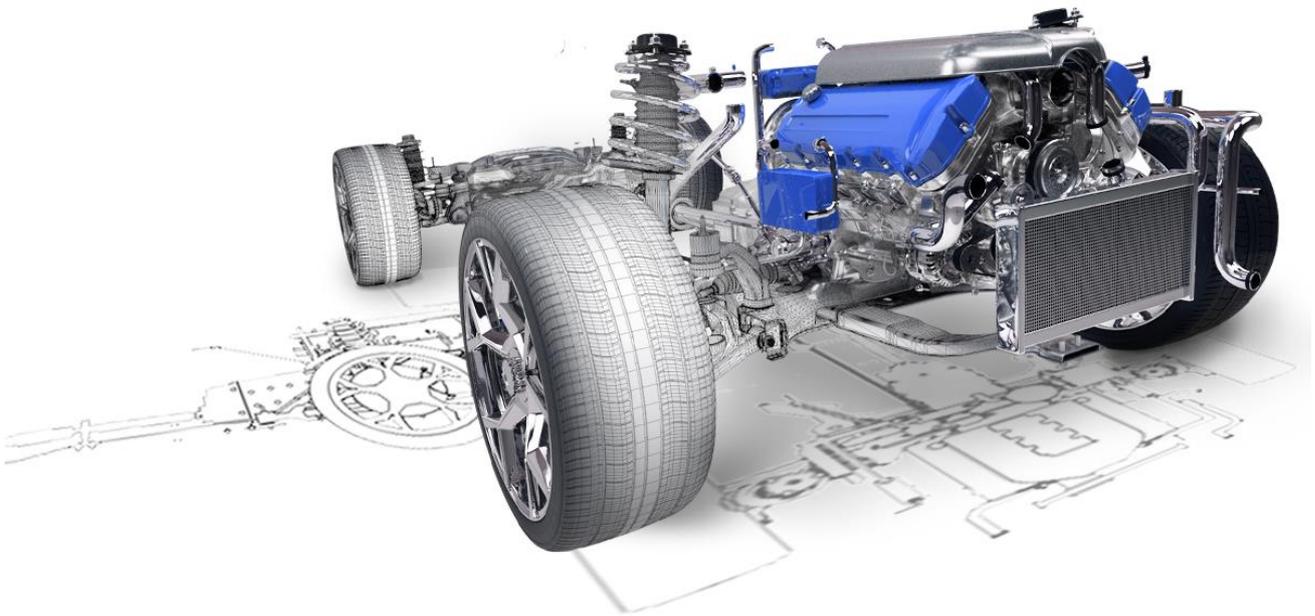


ZW3D



ZW3D From Entry to Master

Weldments

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Weldments

Key Points:

- ✧ Common weldment features
- ✧ Weldment editing functions
- ✧ Create weldment part from line or sketch
- ✧ Create Structural BOM table

Weldment is a necessary module for designing. It can improve the design efficiency. In ZW3D, weldment model is a multi-shape part, but each shape can be treated as one component and list it in the structure BOM. And weldment module includes creation function for path and frame feature, as well as some basic editing tools.

1.1 Frame Functionality

With those frame functions, you can create weldment shapes easily.

1.1.1 Structural Member



Weldments ribbon toolbar->Frame->

Use this command to create a serial of weldment section. Firstly, a sketch or 3D curves should be prepared by Sketch/Line/Polyline command, which must be connected curves or non-parallel lines.

- STEP 01** Select the standard, such as GB
- STEP 02** Define weldment type and section size
- STEP 03** Select the curve to locate the section
- STEP 04** Define the corner treatment, angle and locate

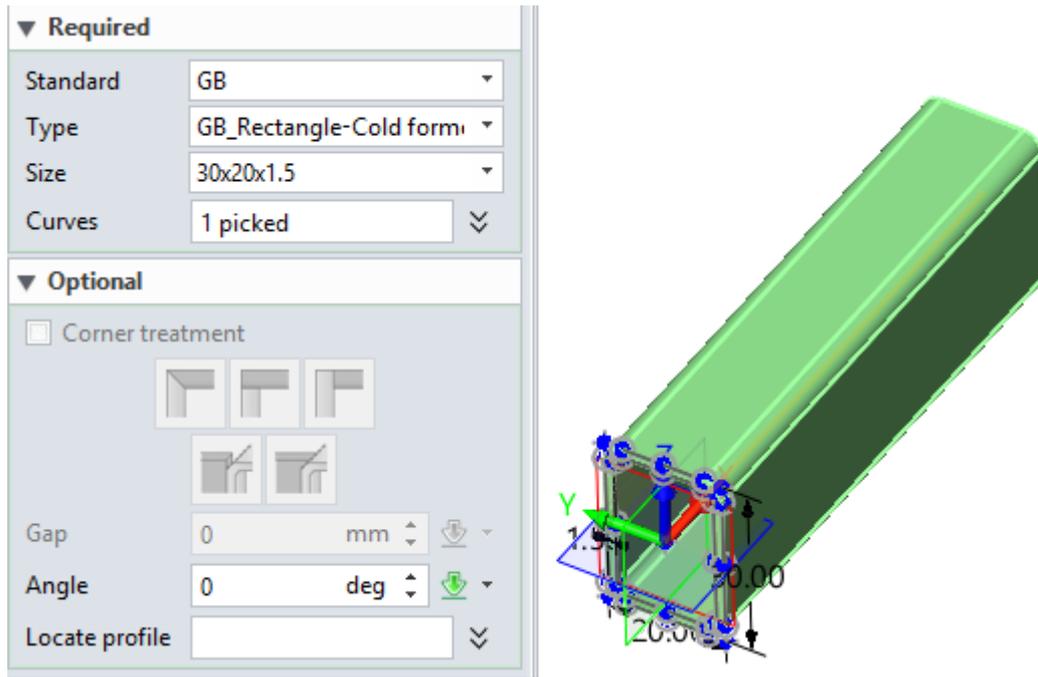


Figure1 Create Weldment Structural Member

Corner Treatment: Use this option to define corner status between two connected structural members.

