Created in COMSOL Multiphysics 5.4

Swept Meshing of a Bracket Geometry

Introduction

Using swept meshing in COMSOL Multiphysics you can create 3D meshes of prism and hexahedral elements. A swept mesh is an example of a semistructured mesh because it is structured in the sweep direction and can be either structured or unstructured orthogonally to the sweep direction. The swept mesher sweeps a face mesh along a domain to generate layers of mesh elements from a source to a destination. Both the source and destination can consist of several connected faces, as long as each destination face corresponds to at least one source face, and each source face corresponds to exactly one destination face or to a subset of it. The faces that connect the source to the destination are called the linking faces.

The swept mesher can automatically analyze the topology of a domain to determine the source, destination, and linking faces, as long as the geometry satisfies some criteria. In addition to the just mentioned requirements on the source and destination faces, the source and destination must also be opposite each other in the domain's topology. The domain must also be bounded by one shell, that is, holes are allowed only if they penetrate both the source and destination. Finally, the cross section of the domain along the direction of the sweep must be topologically constant.

This tutorial demonstrates how to create a swept mesh for a geometry that initially does not satisfy the requirements for swept meshing. You will learn how to use partitioning tools and virtual geometry operations to create domains for swept meshing, how to combine swept and tetrahedral meshes, and how to generate mesh plots to view the various element types.

Model Definition

The geometry shown in Figure 1 represents a bracket, which can be used to install an actuator that is mounted on a pin placed between the two large holes in the bracket arms.

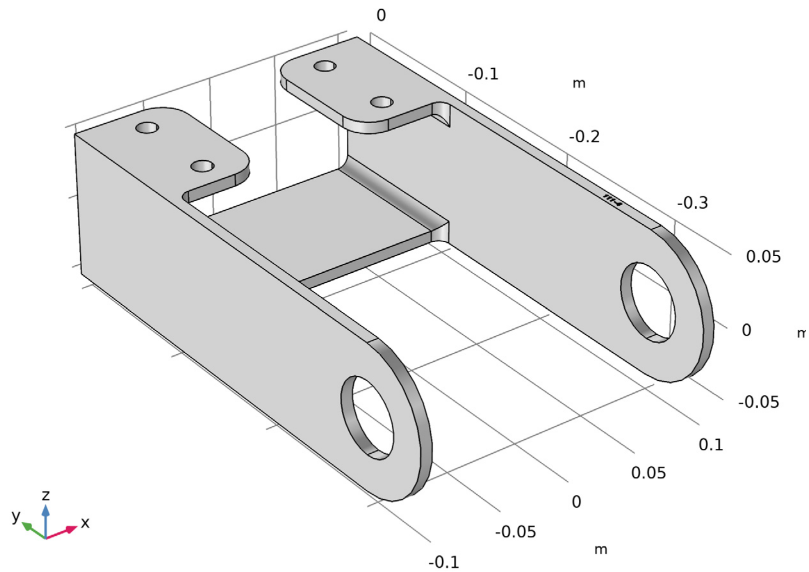


Figure 1: The geometry of the bracket used in this tutorial.

For a structural mechanics analysis of the bracket it is possible to create a free tetrahedral mesh, but for a geometry such as this with large flat regions it can be more efficient to create a swept mesh, or a swept mesh combined with tetrahedron mesh for the region around the fillets.

If you are interested in tutorials for modeling structural mechanics problems using this geometry, look for the series of models titled "Bracket" under Tutorials in the Structural Mechanics Module Application Library. Note, however, that running these models requires additional licenses.

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/bracket_swept_mesh`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

The bracket geometry for this tutorial has been saved in the COMSOL MPHBIN-format.

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `bracket.mphbin`.

5 Click **Import**.

MESH 1

Swept 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

   The swept mesher fails, since the geometry does not satisfy the requirements for generating a swept mesh. By partitioning the geometry we can create several domains that are possible to generate swept mesh for.

