **[Same counter for parts/assemblies]** to use different counter for parts and assemblies.
3. Uncheck option **[Same counter for standard models]** to use different counter for project specific and catalog models.
4. Enter letter **[p]** in **Prefix/Parts** and letter **[a]** in **Prefix/Assemblies** input filed.
5. Enter **[100]** into **Count start/Parts** input field
6. Enter **[1000]** into **Count start standard/Parts** input field.
7. Check option **[Same number for identical models]** to perform recognition of identical models.
8. Click  to perform a new item number calculation.
9. Accept question **Overwrite existing item numbers (y/n)?** with **[y]**.

The item number creation is performed. The item numbers are created as component feature parameters `EFX_ITEM_NR` and for project specific models additionally as model parameters `EFX_ITEM_NR`. You can see that:

- Item numbers for project specific parts have prefix **p** and start with **100**
- Item numbers for catalog parts (like screws, nuts etc.) have prefix **p** and start with **1000**.
- Identical models have the same item number even if the **Creo Parametric** model name differs (e.g. the 4 base plates).

5.3 Creation of BOMs and other reports

Open the drawing steel_frame. **AFX** supports creation of BOMs using **Creo Parametric** repeat region functionality as well as other type of reports typical for beam structure designs like stock length tables and optimized cut lists.

Creating BOMs

BOM information is added to components automatically, when assembling components with **AFX**. In standard configuration the BOM information is in parameters NAMING and DESIGNATION, however this can be configured (see chapter 8.2). BOM information is available in the parts as regular parameter, so BOMs can be created using regular repeat region functionality. In the **AFX** installation directory sub folder demo you can find a **Creo Parametric** table file which fits to standard **AFX** BOM parameters. If you want to create BOMs using your own custom tables, you have to customize **AFX** BOM parameters (see chapter 8.2) to fit to the parameters of your BOM tables repeat region.

To create a BOM of the assembly with **Creo Parametric** functionality place the table profil_bom_english_new.tbl on the drawing.

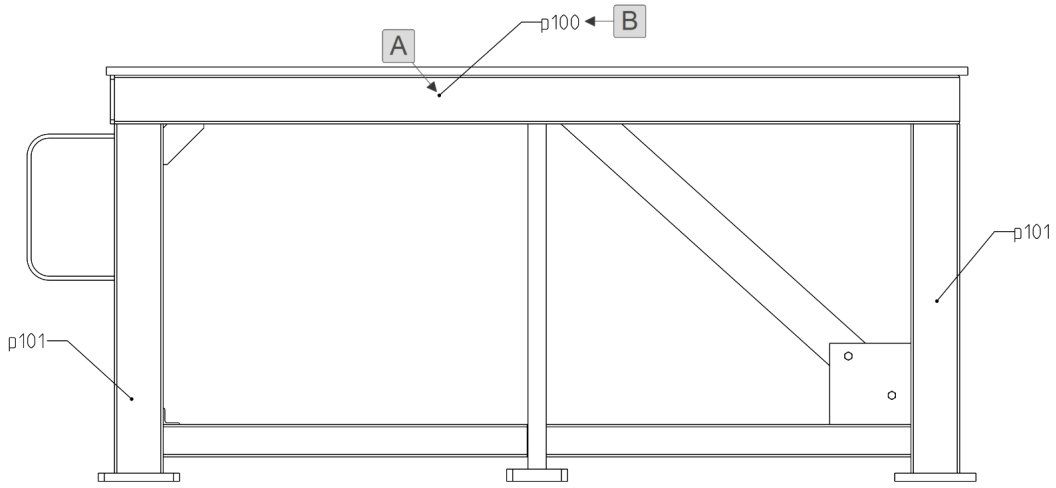
This table does **not** show repeat region parameter rtp.index in column **Pos** but shows parameter EFX_ITEM_NR created in the previous chapter.

Pos	Qty	Name	Description
p100	2	RECTANGULAR TUBE	DIN 2395 120x80x4.00 x 2080
p101	4	RECTANGULAR TUBE	DIN 2395 120x80x4.00 x 860
p102	1	RECTANGULAR TUBE	DIN 2395 120x80x4.00 x 1120
p103	1	RECTANGULAR TUBE	DIN 2395 120x80x4.00 x 1280
p104	2	FLAT STEEL	DIN 1017 25x10
p105	3	FLAT STEEL	DIN 1017 100x50 x 1120
p106	1	U BEAM	DIN 1026 U 100 x 1158.9
p107	2	U BEAM	DIN 1026 U 80 x 850
p108	2	SQUARE PIPE	DIN 2395 80x80x4.00 x 934.5
p109	2	SQUARE PIPE	DIN 2395 80x80x4.00 x 897.5
p110	2	SQUARE PIPE	DIN 2395 80x80x4.00 x 1120
p111	3	PLATE	150x200x20
p112	1	PLATE	150x200x20
p113	1	PLATE	150x150x30
p114	1	angle	DIN 1028 L 40x4
p115	1	PLATE	200x200x10
p116	1	PLATE	100x100x10
p117	1	ENDCAP_RECTANGULAR	120x80x10
p118	1	PLATE	1320x2120x20
p1000	1	Hexagon Head Screw	ISO 4014 - M12 x 65 - 8.8
p1001	4	Hexagon Head Screw	ISO 4014 - M12 x 80 - 8.8
p1002	4	Washer	DIN 125-1-A - 12
p1003	6	Hexagon Nut	ISO 4032 - M12 - 8
p1004	2	Hexagon Head Screw	ISO 4017 - M12 x 30 - 8.8

Place Item number Notes

If parameter `EFX_ITEM_NR` is shown in BOM instead of repeat region parameter an automated placement of BOM-balloons using **Creo Parametric** functionality requires additional configuration. Therefore **AFX** provides functionality to easily place item number notes showing parameter `EFX_ITEM_NR`. To place notes with parameter EFX_ITEM_NR proceed as follows.

1. Click **Create item number note**.
2. Select part for which the note shall be placed **[A]**.
3. Pick location for note **[B]**.
   - The note is created. Repeat step 2 and 3 for other parts.



If you want to place BOM Balloons using **Creo Parametric** functionality proceed as follows.

- Select the BOM table.
- Click Properties and switch to tab **BOM Balloons**
- Select type custom.
- Click **[Browse...]** and select `AFX_POS.sym` in the **steel_frame_start** folder of this guide.
- Click **[OK]** to complete the definition.

It is now possible to place BOM Balloons with the **Creo Parametric** functionality.

Creation of stock length tables

The Stock length table lists the section type, size and total length required for each profile. This information is helpful for ordering raw stock material for profile sections.

To create a stock length table of the assembly proceed as follows.

1. Click  **Create a stock length table**.
2. Pick location for table.
   - The table is created as shown in next picture.

NAME	TYPE	SIZE	LENGTH
RECTANGULAR TUBE	DIN 2395	120x80x4.00	10000.000
FLAT STEEL	DIN 1017	100x50	3360.000
U BEAM	DIN 1026	U 100	1158.907
U BEAM	DIN 1026	U 80	1700.000
SQUARE PIPE	DIN 2395	80x80x4.00	5904.000
angle		DIN 1028 L 40x4	80.000

Creation of optimized cut lists

Optimized cut lists give information about how many raw material pieces with a standard length of 6000mm are required for each profile section and size and which part of the design to cut from which raw material piece to have minimum cut loss.

To create an optimized cut list of the assembly shown on the drawing proceed as follows.

1. Click  **Create a cut list**.
2. Enter number of table columns **[4]**.
3. Pick location for table.
4. Enter whether you want to have separate table for each profile section type and size. In this case enter **[y]**.
   - The table is created as shown in next picture.

RECTANGULAR TUBE DIN 2395 120x80x4.00			
1	SF_RECTANGULAR_TUBE_1 2080.0	SF_RECTANGULAR_TUBE_4 1280.0	SF_RECTANGULAR_TUBE_2 860.0
	SF_RECTANGULAR_TUBE_2 860.0	SF_RECTANGULAR_TUBE_2 860.0	
2	SF_RECTANGULAR_TUBE_1 2080.0	SF_RECTANGULAR_TUBE_3 1120.0	SF_RECTANGULAR_TUBE_2 860.0

FLAT STEEL DIN 1017 100x50			
1	SF_RECTANGULAR_BEAM_6 1120.0	SF_RECTANGULAR_BEAM_6 1120.0	SF_RECTANGULAR_BEAM_6 1120.0

U BEAM DIN 1026 U 100			
1	SF_U_BEAM_16 1158.9		

U BEAM DIN 1026 U 80			
1	SF_U_BEAM_8 850.0	SF_U_BEAM_8 850.0	

SQUARE PIPE DIN 2395 80x80x4.00			
1	SF_SQUARE_TUBE_14 1120.0	SF_SQUARE_TUBE_14 1120.0	SF_SQUARE_TUBE_11 934.5
	SF_SQUARE_TUBE_11 934.5	SF_SQUARE_TUBE_12 897.5	SF_SQUARE_TUBE_12 897.5

angle DIN 1028 L 40x4			
1	SF_1_ANGLE_1 80.0		

The information in the table is as follows. From section **RECTANGULAR TUBE DIN 2395 120x80x4.00** two **6000mm** raw material pieces are required. From 1st raw material cut:

- `SF_RECTANGULAR_TUBE_1` with length 2080
- `SF_RECTANGULAR_TUBE_4` with length 1280
- `3 x SF_RECTANGULAR_TUBE_2` with length 860

From 2nd raw material cut:

- `SF_RECTANGULAR_TUBE_1` with length 2080
- `SF_RECTANGULAR_TUBE_3` with length 1120
- `SF_RECTANGULAR_TUBE_2` with length 860

5.4 Drawing automation tools

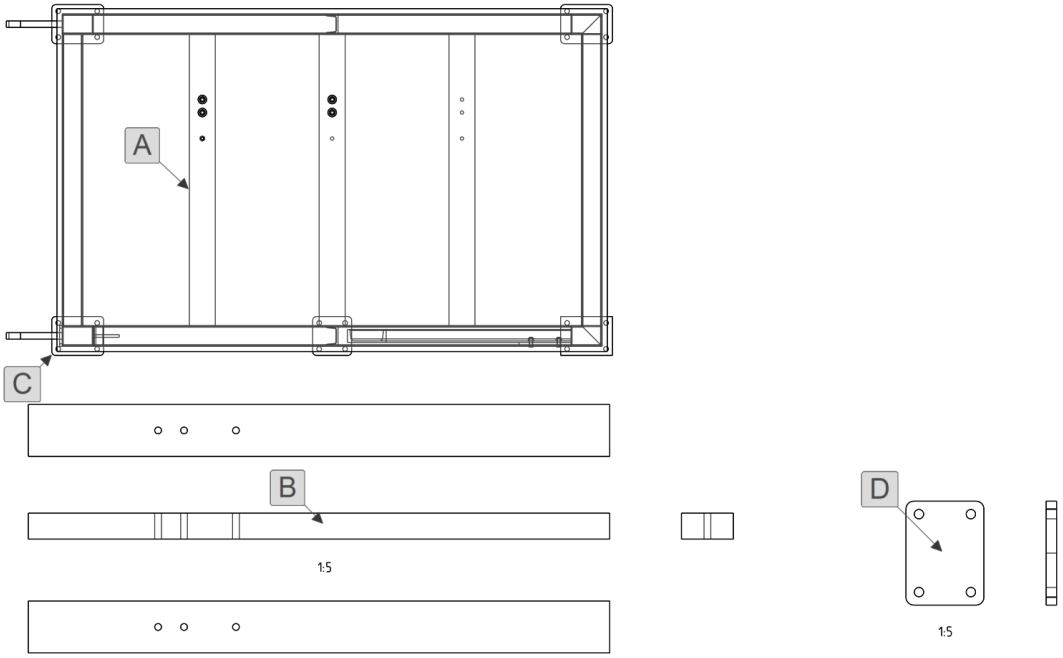
AFX supports creation of drawings for profile and plate parts as views on the assembly drawing as well as separate drawings.

Create profile and plate part views on assembly drawings

AFX offers an easy to use and quick functionality to create profile and plate part views on assembly drawings.

1. Click  **Create component views**.
2. Answer question Enter detail view scale with **[0.2]**.
3. Select one of the 3 patterned rectangular beams **[A]**.

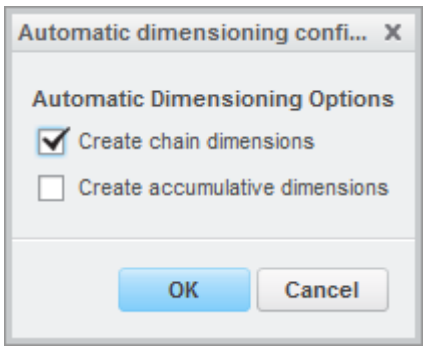
4. Pick location for part view **[B]**.
   - The views for the profile are created as defined in the `drawing_setup.txt` file (see 5.3.4) used for this part.
5. Select one of the 4 corner base plates **[C]**.
6. Pick location for part view **[D]**.
   - The views for the plate are created.



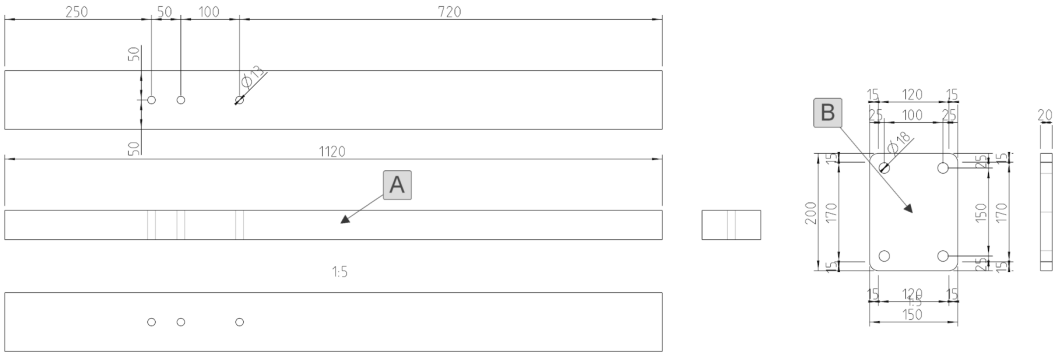
Dimension profile and plate part views on assembly drawings

AFX offers an easy to use and quick functionality to automatically dimension profile and plate part views. The dimension style is typical for simple structural steel parts, like plates and profiles with holes or simple cutouts. It does not work well for more complex parts. If the dimension style is not as desired, it can be modified using regular Creo Parametric functionality. To automatically dimension the profile and plate part views created in previous chapter

1. Click **Dimension the part automatically**.
   - The **Dimension the part automatically** dialog box opens.



2. Check option **[Create chain dimensions]**.
3. Uncheck option **[Create accumulative dimensions]**.
4. Click **[OK]** to accept the definition.
5. Select view of profile part **[A]**.
   - Dimensions for the profile part are created.
6. Select view of plate part **[B]**.
   - Dimensions for the plate part are created.

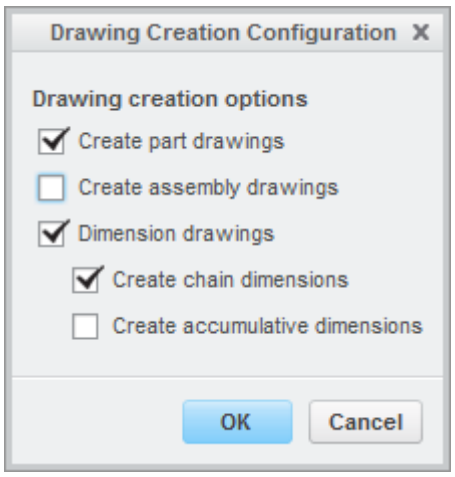


Automatic creation of component drawings

AFX offers functionality to automatically create separate drawings for all profiles, plate parts and project sub-assemblies belonging to an **AFX** assembly. Furthermore it is possible to add dimensions to profiles and plate parts automatically.

To automatically create profile and plate part drawings of all profiles and plates of `steel_frame.asm` proceed as follows.

1. Open the main assembly `steel_frame.asm`.
2. Click **Create component drawings**.
   - The Drawing creation configuration dialog box opens.



3. Check option **[Create part drawings]**.
4. Uncheck option **[Create assembly drawings]**.
5. Check option **[Dimension drawings]**.
6. Check option **[Create chain dimensions]**.
7. Uncheck option **[Create accumulative dimensions]**.
8. Click **[OK]** to accept the definition.
    - All profile part and plate drawings are created and dimensioned as defined in the drawing_setup.txt files (see 5.3.4).

Customizing the look of single part views and single part drawings

The look of single part views and single part drawings created by **AFX** can be customized in a file named drawing_setup.txt. As drawing setup usually depends on the type of component the drawing_setup.txt files are located in **AFX** library and can be defined for single library components or library component folders or library component types. The search order for the drawing_setup.txt file for a specific **AFX** library component is as follows.