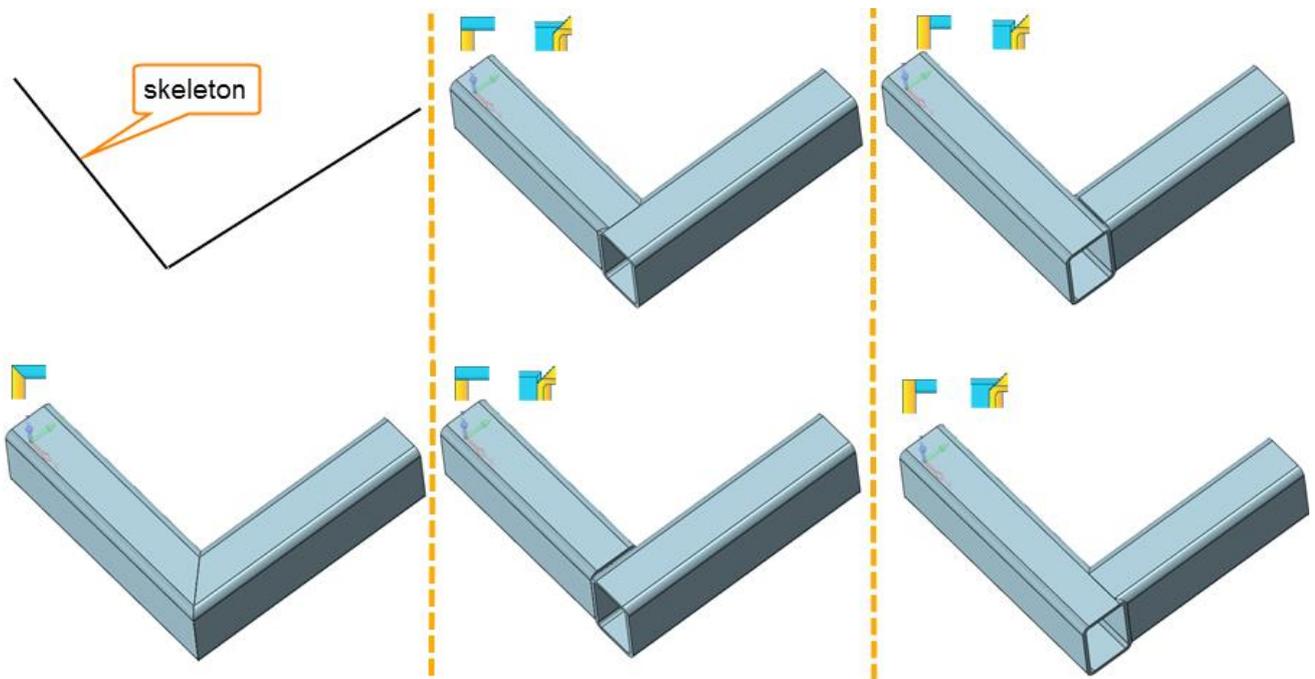


Figure2 Corner Treatment

Angle: Set the rotate angle for the profile.

Locate Profile: Set the a new location for the profile by picking another point of profile.

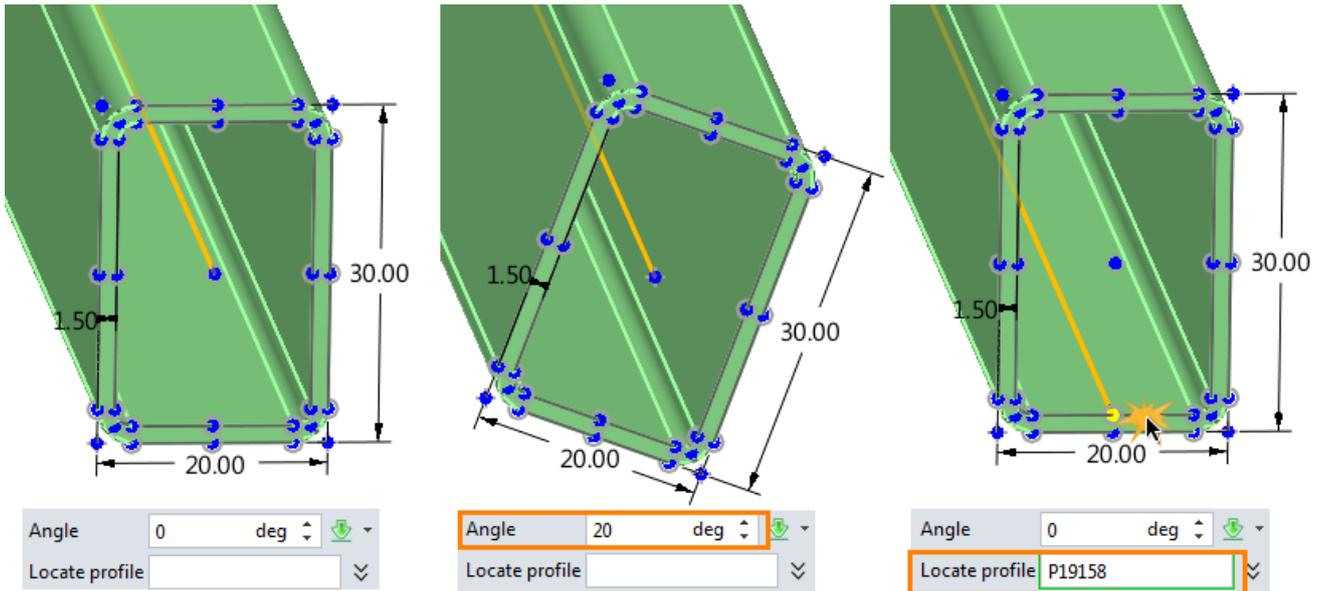


Figure3 Relocate the Profile



Tips: how to define the customized weldment profile?

STEP 01 Create one or more Stand Alone sketches in the file that includes the weldment profile and location point.

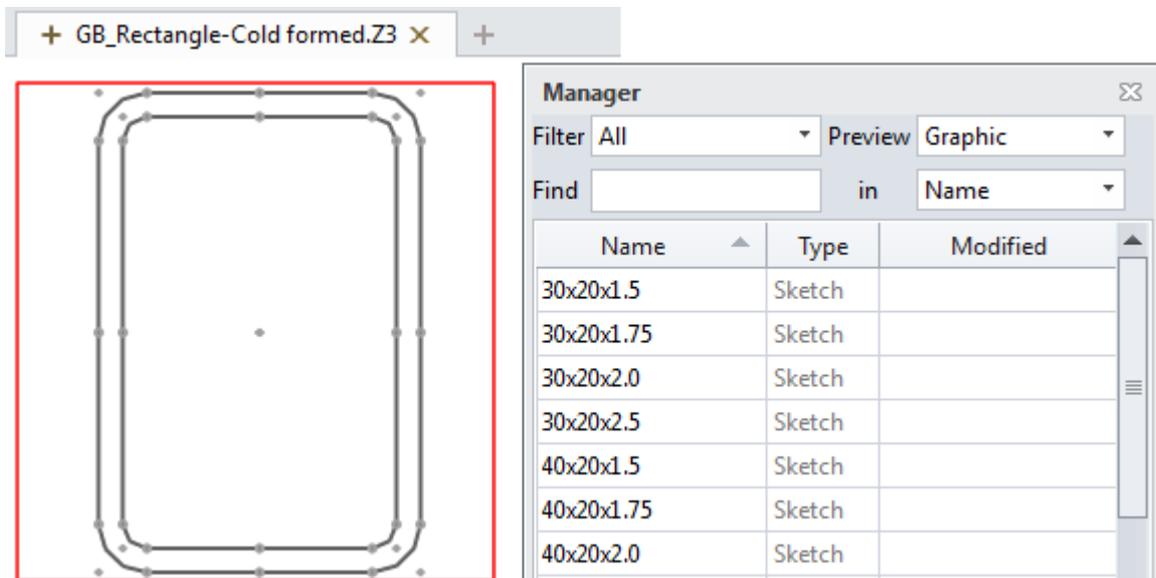


Figure4 Customize Wledment Profile

STEP 02 Save the file to the install dirtory\resource.

You can save this file to the existing folder or new created folder.

STEP 03 Call the customized profile by Structural Member command.

