Model created in COMSOL Multiphysics 6.4

Busbar Assembly Geometry — with Geometry Parts

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Geometry parts provide a way to organize, parameterize, and reuse geometries that you create in COMSOL Multiphysics. They can be used to simplify geometry creation by providing easy-to-use parts with a number of parameters for tailoring the part's shape or dimension when added to a COMSOL Multiphysics geometry.

To create the geometry in a part you use geometry operations just as you would normally do, but these are added to the local geometry sequence of the part. To parameterize the geometry you can define a number of input parameters that will be available when a part instance is inserted into a geometry sequence. In addition, local parameters can help when only local parameterization is needed.

Just as when creating any regular geometry, you can use selections in geometry parts to simplify not only the geometry generation, but also material and physics assignment. You can access selections that you have defined in a part sequence both locally within its sequence or when inserting a part instance into a geometry sequence.

An advantage of breaking up complex geometries with many objects into geometry parts is that you can work in a local coordinate system when creating the geometry within each part. When you are inserting the part into a geometry sequence, you can position it by specifying the coordinates, or by matching a coordinate system defined in the part with a coordinate system in the geometry.

If you create a number of useful geometry parts, it is a good idea to collect them in a user-defined part library. This way you can easily reuse your parts or share them with colleagues.

Follow this tutorial to create the busbar geometry used in the model Electrical Heating in a Busbar Assembly, while learning more about how to:

- Create geometry parts with local and input parameters
- Insert geometry parts into geometry sequences
- Position geometry parts by using work planes
- Create geometry parts that contain other parts
- Use selections defined in geometry parts

Busbar Assembly Geometry — with Group Nodes is the second part of this tutorial that describes how to organize a geometry sequence with a folder-like structure. The two tutorials in this series complement each other, and show methods to structure more complex geometry sequences.

Model Definition

This example contains the detailed steps to create the parameterized geometry used for the model Electrical Heating in a Busbar Assembly. The geometry for this model, displayed in Figure 1, includes the coupling components for one cell, and a section of the intercell busbar that is connected to a cell grid.

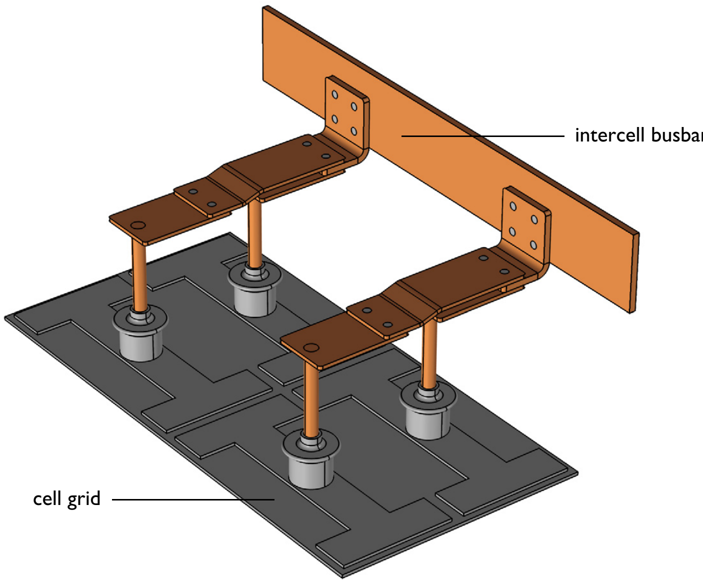


Figure 1: The busbar assembly.

Each component of the busbar is created as a separate geometry part, and a geometry part is also created for the components displayed in Figure 2.

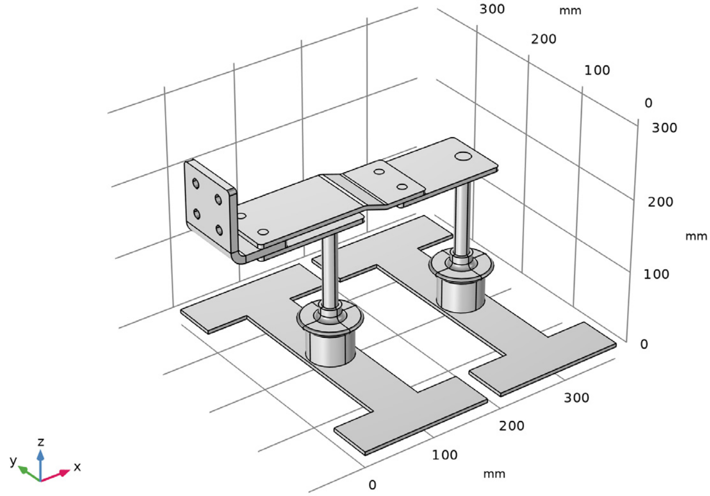


Figure 2: Subunit of the busbar, created in a geometry part.

This example describes only the process of creating the geometry sequence. For the physics setup, follow the instructions in Electrical Heating in a Busbar Assembly.

Application Library path: `COMSOL_Multiphysics/Geometry_Tutorials/busbar_assembly_geometry`

Modeling Instructions

COMSOL DESKTOP

1 From the **File** menu, choose **Open**.

2 Browse to the model's Application Libraries folder and double-click the file `busbar_assembly_geom_subsequence.mph`.

This file contains all but two of the geometry parts for the busbar. In the following you will create the remaining parts and build the busbar geometry. First, check where the geometry parts appear in the model tree.

3 If necessary expand the **Geometry Parts** section under the **Global Definitions** node in the **Model Builder** window.

The geometry parts that appear here are not attached to a specific model component, but can be inserted into the geometry sequence of any model component of the appropriate space dimension. To edit the geometry sequence for a part you can expand the part's node.

Continue by adding a new geometry part.

ANGLE CONNECTOR

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts** > **3D Part**.

2 In the **Settings** window for **Part**, type `Angle Connector` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Default expression	Value	Description
a_c_w_part	90[mm]	0.09 m	Angle connector width

The parameters listed here are available within the part, and can also be specified with new values when you insert the part into a geometry sequence.

4 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Local Parameters

Local parameters are only available within the part. However, they can be defined by expressions containing input parameters.

1 In the **Model Builder** window, expand the **Angle Connector** node, then click **Local Parameters**.

2 In the **Settings** window for **Local Parameters**, locate the **Local Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_h_part	6[mm]	6 mm	Angle connector height
e_c_h_part	10[mm]	10 mm	Elbow connector height
c_g_w_part	400[mm]	400 mm	Cell Grid Top width
b_di_part	20[mm]	20 mm	Bolt to boundary distance
b_r_part	6[mm]	6 mm	Bolt radius

Create the geometry of the angle connector as the intersection of two solid objects: the extrusion of the side view and the extrusion of the top view. Start with drawing and extruding the side view.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Since the work plane is only needed for geometry creation it does not need to be displayed in the part instances.

4 Locate the **Part Instances** section. Clear the **Show work plane in instances** checkbox.

Leave the Sketch mode and create the geometry by entering the polygon coordinates.

5 Click **Go to Plane Geometry** at the top of the **Settings** window.

Work Plane 1 (wp1) > Plane Geometry

In the **Sketch** toolbar, click **Sketch** to toggle the Sketch visualization off.

Work Plane 1 (wp1) > Polygon 1 (pol1)

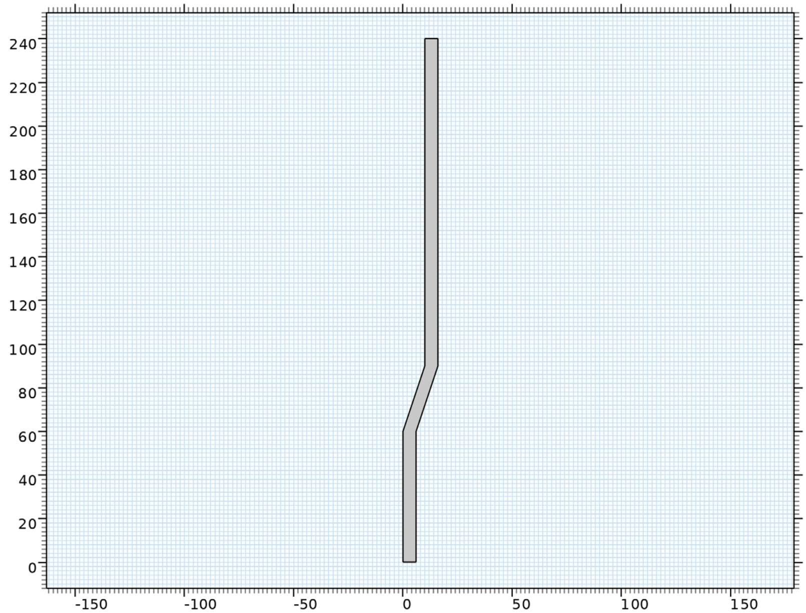
1 In the **Work Plane** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (mm)	**yw (mm)**
0	0
0	60[mm]
e_c_h_part	90[mm]
e_c_h_part	c_g_w_part/2+b_di_part*2
e_c_h_part+a_c_h_part	c_g_w_part/2+b_di_part*2
e_c_h_part+a_c_h_part	90[mm]
a_c_h_part	60[mm]
a_c_h_part	0

4 Click **Build Selected**.



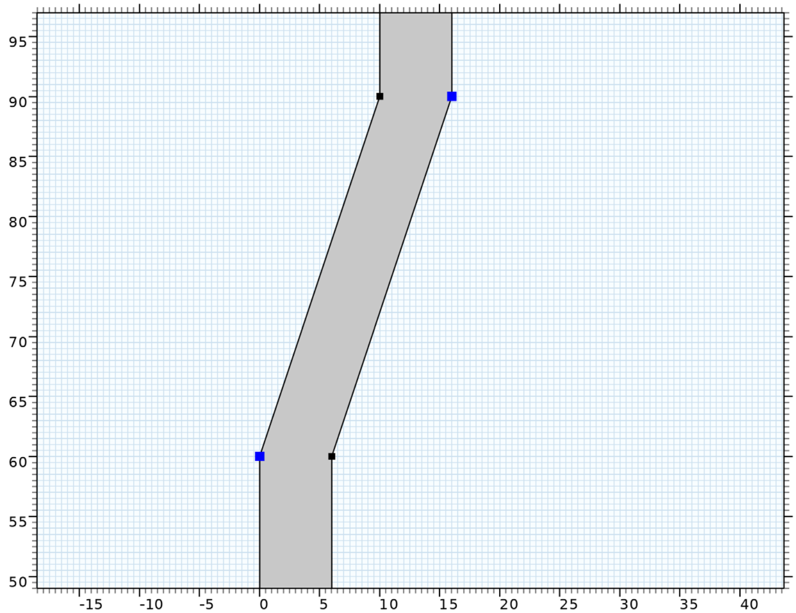
Work Plane 1 (wp1) > Plane Geometry

Add **Fillets** to some of the corners. First, zoom in to the region of the bend on the cross section.

