Created in COMSOL Multiphysics 6.1

# Busbar Assembly Geometry — with Geometry Parts

## *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.

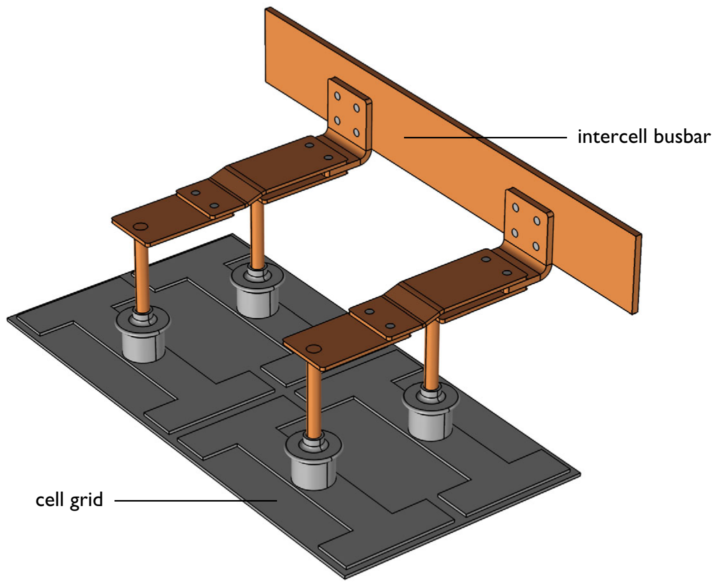



*Figure 1: The busbar assembly.*

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.

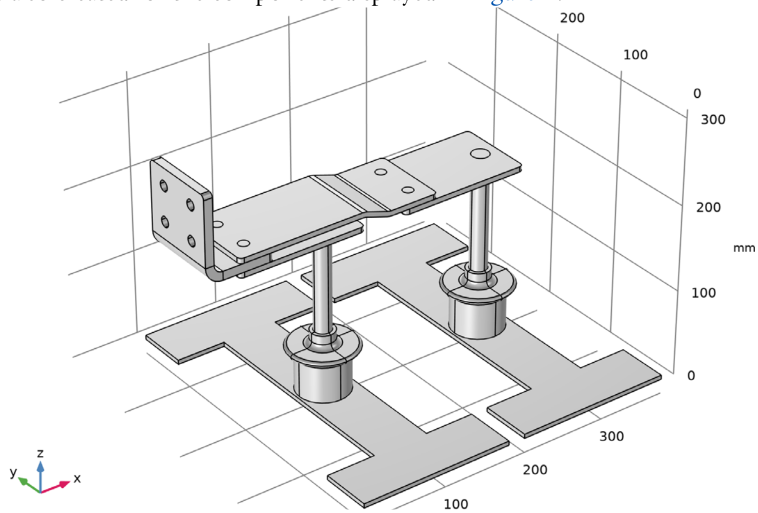



*Figure 2: Subunit of the busbar, created in a geometry part.*

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 In the **Model Builder** window, first expand the **Global Definitions** node, then the **Geometry Parts** node.

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.

### GLOBAL DEFINITIONS

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*

1 In the **Geometry** toolbar, click **Programming** and choose **Local Parameters**.

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

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

*Work Plane 1 (wp1)>Plane Geometry*

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

*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**.


*View 26*

Add **Fillets** to the corners.

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

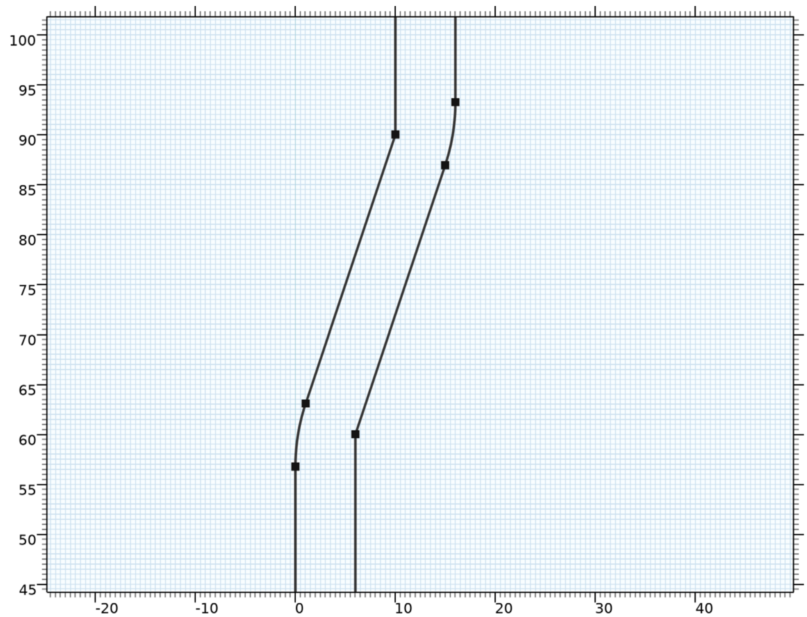
**1** In the **Work Plane** toolbar, click **Fillet**.

**2** On the object **pol1**, select Points 2 and 6 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)*

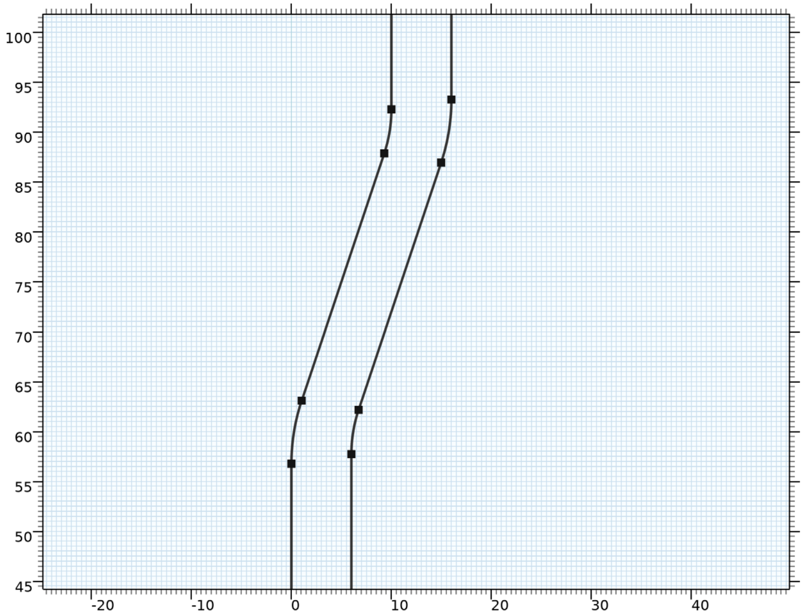
**1** In the **Work Plane** toolbar, click **Fillet**.