3 In the **Error** dialog box, click **OK**.

GEOMETRY 1

Partition Domains 1 (pard1)

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

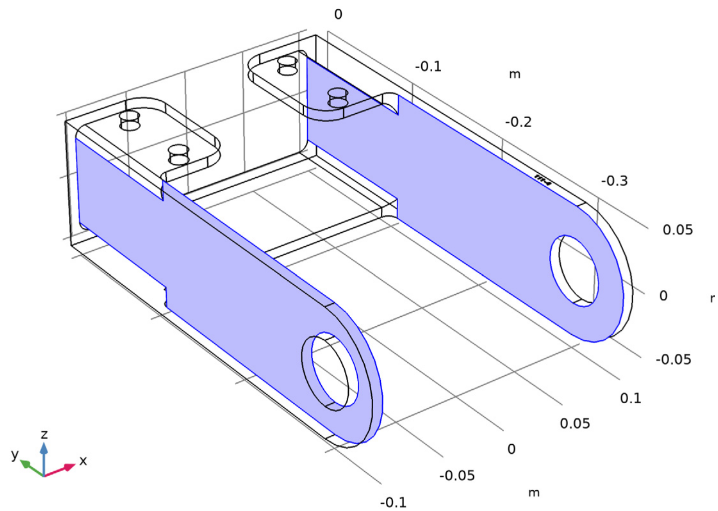
2 On the object **imp1**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended faces**.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

6 On the object **imp1**, select Boundaries 8 and 38 only, highlighted below.



7 Click **Build All Objects**.

The object is now partitioned along the base of the rounded corners into domains for which swept meshing is possible. You can switch to domain selection mode and open the **Selection List** window to step through the domains.

8 In the **Home** toolbar, click **Windows** and choose **Selection List**.

9 Click the **Select Domains** button in the **Graphics** toolbar.

Due to the topology of the object there are limited options for the sweep directions of the domains. For domains 3 and 4 the mesh can be swept only in the direction of the positive z-axis. For domain 2 there are two possibilities, the directions of the negative z-axis and positive y-axis. The remaining two domains can only be swept along the x-axis, domain 1 in the negative and domain 2 in the positive direction.

MESH 1

Now build the mesh again.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.

3 In the **Error** dialog box, click **OK**.

This time the swept mesher succeeds on all but one domain. A linking face in domain 5 contains additional faces that prevent the generation of a structured quad mesh on the linking face. We can get rid of these faces using the Ignore Edges virtual operation.

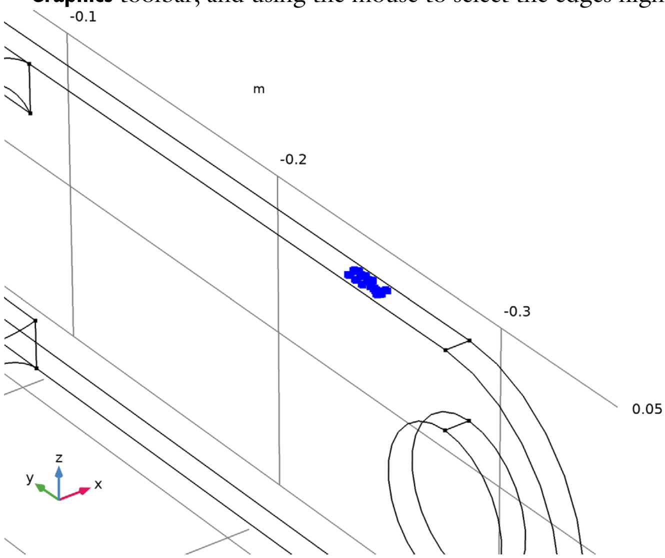
GEOMETRY 1

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

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

3 On the object **fin**, select Edges 123–164 only, by clicking the **Select Box** button on the **Graphics** toolbar, and using the mouse to select the edges highlighted below.