Work Plane 1 (wp1) > Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.

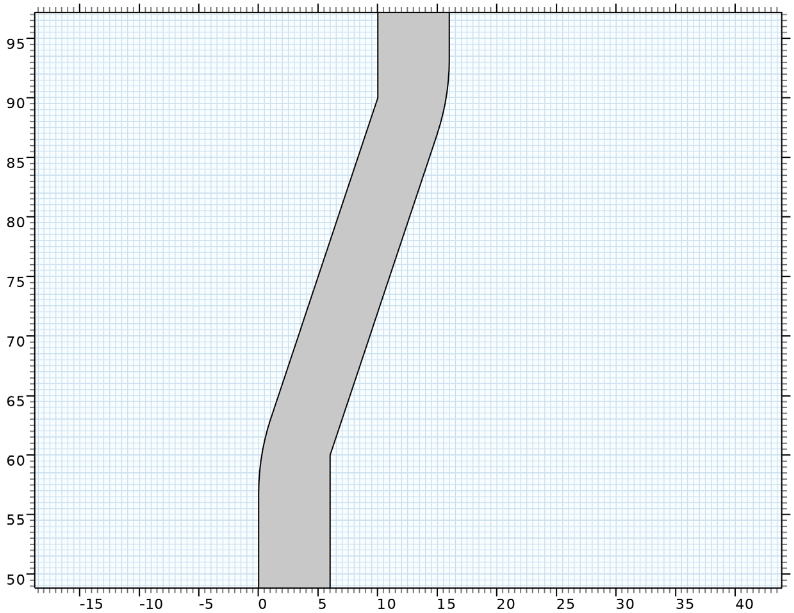
2 On the object **pol1**, select Points 2 and 7 only.



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `20[mm]`.

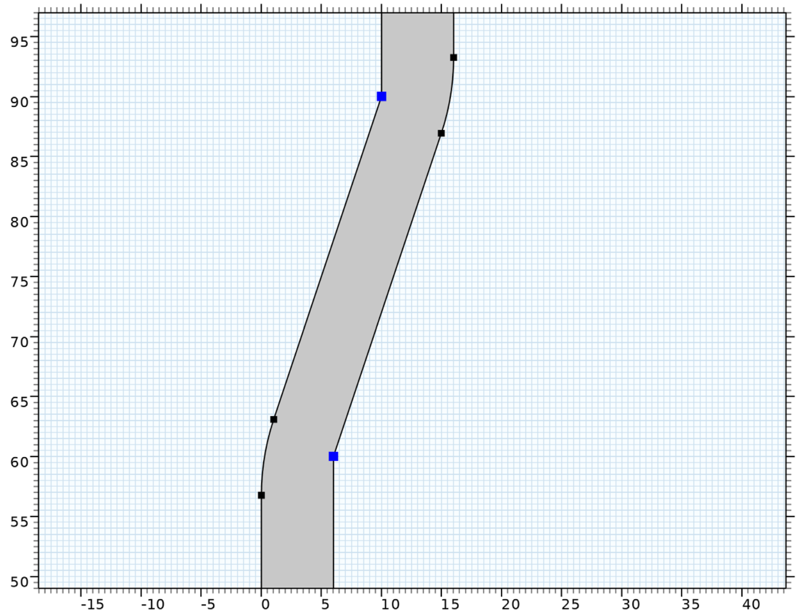
5 Click **Build Selected**.



Work Plane 1 (wp1) > Fillet 2 (fil2)

1 In the **Work Plane** toolbar, click **Fillet**.

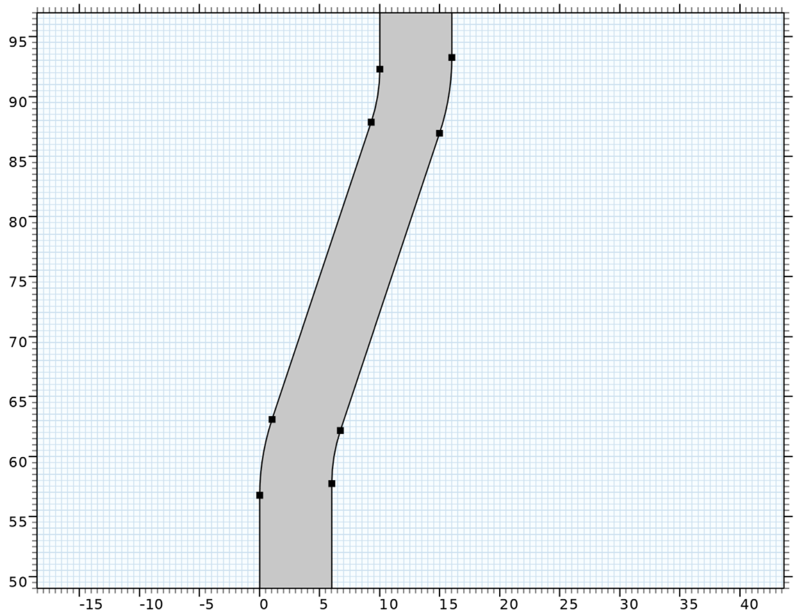
2 On the object **fil1**, select Points 5 and 6 only.



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `20[mm]-a_c_h_part`.

5 Click **Build Selected**.



Extrude 1 (ext1)

1 In the **Model Builder** window, under **Global Definitions** > **Geometry Parts** > **Angle Connector** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

You can use both input and local parameters directly in the feature.

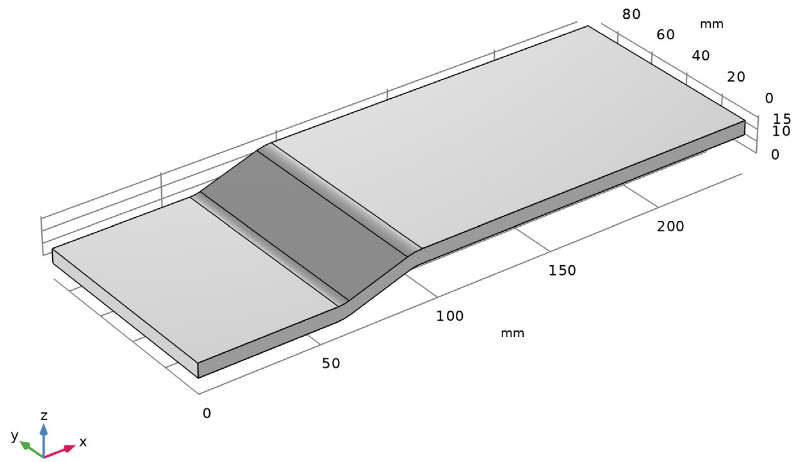
2 Select the first row in the **Distances** table.

3 Click **Insert Expression** below the **Distances** section.

4 Expand the **Input Parameters** section and select **a_c_w_part**.

5 In the **Settings** window for **Extrude**, click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.



The solid object extruded from the side view is now ready. Continue by drawing and extruding the top view of the angular connector.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Part Instances** section.

3 Clear the **Show work plane in instances** checkbox.

Work Plane 2 (wp2) > Plane Geometry

1 In the **Model Builder** window, click **Plane Geometry**.

2 In the **Sketch** toolbar, click **Rectangle**.

Draw a rectangle somewhere on the canvas and then adjust its size and position.

Work Plane 2 (wp2) > Rectangle 1 (r1)

1 In the **Model Builder** window, click **Rectangle 1 (r1)**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `c_g_w_part/2+b_di_part*2`.

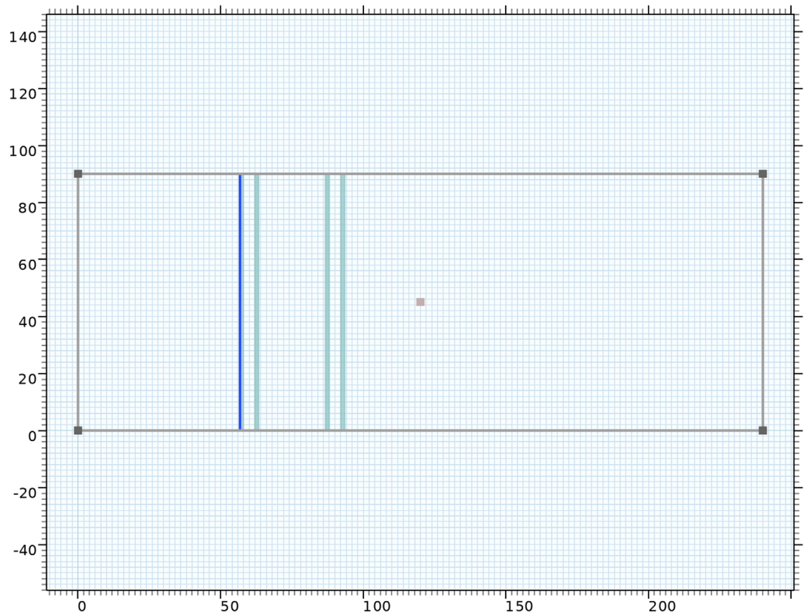
4 In the **Height** text field, type `a_c_w_part`.