**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)*

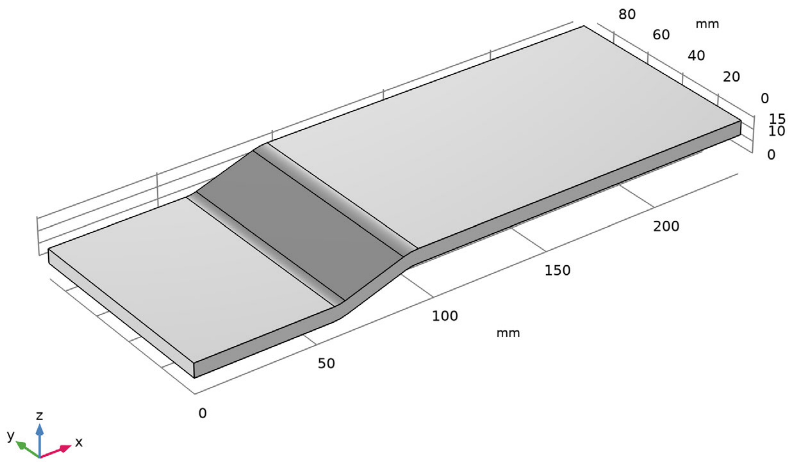
**1** In the **Model Builder** window, under **Global Definitions>Geometry Parts>Angle connector** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

**2** In the **Settings** window for **Extrude**, locate the **Distances** section.

**3** In the table, enter the following settings:

| Distances (mm) |
|---|
| a_c_w_part |

**4** Click **Build Selected**.




The extruded solid for the side view is now ready. Continue by creating the solid for the top view.

*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** check box.

*Work Plane 2 (wp2)>Plane Geometry*

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

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

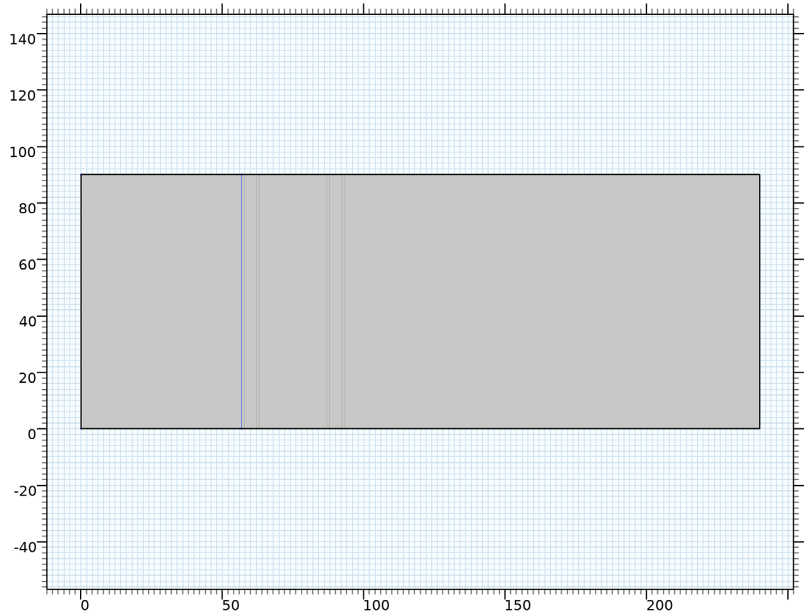
**1** In the **Work Plane** toolbar, click **Rectangle**.

**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** Click **Build Selected**.




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

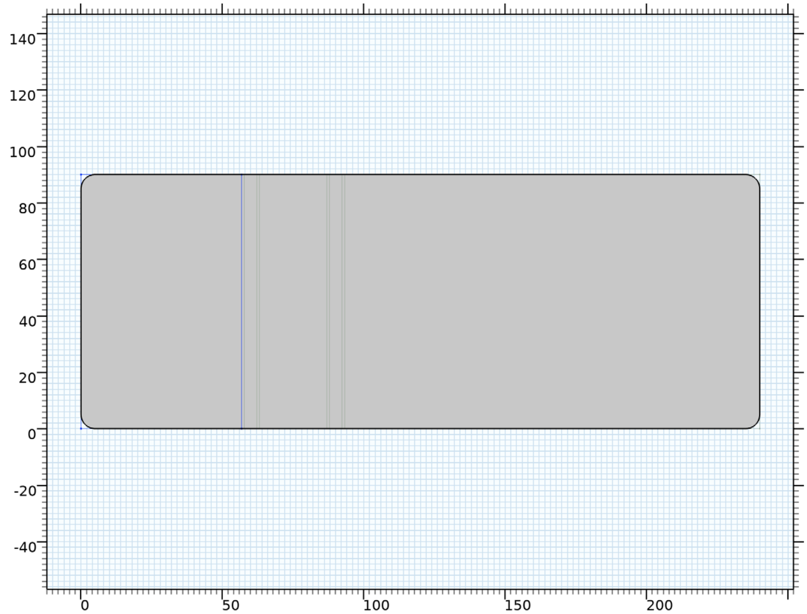
**1** In the **Work Plane** toolbar, click **Fillet**.

**2** On the object **r1**, select Points 1–4 only.

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

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

**5** Click **Build Selected**.




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

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

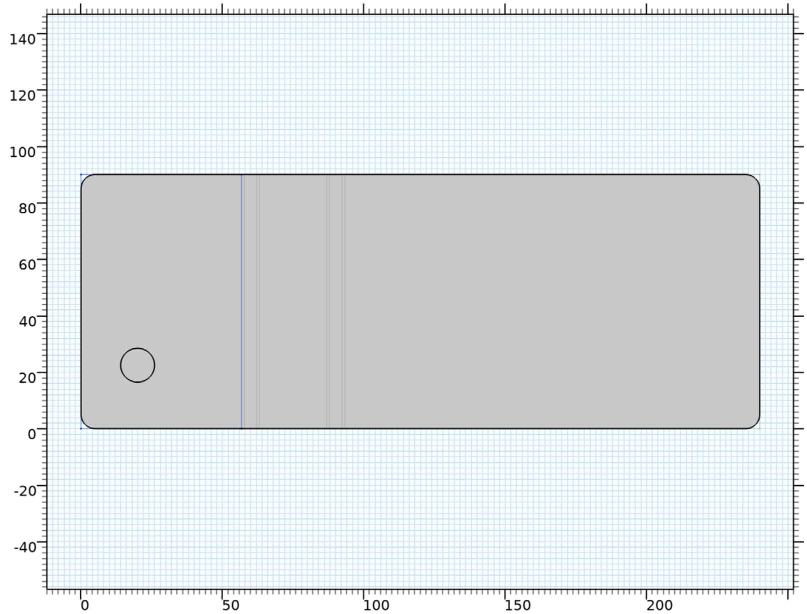
**2** In the **Settings** window for **Circle**, locate the **Size and Shape** section.