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

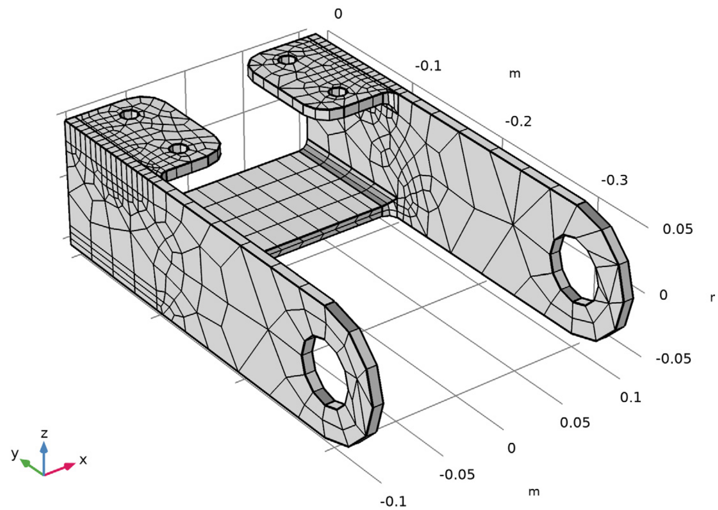
The selected edges are now hidden from the mesher, so that it will be possible to create a swept mesh for the domain.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.

For domain 2 the swept mesher selected the y-axis as the sweep direction. We can easily change the sweep direction for this domain by specifying the source faces for the sweep.



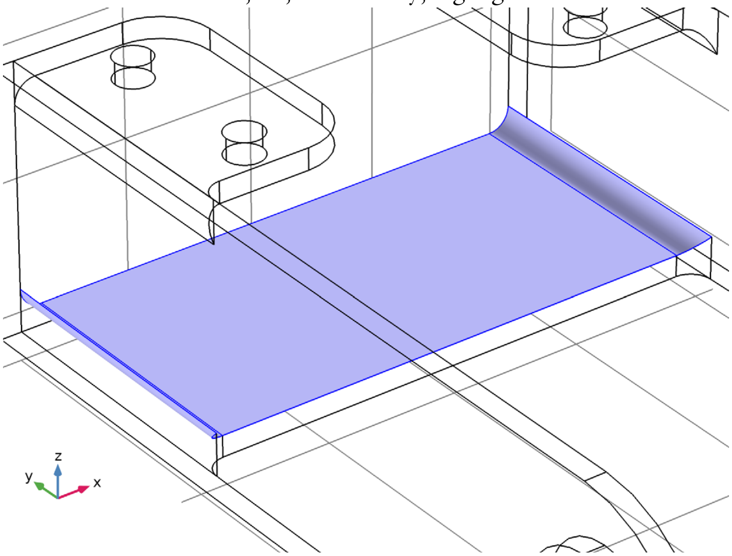
Swept 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept 1**.

2 In the **Settings** window for **Swept**, click to expand the **Source Faces** section.

3 Click the **Mesh Rendering** button in the **Graphics** toolbar.

4 Select Boundaries 12, 20, and 42 only, highlighted below.



Size

Before building the mesh, make some adjustments of the element size. First, change the maximum and minimum allowed element sizes for a better fit of the feature size of the geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `6[mm]`.

5 In the **Minimum element size** text field, type `3[mm]`.

Swept 1

Next, specify the number of element layers in the sweep direction for the swept mesher.

Distribution 1

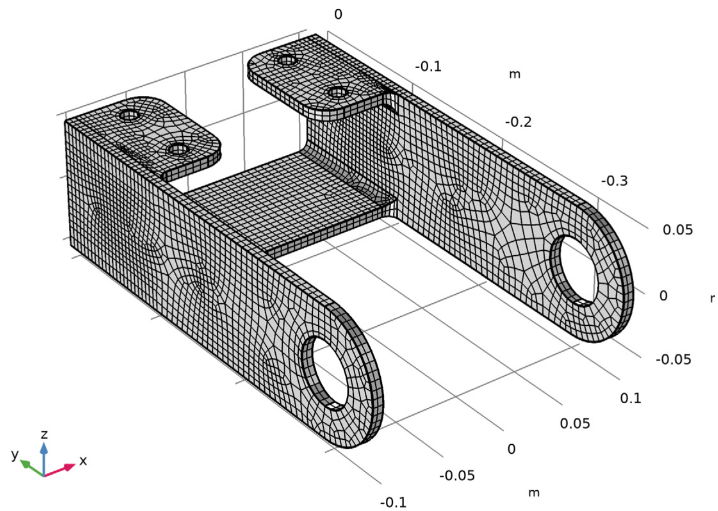
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `2`.