5 Locate the **Position** section. In the **xw** text field, type `0`.

6 In the **yw** text field, type `0`.

7 Click **Build Selected**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.



Fillet all corners of the rectangle.

Work Plane 2 (wp2) > Plane Geometry

1 In the **Sketch** toolbar, click **Fillet**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Model Builder** window, click **Plane Geometry**.

4 Select one vertex of **r1** and drag it inward to create the fillets.

Adjust the fillet radius.

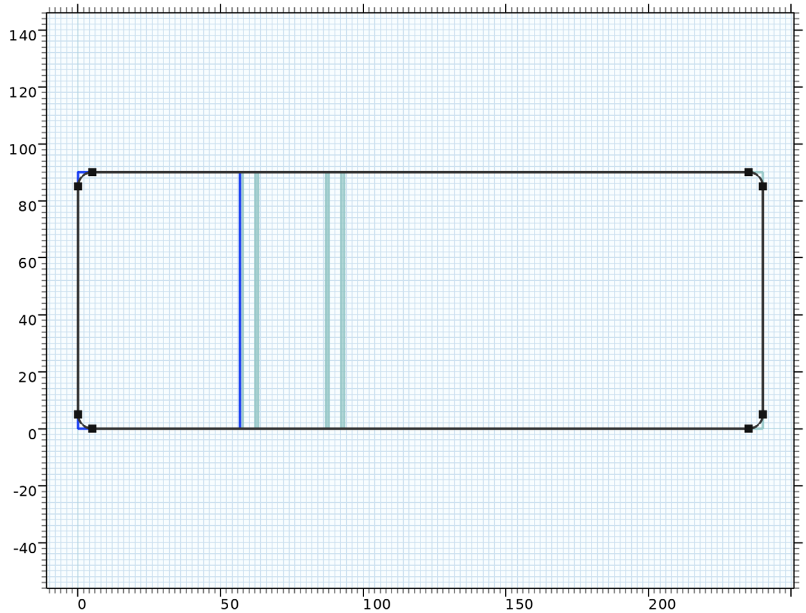
Work Plane 2 (wp2) > Fillet 1 (fil1)

1 In the **Model Builder** window, click **Fillet 1 (fil1)**.

2 In the **Settings** window for **Fillet**, locate the **Radius** section.

3 In the **Radius** text field, type `5[mm]`.

4 Click **Build Selected**.



Add the next feature from the **Work Plane** toolbar. This allows you to enter the parameters for size and shape directly in the feature.

Work Plane 2 (wp2) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

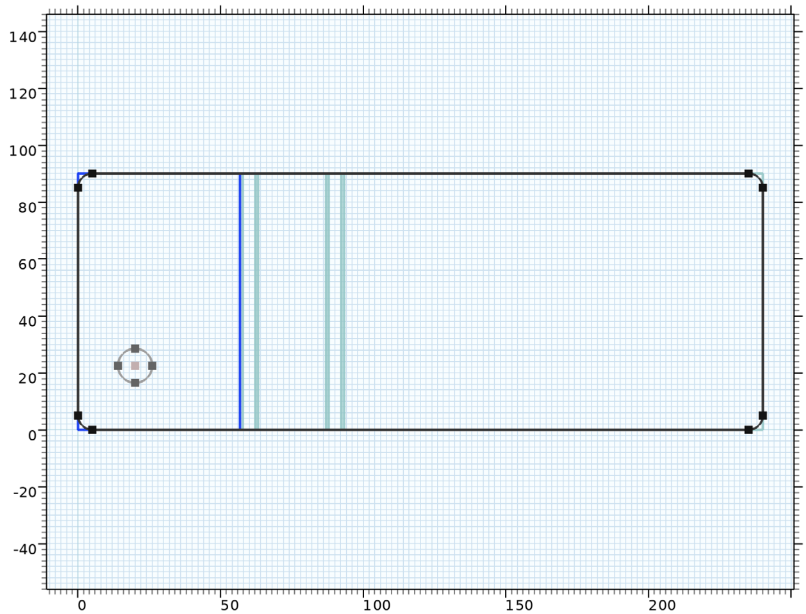
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `b_r_part`.

4 Locate the **Position** section. In the **xw** text field, type `b_di_part`.

5 In the **yw** text field, type `a_c_w_part/4`.

6 Click **Build Selected**.



Work Plane 2 (wp2) > Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **c1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

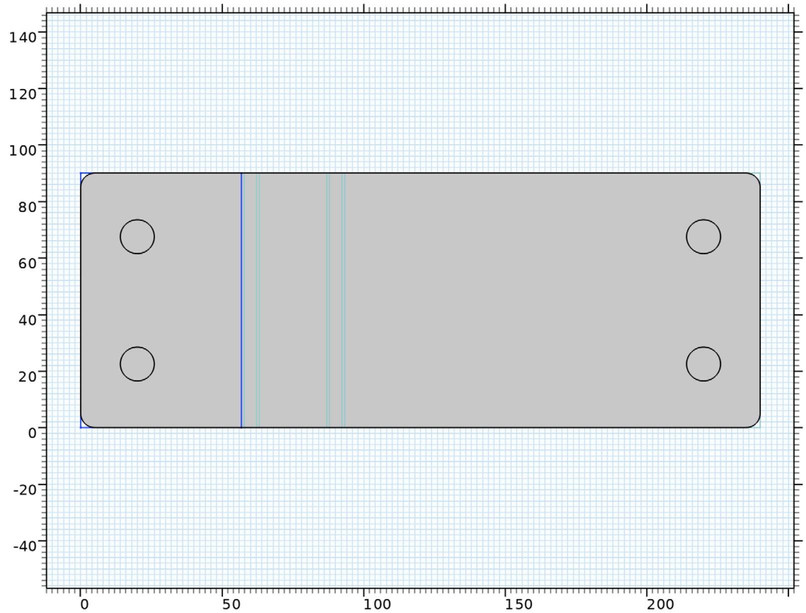
4 In the **xw size** text field, type `2`.

5 In the **yw size** text field, type `2`.

6 Locate the **Displacement** section. In the **xw** text field, type `c_g_w_part/2`.

7 In the **yw** text field, type `a_c_w_part/2`.

8 Click **Build Selected**.



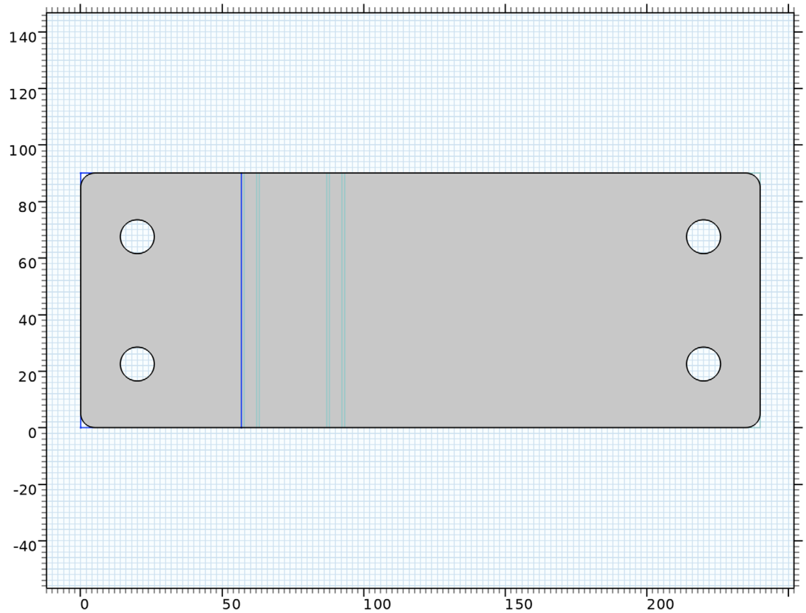
Work Plane 2 (wp2) > Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **fil1** as **Objects to add**.

3 Activate the **Objects to subtract** section, and select the objects **arr1(1,1), arr1(1,2), arr1(2, 1), arr1(2,2)**, which are the four circles.

4 In the **Settings** window for **Difference**, click **Build Selected**.



Extrude 2 (ext2)

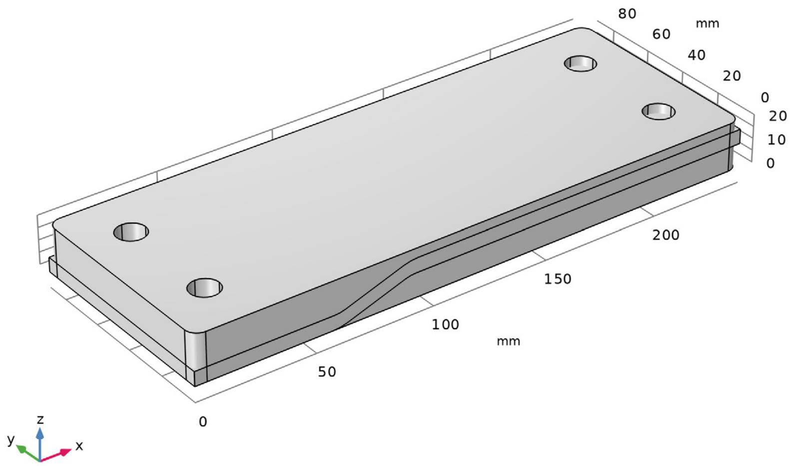
1 In the **Model Builder** window, under **Global Definitions** > **Geometry Parts** > **Angle Connector** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the table, enter `2*`.