1.1.2 Trim



Weldments ribbon toolbar->Frame->

This is an independent command for corner treatments which you can perform onto linear structure members. It supports three different corner treatments and two different extensions. You can choose which side of the structure members to keep.

STEP 01 Select the corner treatment type

STEP 02 Select the trim entity, and reverse the direction if necessary

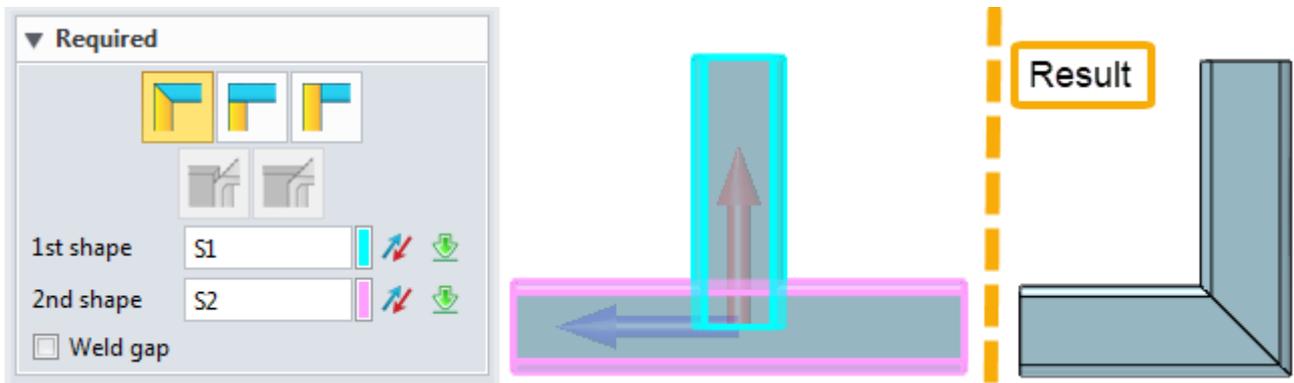


Figure5 Trim

The rest options are same with that in Structure member.

1.1.3 Gusset



Weldments ribbon toolbar->Frame->

Use this command to create gusset. Gusset is used to reinforce structure strength around the corner portion of two intersection structural members.

STEP 01 Select two surfaces around one corner

STEP 02 Define gusset type and dimension parameters

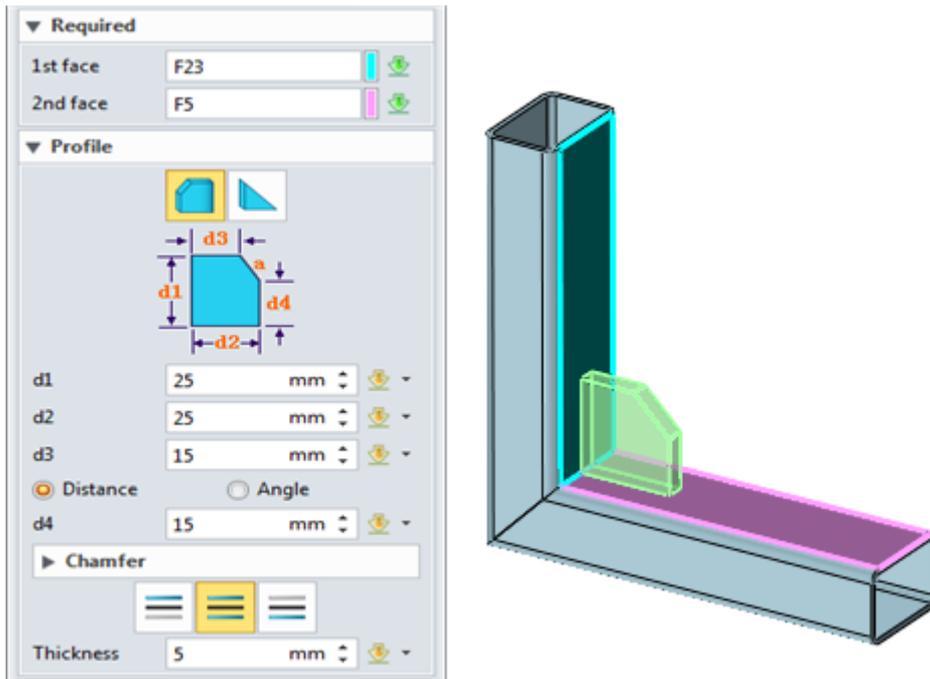


Figure6 Gusset

There are two types gusset: Polygonal and Triangular.

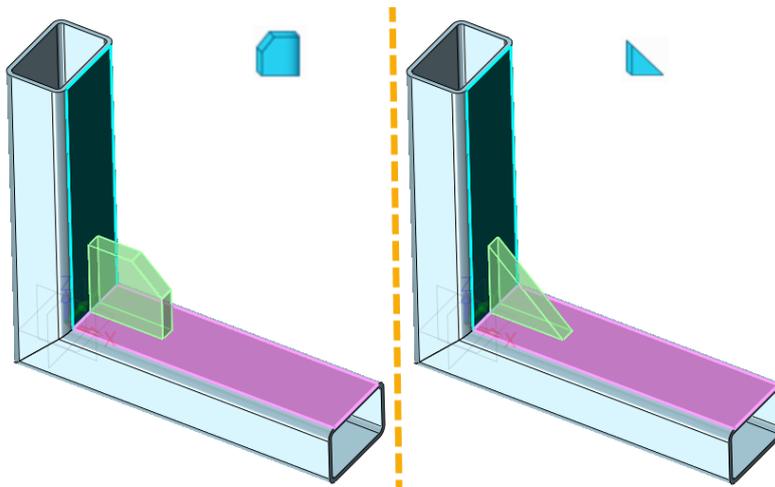


Figure7 Polygonal and Triangular gusset

Gusset position: There have 3 types: inner, middle and side.

Inner: the inner face is coplanar with middle plane of selected face.

Middle: the middle face of gusset is coplanar with middle plane of selected face.

Side: The other side face is coplanar with middle plane of selected face.