4 Click **Build All**.

5 Click the **Mesh Rendering** button in the **Graphics** toolbar.



The information in the **Messages** window shows that the mesh has close to 7000 elements. For more detailed information, look at the mesh statistics.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Statistics**.

The **Statistics** window displays information about the mesh quality including a mesh quality histogram. The latest mesh has a quite low minimum quality, around 0.01, and the histogram reveals that although the quality of the majority of the elements is high, there is a thin tail of low quality elements.

To find the location of the lowest quality elements, generate a mesh plot.

7 In the **Mesh** toolbar, click **Plot**.

RESULTS

Mesh 1

1 In the **Model Builder** window, under **Results>3D Plot Group 1** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click to expand the **Element Filter** section.

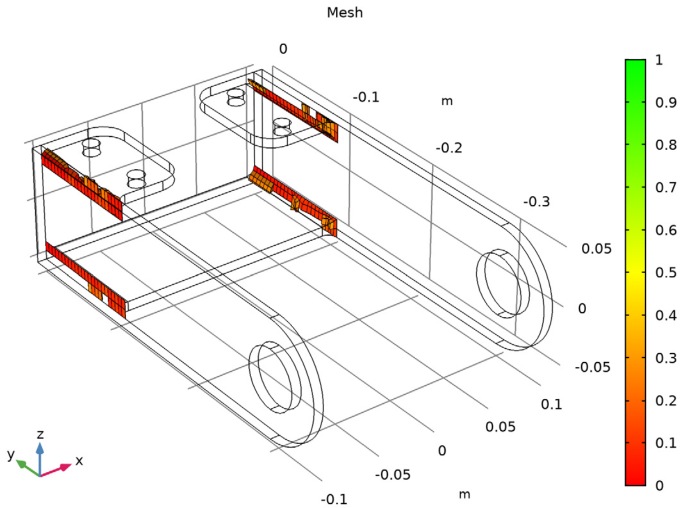
3 Select the **Enable filter** check box.

4 From the **Criterion** list, choose **Worst quality**.

5 In the **Fraction** text field, type `0.03`.

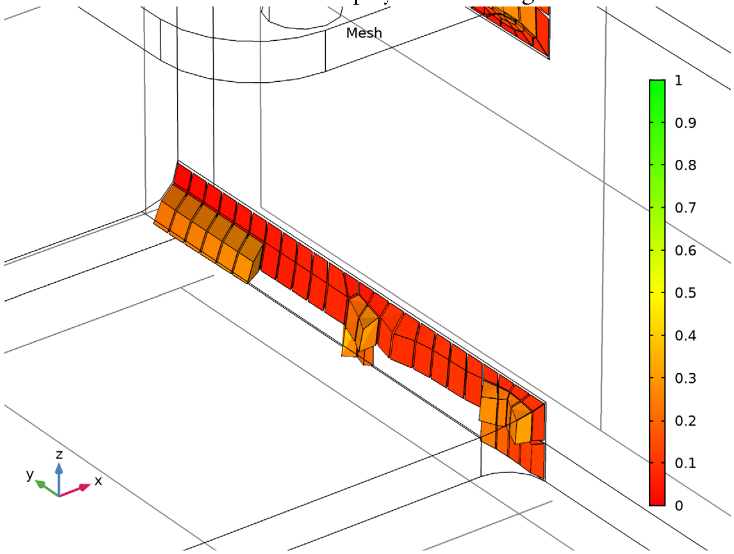
6 Click to expand the **Shrink Elements** section. In the **Element scale factor** text field, type `0.9`.