1. File named `library_component_name_drawing_setup.txt` in library folder of component, e.g. `afx_install_dir/parts/profiles/steel_beams_mm/i_beam_drawing_setup.txt`
2. File named `drawing_setup.txt` in library folder of component, e.g. `afx_install_dir/parts/profiles/steel_beams_mm/drawing_setup.txt`
3. File named `drawing_setup.txt` in **AFX** component library type, e.g. `afx_install_dir/parts/profiles/drawing_setup.txt` for all profile parts.

The content of a `drawing_setup.txt` file looks as follows.

```
DRAWING_FORMAT_NAME efx_a3

GENERAL_VIEW +YZ

TOP_VIEW 1

RIGHT_VIEW 1

BOTTOM_VIEW 1
```

DRAWING_FORMAT_NAME	means the name of the drawing format to be used for single part drawing creation.
GENERAL_VIEW	means the name of a model view which shall be used as general (base) view of the drawing.
TOP_VIEW, RIGHT_VIEW, BOTTOM_VIEW, LEFT_VIEW, TOP_TOP_VIEW, BOTTOM_BOTTOM_VIEW	means that a top, bottom, right, left etc. view of the general view shall be created. The number controls the distance between views relative to view size.

When placing the views **AFX** automatically selects proper drawing scale by choosing from a list of drawing scales in file

`afx_install_dir/configuration/drawing_scale_list.txt`. Instead of defining drawing format and views for the drawing also a drawing template can be used by line `DRAWING_TEMPLATE_NAME`.

Notes:

6

Design Assemblies with Flat Plates

Overview
Plates on curves
Plates on points
Plates on planar
Plate Joints

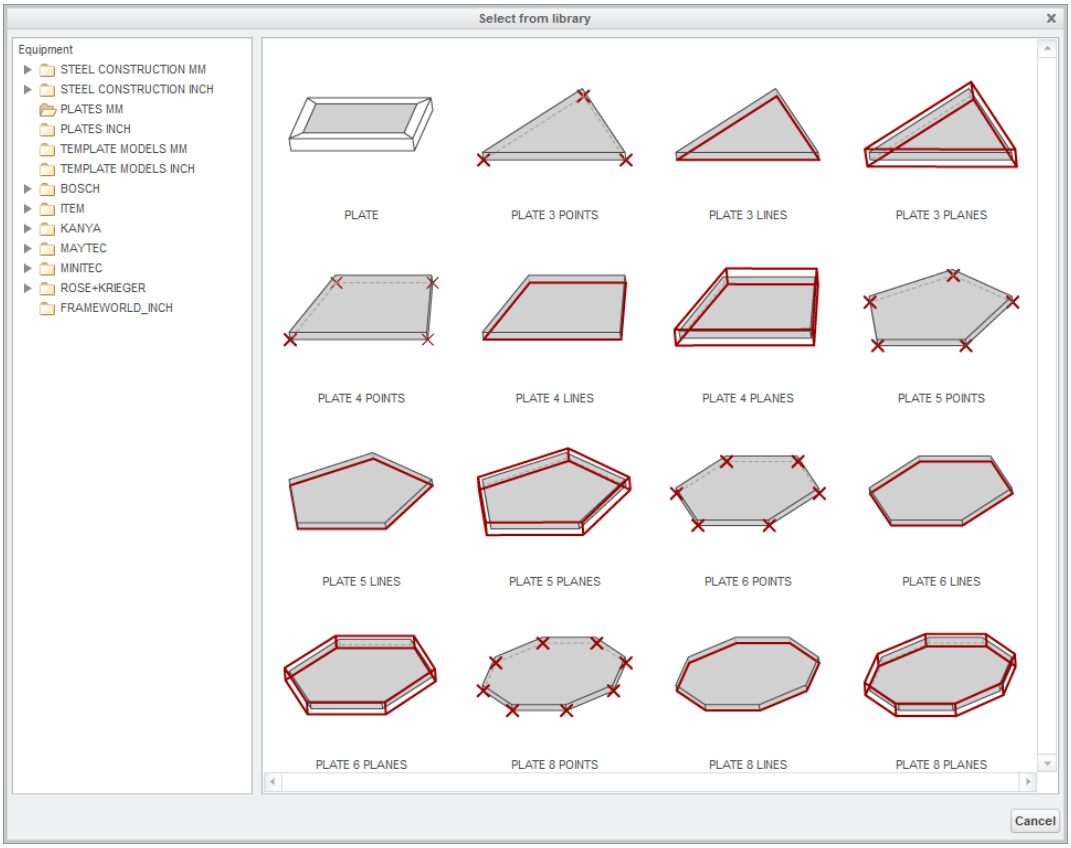
6.1 Overview

AFX has an extended library of flat, N-sided plates which allow you to design simple weldments of flat plates (like shown in next picture) with a top down design process similar to design of frames with beams:

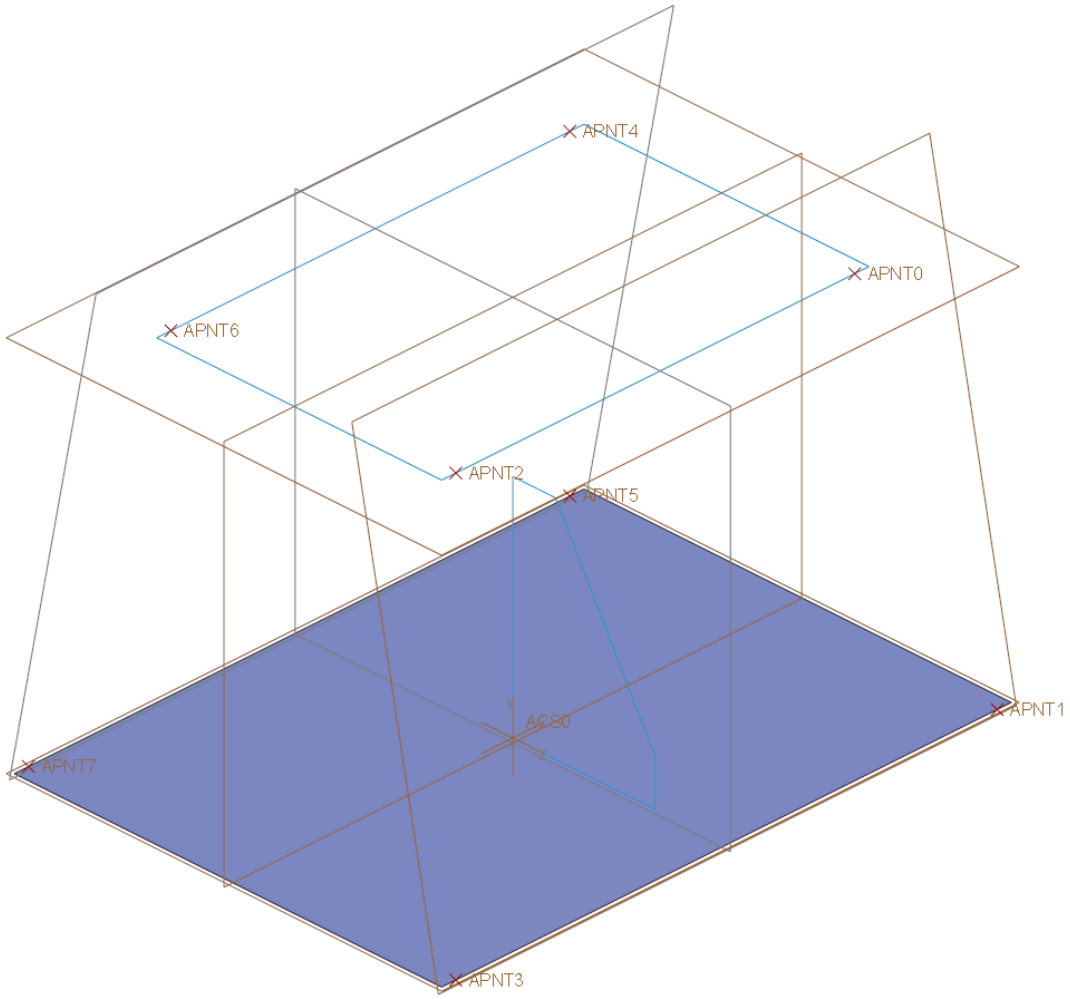
- Create a skeleton with curves, planar surfaces, points etc.
- Assemble flat plates using the curves, planar surfaces, points etc. as references.
- Define joints for side surfaces of plates if required.

It is possible to combine **AFX** flat plates with beams and also regular **Creo Parametric** parts in one assembly.

AFX flat plate library looks as shown in next picture. You can see plates with different number of sides which can be placed using lines, points or surfaces as reference.



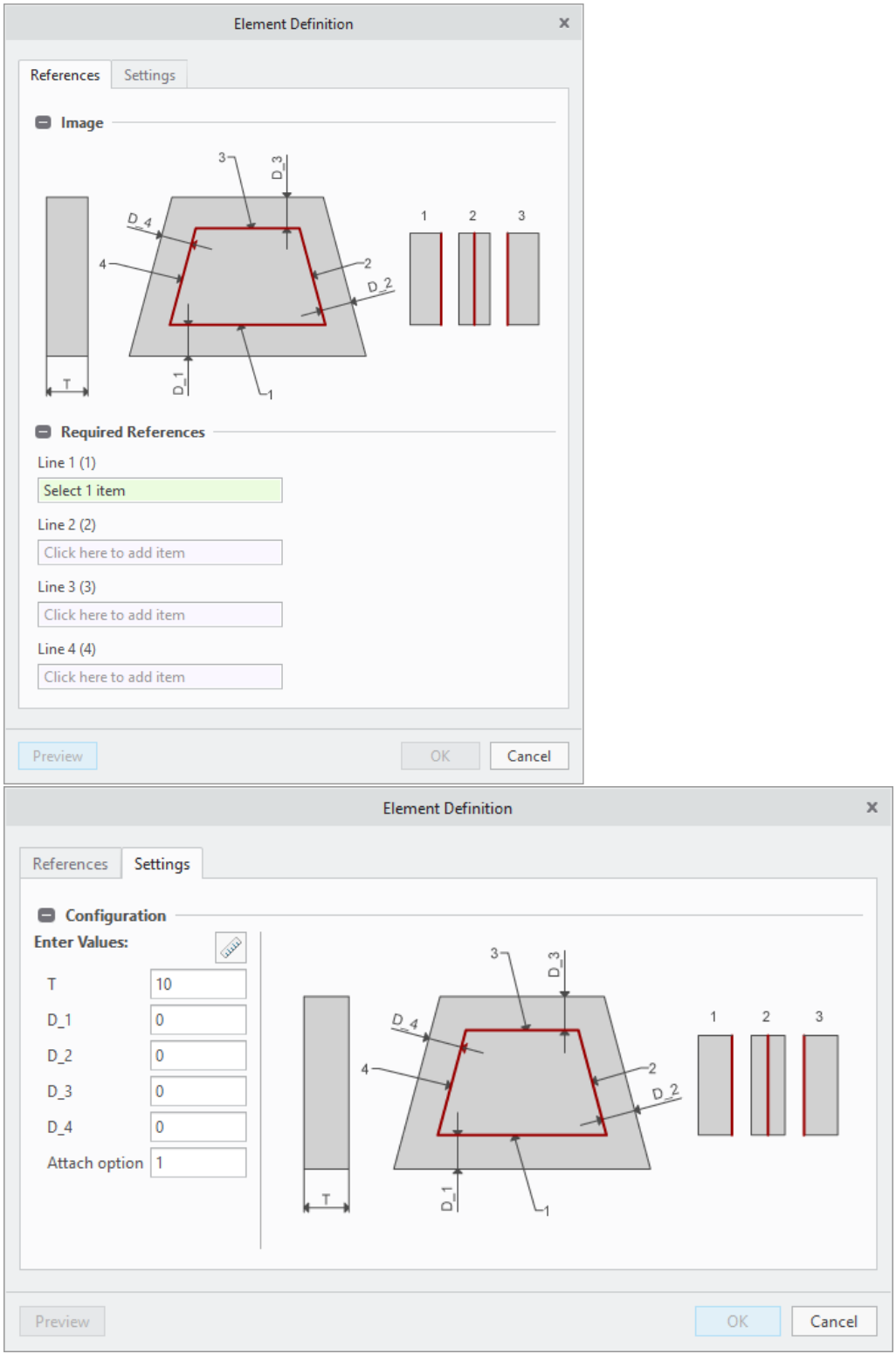
Open assembly `plate_weldment.asm` which already contains a skeleton of planar surfaces, curves, points etc. An **AFX** project already is defined for this assembly.



6.2 Plates on curves

To assemble the base plate of the weldment using the 4 edges of the bottom planar surface proceed as follows.

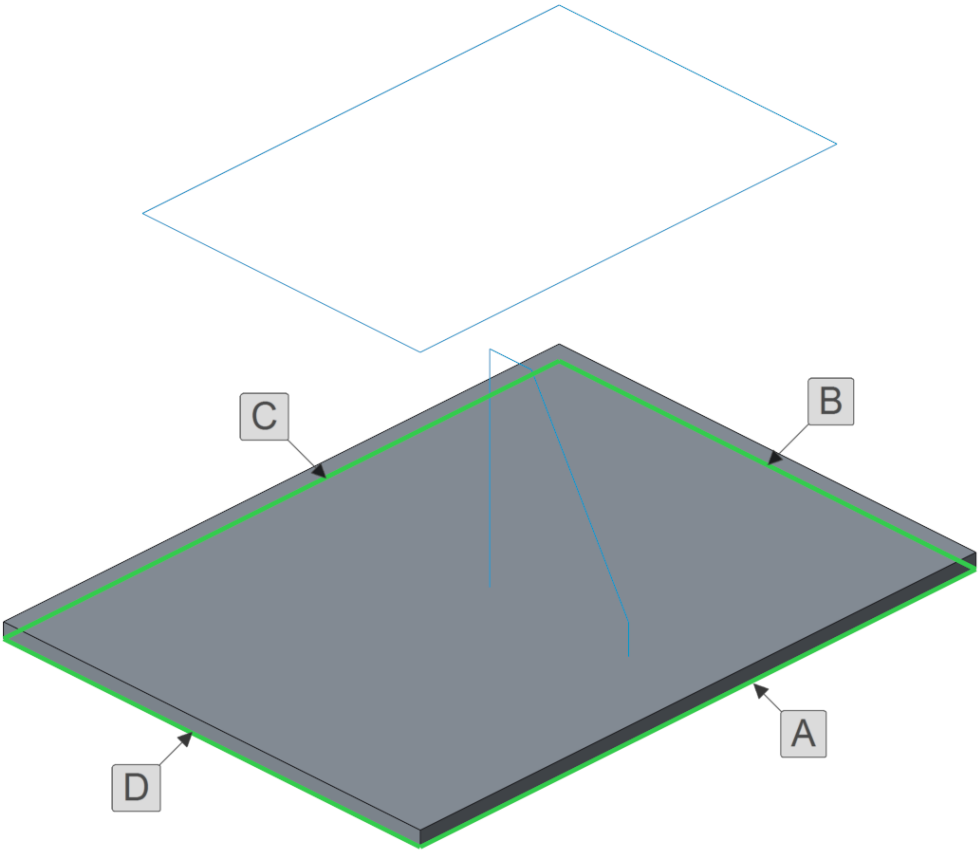
1. Click **New Equipment Elements**.
2. Click to open **Select from library** dialog box.
3. Select **[PLATES MM] > [PLATE 4 LINES]**.
   - The **Element definition** dialog box of the **PLATE 4 LINES** opens.



Hint 13 — Tips for plates. You should be careful, when selecting references. Try to remember what curve you selected first to understand the possible modifications via the dimensions **D_1, D_2, D_3, D_4**. Also be sure to select the references in the correct order (here counter-clockwise) and to configure the **Attach option** accordingly. Unfortunately it is not possible to redefine the **Attach option** later on.

Now select the lines which determine the size of the plate in counter-clockwise order. The lines can be edges or curves. It is not possible to mix edges and curves as references for one plate.

4. As reference **Line 1 (1)** select the front edge of the lower datum surface **[A]**.
5. As reference **Line 2 (2)** select the right edge of the lower datum surface **[B]**.
6. As reference **Line 3 (3)** select the back edge of the lower datum surface **[C]**.
7. As reference **Line 4 (4)** select the left edge of the lower datum surface **[D]**.
8. Toggle to **Settings** tab and keep the default values in the dialog.
9. Click **[Preview]** or middle mouse button to get a preview.
10. Click **[OK]** or middle mouse button to close the **Element definition** dialog box.
    - The plate is assembled.