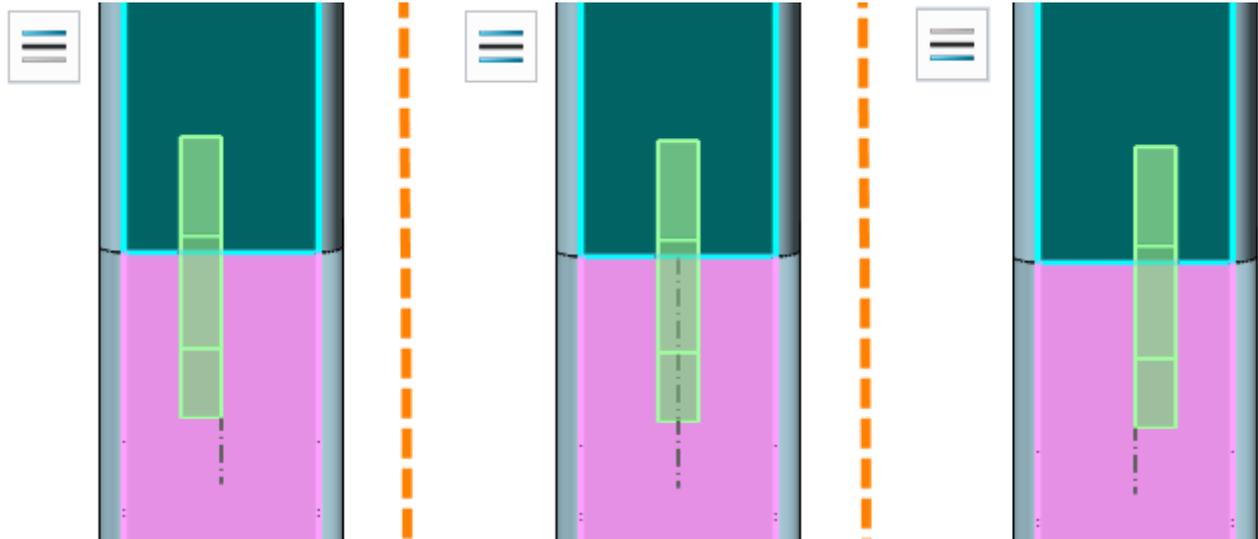


Figure8 Gusset Position

Location: Define the location of the gusset. There are three types: start, middle, end.

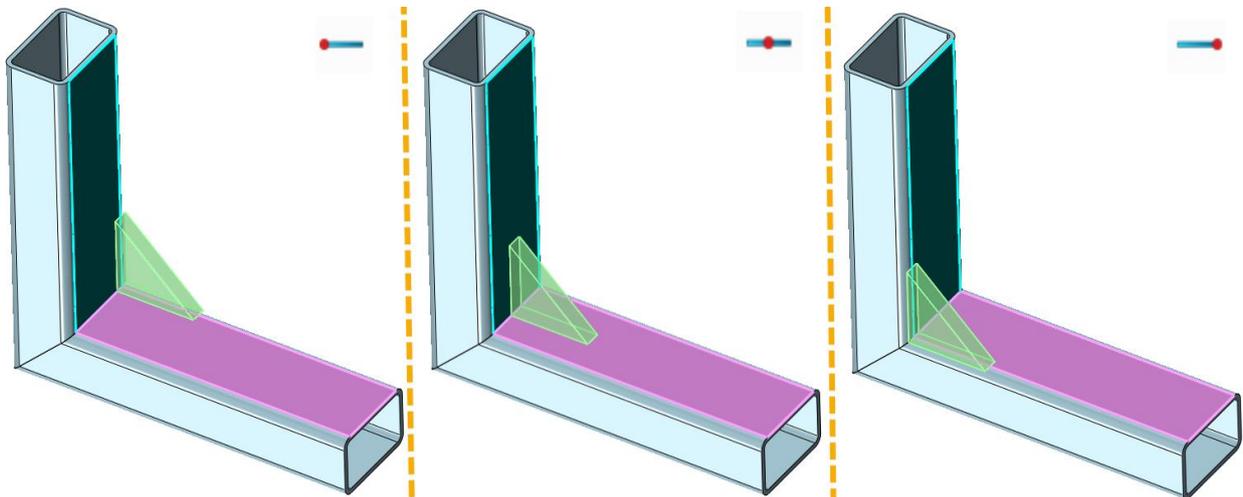


Figure9 Gusset Position

Offset: When this option is checked, it's allowed to set offset value to locate the gusset.

Note: Gusset only can be built on planar faces which intersect each other.

1.1.3 Endcap



Weldments ribbon toolbar->Frame->

Use this command to create end cap. End cap is used to close the open end of the structure member. Only planar faces with one inside loop can be used to add an end cap.

STEP 01 Select the cap face

STEP 02 Set direction and thickness value

STEP 03 Define the offset and chamfer corner

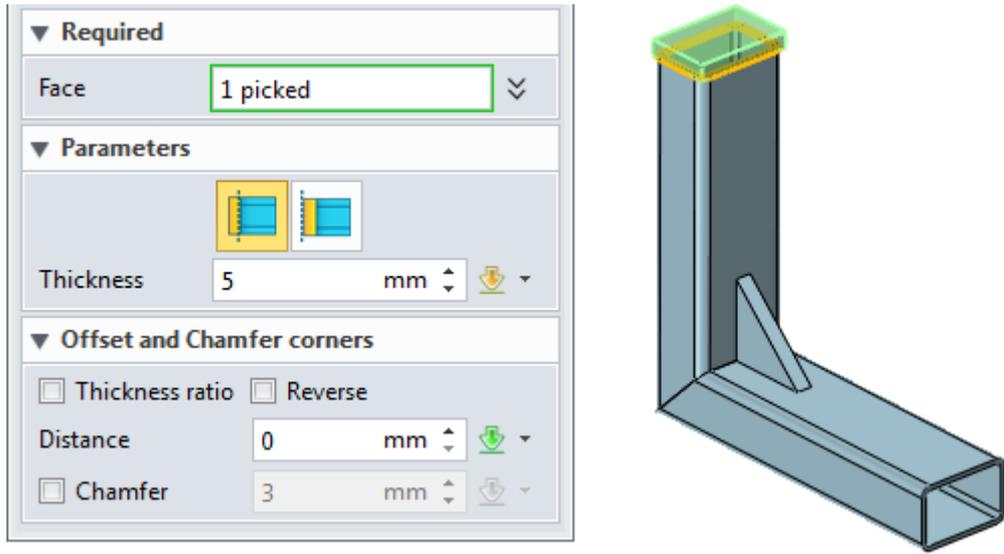


Figure10 Endcap

Parameter: Set the endcap direction and thickness.

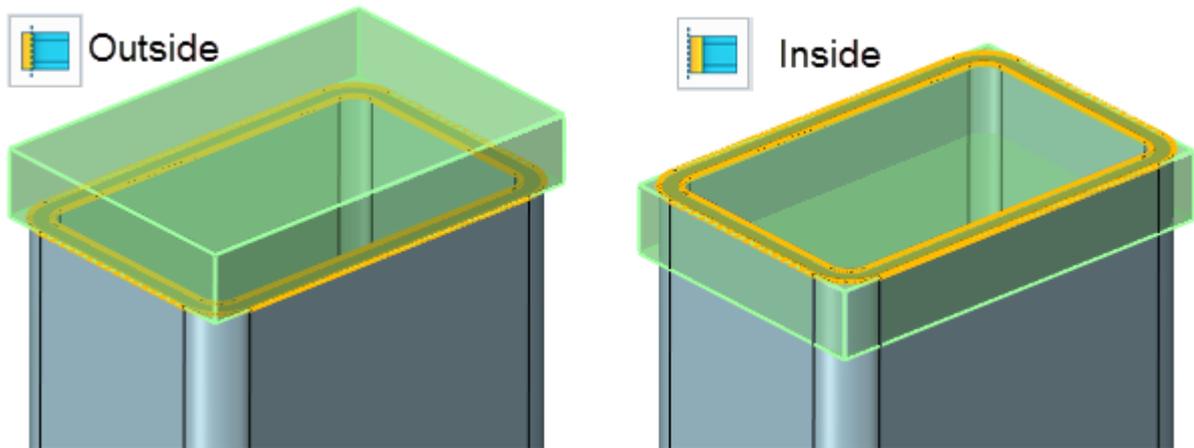


Figure11 Endcap side

Offset & Chamfer: define the offset distance and corner chamfer.