3 Press `Ctrl` + `Space` to open the **Insert Expression** menu.

4 Expand the **Local Parameters** section and select **e_c_h_part**

5 In the **Settings** window for **Extrude**, click **Build Selected**.



Intersection 1 (int1)

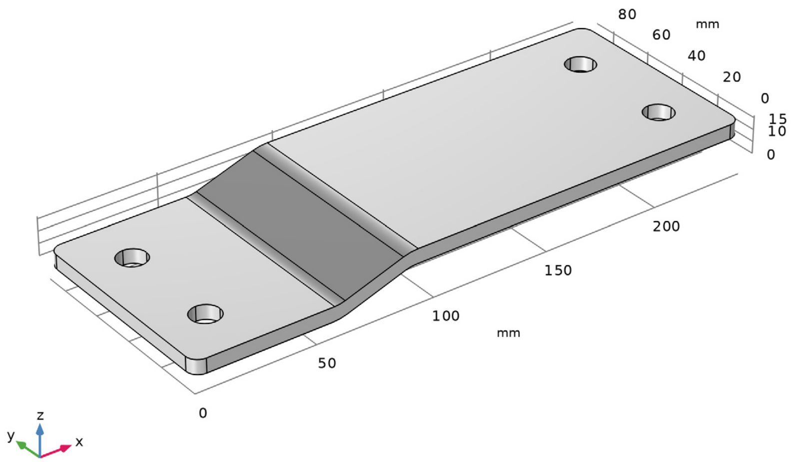
1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Intersection**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting objects selection** checkbox, to make sure that this selection will be accessible from an instance of the geometry part when inserted into a geometry sequence.

5 Click **Build Selected**.



Continue with creating two work planes for the positioning of this connector part. The work planes will be available in the part instances, and we will use them later on when inserting the part into a geometry sequence.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar, for a better view of the edges without rotating the geometry.

Elbow Connector Position

1 In the **Geometry** toolbar, click **Work Plane**.

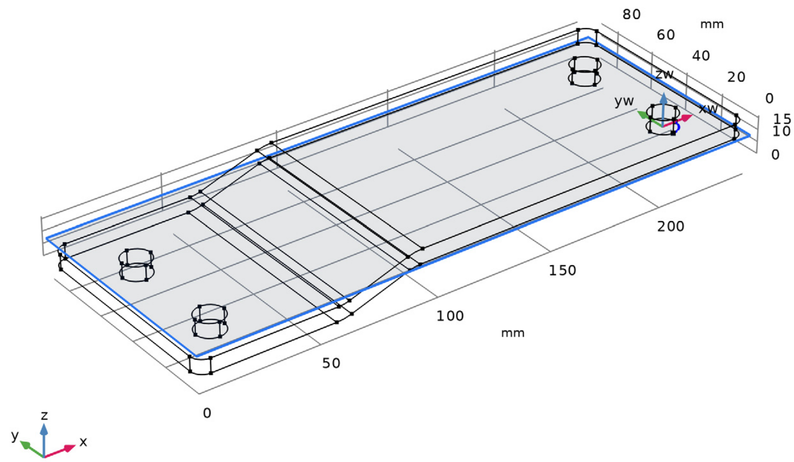
2 In the **Settings** window for **Work Plane**, type `Elbow Connector Position` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Edge parallel**.

4 On the object **int1**, select Edge 74 only. This is one of the lower edges of the front right hole.

5 Click to expand the **Local Coordinate System** section. In the **yw-displacement** text field, type `b_r_part`.

Click somewhere in the **Graphics** window to update the local coordinate system.



Bolt Position

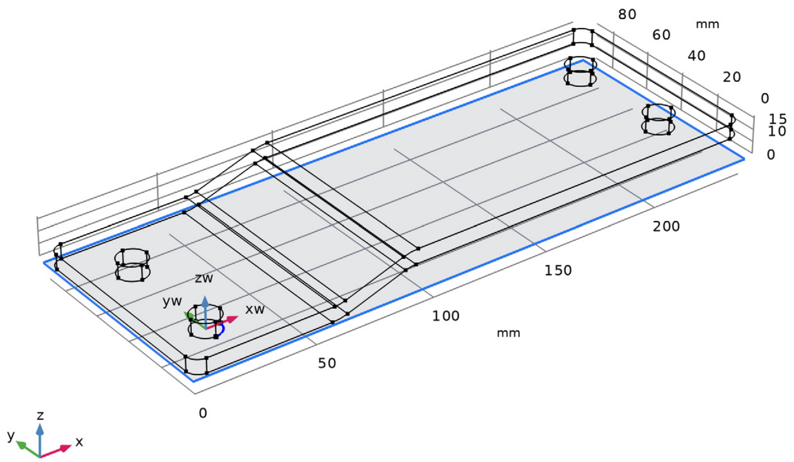
1 Right-click **Elbow Connector Position** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, type `Bolt Position` in the **Label** text field.

The edit field is already activated. Click on the edge located at the front left hole to place the Work Plane.

3 On the object **int1**, select Edge 26 only.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.



The geometry part for the angle connector is now ready. Continue with adding one more geometry part where you will insert several of the already created parts to create a subassembly.

GEOMETRY PARTS

In the **Geometry** toolbar, click **Create Part**.

ANODE TOP ASSEMBLY

1 In the **Settings** window for **Part**, type `Anode Top Assembly` in the **Label** text field.

2 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Default expression	Value	Description
`a_c_w_asm`	`90[mm]`	0.09 m	`Angle connector width`
`r_d_asm`	`20[mm]`	0.02 m	`Rod diameter`

3 Locate the **Units** section. From the **Length unit** list, choose **mm**.

Local Parameters

1 In the **Model Builder** window, expand the **Anode Top Assembly** node, then click **Local Parameters**.

2 In the **Settings** window for **Local Parameters**, locate the **Local Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
c_g_w_asm	400[mm]	400 mm	Cell Grid Top width
r_c_h_asm	6[mm]	6 mm	Rod Connector height

The individual geometry parts contain different selections, open the **Selection List** window for a quick access to these selections as we work with the geometry part.

SELECTION LIST

In the **Geometry** toolbar, click **Selection List** to open the **Selection List** window.

ANODE TOP ASSEMBLY

Spine 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Spine**.

2 In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.

3 Click **New Cumulative Selection**.

Cumulative selections are useful when the output of several geometry operations is contributing to the same selection. Here the cumulative selections will collect the domains with similar material assignments.

4 In the **New Cumulative Selection** dialog, type `Titanium` in the **Name** text field.

5 Click **OK**.

6 Select **Titanium** from the **Contribute to** list to add the part to this selection.

As soon as a new selection is added and the corresponding entity level is selected, it is displayed in the lower part of the **Selection List** window.

7 In the **Settings** window for **Part Instance**, click **Build Selected**.

Central Column 1 (pi2)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Central Column**.

Link the input parameter of this part instance of *Central Column 1* with a parameter of the part *Anode Top Assembly*.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

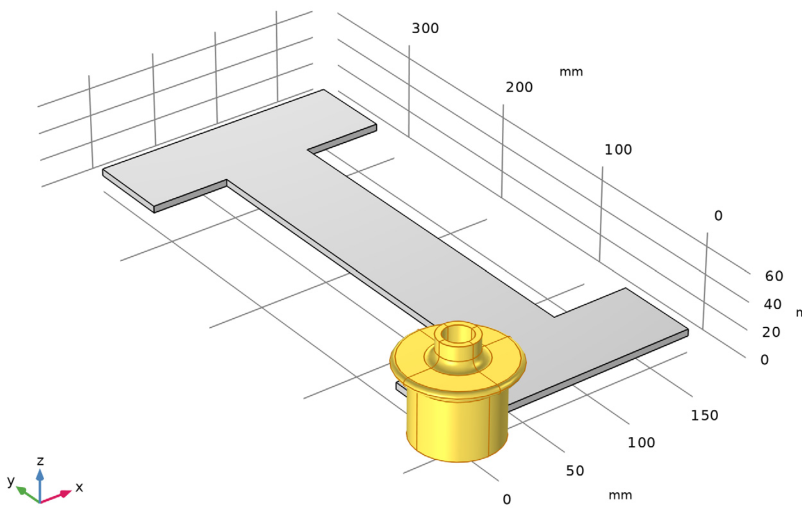
Name	Expression	Value	Description
r_d_part	`r_d_asm`	20 mm	Rod diameter

4 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Revolve 1		√	Titanium

5 Click **Build Selected**.

6 Click **Highlight Result** to make it easier to identify the output of the geometry features.

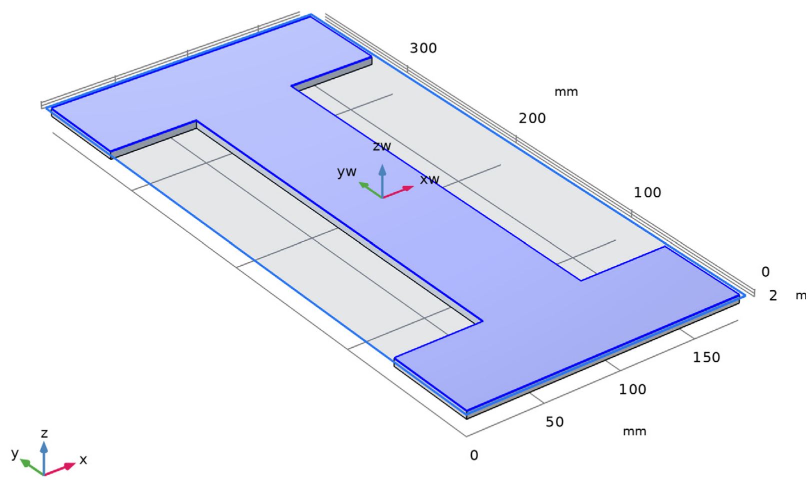


To position the part for the central column at the center of the spine, use a coordinate system already defined by the appropriate work plane in the geometry part for the spine.

SPINE

Central Column Position (wp2)

1 In the **Model Builder** window, expand the **Global Definitions** > **Geometry Parts** > **Spine** node, then click **Central Column Position (wp2)**.



Returning to the geometry sequence of the *Anode Top Assembly*, use this work plane to position the part for the central column.