**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`.

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

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

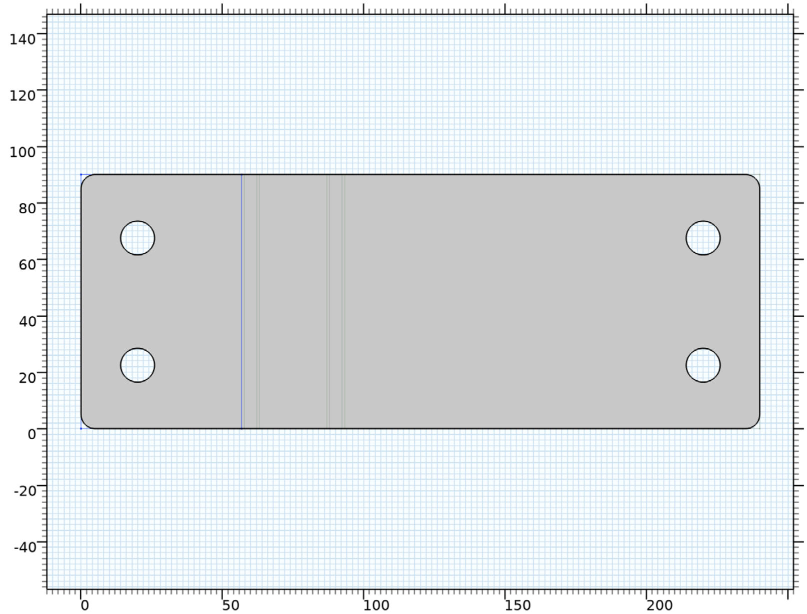
**2** Select the object **fil1** only.

**3** In the **Settings** window for **Difference**, locate the **Difference** section.

**4** Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

**5** Select the objects **arr1(1,1)**, **arr1(1,2)**, **arr1(2,1)**, and **arr1(2,2)** only.

6 Click **Build Selected**.




*Extrude 2 (ext2)*

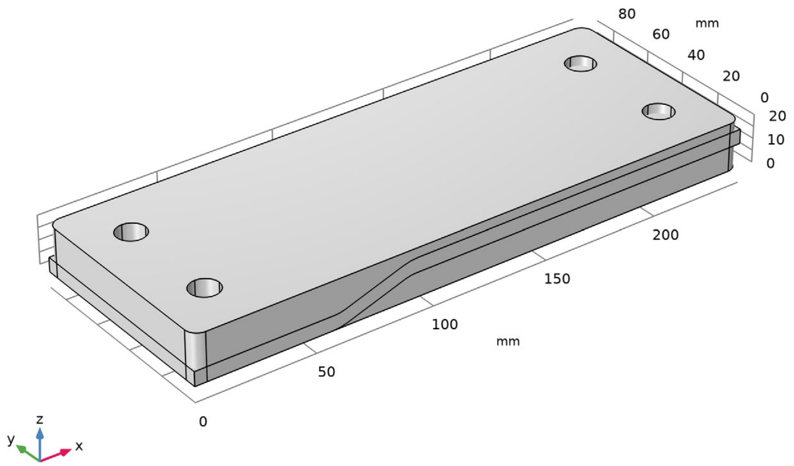
1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Angle connector** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

| Distances (mm) |
| --- |
| 2*e_c_h_part |

**4** Click **Build Selected**.




*Intersection 1 (int1)*

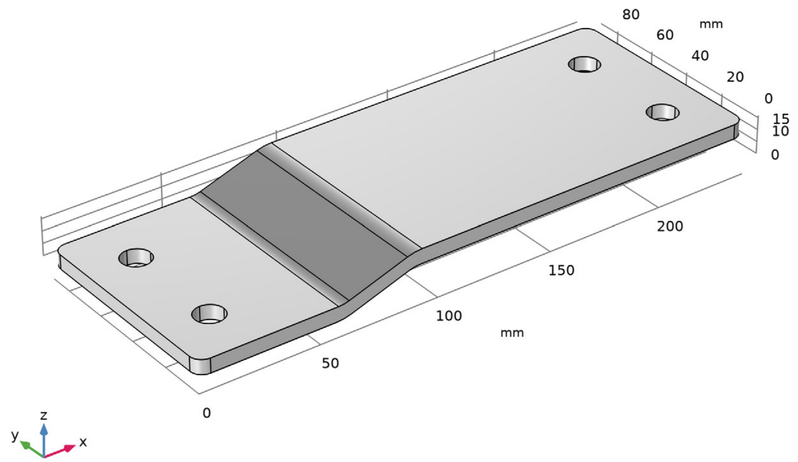
**1** In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

**2** Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

**4** Select the **Resulting objects selection** check box, to access this selection from an instance of the part inserted into a geometry sequence.

5 Click Build Selected.

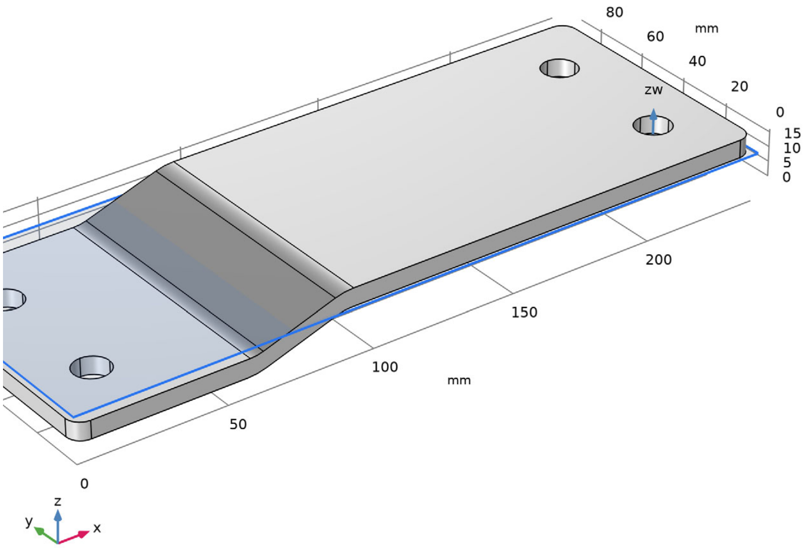



For easy positioning of this connector part, create two **Work Planes** and orient the associated coordinate system.