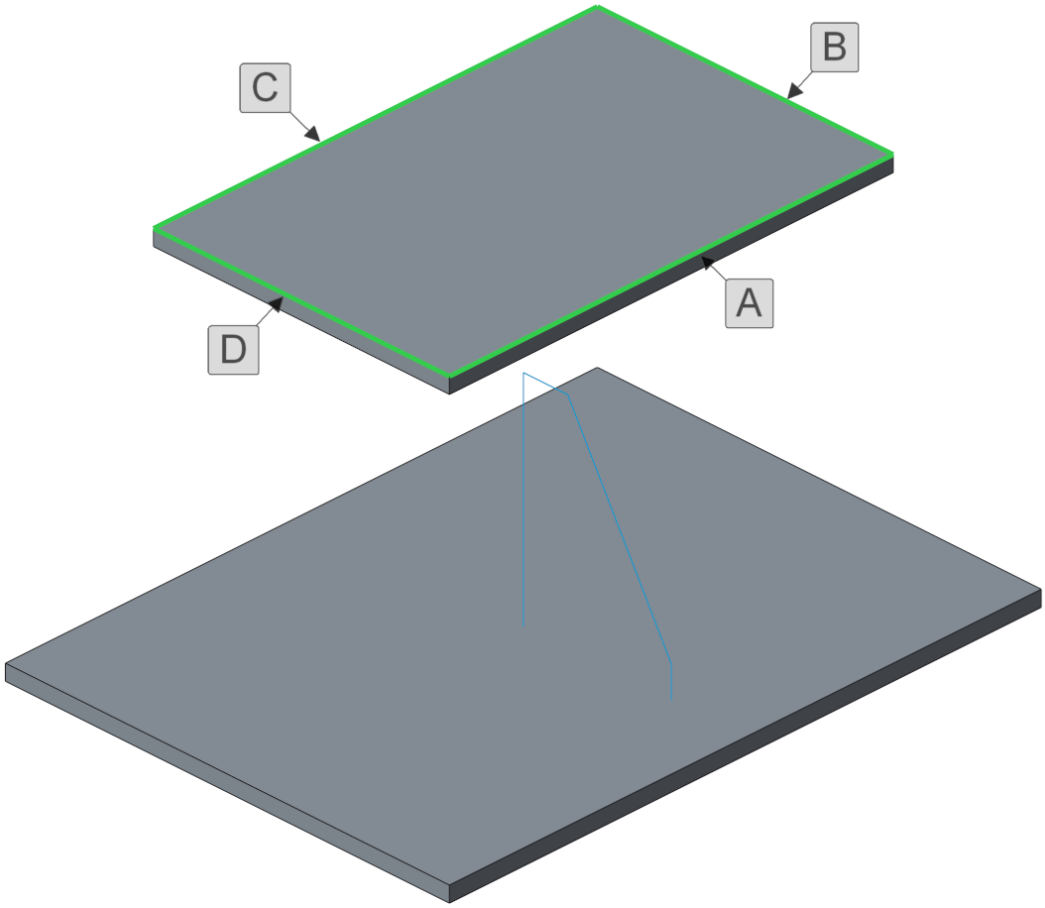


Hint 14 — Select all references. As you learned in one of the previous chapters you can also select all references in a loop. To do so press and select the references one after another. The required reference will also be displayed in the **Creo Parametric** message area.

To assemble the top plate of the weldment using the 4 curves on the top proceed as follows.

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

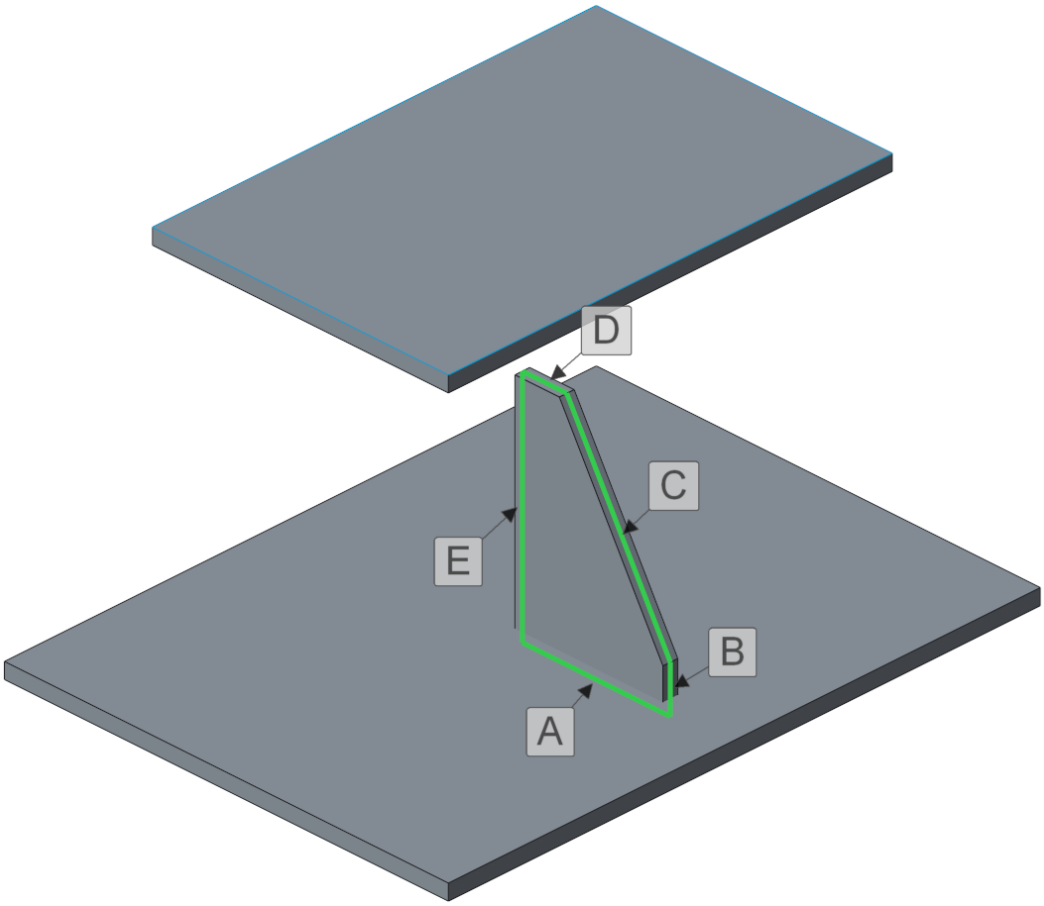
3. Select **[PLATES MM] > [PLATE 4 LINES]**.
   - The **Element definition** dialog box of the **PLATE 4 LINES** opens.
4. As reference **Line 1 (1)** select the front edge of the lower datum surface **[A]**.
5. As reference **Line 2 (2)** select the right edge of the lower datum surface **[B]**.
6. As reference **Line 3 (3)** select the back edge of the lower datum surface **[C]**.
7. As reference **Line 4 (4)** select the left edge of the lower datum surface **[D]**.
8. Toggle to **Settings** tab and keep the default values in the dialog.
9. Enter desired attach option **[3]** in field **Attach option** as the plate shall be placed below/ behind the plane defined by the 4 curves.
10. Click **[Preview]** or middle mouse button to get a preview.
11. Click **[OK]** or middle mouse button to close the Element definition dialog box.
    - The plate is assembled.



To assemble the stiffening plate of the weldment proceed as follows.

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

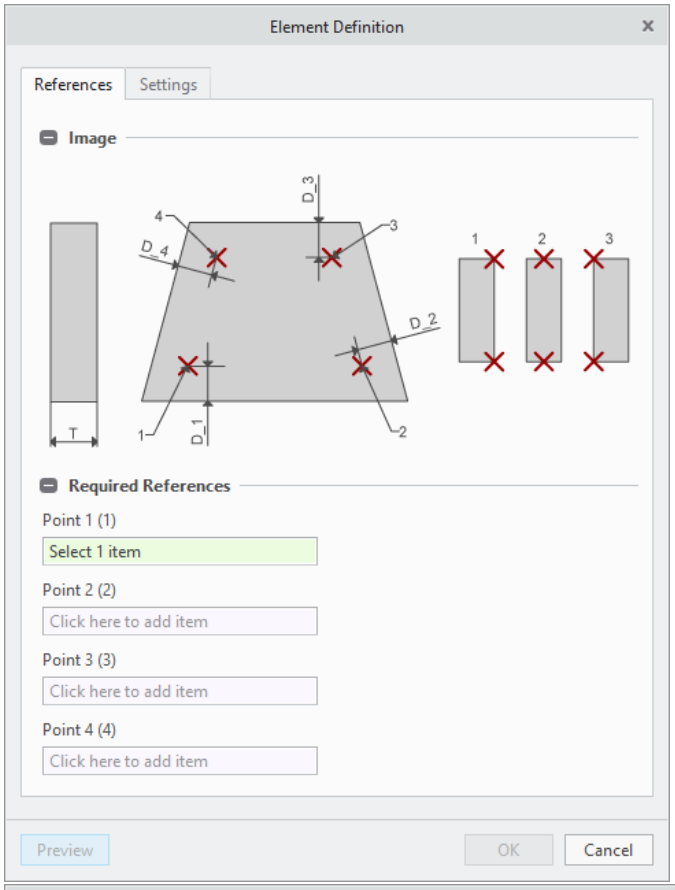
3. Select **[PLATES MM] > [PLATE 5 LINES]**.
   - The **Element definition** dialog box of the **PLATE 5 LINES** opens.
4. As reference **Line 1 (1)** select the curve marked with **[A]**.
5. As reference **Line 2 (2)** select the curve marked with **[B]**.
6. As reference **Line 3 (3)** select the curve marked with **[C]**.
7. As reference **Line 4 (4)** select the curve marked with **[D]**.
8. As reference **Line 5 (5)** select the curve marked with **[E]**.
9. Toggle to **Settings** tab and keep the default values in the dialog.
10. Enter desired attach option **[2]** in field **Attach option** as the plate shall be placed centered to the plane defined by the 5 curves.
11. Click **[Preview]** or middle mouse button to get a preview.
12. Click **[OK]** or middle mouse to close the Element definition dialog box. The plate is assembled.

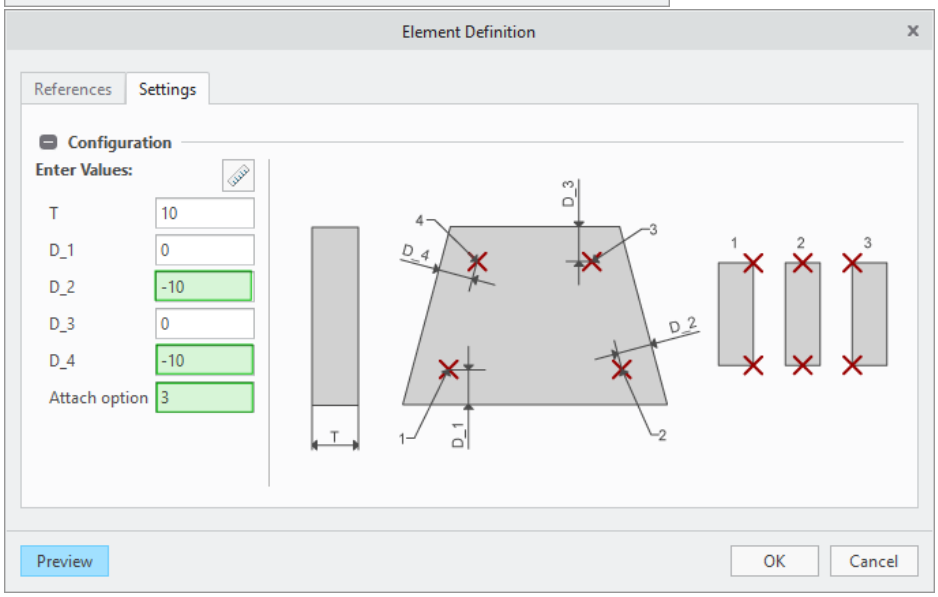


6.3 Plates on points

To assemble the right side plate of the weldment using the 4 datum points on the right side proceed as follows.

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



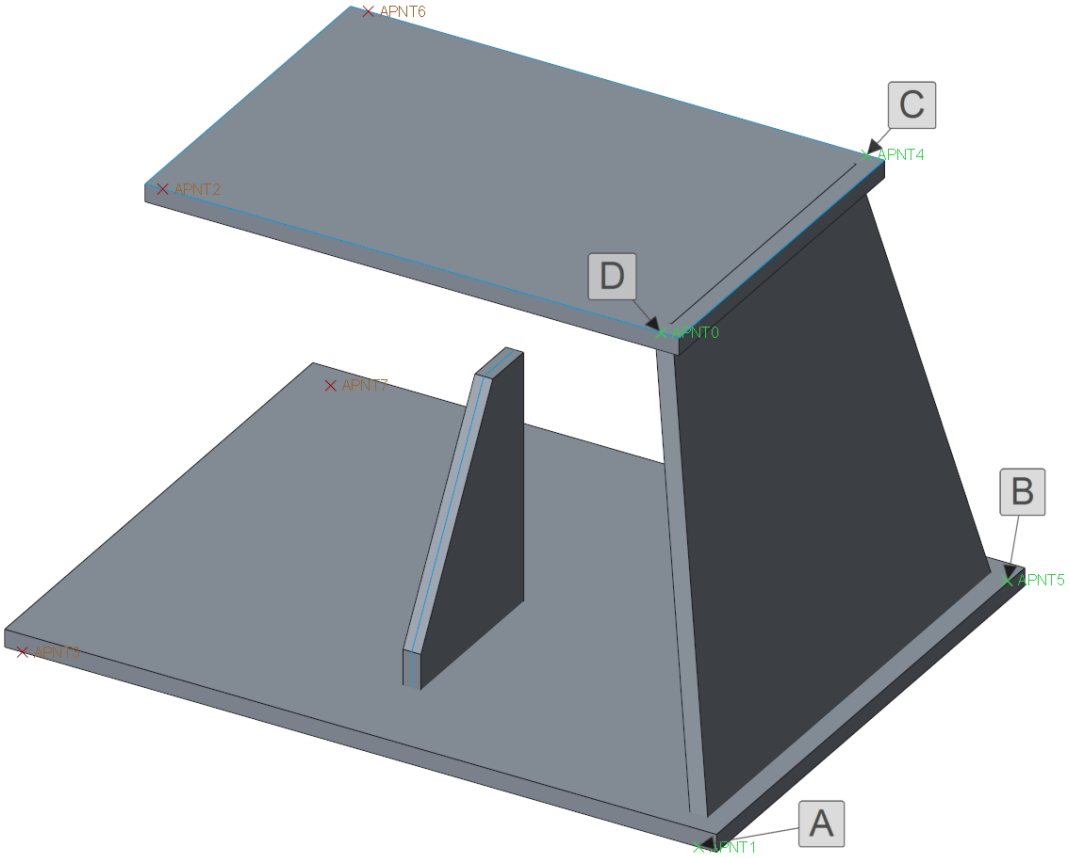


4. As reference **Point 1** (1) select datum point **APNT1 [A]**.
5. As reference **Point 2** (2) select datum point **APNT5 [B]**.
6. As reference **Point 3** (3) select datum point **APNT0 [C]**.
7. As reference **Point 4** (4) select datum point **APNT4 [D]**.
8. Toggle to **Settings** tab.
9. Enter offset values **[-10]** in **D_2** and **D_4** to have some space for weldment lines.

10. Enter desired attach option **[3]** in field **Attach option**.

    Hint 15 — Attach Options. The points you select as references in counterclockwise sense form a plane. Attach option 1 means in front of this plane, 2 means centered to this plane, 3 means behind this plane.