When “Thickness ratio” is checked, the offset distance is equal to structure member thickness multiply the ratio. The ratio range is from 0 to 1. Otherwise, offset distance can be directly defined.

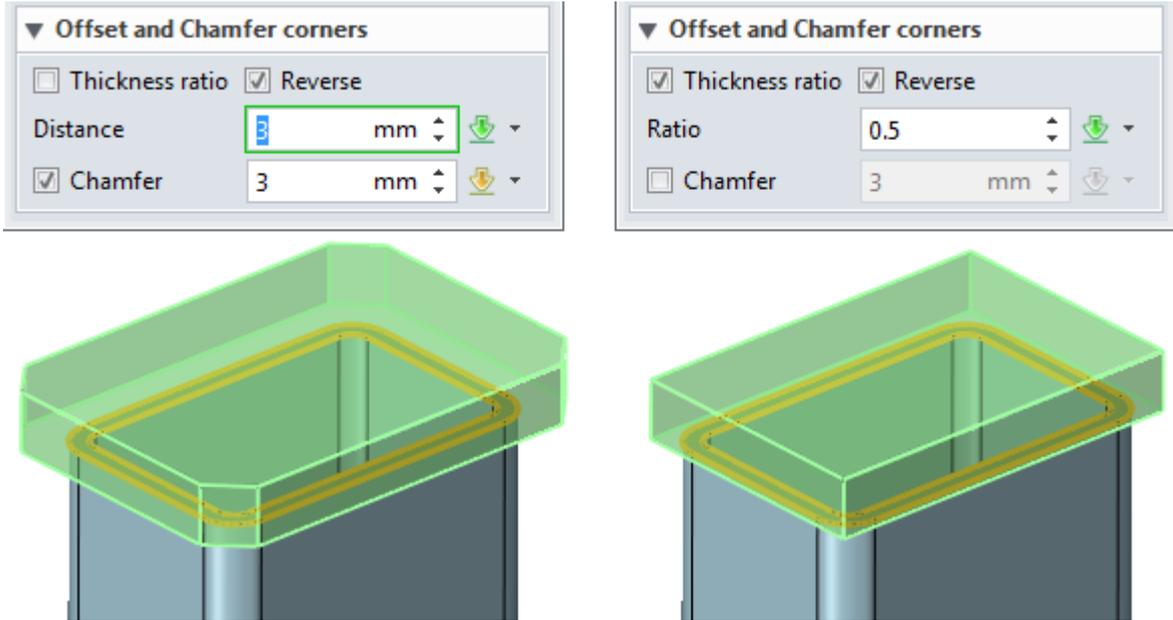


Figure12 Endcap Offset and Chamfer Corner

1.1.4 Weldbead

Weldments ribbon toolbar->Frame-> 

Use this command to create weld bead. The weld bead is used to join structure member together. It can be continuous or intermittent.

STEP 01 Select the edge or face that want to add weld bead.

STEP 02 Define the start and length.

STEP 03 Set the intermittent if necessary.

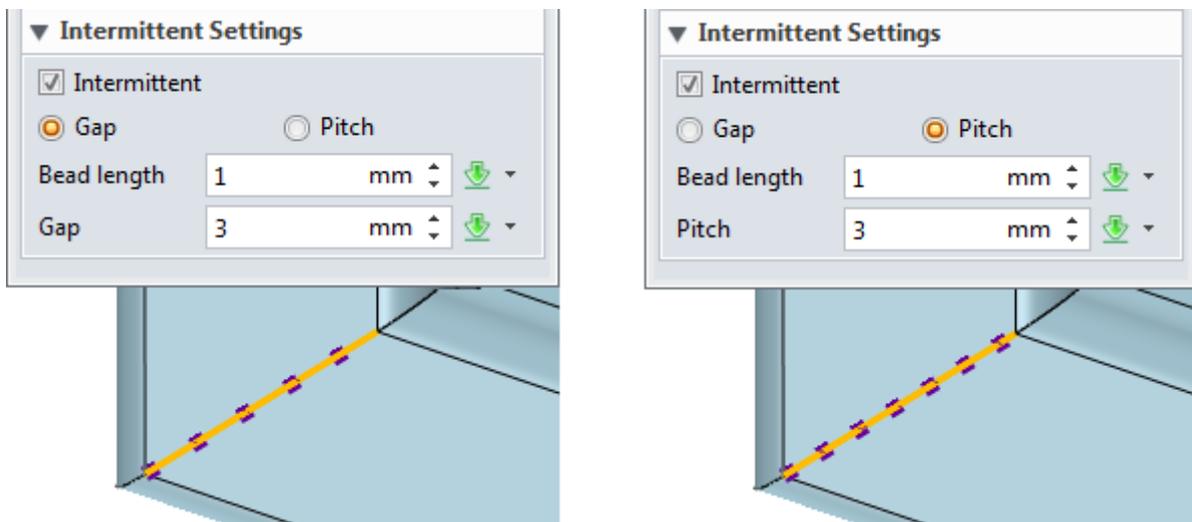


Figure13 Intermittent Settings

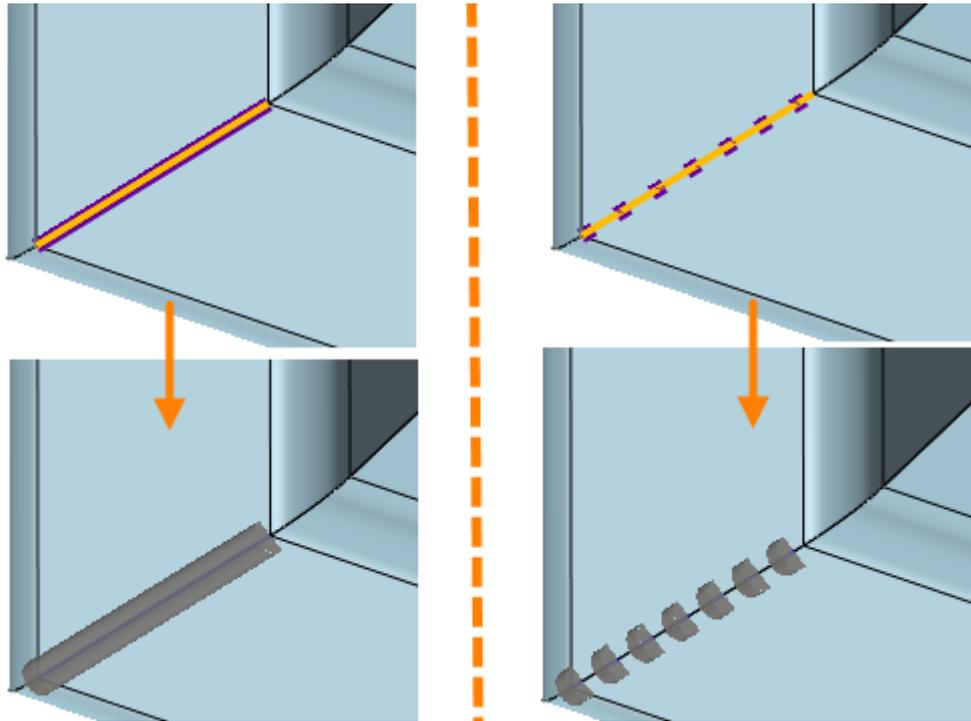


Figure14 Weld Bead

1.1.5 Structure BOM

Weldments ribbon toolbar->Frame-> 

Use this command to generate a table of all weldment elements. The table supports the features as below.