*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 **Transformed**.
4 From the **Work plane to transform** list, choose **Work Plane 2 (wp2)**.
5 Find the **Displacement** subsection. In the **xw** text field, type `c_g_w_part/2+b_di_part`.
6 In the **yw** text field, type `a_c_w_part/4`.

**7** In the **zw** text field, type `e_c_h_part`.




*Bolt Position*

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

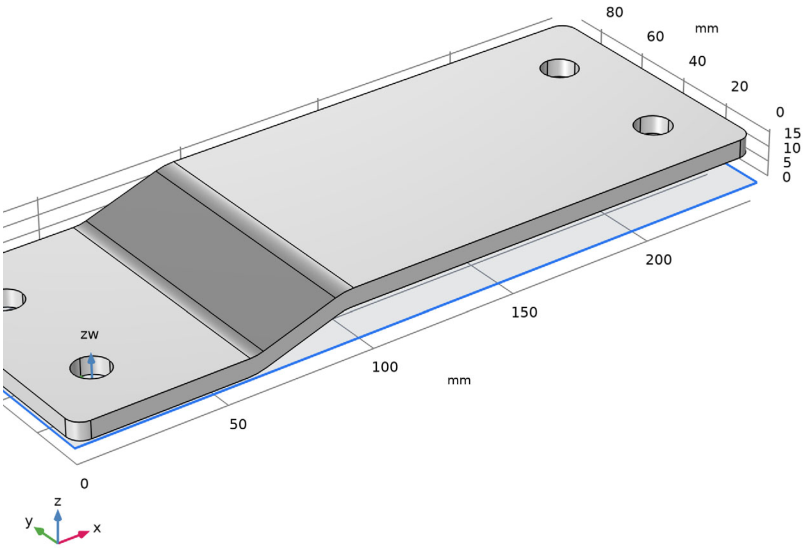
**2** In the **Settings** window for **Work Plane**, type `Bolt Position` in the **Label** text field.

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

**4** From the **Work plane to transform** list, choose **Work Plane 2 (wp2)**.

**5** Find the **Displacement** subsection. In the **xw** text field, type `b_di_part`.

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




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

The geometry part for the angle connector is now ready. We will continue with adding one more geometry part where we will insert geometry parts to built a repeating subassembly of the busbar.

## ANODE TOP ASSEMBLY

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

2 In the **Settings** window for **Part**, type `Anode top assembly` 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_asm` | `90[mm]` | 0.09 m | `Angle connector width` |
| `r_d_asm` | `20[mm]` | 0.02 m | `Rod diameter` |

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

### *Local Parameters*

1 In the **Geometry** toolbar, click **Programming** and choose **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 |

*Spine 1 (pi1)*

1 In the **Geometry** toolbar, click **Parts** 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 we want the output of several geometry operations to contribute to a selection. Here the cumulative selections will collect the domains for assigning the different materials.

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

5 Click **OK**.

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

7 In the table, enter the following settings:

| Name | Keep | Instances | Contribute to |
| --- | --- | --- | --- |
| Extrude 1 | | √ | Titanium |

8 Click **Highlight Result** to make it easier to identify the output of the various features.

9 Click **Build Selected**.

*Central column 1 (pi2)*

1 In the **Geometry** toolbar, click **Parts** and choose **Central column**.

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

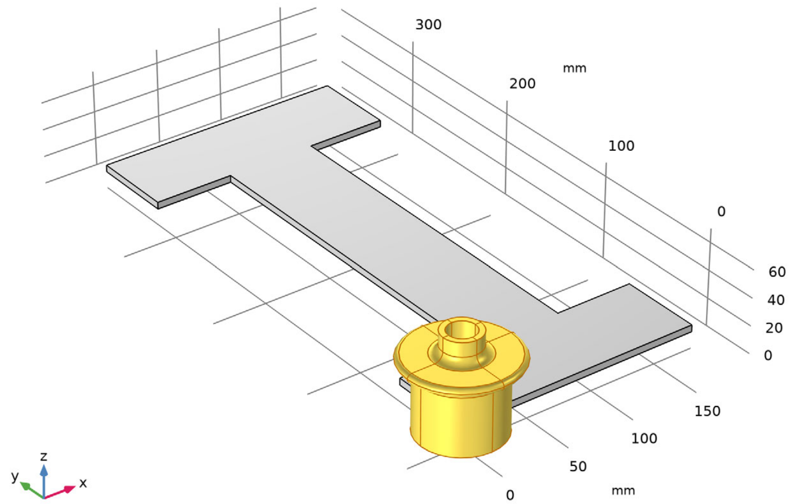
3 In the table, enter the following settings:

| 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**.



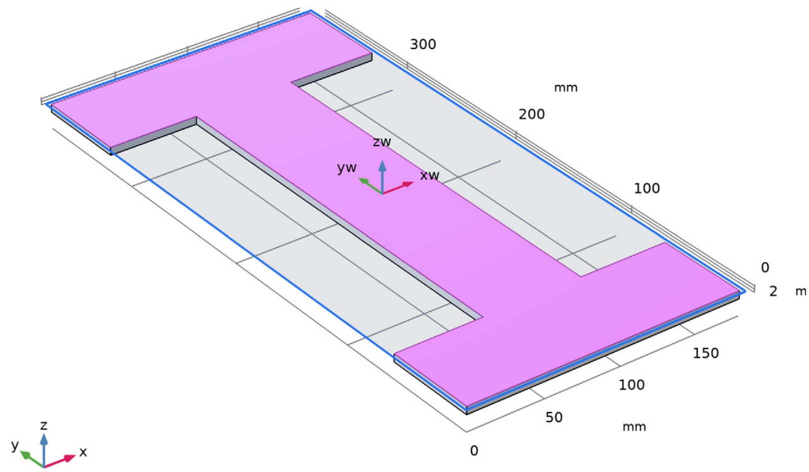

To position the part for the central column at the center of the spine, use a coordinate system defined by a work plane the geometry part for the spine.

### SPINE

**1** In the **Model Builder** under **Geometry Parts**, expand **Spine**.

**2** Select *Central column Position* to see where the work plane is located.

*Central column Position (wp2)*




Returning to the geometry sequence of the Anode top assembly, we can use this work plane to position the part for the central column.