7 In the **3D Plot Group 1** toolbar, click **Plot**.



The elements with the lowest quality are all located in the region of the rounded corners, at the base of the fillet where the interior faces separate the domains. At this location the domains that contain the fillets become very thin, resulting in these distorted elements.

8 Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in to the corner on the bracket displayed below to get a better view of the elements.



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

A better strategy for partitioning would be to create domains that include the fillets together with a small region of the surrounding volume. This will also make it possible to generate a tetrahedral mesh for the region including the fillets, while using swept mesh in the remaining domains. Continue by testing this strategy.

GEOMETRY 1

To generate the domains around the fillets, first create solid objects by extruding a 2D drawing. These solid objects will become the tools that you will use to partition the bracket.

Work Plane 1 (wp1)

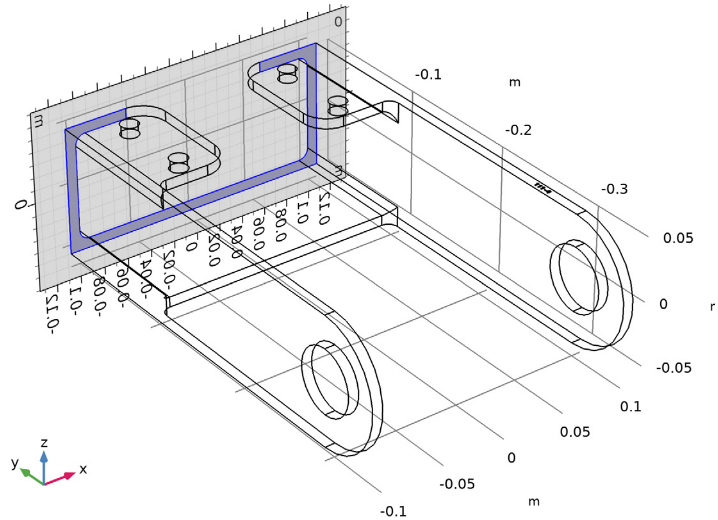
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Partition Domains 1 (pard1)** and choose **Delete**.

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

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

4 From the **Plane type** list, choose **Face parallel**.

5 On the object **imp1**, select Boundary 7 only.



6 In the **Offset in normal direction** text field, type `0.02`.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.033`.

4 In the **Height** text field, type `0.024`.

5 Locate the **Position** section. In the **xw** text field, type `-0.06`.

6 In the **yw** text field, type `0.086`.

Work Plane 1 (wp1)>Rectangle 2 (r2)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Rectangle 1 (r1)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **xw** text field, type `0.027`.

4 Click **Build Selected**.

Work Plane 1 (wp1)>Rectangle 3 (r3)

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.
2 In the **Settings** window for **Rectangle**, locate the **Position** section.
3 In the **yw** text field, type `-0.11`.
4 Click **Build Selected**.

Work Plane 1 (wp1)>Rectangle 4 (r4)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Rectangle 3 (r3)** and choose **Duplicate**.
2 In the **Settings** window for **Rectangle**, locate the **Position** section.
3 In the **xw** text field, type `0.027`.
4 Click **Build Selected**.

Work Plane 1 (wp1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.
2 Click **Build Selected**.

Extrude 1 (ext1)

1 In the **Geometry** toolbar, click **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (m)
0.14

4 Select the **Reverse direction** check box.
5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

   This option generates a selection that contains the output objects of the extrude operation. The selection can be used as input to subsequent operations and eliminates the need to select the objects, and its entities, by clicking in the **Graphics** window.
6 Click **Build All Objects**.

Partition Objects 1 (par1)

Using the extruded solids, you can now partition the bracket. For this purpose, use the Partition Objects operation which is similar to the Partition Domains operation you used earlier. Beside the difference that one operates on objects and the other on domains, the

Partition Domains operation provides more choices for the partitioning tools, and it can be added to the geometry sequence after the **Form Union** node.