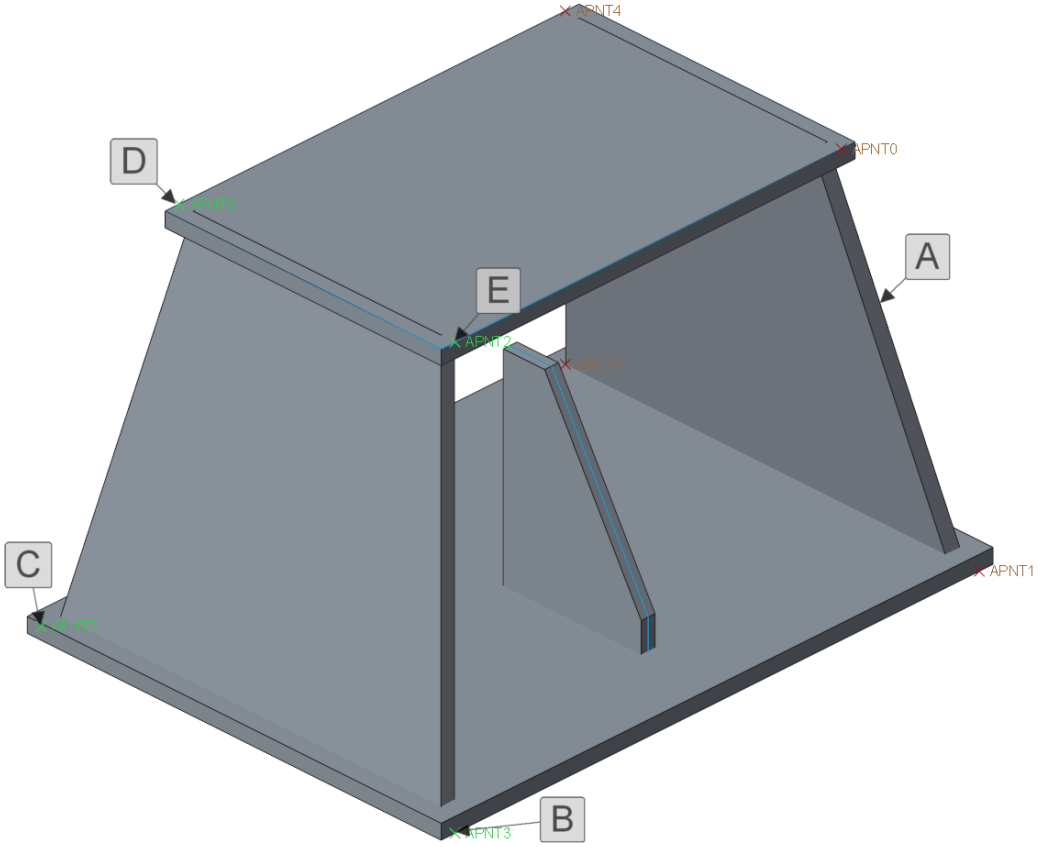
11. Click **[Preview]** or middle mouse button to get a preview.
12. Click **[OK]** or middle mouse button to close the **Element Definition** dialog box.
    - The plate is assembled.



To assemble the right side plate again on the left side of the weldment using the 4 datum points on the left side.

1. Click **Reuse**.
2. Select the plate assembled before **[A]** as element you want to assemble again.
   - The **Element definition** dialog box of the **PLATE 4 POINTS** opens on **References** tab.
3. As reference **Point 1** (1) select datum point **APNT3 [B]**.
4. As reference **Point 2** (2) select datum point **APNT7 [C]**.
5. As reference **Point 3** (3) select datum point **APNT6 [D]**.
6. As reference **Point 4** (4) select datum point **APNT2 [E]**.
7. Toggle to **Settings** tab.

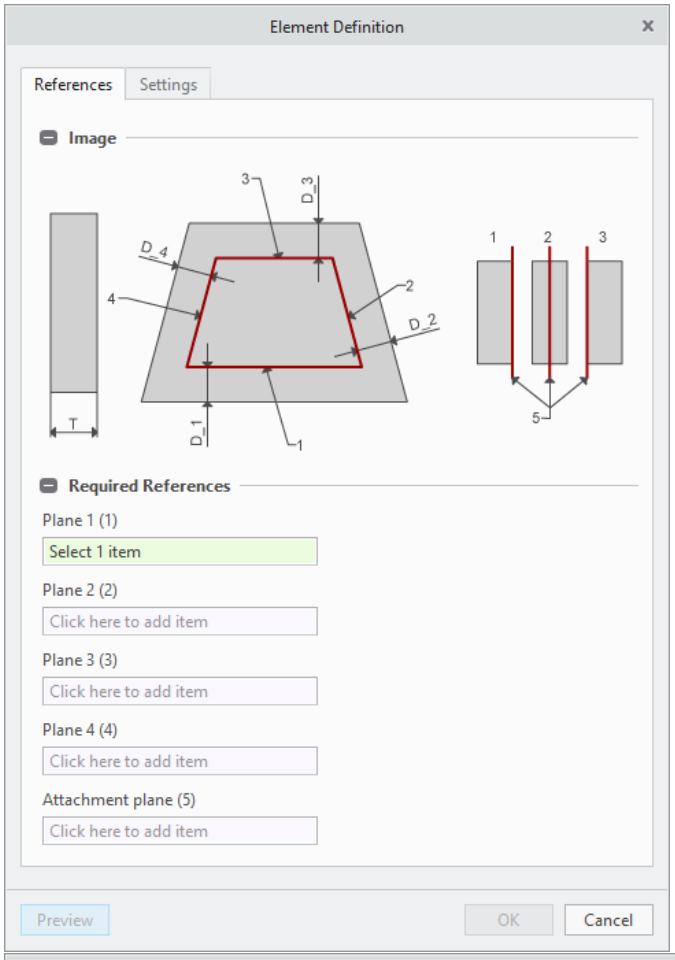
8. Enter desired attach option **[1]** in field **Attach option** to place the plate in front of the plane defined by the 4 points in counterclockwise sense.
9. Click **[Preview]** or middle mouse button to get a preview.
10. Click **[OK]** or middle mouse button to close the **Element Definition** dialog box.
    - The plate is assembled.

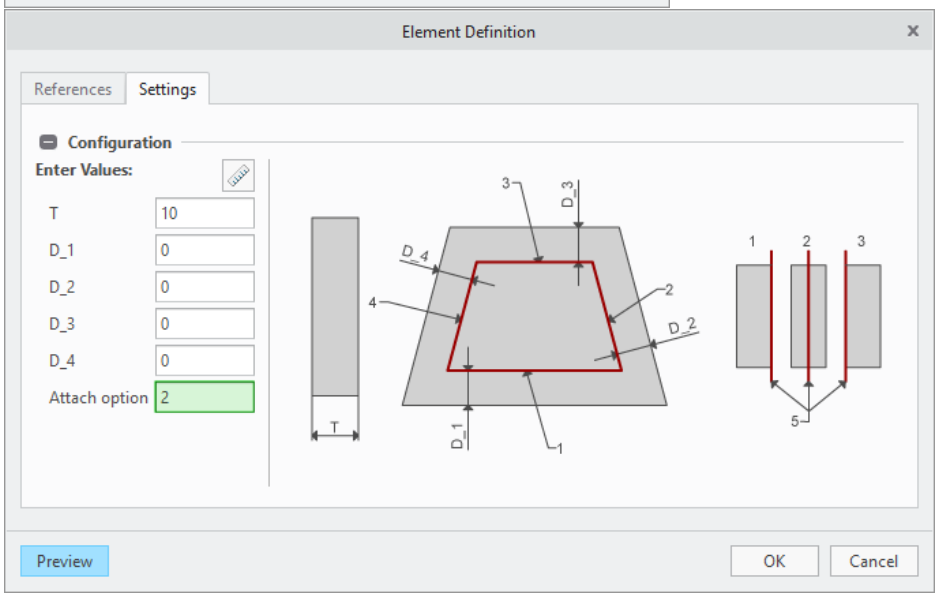


6.4 Plates on planar

To assemble the center plate of the weldment using the 4 side surfaces of bottom, right, top and left plate.

1. Click to open **Select from library** dialog box.
2. Select **[PLATES MM] > [PLATE 4 PLANES]**.
    - The  **Element definition** dialog box of the **PLATE 4 PLANES** opens on **References** tab.

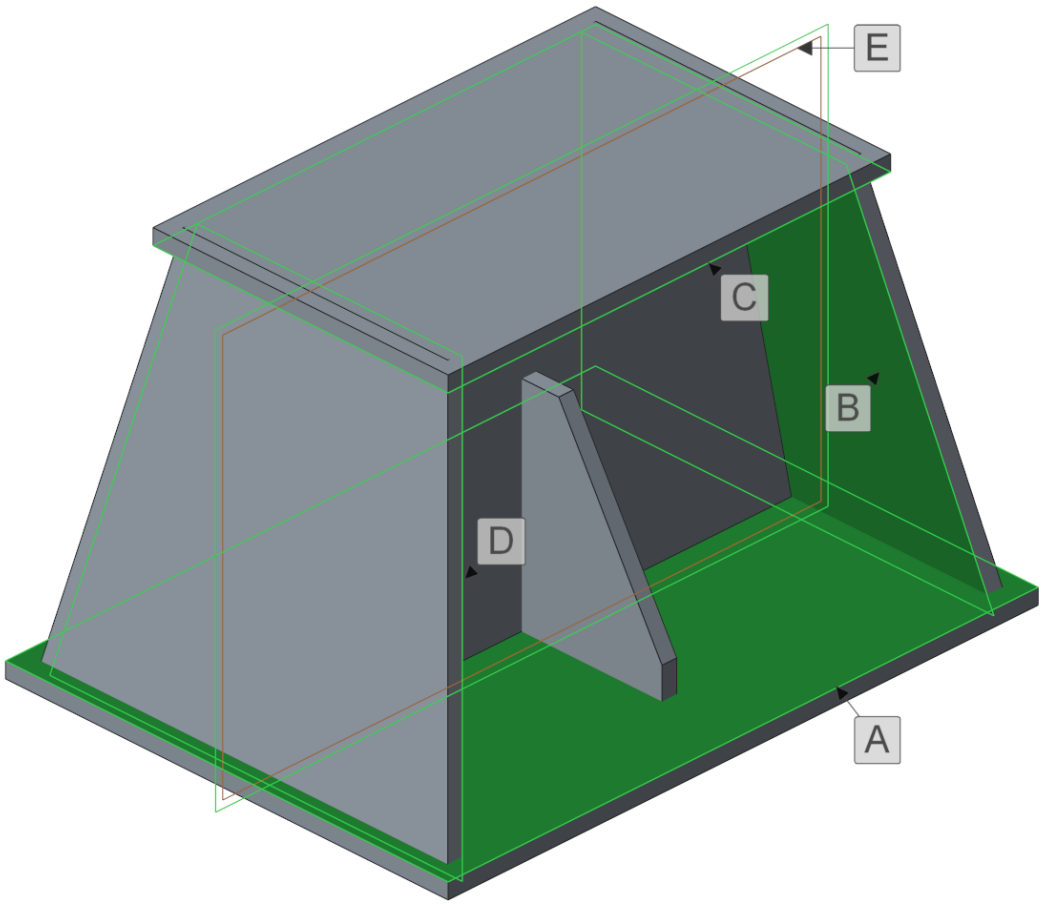




Now select the planar surfaces which determine the size of the plate in counterclockwise sense.

3. As reference **Plane 1** (1) select upper surface of bottom plate **[A]**.
4. As reference **Plane 2** (2) select left surface of right plate **[B]**.
5. As reference **Plane 3** (3) select lower surface of top plate **[C]**.

6. As reference **Plane 4** (4) select right surface of left plate **[D]**.
7. As reference **Attachment plane** (5) and select datum plane **[E]**.
8. Toggle to **Settings** tab.
9. Enter desired attach option **[2]** in field **Attach option**.
10. Click **[Preview]** or middle mouse button to get a preview.
11. Click **[OK]**or middle mouse button to close the **Element definition** dialog box.
    - The plate is assembled.



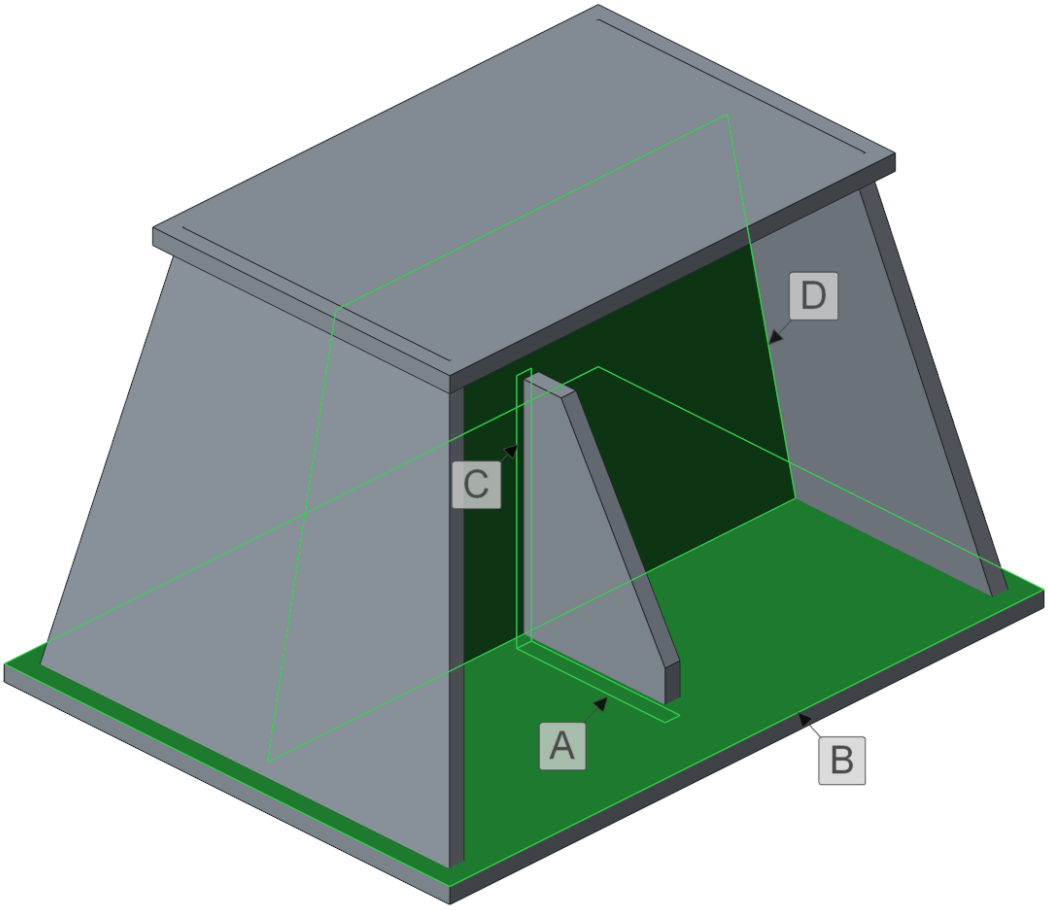
6.5 Plate Joints

As you can see at the current state of the model the plates interfere as the planes, curves and points of the skeleton did not consider plate thickness. This can be solved by creating a **replace surface** joint. It works similar to the **to selected surface** joint for profiles.

To create regular **replace surface** joints for the 5-sided stiffening plate proceed as follows:

1. Open the **Advanced Joints** dialog box.
2. Click to create regular **replace surface** joints without offset.
3. Select the lower surface of the 5-sided stiffening plate as **Surface to replace [A]**.

4. Select the upper surface of the bottom plate as **Attachment surface [B]**.
   - The side surface of the 5-sided plate is attached to the upper surface of the lower plate.
5. Click **[Repeat]** or middle mouse button to complete joint creation and continue with next joint.
6. Select the back surface of the 5-sided stiffening plate as **Surface to replace [C]**.
7. Select the front surface of the center plate as attachment surface **[D]**.
   - The side surface of the 5-sided plate is attached to the front surface of the center plate.
8. Click **[OK]** to complete joint creation and close dialog box.



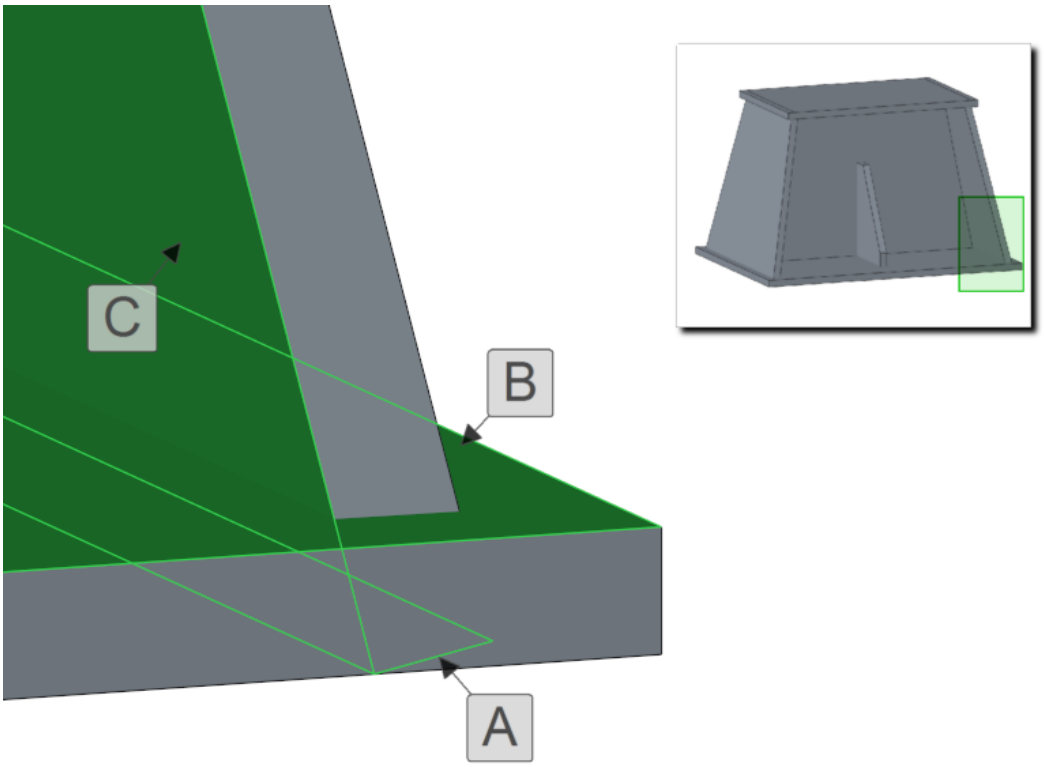
In some cases a plate side surface shall not be attached directly to another surface, but it shall remain perpendicular to the plates plane and only cut along an intersection line of the plate surface and the attachment surface. The lower and upper side surface of the right plate is an example for this.