### ANODE TOP ASSEMBLY

*Central column 1 (pi2)*

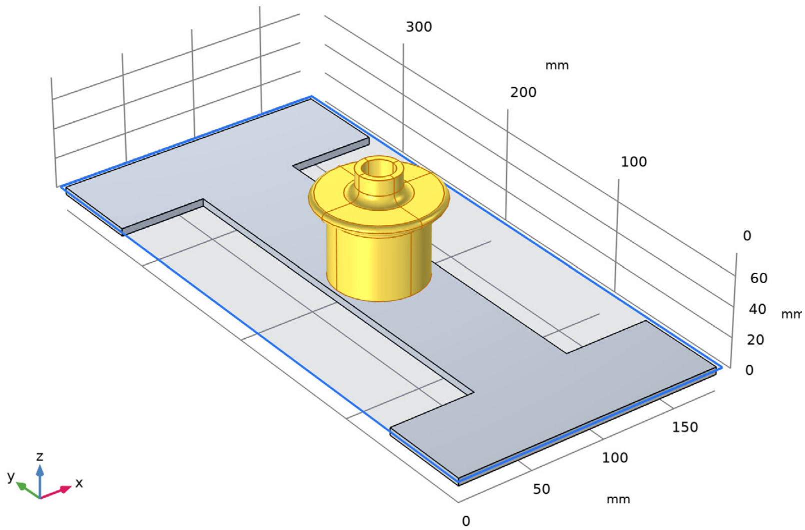
1 In the **Model Builder** window, expand the **Global Definitions>Geometry Parts>Spine** node, then click **Global Definitions>Geometry Parts>Anode top assembly> Central column 1 (pi2)**.

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 **Parts** and choose **Rod**.

**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 **Central column 1 (pi2)**.

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

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

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

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

**7** In the **New Cumulative Selection** dialog box, 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 **Parts** and choose **Rod connector**.

**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 **Rod 1 (pi3)**.

**4** From the **Work plane** list, choose **Rod connector Position (wp1)**.

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

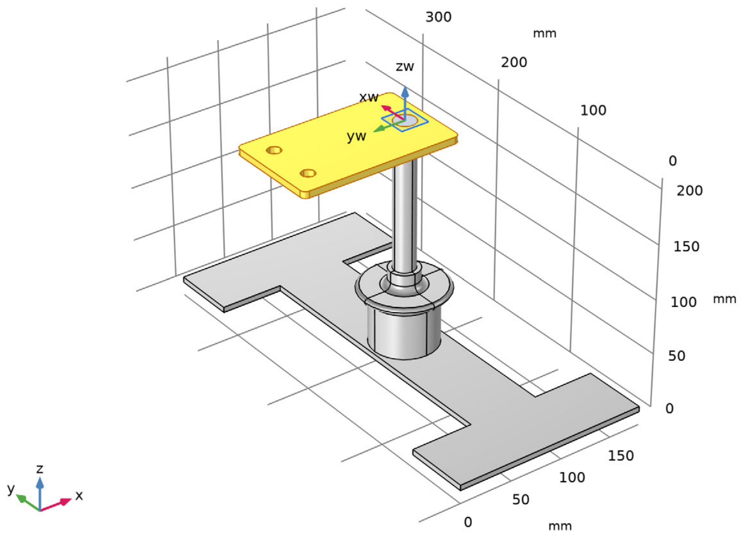
| Name | Expression | Value | Description |
|---|---|---|---|
| a_c_w_part | a_c_w_asm | 90 mm | Angle connector width |
| r_d_part | r_d_asm | 20 mm | Rod diameter |
| r_c_h_part | r_c_h_asm | 6 mm | Rod connector height |

**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 you will use the Copy operation.

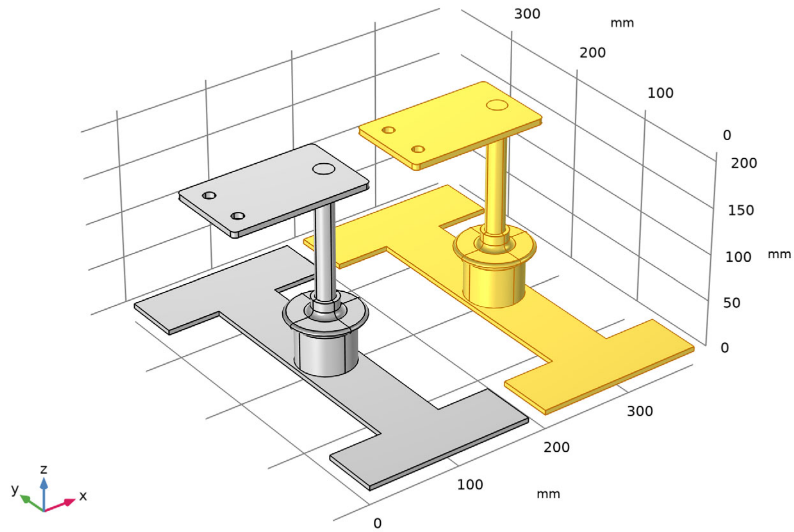
**1** In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

**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.




*Elbow connector 1 (pi5)*

1 In the **Geometry** toolbar, click **Parts** and choose **Elbow connector**.

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

3 Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Rod connector Position (wp4)**.

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

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

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

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

| Name | Expression | Value | Description |
|---|---|---|---|
| a_c_w_part | `a_c_w_asm` | 90 mm | Angle connector width |

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

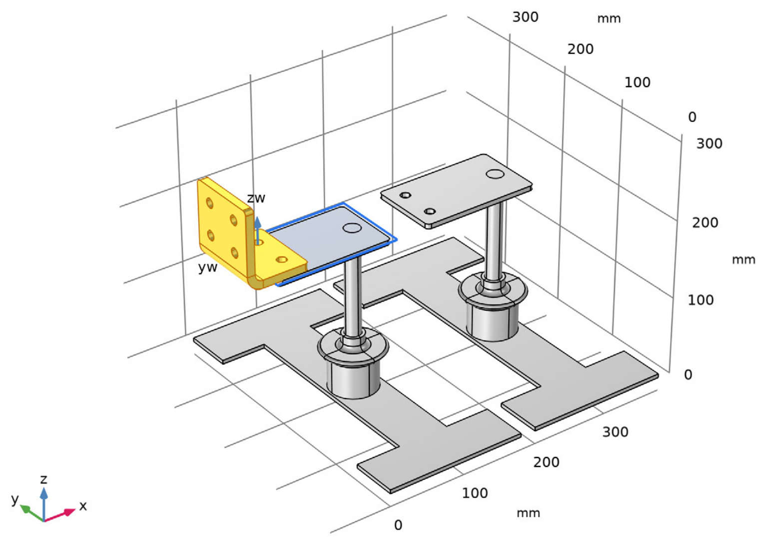
| Name | Keep | Instances | Contribute to |
|---|---|---|---|
| Union 1 | | √ | Copper |

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

| Name | Keep | Instances | Contribute to |
|---|---|---|---|
| Bolt medium Position | √ | √ | None |