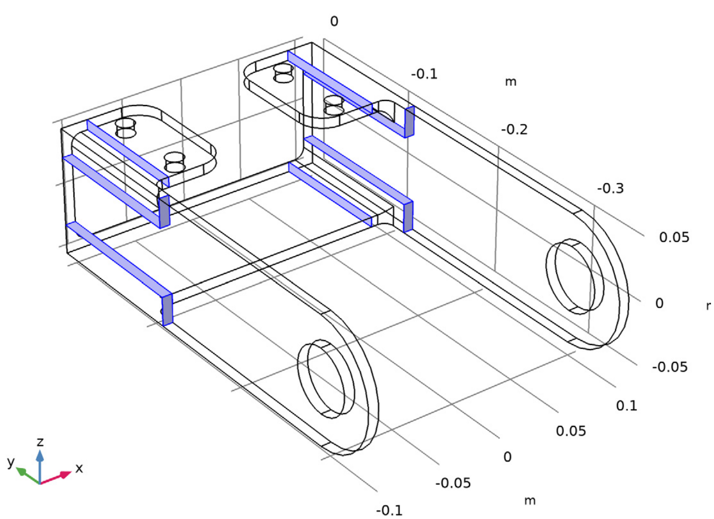
1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2 Select the object **imp1** only.
3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
4 From the **Tool objects** list, choose **Extrude 1**.
5 Click **Build All Objects**.

Mesh Control Faces 1 (mcf1)

While it is advantageous to be able to partition the geometry for meshing, the additional domains often require extra steps during the simulation setup, for example to define material properties. To avoid this, use the Mesh Control Faces feature to remove the interior faces that separate the domains. The faces selected in the mesh control feature will be available for meshing, but removed from the geometry as soon as they are meshed.

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Faces**.
2 In the **Settings** window for **Mesh Control Faces**, locate the **Input** section.
3 From the **Faces to include** list, choose **Extrude 1**.

The **Extrude 1** selection contains the faces that remain in the geometry from the **Extrude 1** operation. These faces are the interior faces that separate the domains.



The selected faces will be removed after meshing.

MESH 1

The geometry is now ready, and you can continue with setting up the mesh. Because you are combining the swept mesh with a tetrahedral mesh, first remove the domains containing the fillets from the **Swept 1** operation.

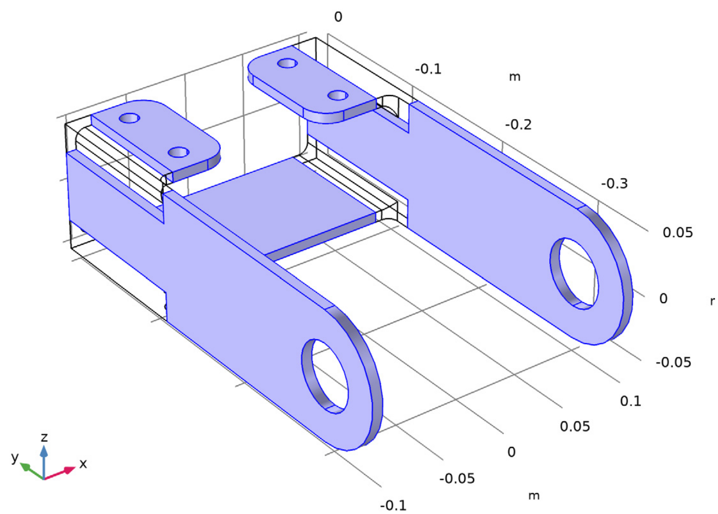
Swept 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept 1**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 4–6, and 9 only, highlighted below.