To create a **Replace Surface joint normal to selected surface along intersection line** for the right side plate proceed as follows.

1. Open the **Advanced Joints** dialog box.
2. Click to create a **Replace Surface joint normal to selected surface along intersection line**.
3. Select the lower surface of the right side plate as **Surface to replace [A]**.

4. Select the upper surface of the bottom plate as **Attachment surface [B]**.
5. Select left surface of the right side plate **[C]** as **Planar surface to determine intersection** line with attachment surface.
6. Click **[Repeat]** or middle mouse button to complete joint creation and continue with next joint.

The lower surface of the right side plate is attached to the upper surface of the lower plate however it still is perpendicular to the plates plane.

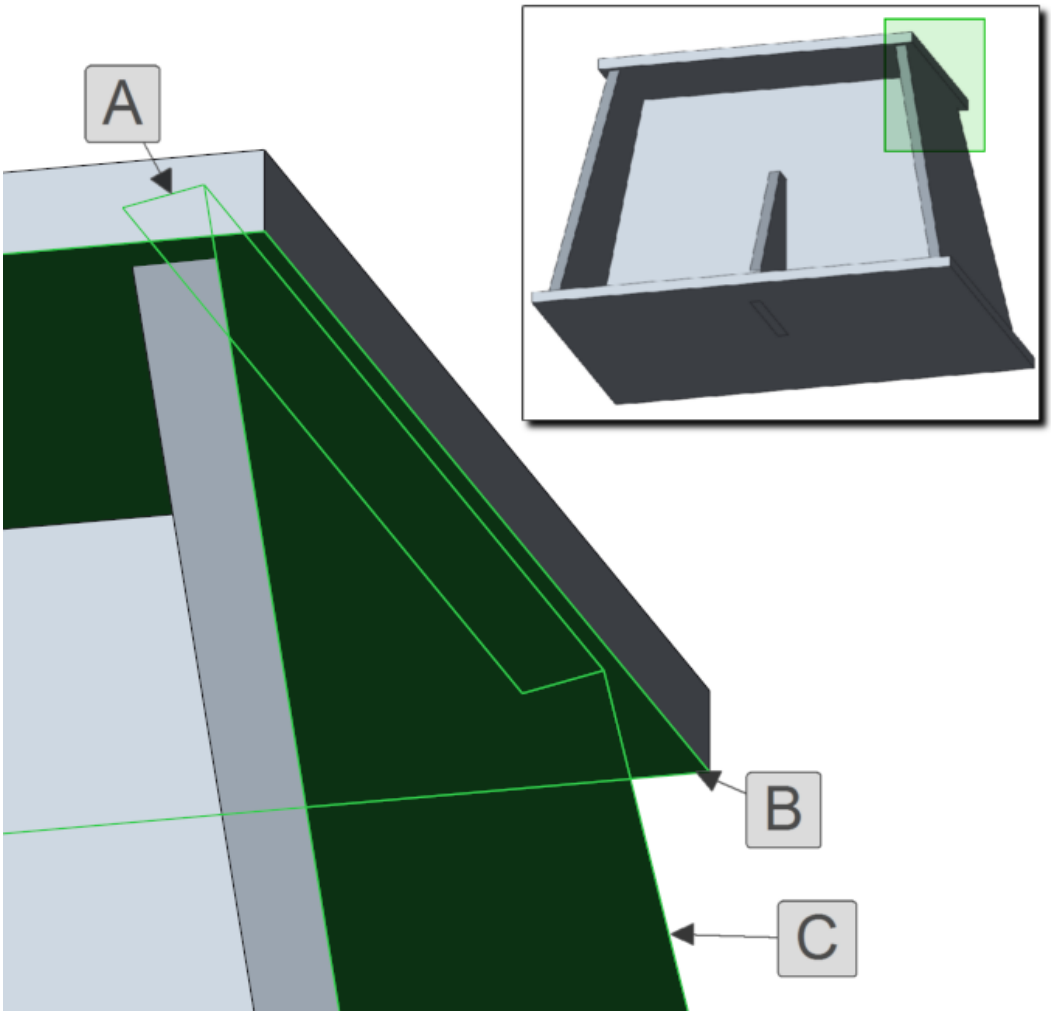


Hint 16 — Delete Replace surface joints. To delete a **replace surface** joint press ✕ and select the surface that has been modified by the joint definition. In this case the selected surface **[A]**.

Continue with the joint definition on the upper side of the assembly.

1. Select the upper surface of the right side plate as surface to replace **[A]**.
2. Select the lower surface of the top plate as attachment surface **[B]**.
3. Select right surface of the right side plate **[C]** as surface to determine intersection line with attachment surface.
4. Click **[OK]** to complete joint creation and close dialog box.

The upper surface of the right side plate is attached to the lower surface of the upper plate however it still is perpendicular to the plates plane. As the right plate is assembled as identical model on the left side of the assembly, the modifications also appear on left side.


A
B
C

7

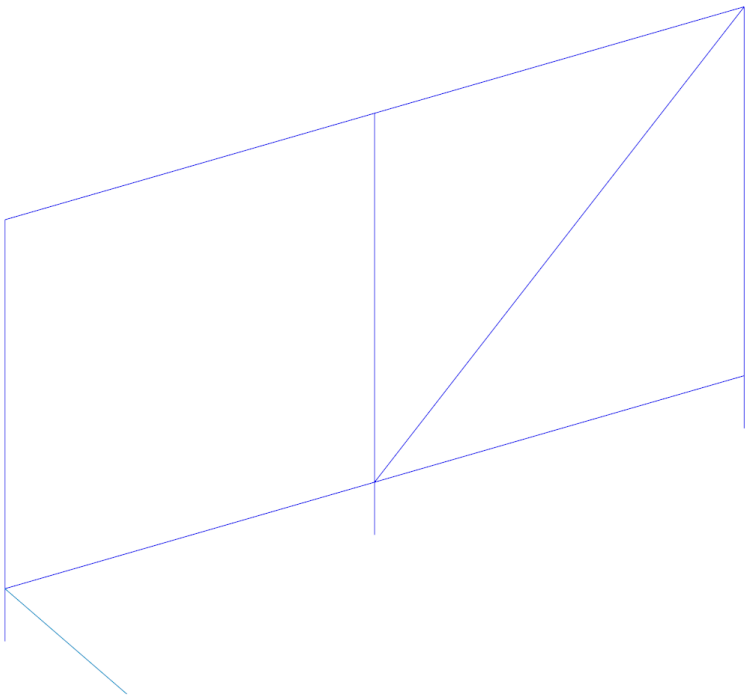
Aluminum profile systems

Overview
Aluminium profiles
Connector elements
Equipment elements
Simplified representation

7.1 Overview

Besides profiles and components for steel construction, the **AFX** library contains many profiles and components of different aluminum profile systems (e.g. Bosch, Item, etc.). In this chapter, a simple framework using Item profiles will be created.

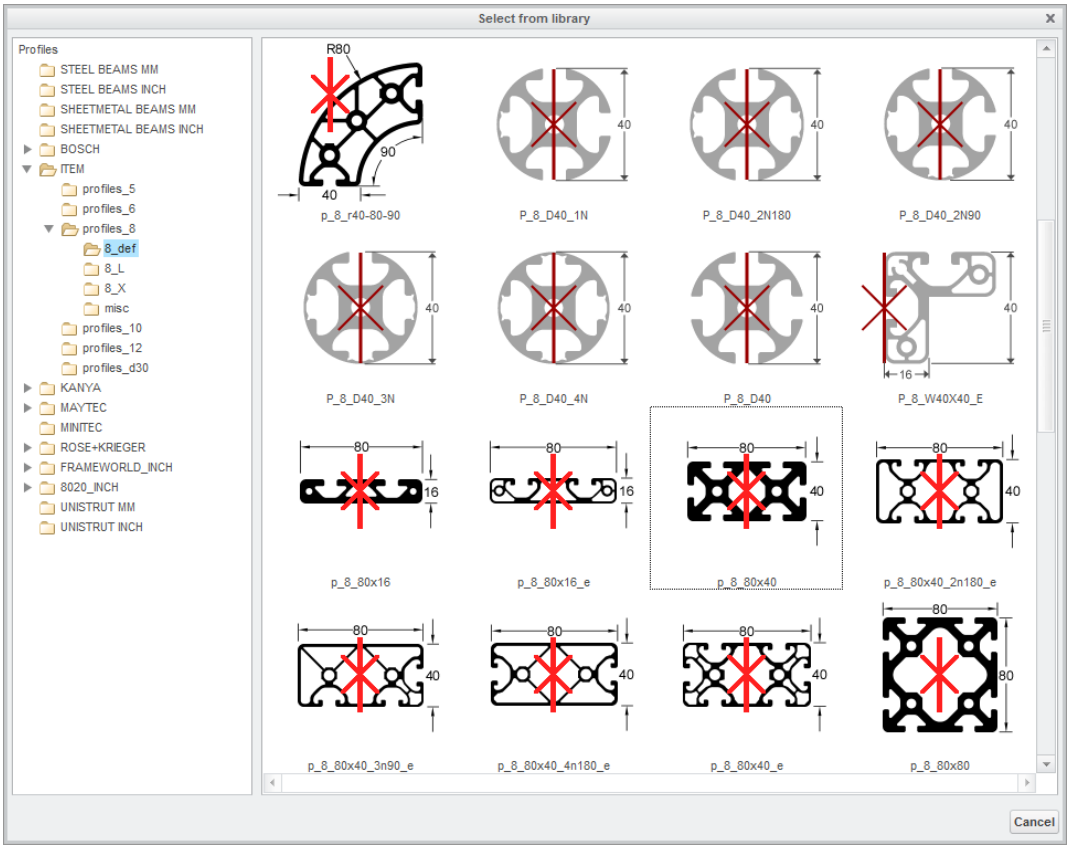
1. Open the assembly `alu_frame.asm`.
   - The reference curves for the frame already are created.
2. Press **Create new project** and enter **[af]** as project shortcut.



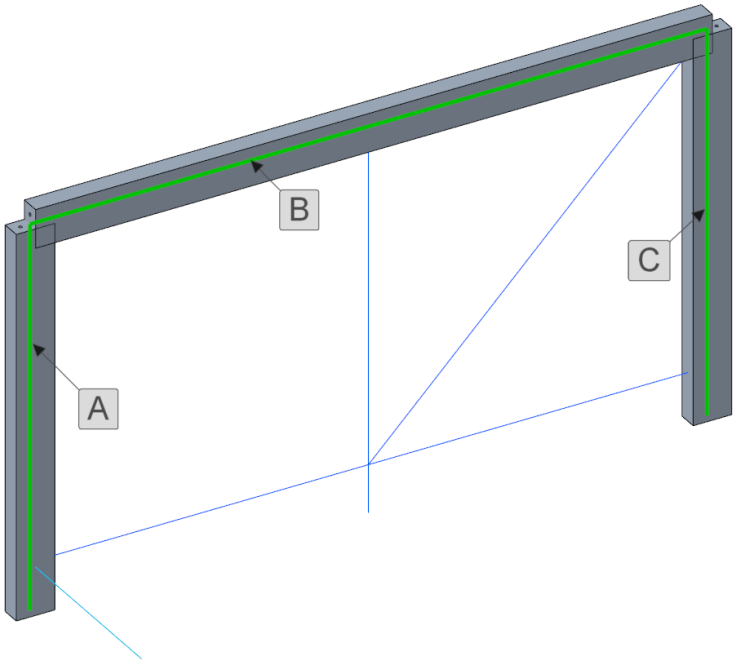
7.2 Aluminium profiles

Assembling aluminum profiles using **AFX** is similar to steel beams, but it is not necessary to choose standard and size. The profile is directly chosen in the library dialog box. In this example some Item profiles 8 80x40, then some profiles 40x40, and finally a cover profile 8 32x4 will be assembled to the frame.

1. Click to open the Profiles dialogbox.
2. Click and select **[ITEM] > [profiles_8] > [8_def]**.
3. Select profile **[p_8_80x40]**.

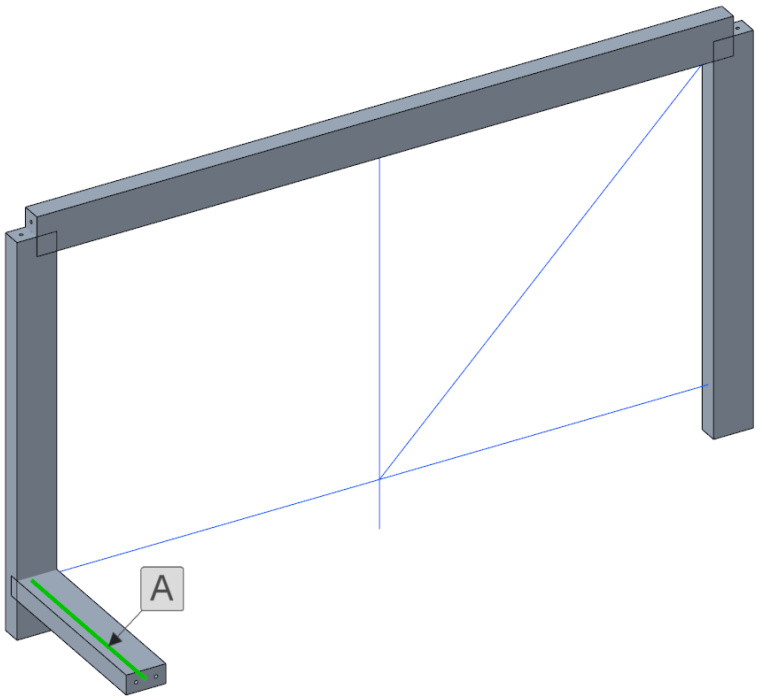


4. Select the reference curves **[A]**, **[B]** and **[C]**. Rotate the profiles if necessary.
   - A new profile is assembled on each curve.



Keep the previously selected profile active and proceed as follows.

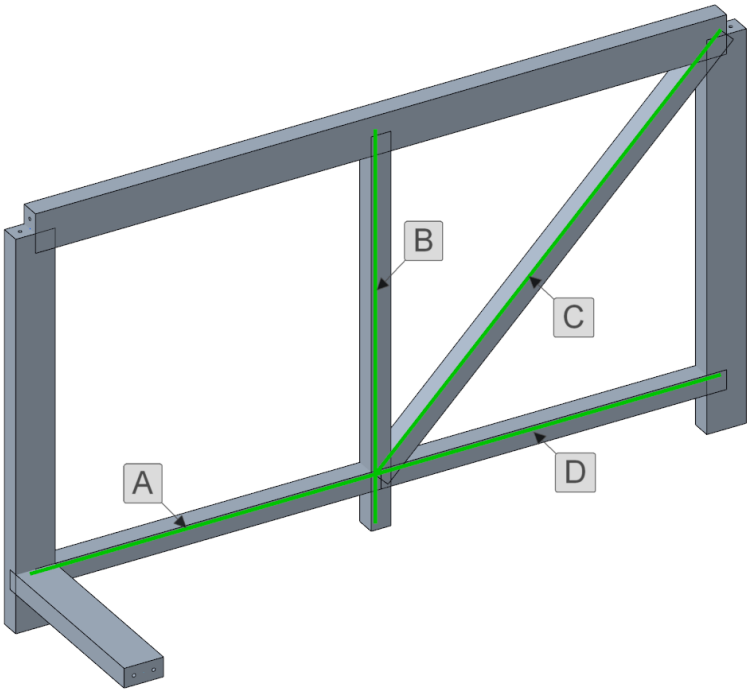
1. Select reference curve **[A]**.
   - A new profile is assembled.



On the remaining curves, profiles 8 40x40 will be assembled.

1. Click  to open the **Select from library** dialog box.
2. Select profile **[p_8_40x40]** from folder **[ITEM] > [profiles_8] > [8_def]**.
3. Select the reference curves **[A]**, **[B]**, **[C]** and **[D]**.