**10** Click **Build Selected**.

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




*Angle connector 1 (pi6)*

**1** In the **Geometry** toolbar, click **Parts** 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)**.

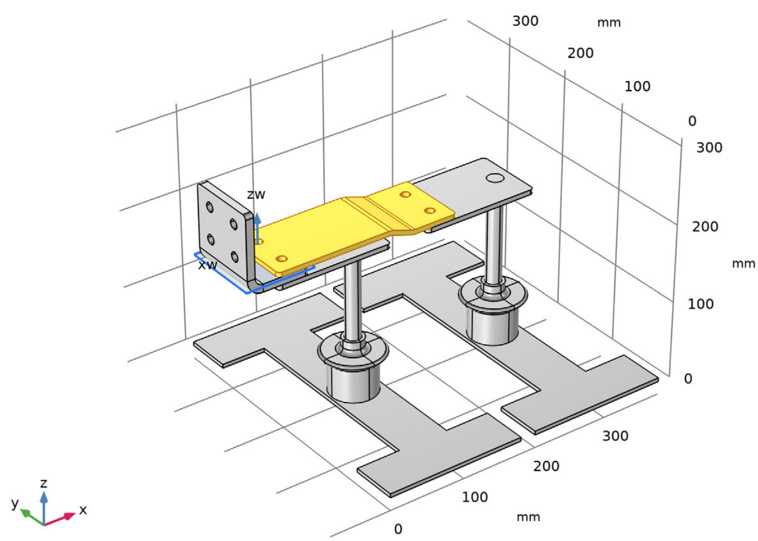
5 Find the **Coordinate system to match** subsection. From the **Take work plane from** list, choose **Elbow connector 1 (pi5)**.

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 **Parts** and choose **Bolt**.

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

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

| Name | Expression | Value | Description |
|---|---|---|---|
| size | 0 | 0 | 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 **Angle connector 1 (pi6)**.

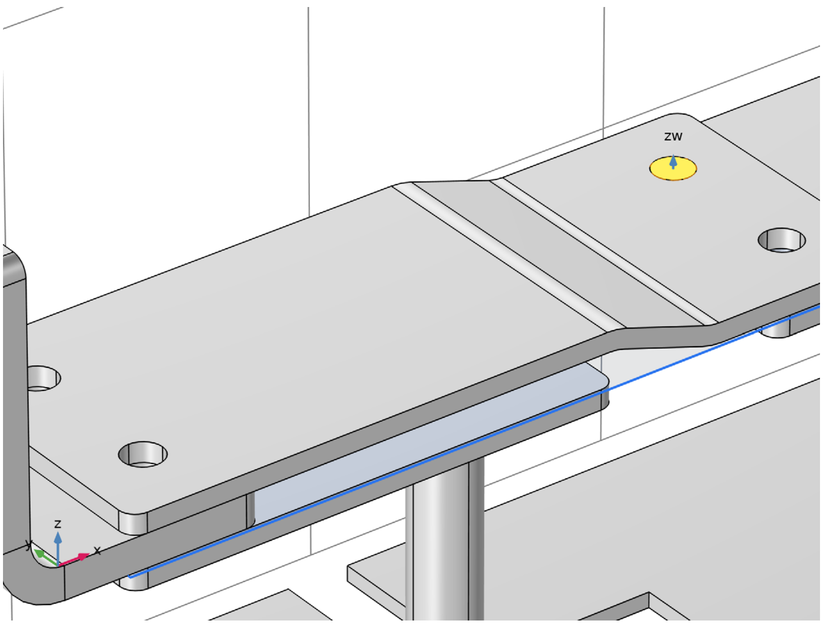
5 From the **Work plane** list, choose **Bolt Position (wp4)**.

6 Find the **Displacement** subsection. In the **zw** 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**.




*Bolt large*

1 In the **Geometry** toolbar, click **Parts** 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)**.

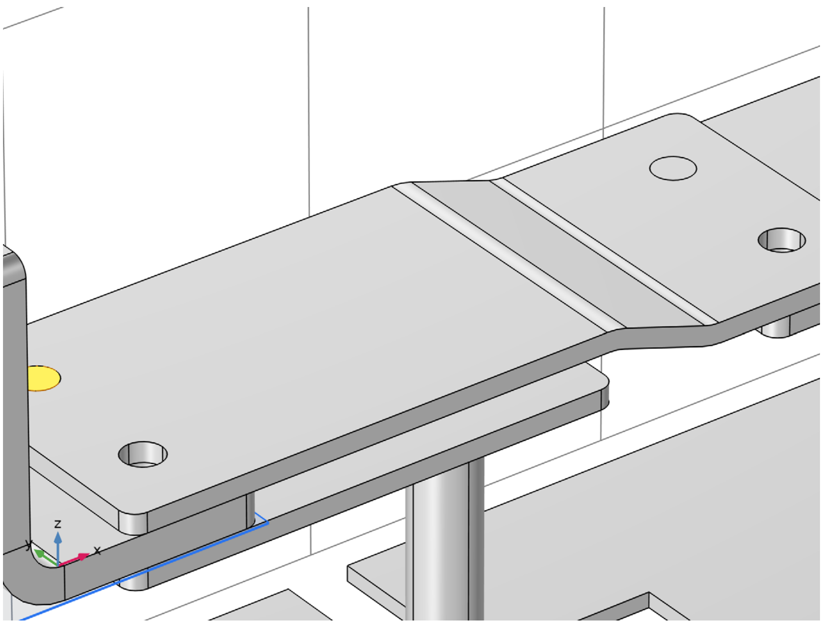
5 From the **Work plane** list, choose **Rod connector Position (wp4)**.

6 Find the **Displacement** subsection. In the **zw** 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.

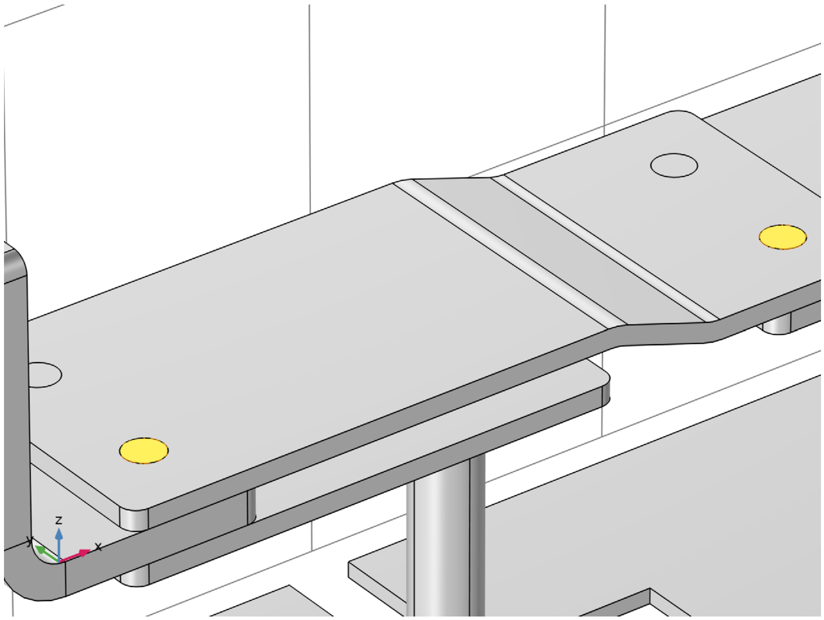
4 Select the **Keep input objects** check box.