Weld structural list

Structural Attributes
Structural Table

Structural Attributes	Structural Table	
Property name	Value	
Structural1.0	Spec	GB_Rectangle-Cold formed,30x20x1.5
Structural1.1	Material	Aluminum
Gusset1	Mass	0.0357(kg)
EndCap1	Length	110[mm]
	Description	Structural
	List in BOM	<input checked="" type="checkbox"/>

Add
Delete

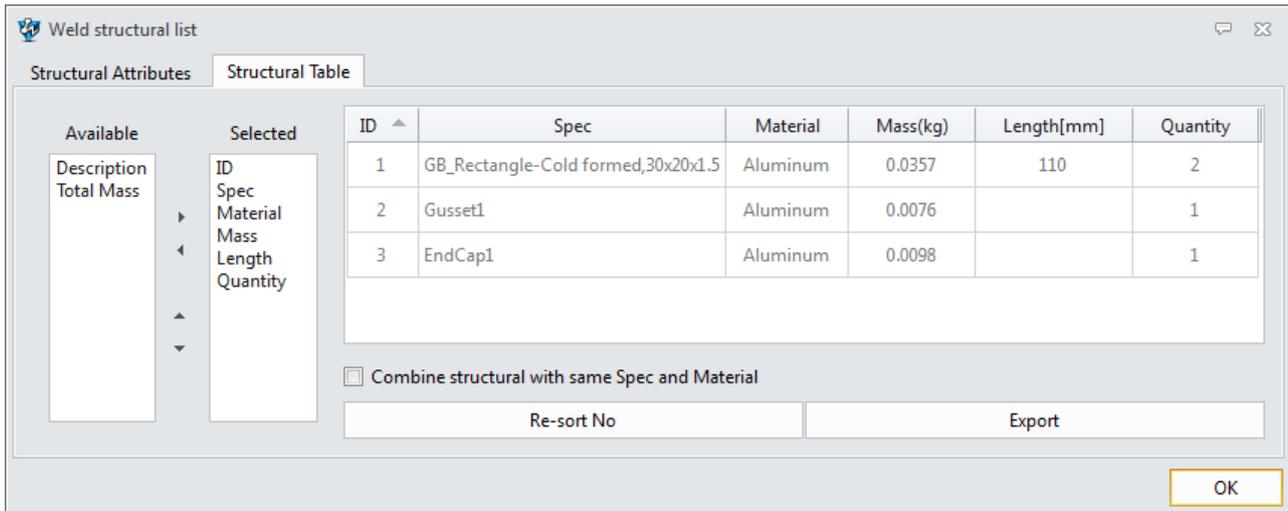


Figure15 Structure BOM

If the option “Combine structural with same Spec and Material” is checked, the rows with same attributes will be merged together.



Tips: How to show the Mass for structure member?

STEP 01 Set the material for all the structure member shapes.

STEP 02 Inquire the mass properties.

1.2 Case---Weldment

In this module, you can learn how to use those weldment functions to design your own product. The following case will show you the general process of weldment in ZW3D.

Through below example to, you will learn how to use Structure member, endcap, gusset, structure BOM, etc.to create weldment part.

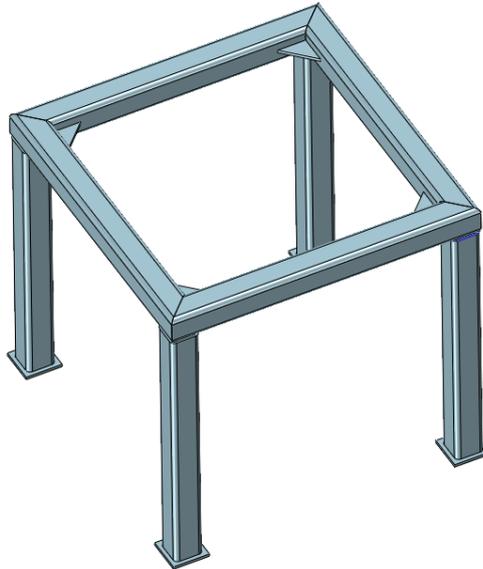


Figure16 Case- Table Support

1. Create the Frame sketch

STEP 01 In the Shape ribbon tab, select 3D sketch function. Then create a sketch like below.

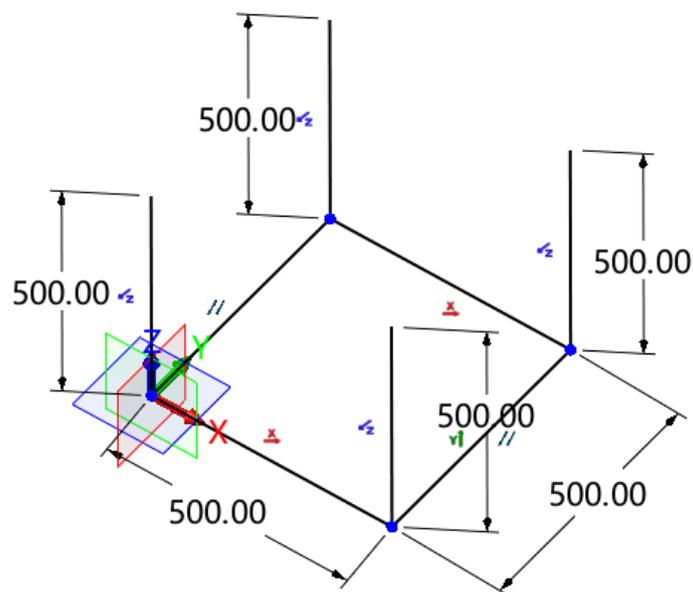


Figure17 3D Sketch1

2. Create Structure Member

STEP 01 Click Structure member command and select 4 lines, as shown in the image below.

STEP 02 Define the standard to ISO, and type to ISO_Square-Cold formed, set the size to 50x50x4. And select the corner treatment to end miter with default parameters.