- A new profile is assembled on each curve.

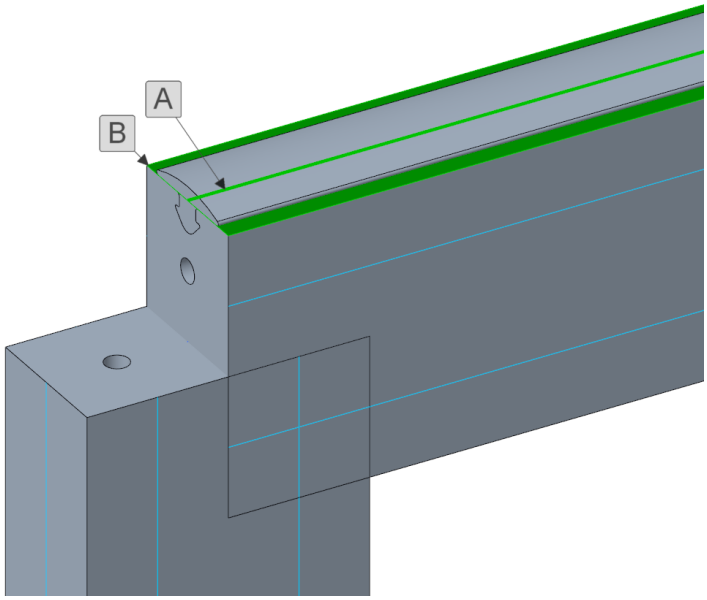


As aluminum profile systems sometimes have profiles that are assembled into the slots of other system profiles, these system profiles have predefined curves at the slots that can be used to assemble other profiles.

To use slot curves as reference for other profiles proceed as follows.

1. Show layer 00_PROFIL_SLOT_CURVE.
   - The location of slots will become visible on the profiles
2. Click to open the Profiles dialog box.
3. Click and select **[ITEM]** > **[profiles_8]** > **[misc]**.
4. Select profile **[cover_profile_8_32x4]**.

5. Select the upper surface of the horizontal profile as orientation plane **[A]**.
6. Select the profile slot reference curve **[B]**.
   - The cover profile is assembled.



Hint 17 — Simplified representation of aluminum profiles. The aluminum profiles are assembled in simplified representation by default. If desired they can be shown in a more detailed representation (see chapter 7.5).

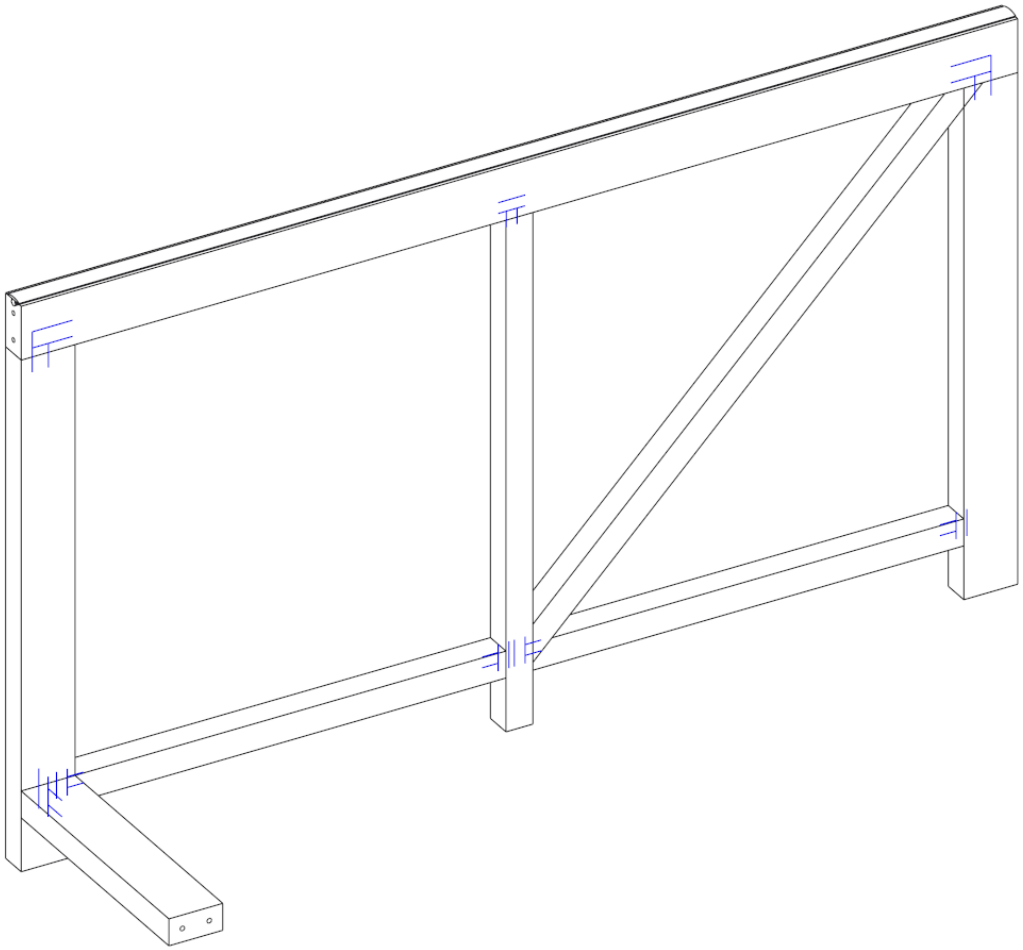
Now, create the joints between the profiles like shown in the next picture. Do not apply a joint to the diagonal profile yet. This will be done later on.

For this task you will need the following joints.

- 

Corner Joint
- 

T-joint



7.3 Connector elements

Assembling connector elements for aluminum profile systems using **AFX** works like described in chapter 3.

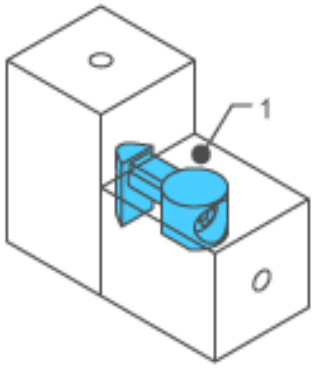
Unlike steel profile connections this connector elements usually have no configuration options since most connector and equipment elements are not variable in size and do not have different assembling options. The necessary selections for the element are shown in the Connector Elements dialog box. Using some parts of the item system, the procedure is demonstrated, connector and equipment elements of other vendors work similar.

To open the **New Connector Elements** dialog box click **New Connector Elements**.

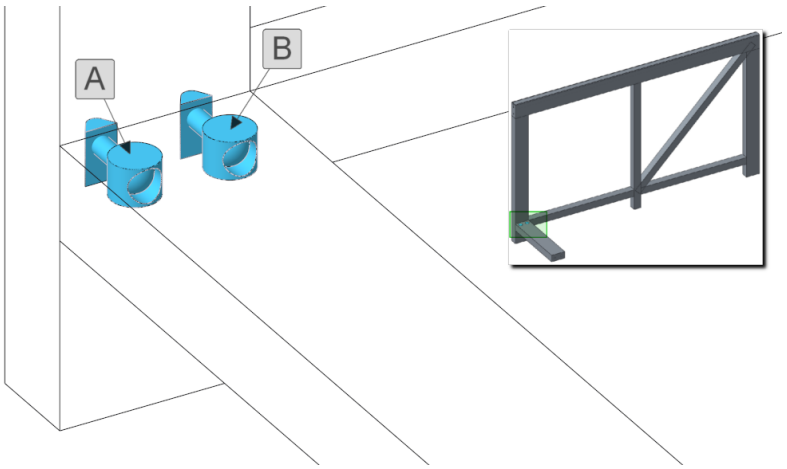
Universal connectors

To assemble an **Item** universal connector proceed as follows.

1. Click  and select **[ITEM] > [connector_sets] > [universal_connector] > [uc_8]**.
   - The **Element definition** dialog box of the **uc_8** opens. The required pick references are shown in the picture. If a profile has multiple slots you must choose the approximate location. **AFX** will use the closest slot according to your selection.



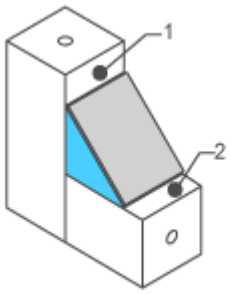
2. As reference **ASSEMBLY-POSITION** select the lower left profile at the left assembling position **[A]** of the connector element.
   - The universal connector is assembled and the necessary bore is created in the profile.
3. Click **[OK]** or middle mouse to close the Element definition dialog box.
4. Click **[Next]** or middle mouse in **New Component** dialog box to assemble connector again.
5. As reference **ASSEMBLY-POSITION** select the profile at the right assembling position **[B]** of the connector element.
   - The universal connector is assembled and a bore is created in the profile.
6. Click **[OK]** or middle mouse to close the Element definition dialog box.



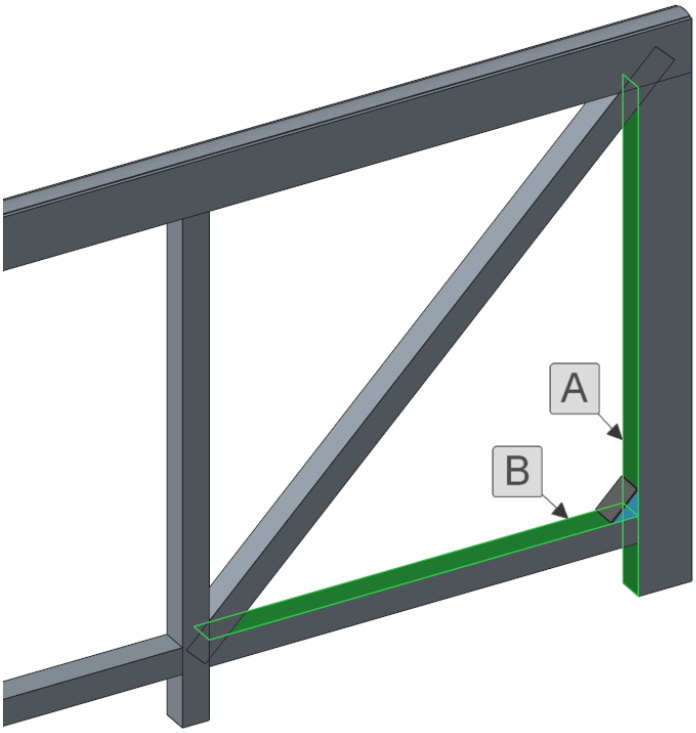
Creation of optimized cut lists

Unlike the universal connector the angle bracket needs two references. Assemble the element as follows.

1. Click  and select **[ITEM] > [bracket_connectors] > [brack_set_8_40x40]**.
   - The **Element definition** dialog box of the **brack_set_8_40x40** opens. The required pick references are shown in the picture.

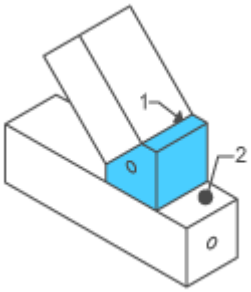


2. As reference **ASSEMBLY-POSITION1** select the left surface of the right vertical profile **[A]**.
3. As reference **ASSEMBLY-POSITION12** select the upper surface of the bottom right profile **[B]**.
   - The bracket connectors is assembled.
4. Click **[OK]** or middle mouse to close the Element definition dialog box.

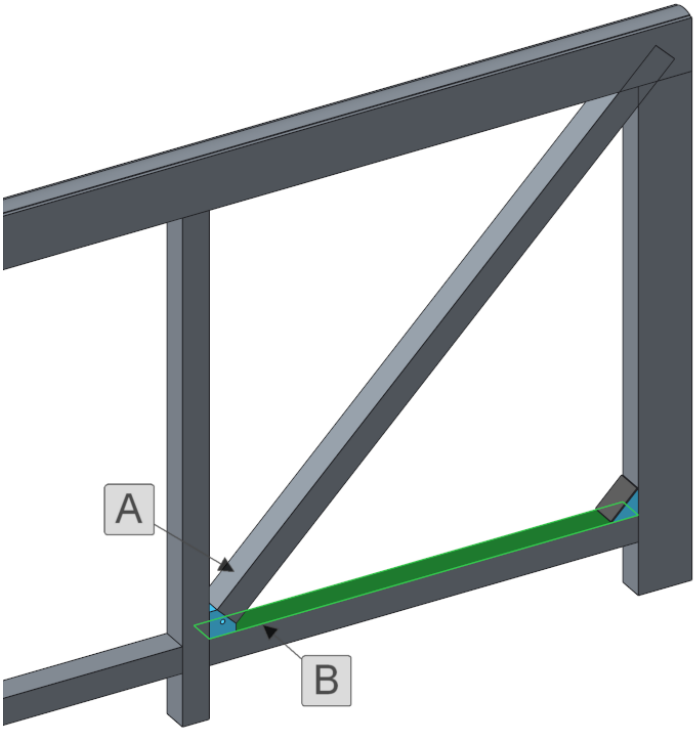


In the next step we will assemble another bracket connector. This one can be used to attach the diagonal profile. Assemble the element as follows.

1. Click  and select **[ITEM] > [bracket_connectors] > [brack_element_8_t1_40]**.
   - The **Element definition** dialog box of the **brack_element_8_t1_40** opens. The required pick references are shown in the picture.



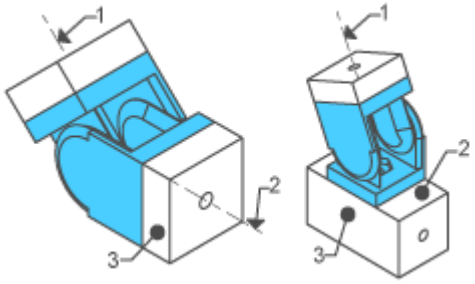
2. As reference **PROFILE END** select the diagonal profile near the end **[A]**.
3. As reference **ATTACH PLANE** select the lower right profile at the upper surface **[B]**.
   - The bracket connectors is assembled.
4. Click **[OK]** or middle mouse to close the Element definition dialog box.



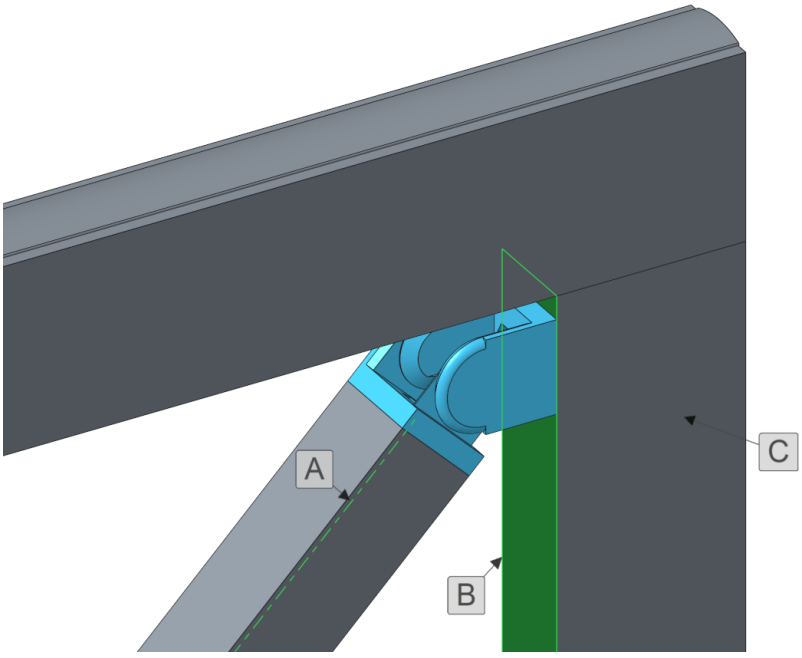
Joints

In this section a joint will be assembled. When you use this component **AFX** will automatically measure the required angle and create a new instance. Assemble the element as follows.

1. Click  and select **[ITEM] > [joints] > [joint_8_40x40_flex]**.
   - The **Element definition** dialog box of the **joint_8_40x40_flex** opens. The required pick references are shown in the picture.



2. As reference **PROFILE_1_BORE_AXIS** select the axis of the diagonal profile **[A]**.
3. As reference **PROFILE_2_BORE_AXIS_OR_ATTACH_FACE** select the right vertical profile at the left surface **[B]**.
4. As reference **PROFILE_SIDE_FACE** select the right vertical profile at the front surface **[C]**.
   - The joint is assembled and the profile is shortened.
5. Click **[OK]** or middle mouse to close the Element definition dialog box.