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 with the positioning of this part.

*Intercell busbar Position*

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

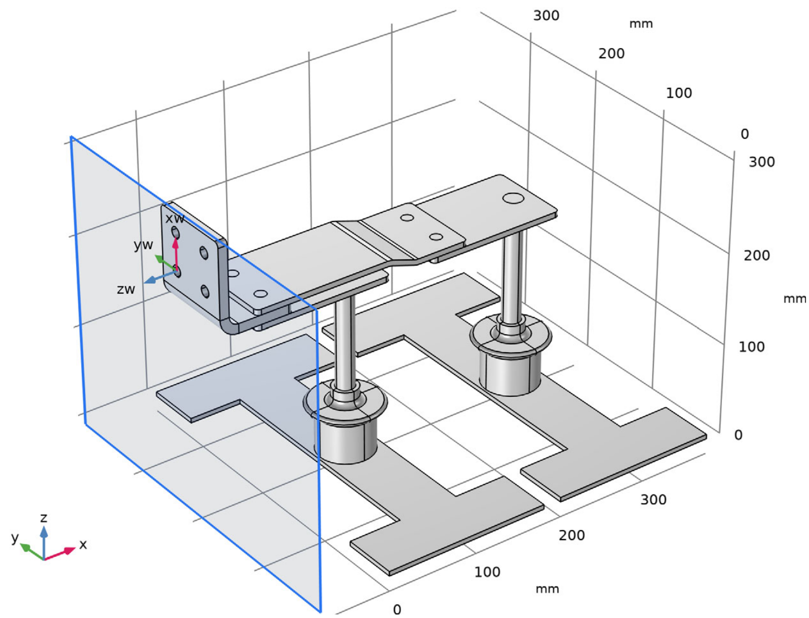
**2** In the **Settings** window for **Work Plane**, type `Intercell busbar Position` in the **Label** text field.

**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 the dimensions to control in the parametric sweep. 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)*

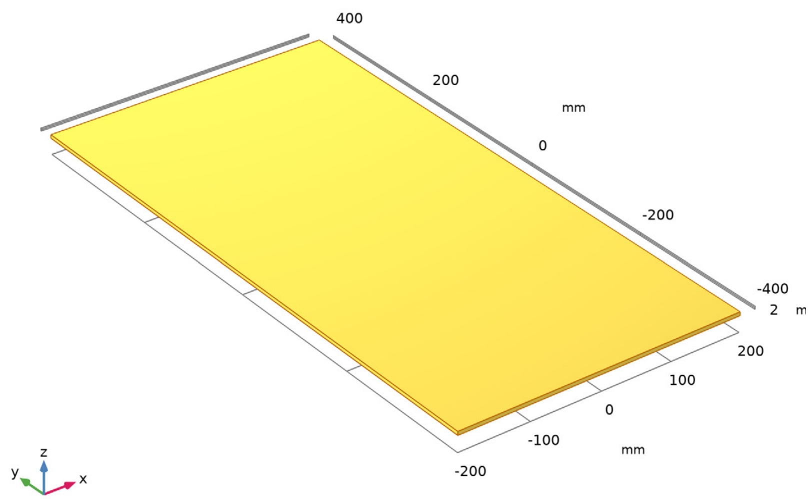
1. In the **Geometry** toolbar, click **Parts** and choose **Cell grid top**.
2. In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.
3. Click **New Cumulative Selection**.
4. In the **New Cumulative Selection** dialog box, Define Cumulative Selections for the two materials to collect all parts with the same material.
5. type `Titanium` in the **Name** text field.
6. Click **OK**.
7. In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.
8. Click **New Cumulative Selection**.
9. In the **New Cumulative Selection** dialog box, type `Copper` in the **Name** text field.
10. Click **OK**.
11. In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.
12. In the table, enter the following settings:

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| Block 1 | | √ | Titanium |

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

| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| Electrolyte boundary | √ | √ | None |

**14** Click **Build Selected**.




*Anode top assembly 1 (pi2)*

**1** In the **Geometry** toolbar, click **Parts** 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 |

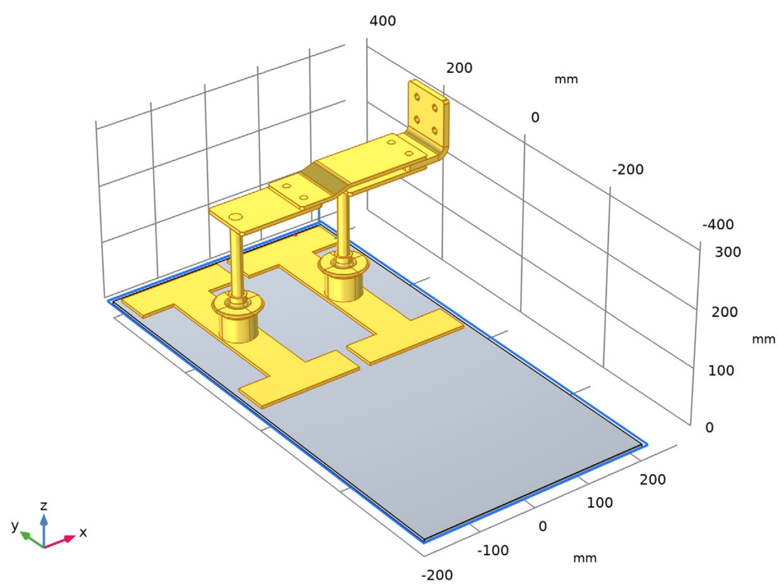
**7** Locate the **Domain Selections** section. In the table, enter the following settings:

| 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**.