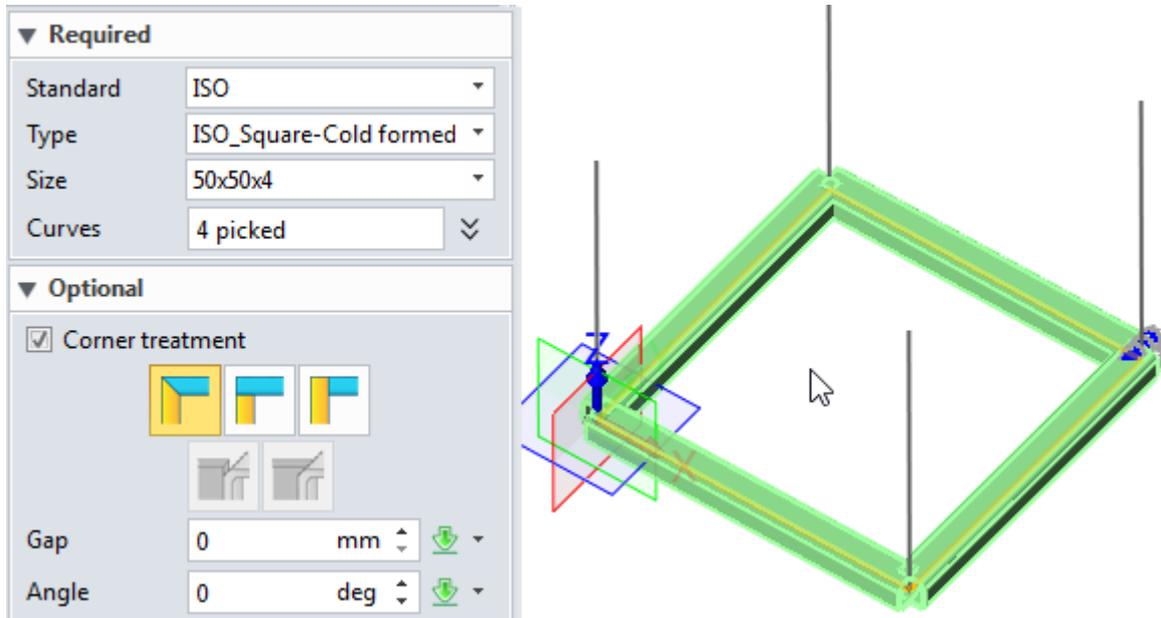


Figure18 Structure Member

3. Create Table Supports

STEP 01 Middle click to repeat the last operation -Structural member.

STEP 02 Select 4 vertical lines and set the parameters which are the same with the last operation.

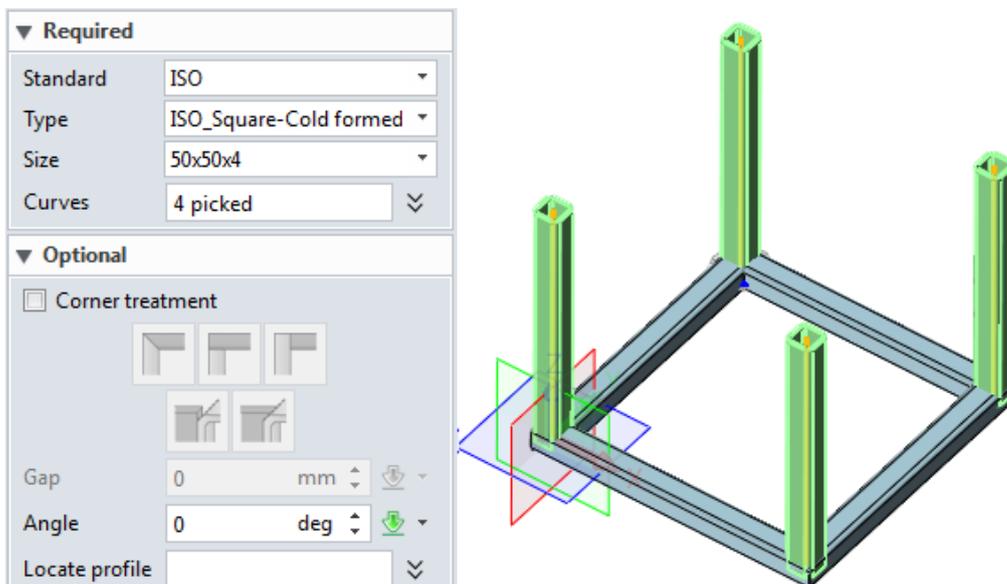


Figure19 Supports

4. Trim Structure Member

STEP 01 Select trim function from Shape Ribbon tab.

STEP 02 Refer to the image below to trim supports.

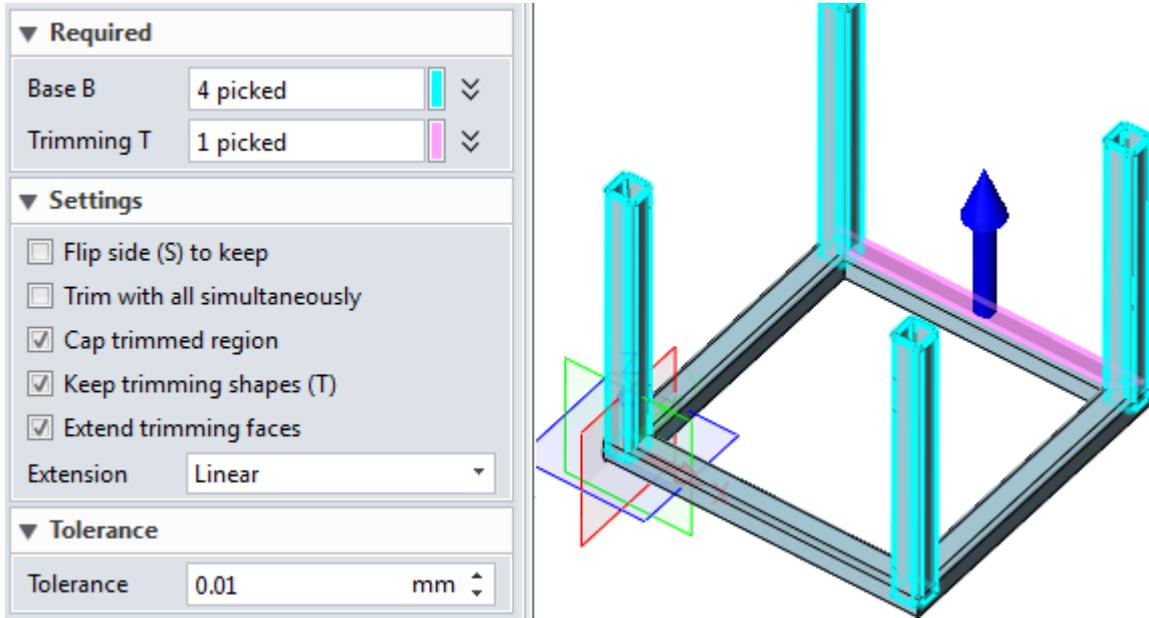


Figure20 Trim Structure Member

STEP 03 Check “Extend Trimming Faces” option to get the following result.

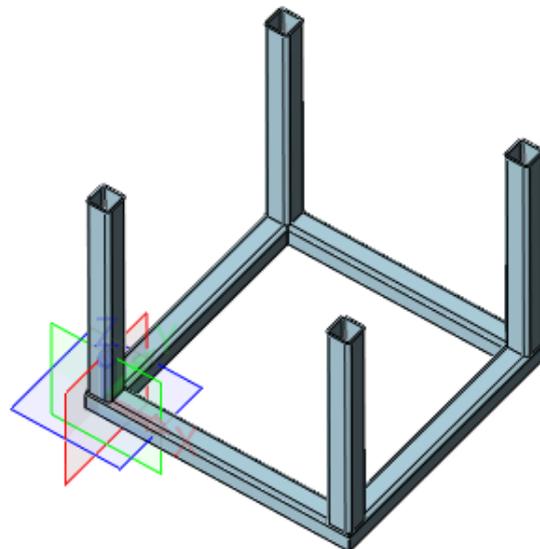


Figure21 Trim Result

5. Create Endcap

STEP 01 Select endcap function.