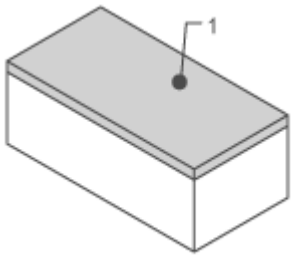
7.4 Equipment elements

Assembling equipment elements for aluminum profile systems with **AFX** works similar to connector elements. To open the **New Equipment Elements** dialog box click **New Equipment Elements**.

Profile end caps

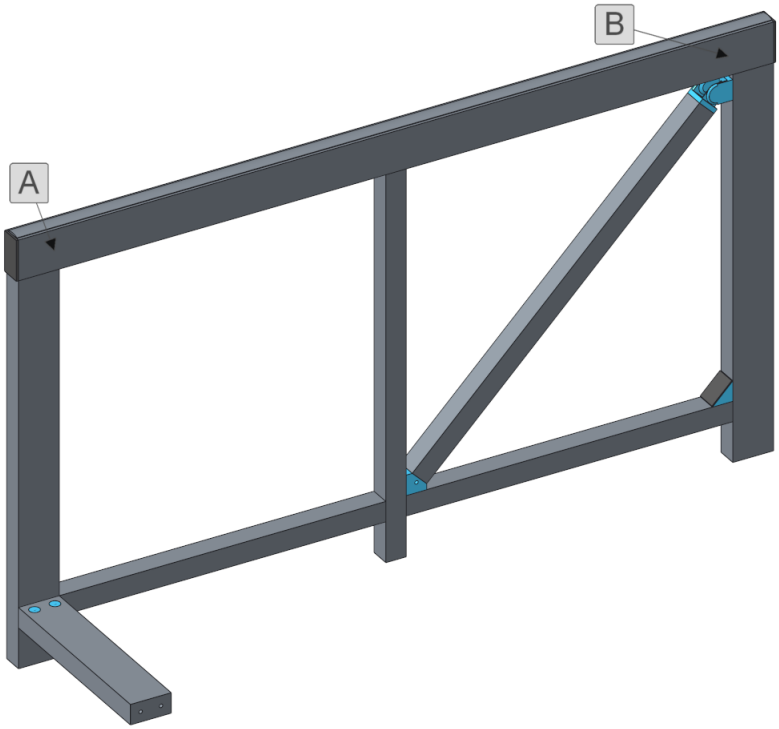
To assemble profile end caps proceed as follows.

1. Click and select **[ITEM] > [caps] > [caps_8] > [cap_8_80x40]**.
   - The **Element definition** dialog box of the **cap_8_80x40** opens. The required pick references are shown in the picture.



cap_8_80x40

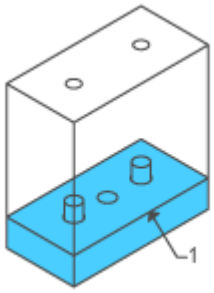
2. As reference **PROFILE-END** select the upper horizontal profile close to the left end **[A]**.
   - The cap is assembled.
3. Click **[OK]** or middle mouse to close the Element definition dialog box.
4. Click **[Next]** or middle mouse in **New Component** dialog box to assemble connector again.
5. As reference **PROFILE-END** select the upper horizontal profile close to the right end **[B]**.
   - The cap is assembled again.
6. Click **[OK]** or middle mouse to close the Element definition dialog box.



Transport and foot plates

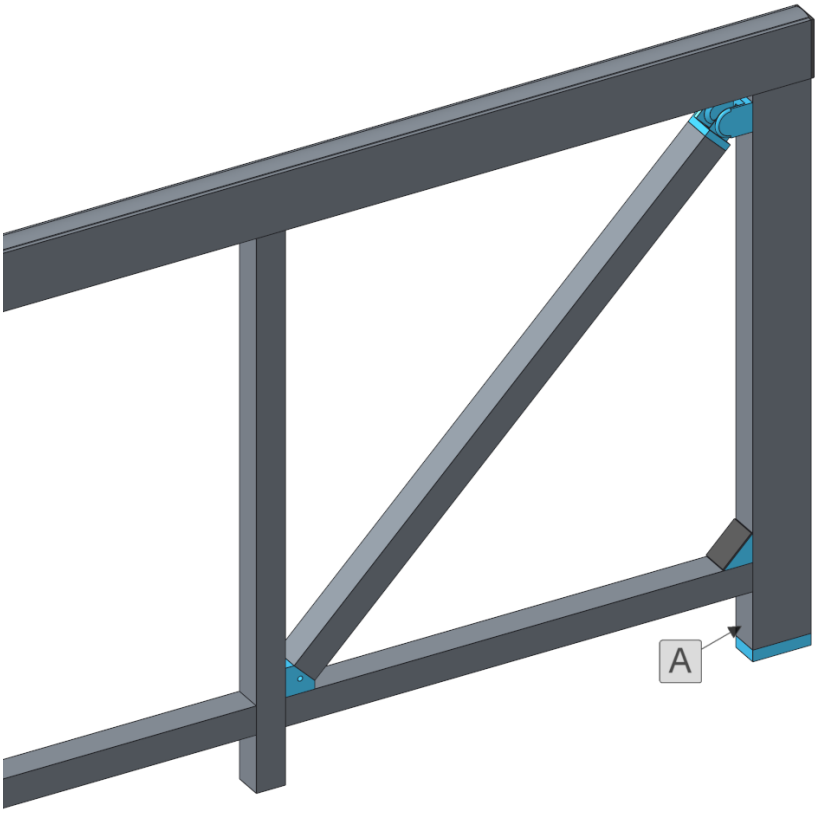
To assemble transport and footplates proceed as follows.

1. Click  and select **[ITEM] > [transport_foot_plates] > [transport_footplates_8] > [tf_pl_8_80x40_m10]**.
   - The **Element definition** dialog box of the **tf_pl_8_80x40_m10** opens. The required pick references are shown in the picture.



tf_pl_8_80x40_m10

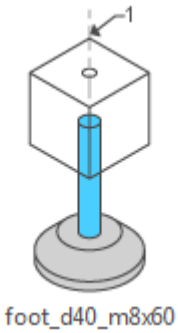
2. As reference **PROFILE-END** select the right vertical profile close to the lower end **[A]**.
   - The plate is assembled with screws and the threads are created in the profile.
3. Click **[OK]** or middle mouse to close the Element definition dialog box.



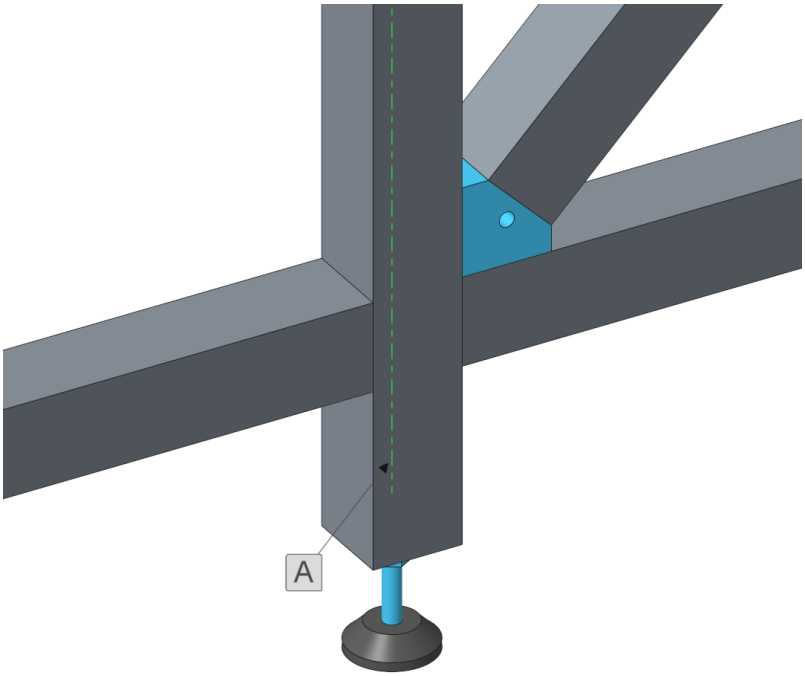
Feet

To assemble transport and footplates proceed as follows.

1. Click  and select **[ITEM] > [feet] > [foot_d40_m8x60]**.
   - The **Element definition** dialog box of the **foot_d40_m8x60** opens. The required pick references are shown in the picture.



2. As reference **PROFILE BORE AXIS** select the bore axis of the middle vertical profile close to the lower end **[A]**.
   - The foot is assembled and the thread is created in the profile.
3. Click **[OK]** or middle mouse to close the Element definition dialog box.



Now assemble another foot on the previously assembled transport and footplate.

1. Click  and select **[ITEM] > [feet] > [foot_d40_m10x80]**.
2. As reference **PROFILE BORE AXIS** select as **[PROFILE BORE AXIS]** the axis of the middle bore of the foot plate that was assembled in the previous chapter **[A]**.
   - Since the axis does not belong to a profile, **AFX** can't find the proper profile end surface. Therefore you are prompted to select the **PROFILE END SURFACE**
3. Select the bottom side of the plate **[B]**.
   - The foot is assembled.
4. Click **[OK]** or middle mouse to close the Element definition dialog box.

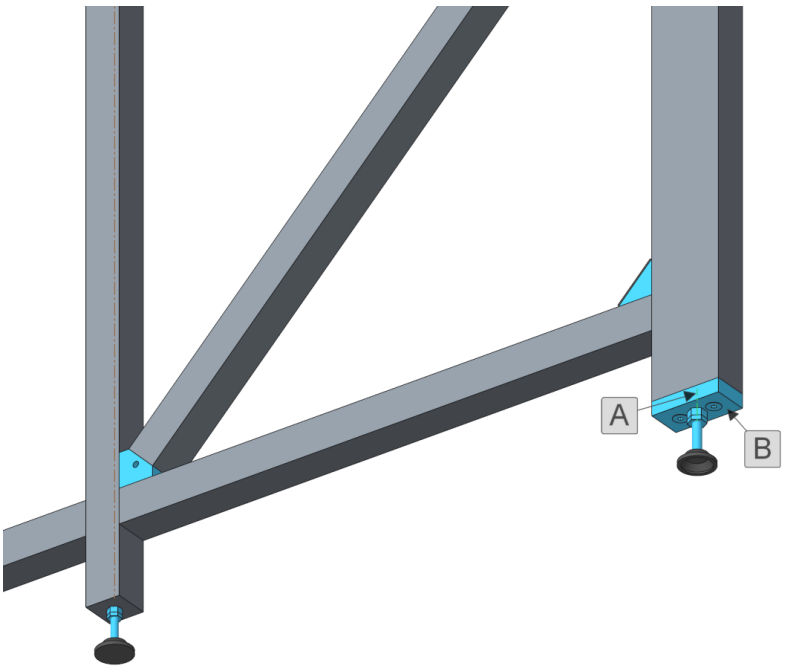
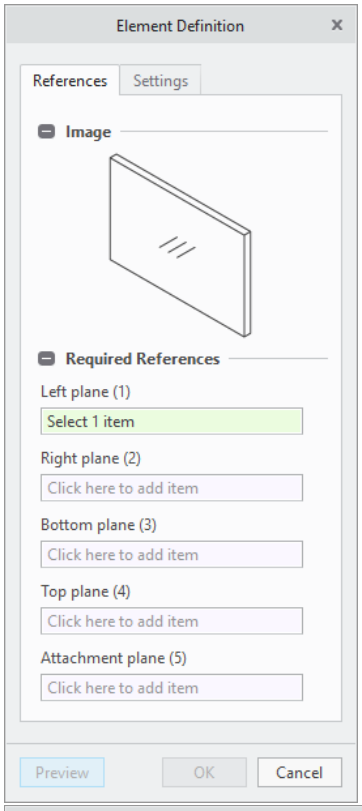
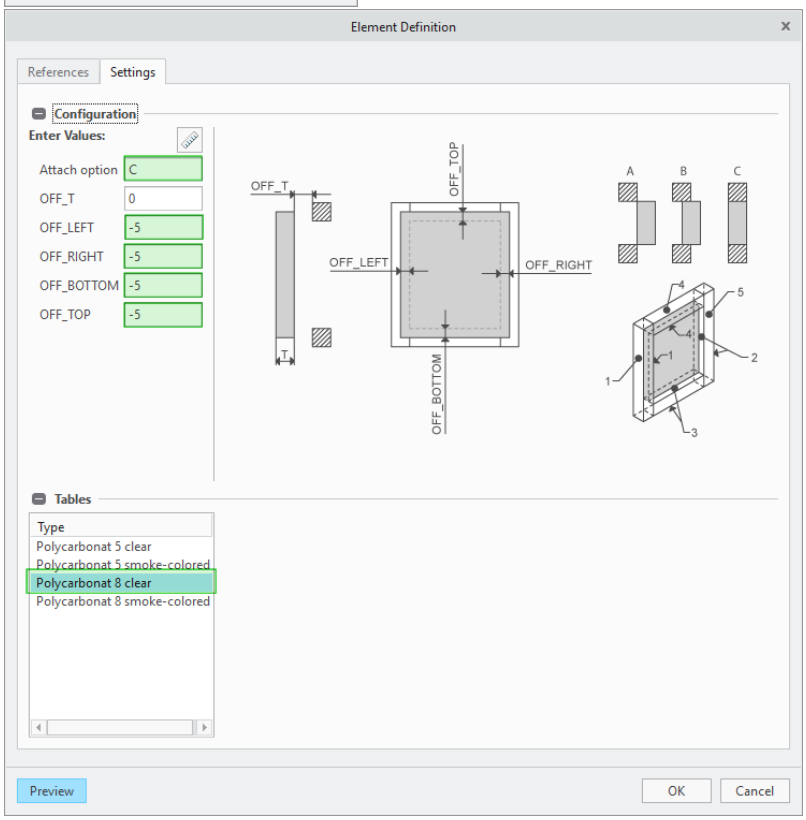


Plate elements

To assemble Item plate elements proceed as follows.

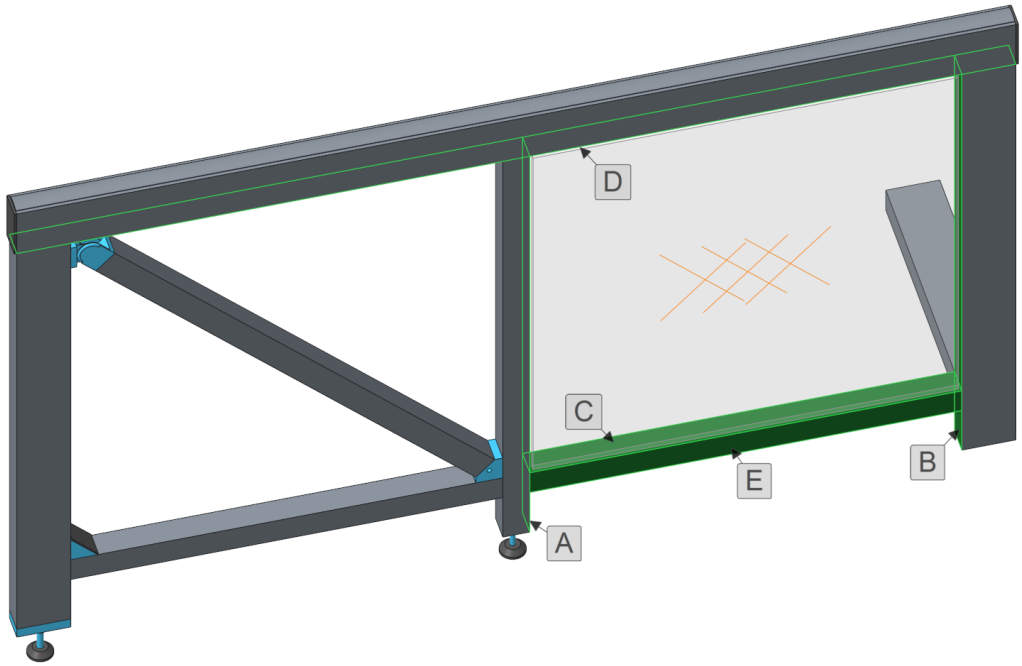
1. Click  and select **[ITEM] > [plate_elements] > [polycarbonat]**.
   - The **Element definition** dialog box opens on **References** tab.





2. As reference **Left plane (1)** select the right side surface of the left vertical profile **[A]**.
3. As reference **Right plane (2)** select the left side surface of the right vertical profile **[B]**.