ANODE TOP ASSEMBLY

Central Column 1 (pi2)

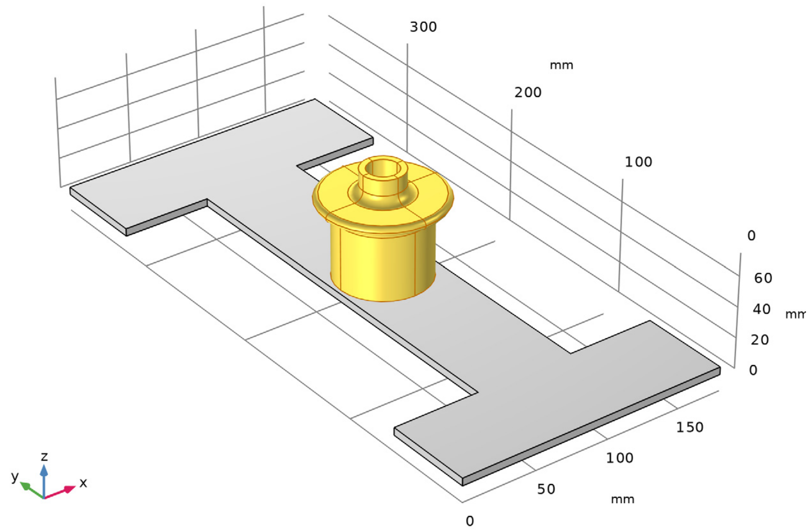
1 In the **Model Builder** window, under **Global Definitions** > **Geometry Parts** > **Anode Top Assembly** click **Central Column 1 (pi2)**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Spine 1 (pi1)**.

4 From the **Work plane** list, choose **Central Column Position (wp2)**.

5 Click **Build Selected**.



Rod 1 (pi3)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Rod**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
r_d_part	`r_d_asm`	20 mm	Rod diameter

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Central Column 1 (pi2)**.

5 From the **Work plane** list, choose **Rod Position (wp2)**.

6 Locate the **Domain Selections** section. Click **New Cumulative Selection**.

7 In the **New Cumulative Selection** dialog, type `Copper` in the **Name** text field.

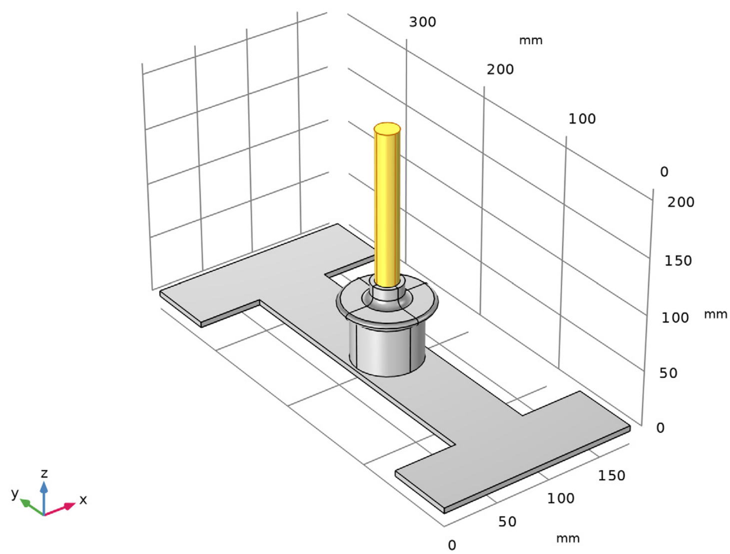
8 Click **OK**.

9 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

10 In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Cylinder 1		√	Copper

11 Click **Build Selected**.



Rod Connector 1 (pi4)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Rod Connector**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_w_part	a_c_w_asm	90 mm	Angle connector width
r_d_part	r_d_asm	20 mm	Rod diameter
r_c_h_part	r_c_h_asm	6 mm	Rod connector height

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Rod 1 (pi3)**.

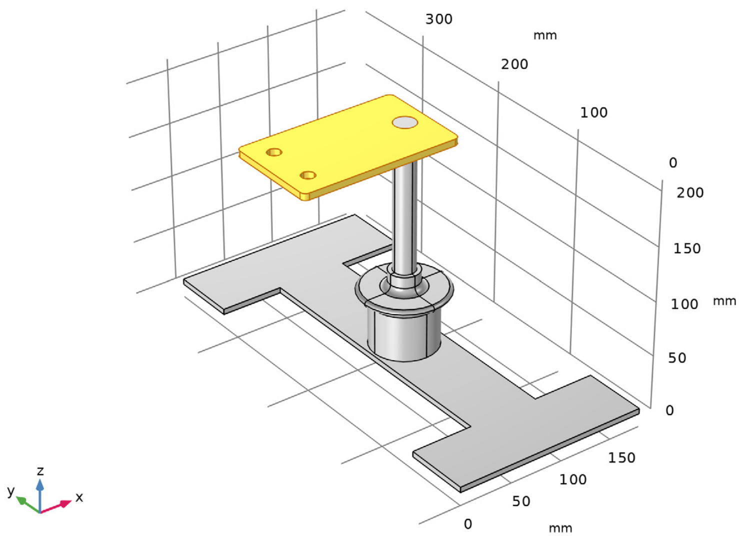
5 From the **Work plane** list, choose **Rod Connector Position (wp1)**.

6 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Extrude 1		√	Copper

7 Click **Build Selected**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.



Copy 1 (copy1)

To obtain another copy of the already inserted geometry parts we can use the **Copy** operation.

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

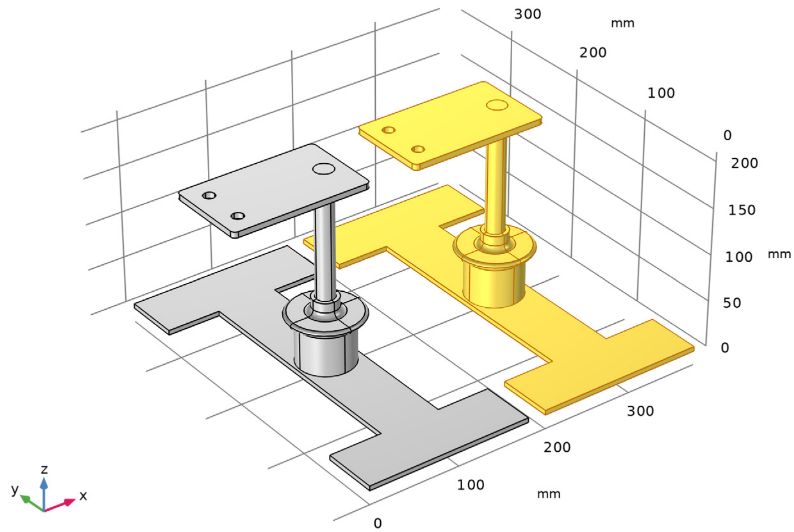
2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `c_g_w_asm/2`.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.



Elbow Connector 1 (pi5)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Elbow Connector**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_w_part	a_c_w_asm	90 mm	Angle connector width

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Rod Connector Position (wp4)**.

5 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Rod Connector 1 (pi4)**.

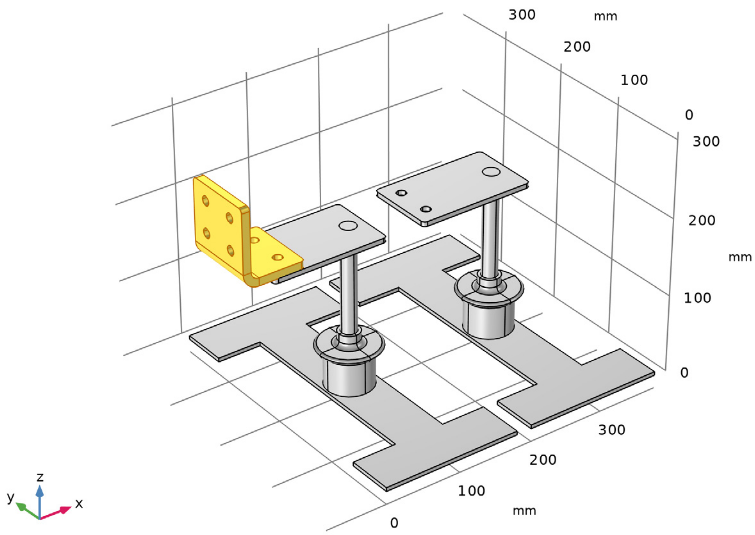
6 From the **Work plane** list, choose **Elbow Connector Position (wp2)**.

7 Find the **Rotation** subsection. In the **Rotation angle** text field, type `90[deg]`.

8 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Union 1		√	Copper

9 Click **Build Selected**.



10 Locate the **Selection Settings** section. Select the **Keep noncontributing selections** checkbox, to make selections from this part available in instances of the Anode Top Assembly part.

The selection we are interested in is defined on the point level in the Elbow Connector.

11 In the **Graphics** window toolbar, click next to **Select Domains**, then choose **Select Points**. You can now see the named selection **Bolt Medium Position (Elbow Connector 1)** listed in the lower part of the **Selection List** window.

12 Click **Highlight Result**.

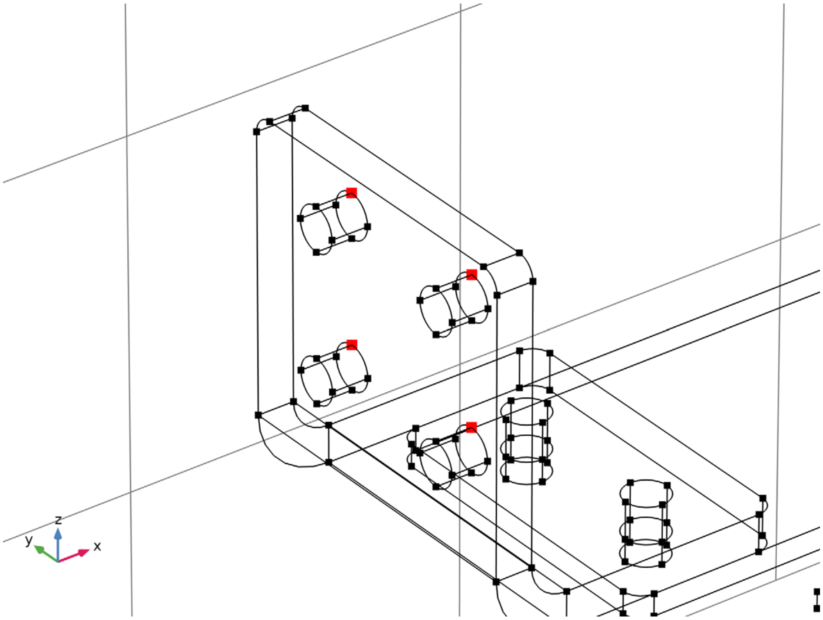
13 Click the **Wireframe Rendering** button in the **Graphics** toolbar for a better view of this selection.