STEP 02 Select end faces from supports as below.

STEP 03 Check “Reverse” option and set the distance value to 10.

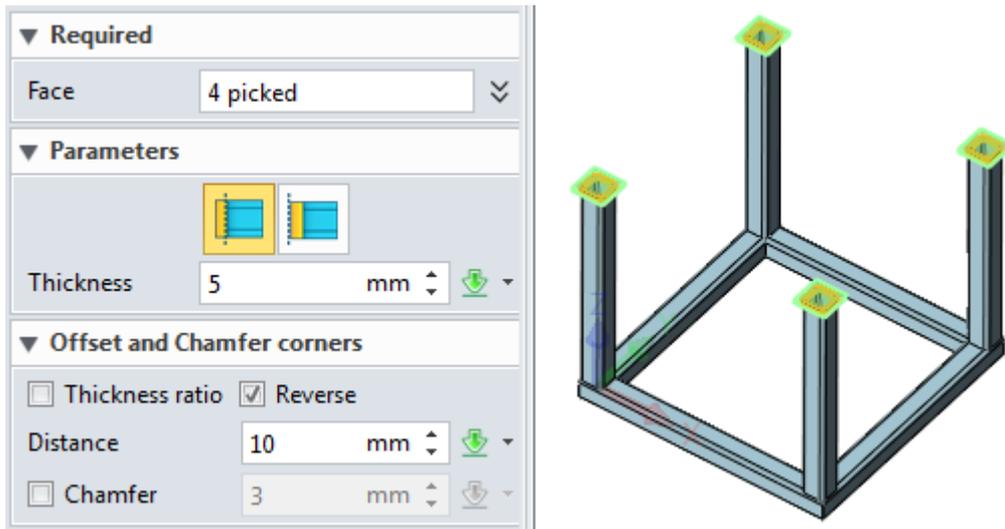


Figure22 Endcap

6. Create Weld Bead

STEP 01 Select weld bead function.

STEP 02 Select loop curves like below.

STEP 03 Set the radius to 1 and keep other parameters in default.

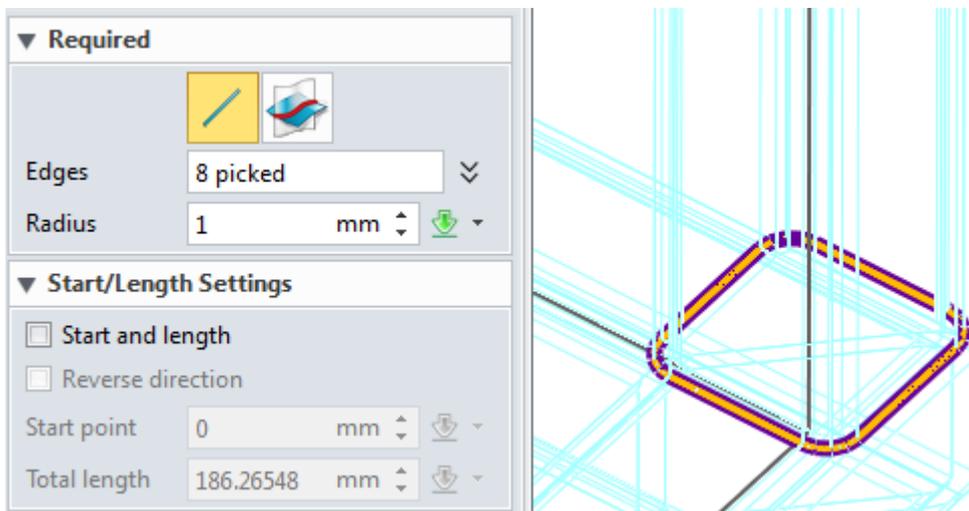


Figure23 Weld Bead

Use the same method to create the other three weld beads.

7. Create Gusset

STEP 01 Select gusset function.

STEP 02 Select the cross faces, and set the type to triangular, and set d1 & d2 to 50.

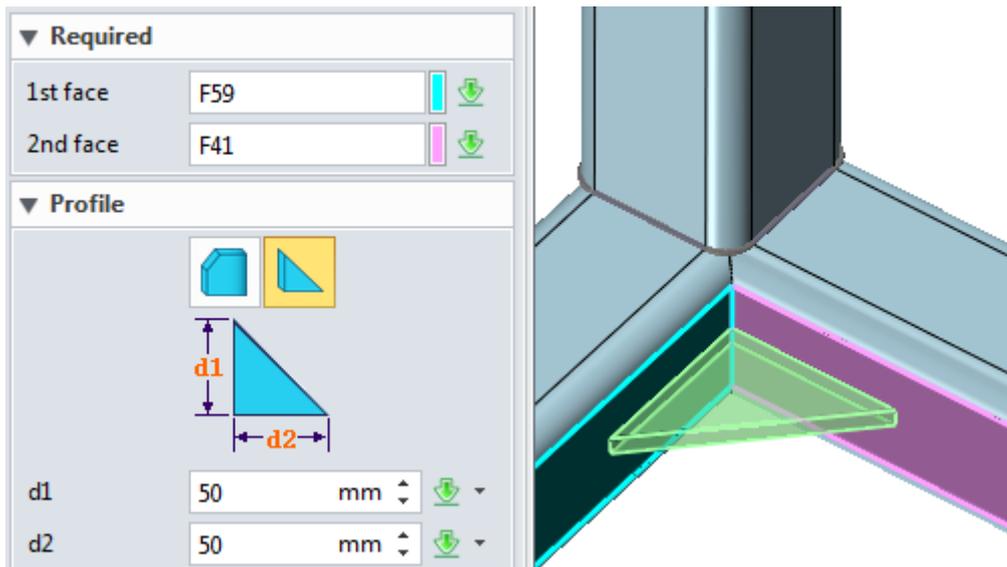


Figure24 Gusset

Then we can use the same method to create other 3 gussets. The final model is shown in the following image.

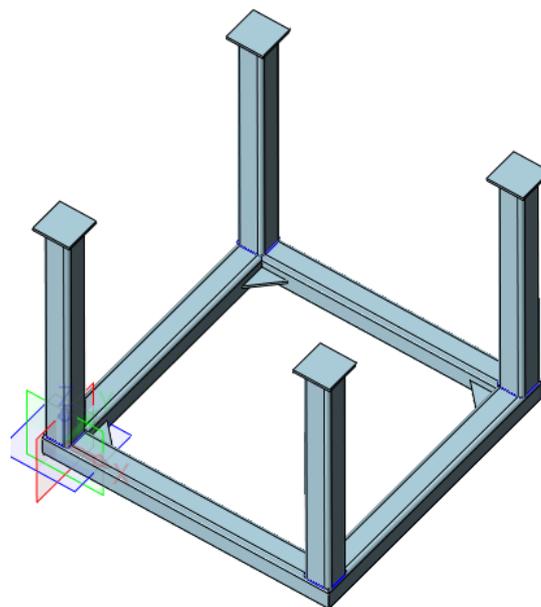


Figure25 Result