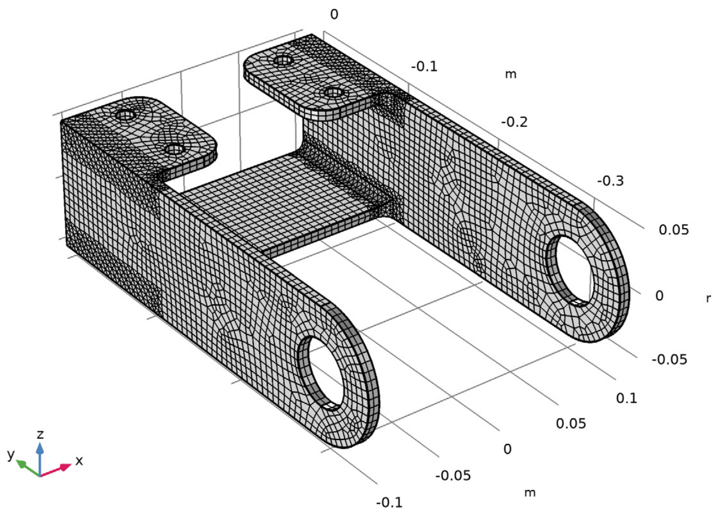


5 Click **Build All**.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

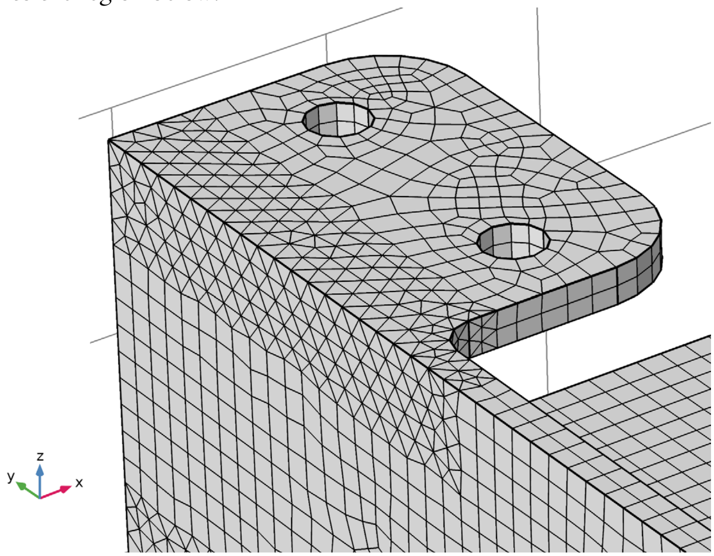
2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.



Although not visible from the surface mesh rendered in the **Graphics** window, the mesher automatically inserted a layer of pyramid elements between the hexahedral and tetrahedral elements. Further ahead, you will generate a mesh plot to visualize these elements.

By default, the mesher also smoothes the transition in element size across the removed mesh control faces. The effect of the smoothing is detectable by looking at the surface elements at the location of the removed edges of the interior faces.

3 Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in to the region below.



In the figure above, where the quadrilateral elements meet the triangular elements, the elements do not follow the straight line of the removed edges.

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

5 Right-click **Mesh 1** and choose **Statistics**.

According to the information in the **Statistics** window, the minimum element quality of the mesh is now around 0.2, which is considered good for most applications. It is also possible to display information for the individual element types in the mesh.

6 From the **Element type** list, choose **Tetrahedron**.

The minimum quality of the tetrahedral elements corresponds to the overall minimum quality of the mesh, that is, the element with the lowest quality is a tetrahedron.

Some simulation types require a better resolution of the curved faces of the fillets. For example, when you are interested in accurately determining the stresses in a structural mechanics analysis. You can specify smaller elements for these regions by using a Size attribute for the **Free Tetrahedral 1** node.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