SELECTION LIST

1 Go to the **Selection List** window.

2 In the **Point selections** tree, select **Bolt Medium Position (Elbow Connector 1)**.

3 Zoom to the highlighted vertices.



4 Reset the view rendering by activating **Highlight Result** and deactivating **Wireframe Rendering**.

ANODE TOP ASSEMBLY

Angle Connector 1 (pi6)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Angle Connector**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_w_part	a_c_w_asm	90 mm	Angle connector width

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Elbow Connector Position (wp3)**.

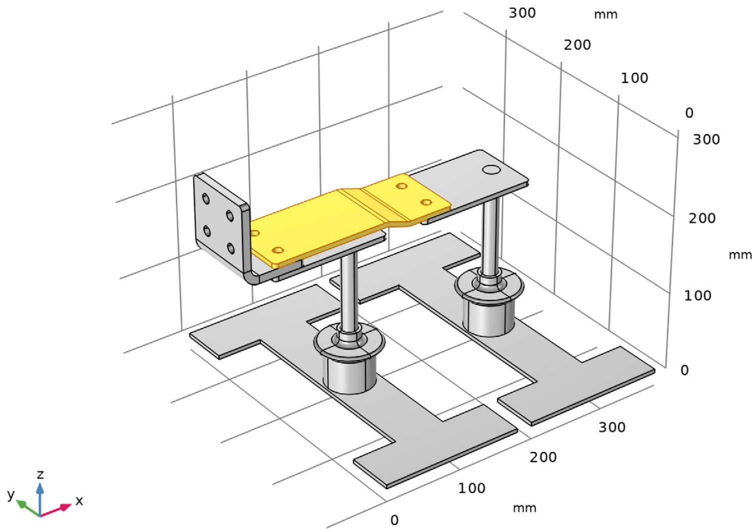
5 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Elbow Connector 1 (pi5)**.

6 From the **Work plane** list, choose **Angle Connector Position (wp6)**.

7 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Intersection 1		√	Copper

8 Click **Build Selected**.



Bolt Small

1 In the **Geometry** toolbar, click **Part Instance** and choose **Bolt**.

2 In the **Settings** window for **Part Instance**, type `Bolt Small` in the **Label** text field.

3 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Angle Connector 1 (pi6)**.

4 From the **Work plane** list, choose **Bolt Position (wp4)**.

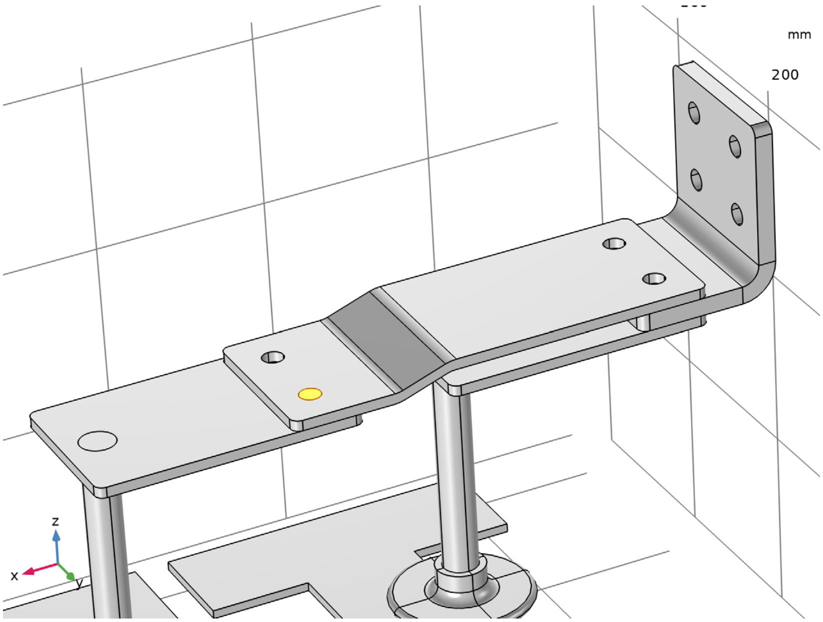
5 Find the **Displacement** subsection. In the **zwi** text field, type `-r_c_h_asm`.

6 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Cylinder 1		√	Titanium

7 Click **Build Selected**.

Use the mouse to get a better view of the inserted part.



Bolt Large

1 In the **Geometry** toolbar, click **Part Instance** and choose **Bolt**.

2 In the **Settings** window for **Part Instance**, type `Bolt Large` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
size	2	2	Size selection 0 = small, 1 = medium, 2 = large

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Elbow Connector 1 (pi5)**.

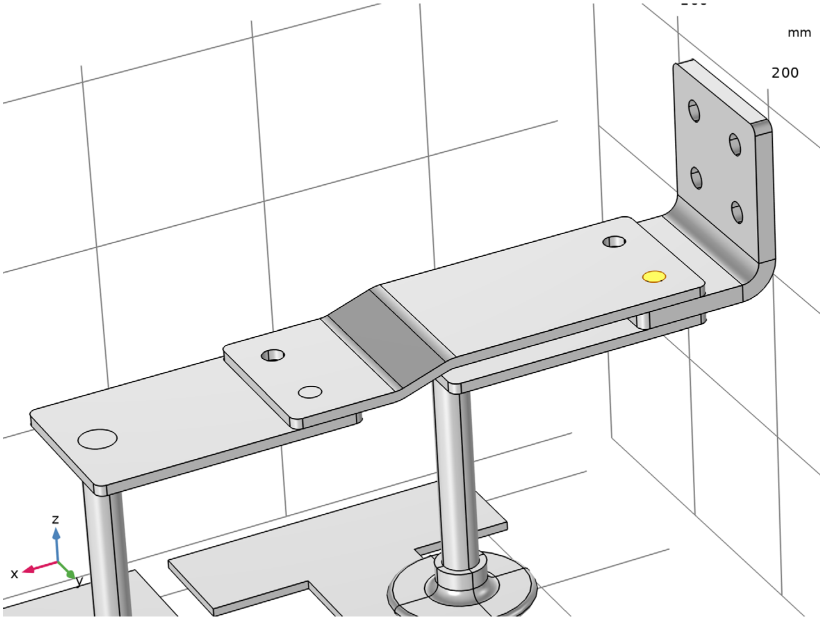
5 From the **Work plane** list, choose **Rod Connector Position (wp4)**.

6 Find the **Displacement** subsection. In the **zwi** text field, type `-r_c_h_asm`.

7 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Instances	Contribute to
Cylinder 1		√	Titanium

8 Click **Build Selected**.



Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **pi7** and **pi8** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

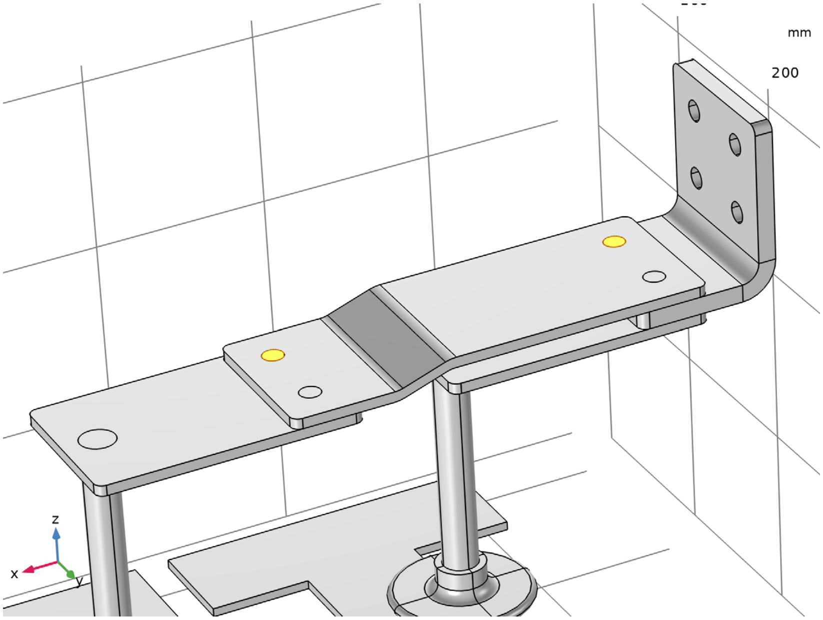
4 Select the **Keep input objects** checkbox.

5 Locate the **Point on Plane of Reflection** section. In the **y** text field, type `190[mm]`, which is half of the length of the spine.

6 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

7 In the **z** text field, type `0`.

8 Click **Build Selected**.



Next, add a work plane that will help position this part.