### *Move 1 (mov1)*

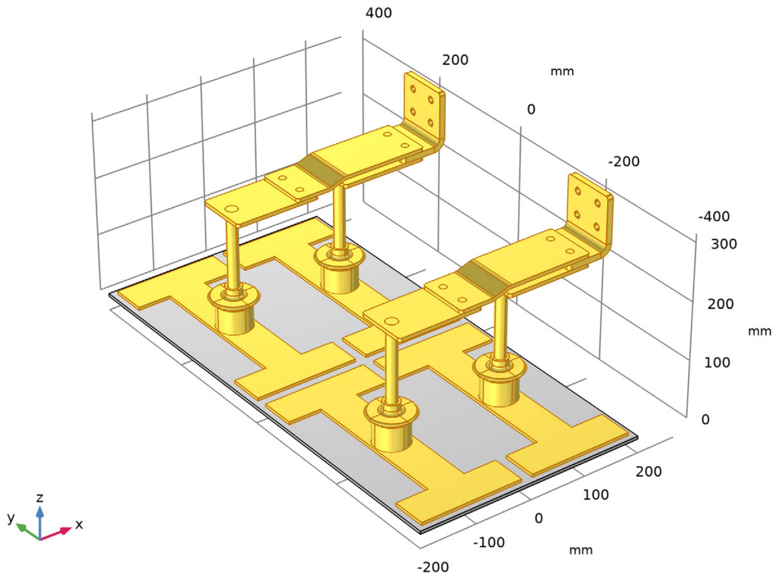
**1** In the **Geometry** toolbar, click **Transforms** and choose **Move**.

**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 values specified by the vector.

**5** Click **Build Selected**.




*Intercell busbar 1 (pi3)*

**1** In the **Geometry** toolbar, click **Parts** 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)**.

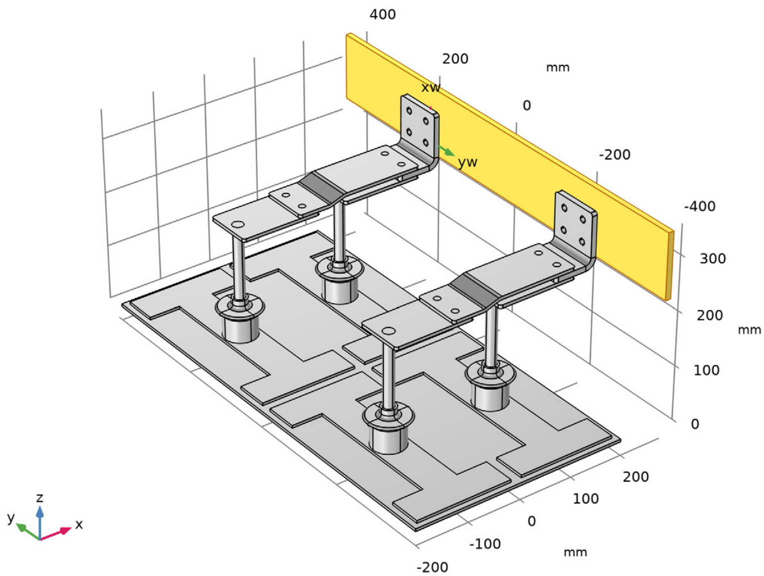
**7** Locate the **Domain Selections** section. In the table, enter the following settings:

| 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 **Parts** 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 **yw-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 |

*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.

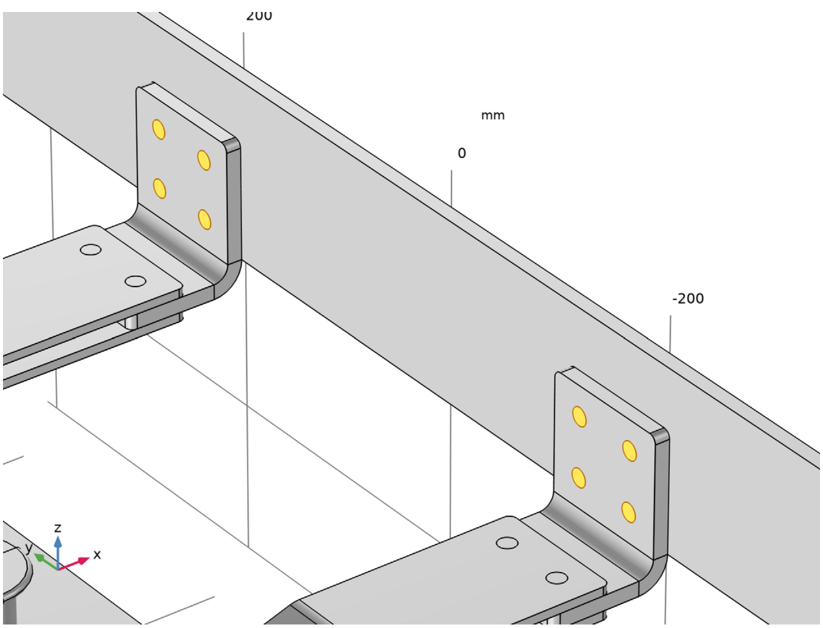
**4** From the **Specify** list, choose **Positions**.

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

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

**7** From the **Vertices to move to** list, choose **Bolt medium Position (Elbow connector 1) (Anode top assembly 1)**.

**8** Click **Build Selected**.

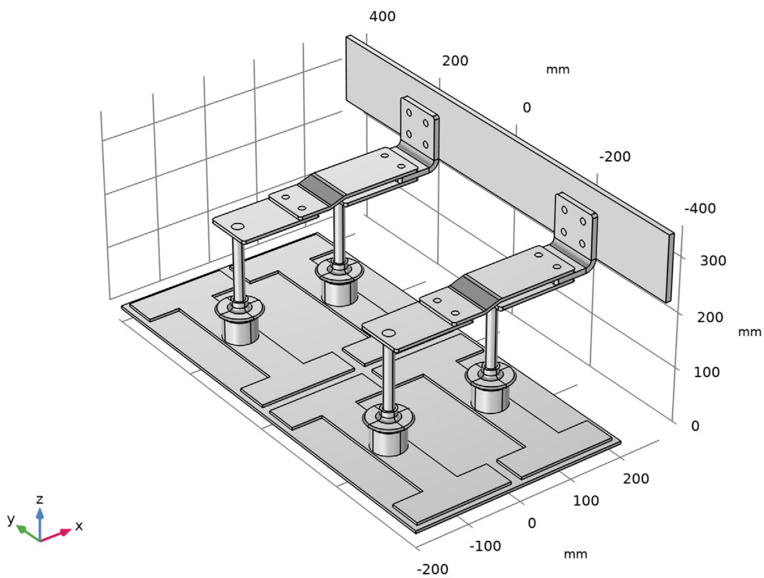


*Form Union (fin)*

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

3 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.

*Adjacent Selection 1 (adjsel1)*

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

2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

3 Click **Add**.

4 In the **Add** dialog box, in the **Input selections** list, choose **Titanium** and **Copper**.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, click **Build Selected**.

*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 + **Add**.

5 In the **Add** dialog box, 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 + **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Electrolyte boundary (Cell grid top 1)** and **Grounded boundary (Intercell busbar 1)**.

10 Click **OK**.