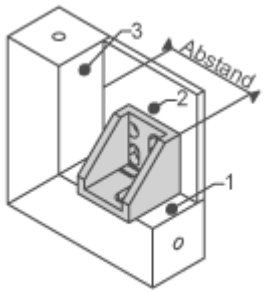
4. As reference **Bottom plane (3)** select the upper surface of the down horizontal profile **[C]**.
5. As reference **Top plane (4)]** select the lower surface of the upper horizontal profile **[D]**.
6. As reference **Attachment plane (5)** select the rear surface of the lower horizontal profile **[E]**.
7. Toggle to the **Settings** tab.
8. Enter **Attach option [C]** to place the plate aligned to the attach face (A = on the surface, B = in the middle).
9. Select type **[polycarbonat 8mm, clear]** from the table.
10. Enter the desired offset values **[-5]** into the four fields **OFF_LEFT** (offset at the left), **OFF_RIGHT** (offset at the right), **OFF_BOTTOM** (offset at the bottom), **OFF_TOP** (offset at the top).
11. Click **[Preview]** or middle mouse button to get a preview.
12. Click **[OK]** or middle mouse button to close the **Element definition** dialog box.
    - The plate is assembled.



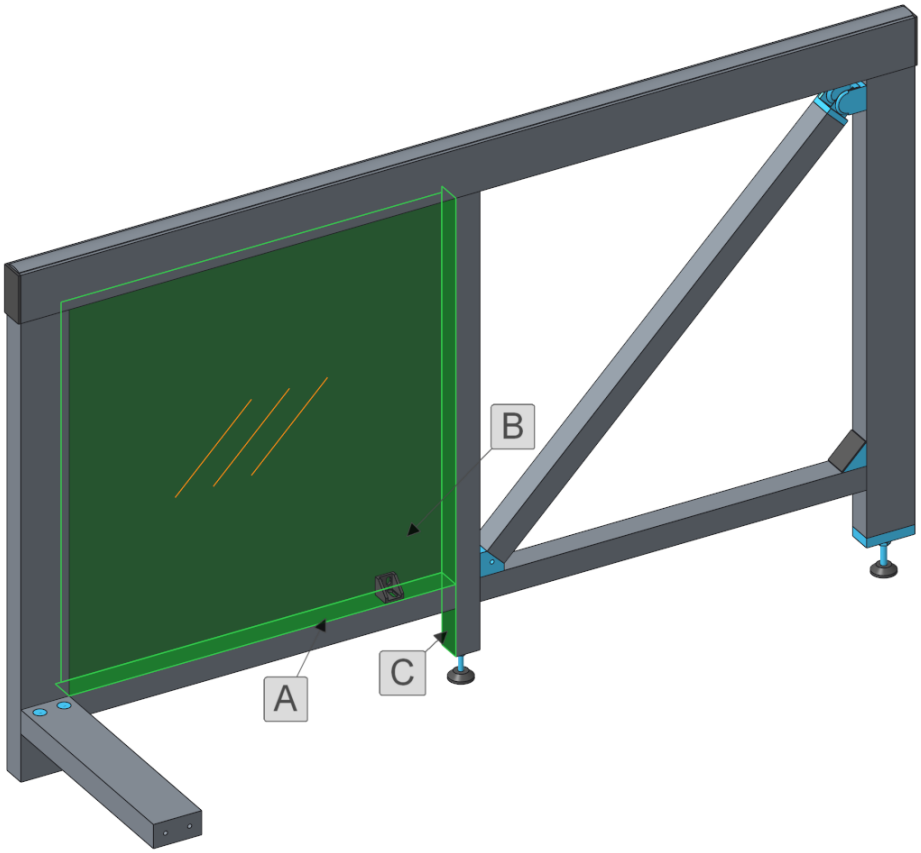
Multi blocks

To assemble multi blocks proceed as follows.

1. Click and select **[ITEM] > [multi_blocks] > [multi_block_8_zn]**.
    - The **Element definition** dialog box of the **multi_block_8_zn** opens. The required pick references are shown in the picture. In the message area you will be prompted to select **PLACEMENT PLANE_PROFILE**, **PLACEMENT_PLANE_SURFACE_ELEMENT** and **DISTANCE_PLANE**.



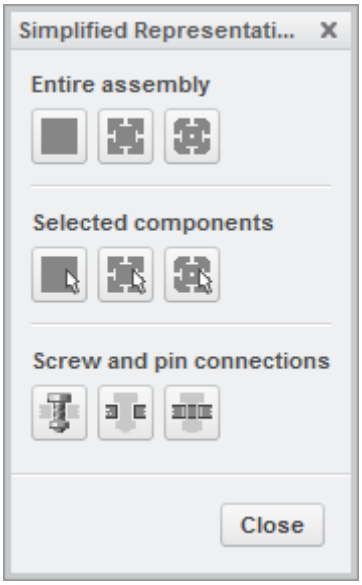
2. As reference **PLACEMENT_PLANE_PROFILE** select the upper surface of the lower, horizontal profile **[A]**.
3. As reference **PLANE_SURFACE_ELEMENT** select the surface of the poly-carbonate plate **[B]**.
4. As reference **DISTANCE_PLANE** select the right surface of the left, vertical profile **[C]**.
5. Toggle to the **Settings** tab.
6. Enter as **DISTANCE [100]**.
7. Click **[Preview]** or middle mouse button to get a preview.
8. Click **[OK]** or middle mouse button to close the **Element definition** dialog box.
   - The multi block is assembled.



7.5 Simplified representation

As you might have realized, the profiles are not in a very high level of detail. This an intended behavior to reduce regeneration and retrieving time of the assembly. This technique also allows you to create huge assemblies without slowing the system down.

AFX comes with automatic simplified representations. To switch between different levels of quality open the **Simplified Representations** dialog box.

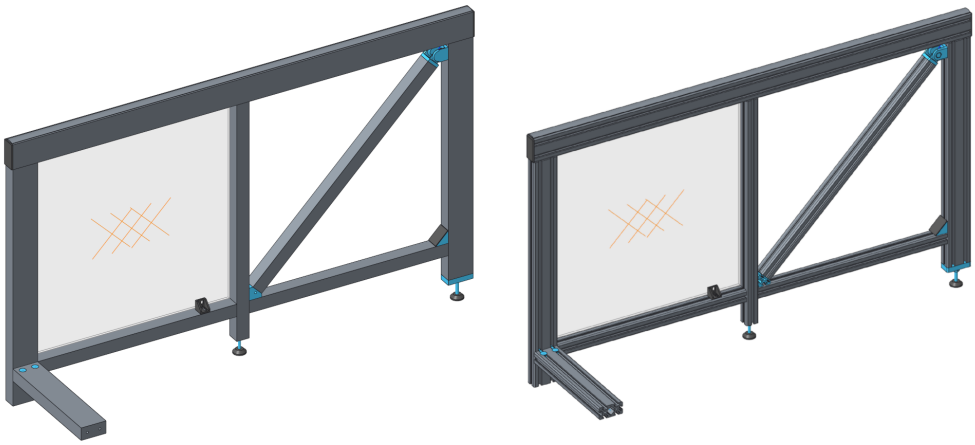


In the dialog you have the following possibilities.

- Change the level of detail for the entire assembly
- Change the level of detail for selected components
- Change the level of detail for screws and pins

Each of these groups has three different levels of detail. The first option is the default value. If you change the settings all new assembled profiles will be assembled in the level you previously chose.

Press to show the assembly in full detailed level.



8

Configuration

Overview
BOM Parameters
Component Naming Scheme
Start part configuration

8.1 Overview

This chapter will give you a quick information about the **AFX** configuration. It is possible to configure BOM Parameters, the **AFX** part naming scheme and allows you to configure the start model.

8.2 BOM Parameters

Each profile assembled with **AFX** has six system parameters by default. All other **AFX** elements have three system parameters by default. These parameters are either stored as fixed values in parameters of the profiles or elements or they are controlled by **AFX** during assembly of the profile or element or during regeneration of the complete assembly:

1. **BUW_NAME** - Contains the name of the profile or element, such as angle, pipe, or end plate. This value is stored as a fixed value in the profile, connector or equipment element and is not changed.
2. **BUW_TYPE** - Contains a description of the type of profile or element, for example, supplier and ordering number of elements from aluminum profile systems. This value is also stored as a fixed value in the library parts and is not changed.
3. **BUW_SIZE** - Contains the size information for the profile or element, such as **L4x4x1/2** for an angle, or **extra strong 3** for a pipe. The value is either saved as a fixed value in the library part or it is controlled by **AFX** if during selection of the element a list with the different sizes appears.
4. **BUW_LENGTH** - Contains the cutting length of a profile, rendered a as number. This value is adjusted after regeneration.
5. **BUW_LENGTH_STR** - Contains the cutting length of a profile, rendered as text. This value is adjusted after regeneration.
6. **BUW_PROFIL_TYPE** - Contains the part name of a profile in the library. This value may not be changed.

You can create your own **BOM** parameters from these basic system parameters using **Creo Parametric** relations. Write the required relations in a text file named `parameter_relations.txt`. The relations in this file are added to the profile or element during the assembly process so that the part has the required parameters as a combination of the values of the six system parameters. The following example describes this process using the standard relation file for profiles (`param_relations.txt` in sub directory parts/profiles):

```
NAMING = BUW_NAME

DESIGNATION = BUW_TYPE + " " + BUW_SIZE + " x " + BUW_LENGTH_STR

LENGTH = BUW_LENGTH
```

During the assembly process, the contents of the parameters are translated and the relation file is added, so the following **BOM** parameters appear in the part:

```
NAMING = "ANGLE"

DESCRIPTION = "L"+" "+"4 x 4 x 1/2"+" x "+"50"="L 4 x 4 x 1/2 x 50"

LENGTH = 50
```

AFX uses the following rule when adding relation files:

- If a file named `param_relations.txt` in the sub-directory of the library from which the profile or element is retrieved exists, this `param_relations.txt` is added to the part.
- If a file named `param_relations.txt` in the sub-directory of the library from which the profile or element is retrieved does not exist, the default `param_relations.txt` from the library parts/profiles for profiles, from the library parts/bend_profiles for bent profiles, from the library parts/connectors for connectors, and from the library parts/equipment for equipment is added.

For screws, washers and nuts the file `param_relations.txt` is located in **parts/ifx_fastener_data**:

You can configure the parameters and their contents in the default `param_relations.txt` and in the profile or element specific `param_relations.txt` in the various sub-directories, according to your needs.

8.3 Component Naming Scheme

For standard configuration, **AFX** names components using the following naming scheme:

`<PROJECT_SHORT>_<ELEMENT_NAME>_<ELEMENT_NUMBER>`

You can overwrite this standard configuration for individual component sub-directories in the library or globally for each of the component groups (profiles, bend_profiles, connectors, and equipment). Create a file named element_name.txt in the desired folder (e.g. profiles) and describe your own naming scheme using the following principles. The element name can be composed by the following components:

`<PROJECT_SHORT>` - Project shortcut

`<ELEMENT_NAME>` - Element name in the library

`<ELEMENT_NUMBER>` - Incremental number of the element in the current assembly

Arbitrary intermediate text elements #. For example, the default naming scheme in **AFX** can be described by the following formatdescribing line in the file `element_name.txt`:

`PROJECT_SHORT#_#ELEMENT_NAME#_#ELEMENT_NUMBER`

When **AFX** composes an element name with this scheme, `PROJECT_SHORT`, `ELEMENT_NAME`,and `ELEMENT_NUMBER` are replaced with the current values. The _ are intermediate texts and are kept.

To generate file-names via Windchill number-generator write the following:

`WINDCHILL_AUTONUMBER`

8.4 Start part configuration

All components in the **AFX** library are based on the same start part or start assembly. This start part or assembly may be different from the one you like to use. In order to match **AFX** components with your start part or assembly requirements you have the following possibilities:

- Rename datum planes and coordinate system features in the **AFX** library components to your desired names
- Add parameters to **AFX** library components, give them default values and designate them to Windchill
- Create layers in the **AFX** library components
- Create views in the **AFX** library components
- Change the accuracy type and value
- Change the tolerance type for dimensions

You can define these actions in a file named `start_mdl.cfg` in the **AFX** configuration directory. In standard installation of **AFX** you will find a template for this `start_mdl.cfg`. The template file is `named _start_mdl.cfg` (see below). If you remove the _at the beginning of the file name, the modifications in the **AFX** components will be performed.

NOTE: The `start_mdl.cfg` configuration will only apply to **NEW components**!

The syntax for the different modifications is as follows (lines which begin with ! are comments):

```
!FEATURE NAMES OF STANDARD FEATURES
FEATURE_NAME XY START_XY
FEATURE_NAME XZ START_XZ
FEATURE_NAME YZ START_YZ
FEATURE_NAME CS0 START_CSYS
FEATURE_NAME A_XY A_START_XY
FEATURE_NAME A_XZ A_START_XZ
FEATURE_NAME A_YZ A_START_YZ
FEATURE_NAME ACS0 A_START_CSYS
!VIEWs TO CREATE OR TO MODIFY
CREATE_VIEW TEST_VIEW
1 1 0
-1 1 0
0 0 1
!PARAMETERS TO CREATE
CREATE_PARAMETER TEST_INT_PARAM INTEGER 17
CREATE_PARAMETER TEST_DOUBLE_PARAM DOUBLE 17.0
CREATE_PARAMETER TEST_STRING_PARAM STRING XYZ
```

```
!LAYERS TO CREATE
CREATE_LAYER TEST NORMAL
CREATE_LAYER MY_DTM_PLANE BLANK DATUM_PLANE
!SET MODEL ACCURACY SET_ACCURACY ABSOLUT 0.01
!SET DEFAULT TOLERANCE TYPE
SET_DEFAULT_TOL_TYPE DEFAULT
```

Rename Features

A line with the following syntax will rename one of the standard features:

`FEATURE_NAME <original feature name> <new feature name>`

The original feature names are:

- **XY**, **XZ**, **YZ** for the three default datum planes in parts
- **CS0** for the standard coordinate system in parts
- **A_XY**, **A_XZ**, **A_YZ** for the three default datum planes in assemblies
- **ACS0** for the standard coordinate system in assemblies
- The **new feature name** is user defined

Create Parameters

A line with the following syntax will create a parameter in the part or assembly:

`CREATE_PARAMETER <parameter name> <parameter type> <default value> DESIGNATE`

Explanation:

- **parameter name** is the name of the parameter to create.
- **parameter type** is the data type of the parameter. Valid types are `INTEGER`, `DOUBLE` and `STRING`.
- As default value you can write an **INTEGER** number, a **DOUBLE** number or a **STRING** as default value for the parameter.
- The string **DESIGNATE** at the end of the line is optional. If it exists then the created parameter will be designated to **Windchill**.

Create Rule Layers and Combined States

In order to work with rule based layers and combined states you will need to set the following options:

CREATE_RULE_LAYER_PARAM YES

DELETE_LAYER ALL

RULE_LAYER_TEMPLATE afx_start_model

They have the following effect:

CREATE_RULE_LAYER_PARAM: All features will receive a parameter that makes the rule layer template easier to configure

DELETE_LAYER ALL: Deletes all existing layers from the model

RULE_LAYER_TEMPLATE afx_start_model: Copy all layers and combined states from the afx_start_model.

You can find a good example in the folder afx/configuration/buw_start_configuration_example

Manage Views

Create Views:

A block with the following syntax will create a view in the part or assembly:

- `CREATE_VIEW <view name>`
- `trans_matrix[0][0] trans_matrix[0][1] trans_matrix[0][2]`
- `trans_matrix[1][0] trans_matrix[1][1] trans_matrix[1][2]`
- `trans_matrix[2][0] trans_matrix[2][1] trans_matrix[2][2]`

Explanation:

- **View name** is the name of the view to create.
- **trans_matr[i][j]** are the numbers in the transformation matrix from model default coordinate system to the coordinates of the view to create.

You can get the matrix by the following steps:

1. Create an empty **part** or **assembly**.
2. Create a **coordinate system**. It will represent the model **default coordinate system**.
3. If needed create additional features for orientation.
4. Orient model with **Creo Parametric view manager** functions.
5. Create a **coordinate system** using the **default coordinate** system as reference and using the option **Set Z Normal To Screen**
6. Obtain the transformation from this coordinate system to default coordinate system by using the **[Analysis] > [Measure]** function of **Creo Parametric** and choose type Transform. This function gives you a transformation matrix. You have to enter these values as `trans_matr[i][j]` in the view creation block.

Rename Views:

Use the following syntax to rename existing views:

- RENAME_VIEW +XY FRONT
- RENAME_VIEW -XY BACK
- RENAME_VIEW -XZ BOTTOM
- RENAME_VIEW -YZ LEFT
- RENAME_VIEW +YZ RIGHT
- RENAME_VIEW +XZ TOP

Change model accuracy

A block with the following syntax will change the default accuracy:

```
SET_ACCURACY <type> <value>
```

Explanation:

- **type** the type can be set to **RELATIVE** or **ABSOLUT**
- **value** is a double value (e.g. 0.01).

Change the tolerance type for dimensions

A block with the following syntax will change the tolerance type for dimensions:

```
SET_DEFAULT_TOL_TYPE
```

Explanation:

type the type can be set to **LIMITS**, **NORMAL** or **PLUS_MINUS**

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