9 Click the **Go to Default View** button in the **Graphics** toolbar.

Intercell Busbar Position

1 In the **Geometry** toolbar, click **Work Plane**.

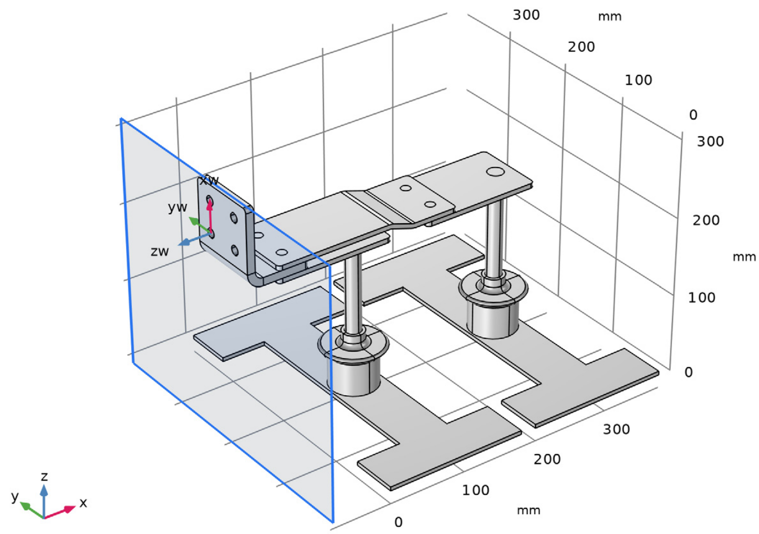
2 In the **Settings** window for **Work Plane**, type `Intercell Busbar Position` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane type** list, choose **Transformed**.

4 From the **Take work plane from** list, choose **Elbow Connector 1 (pi5)**.

5 From the **Work plane to transform** list, choose **Intercell Busbar Position (wp5)**.

6 Click **Build Selected**.



Finally, set up a selection that includes all objects in this geometry part. It will come in handy when building the busbar geometry.

Box Selection 1 (boxsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Object**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

GLOBAL DEFINITIONS

All geometry parts are now ready. Next, add the global parameters for controlling the busbar dimensions. Then, add a 3D model component where you can build the busbar geometry.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`a_c_w`	`90[mm]`	0.09 m	`Angle connector width`
`r_d`	`20[mm]`	0.02 m	`Rod diameter`

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 From the **Length unit** list, choose **mm**.

Cell Grid Top 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Cell Grid Top**.

2 In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.

In the following we will set up separate selections to collect the titanium and copper parts.

3 Click **New Cumulative Selection**.

4 In the **New Cumulative Selection** dialog, type `Titanium` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

7 Click **New Cumulative Selection** again to define a selection for the copper parts.

8 In the **New Cumulative Selection** dialog, type `Copper` in the **Name** text field.

9 Click **OK**.

10 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

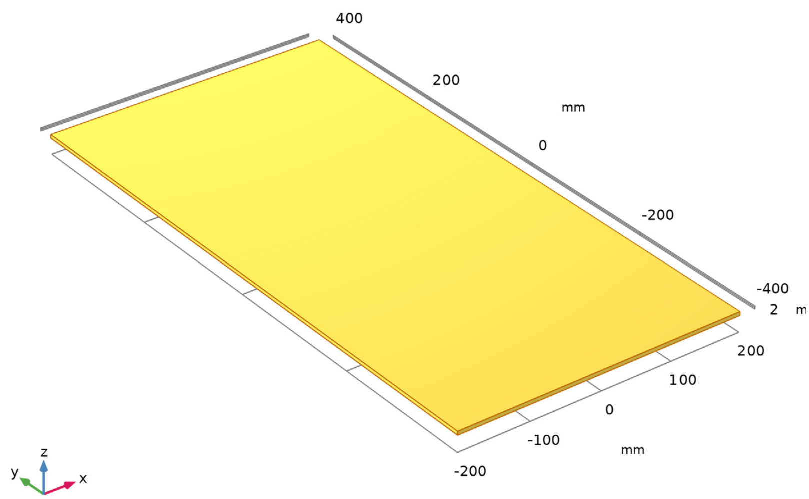
11 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Block 1		√	Titanium

12 Click to expand the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Electrolyte Boundary	√	√	None

13 Click **Build Selected**.



Anode Top Assembly 1 (pi2)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Anode Top Assembly**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_w_asm	a_c_w	90 mm	Angle connector width
r_d_asm	r_d	20 mm	Rod diameter

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Cell Grid Top 1 (pi1)**.

5 From the **Work plane** list, choose **Spine Position (wp1)**.

6 Click to expand the **Object Selections** section. In the table, enter the following settings:

Name	Keep	Contribute to
Box Selection 1	√	None

7 Locate the **Domain Selections** section. In the table, enter the following settings:

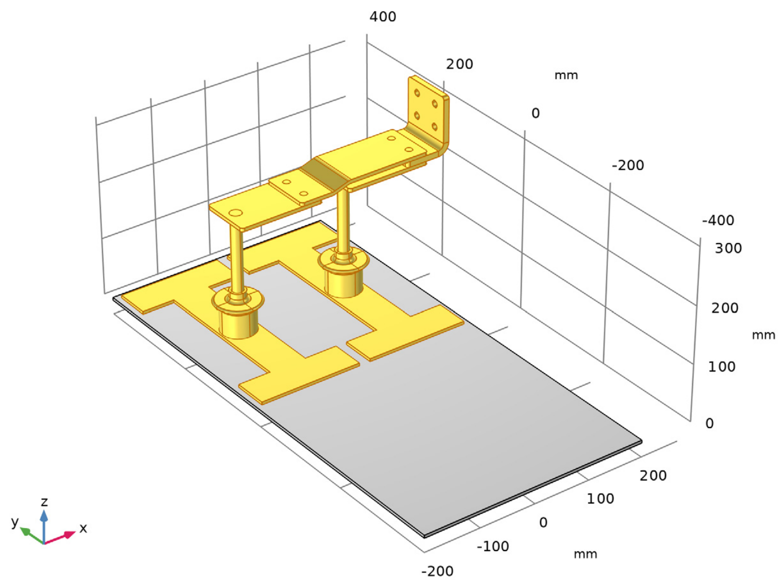
Name	Keep	Physics	Contribute to
Titanium		√	Titanium
Copper		√	Copper

8 Click to expand the **Point Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Bolt Medium Position (Elbow Connector 1)	√		None

9 Click **Build Selected**.

10 In the **Graphics** window toolbar, click next to **Select Points**, then choose **Select Objects**, to highlight the objects.



Move 1 (mov1)

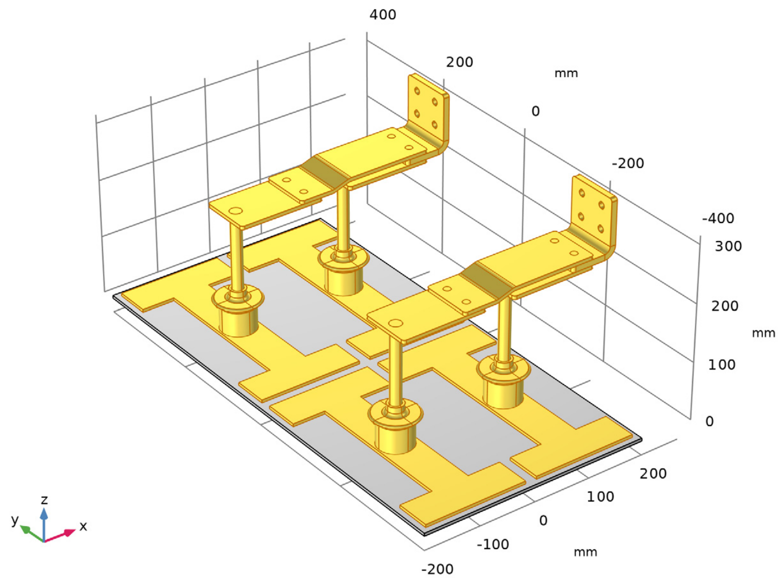
1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Box Selection 1 (Anode Top Assembly 1)**.

4 Locate the **Displacement** section. In the **y** text field, type `0 -400[mm]`. By using a displacement vector, the input objects are moved to each of the positions specified by the vector.

5 Click **Build Selected**.



Intercell Busbar 1 (pi3)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Intercell Busbar**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_c_w_part	`a_c_w`	90 mm	Angle connector width

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Elbow Connector Position (wp2)**.

5 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Anode Top Assembly 1 (pi2)**.

6 From the **Work plane** list, choose **Intercell Busbar Position (wp1)**.

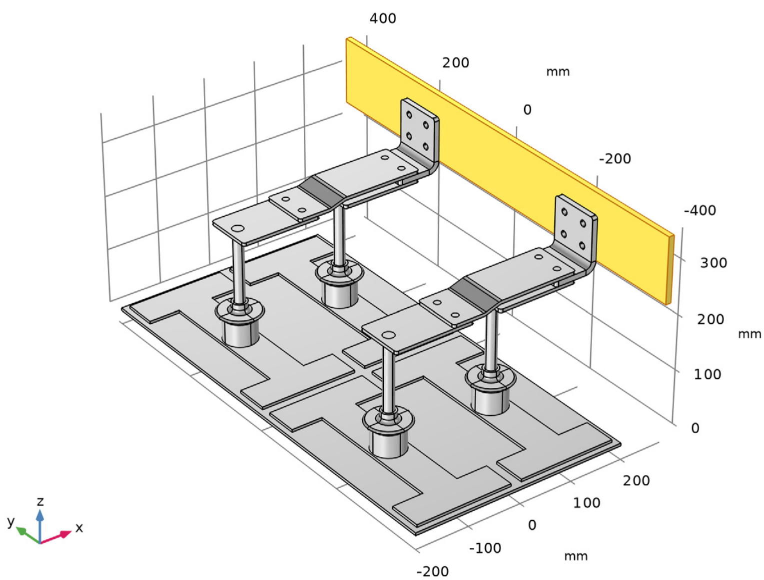
7 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Difference 1		√	Copper

8 Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Grounded Boundary	√	√	None

9 Click **Build Selected**.



Bolt Medium

1 In the **Geometry** toolbar, click **Part Instance** and choose **Bolt**.

2 In the **Settings** window for **Part Instance**, type `Bolt Medium` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
size	1	1	Size selection 0 = small, 1 = medium, 2 = large

4 Locate the **Position and Orientation of Output** section. Find the **Rotation** subsection. From the **Axis type** list, choose **ywi-axis**.