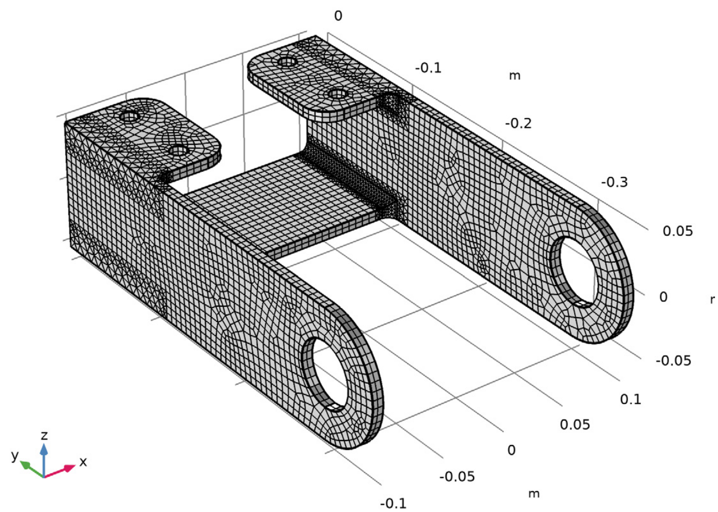
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Free Tetrahedral 1

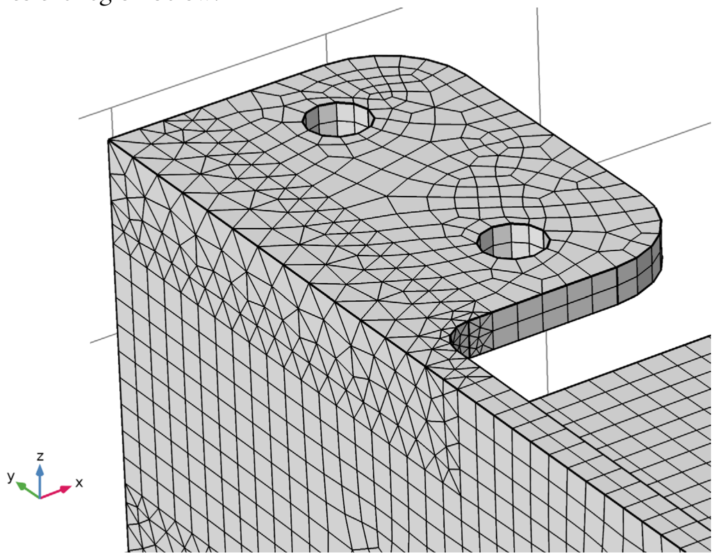
Next, test how to turn off the option to smooth the transition in element size across the removed mesh control faces.

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Control Entities** section.
3 Clear the **Smooth across removed control entities** check box.
4 Click **Build All**.



In this final mesh, which has approximately 20000 domain elements the fillet regions are better resolved by the mesh.

5 Click the **Zoom Box** button on the **Graphics** toolbar and then use the mouse to zoom in to the region below.



With the option to smooth the transition in element size across the removed mesh control faces turned off, the interface between the triangular and quadrilateral elements follows the straight line of the removed edges.

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

7 In the **Model Builder** window, right-click **Mesh 1** and choose **Statistics**.

A look in the **Statistics** window reveals that the minimum element quality is slightly higher than before.

RESULTS

In the final steps of this tutorial, modify the mesh plot you created earlier to be able to view how the layer of pyramid elements interfaces the hexahedral elements in the mesh.

Mesh 1

1 In the **Model Builder** window, under **Results>3D Plot Group 1** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Level** section.

3 From the **Element type** list, choose **Pyramid**.

4 Locate the **Element Filter** section. Clear the **Enable filter** check box.

5 Locate the **Color** section. From the **Element color** list, choose **Magenta**.

Filter 1

1 Right-click **Results>3D Plot Group 1>Mesh 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `(x<-0.08)*(z>0.02)*(y>-0.13)`.

4 In the **3D Plot Group 1** toolbar, click **Plot**.

Mesh 2

1 Right-click **Mesh 1** and choose **Duplicate**.

2 In the **Settings** window for **Mesh**, locate the **Level** section.

3 From the **Element type** list, choose **Hexahedron**.

4 Locate the **Color** section. From the **Element color** list, choose **Gray**.

5 In the **3D Plot Group 1** toolbar, click **Plot**.

6 Click the **Zoom Box** button on the **Graphics** toolbar and then use the mouse to zoom in to get a better view of the elements.