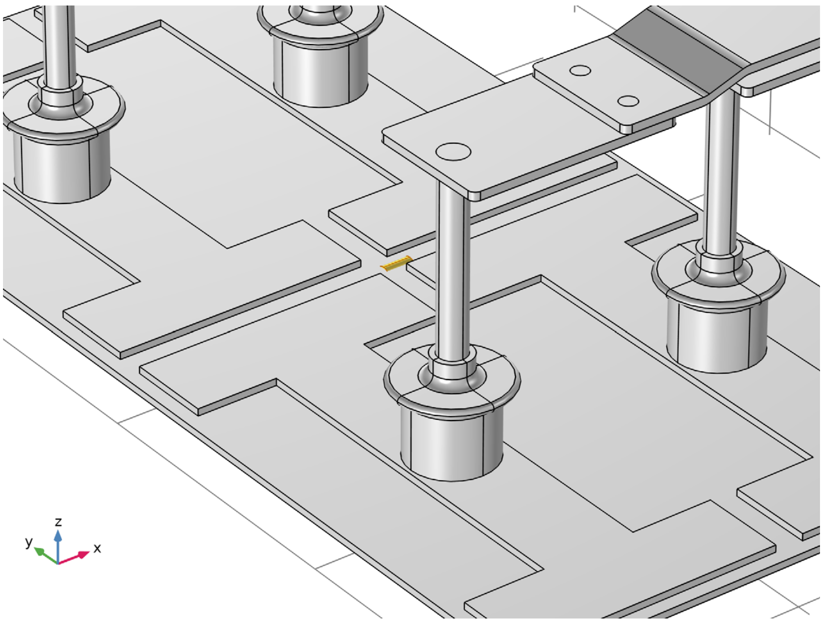
5 In the **Rotation angle** text field, type 90.

6 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Cylinder 1		√	Titanium

7 Click **Build Selected**.

The part is placed at the origin of the component. Zoom to the center to see the bolt.



Move 2 (mov2)

Position the bolt by adding a Move transform operation, and using the option to specify the positions to move to by selecting vertices.

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **pi4** only.

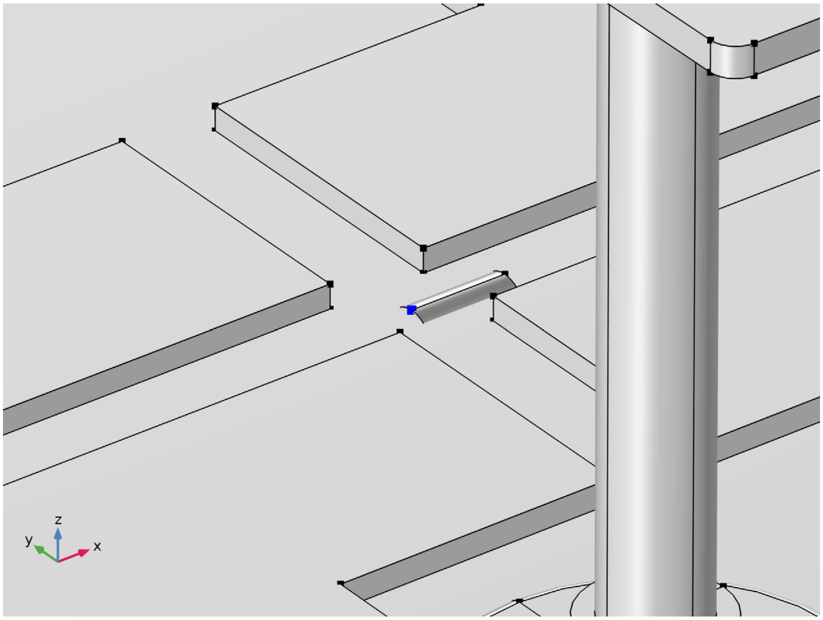
3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 From the **Specify** list, choose **Positions**.

5 Click to select the **Activate Selection** toggle button for **Vertex to move**.

6 On the object **pi4**, select Point 1 only.

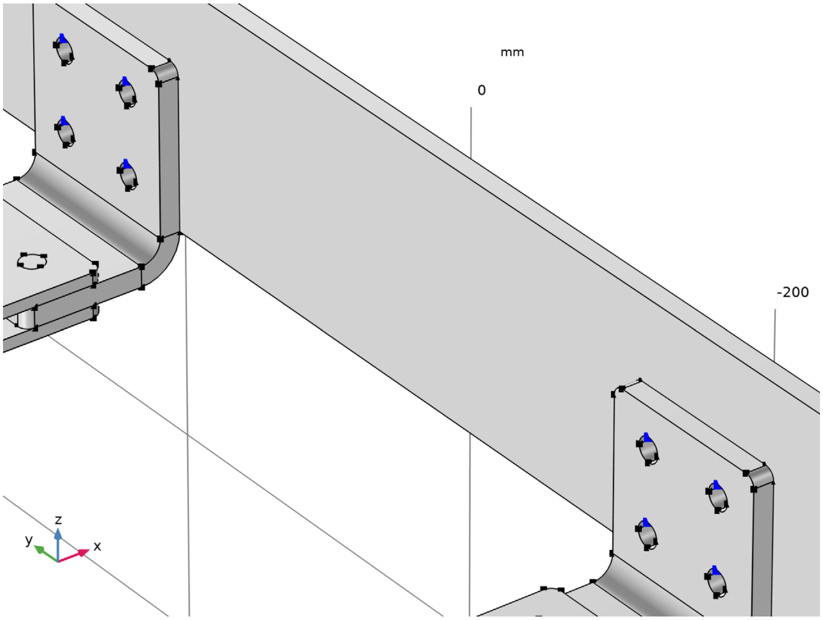
7 Click **Zoom to Selection**.



8 Click to select the **Activate Selection** toggle button for **Vertices to move to**.

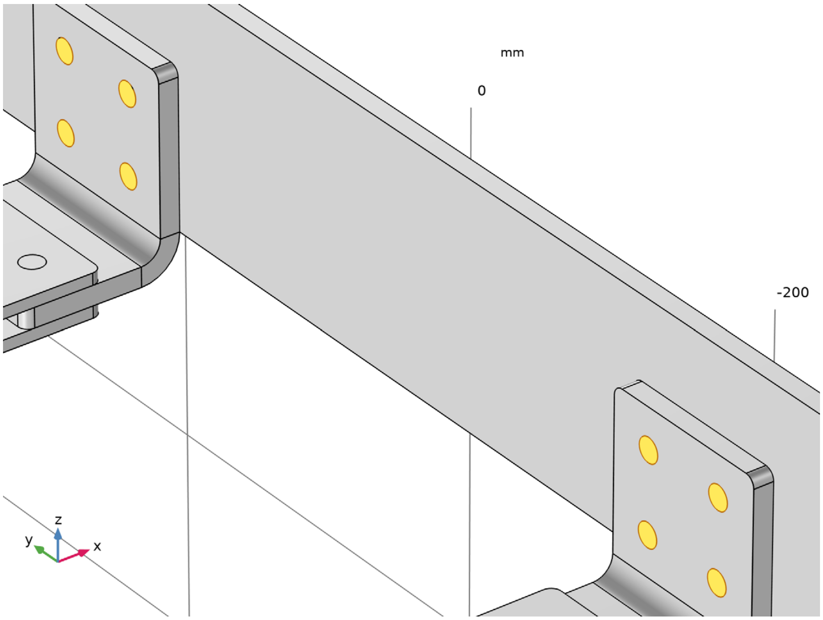
9 From the **Vertices to move to** list, choose **Bolt Medium Position (Elbow Connector 1) (Anode Top Assembly 1)**.

10 Click the Zoom to Selection button for **Vertices to move to**.



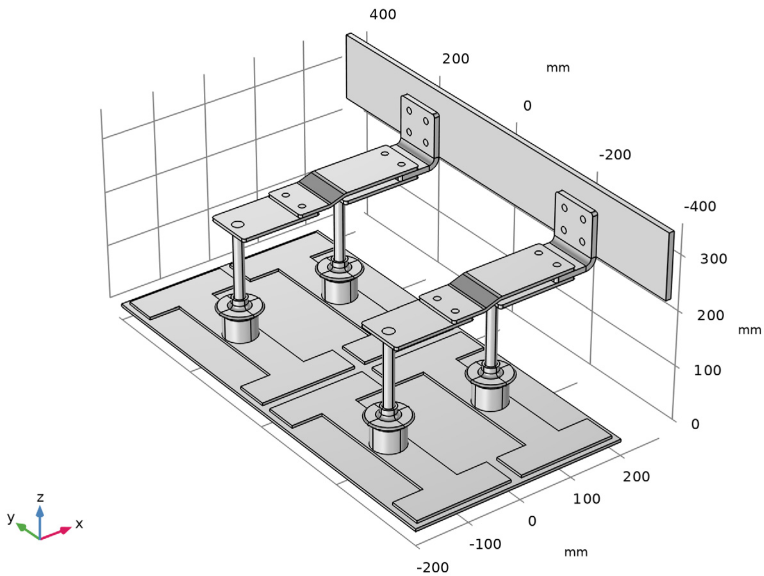
11 Click **Build Selected**.

12 In the **Graphics** window toolbar, click ▾ next to **Select Points**, then choose **Select Objects**, to highlight the objects.



13 In the **Geometry** toolbar, click **Build All**.

14 Click the **Zoom Extents** button in the **Graphics** toolbar.



As the busbar geometry is now ready, set up selections to use for the physics definitions.

15 In the **Graphics** window toolbar, click ▼ next to **Select Objects**, then choose **Select Domains**.

SELECTION LIST

1 Go to the **Selection List** window.

2 In the **Domain selections** tree, select **Cumulative Selections** > **Copper** and **Cumulative Selections** > **Titanium**.

3 Right-click **Create Selection** and choose **Adjacent Selection**.

4 In the **Create Selection** dialog, click **OK**.

GEOMETRY 1

Heat Flux Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Heat Flux Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the + **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Adjacent Selection 1** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the + **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Electrolyte Boundary (Cell Grid Top 1)** and **Grounded Boundary (Intercell Busbar 1)**.

10 Click **OK**.

11 In the **Geometry** toolbar, click **Selection List** to close the **Selection List** window.