Model created in COMSOL Multiphysics 6.4

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 surface 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 to each other in the domain's topology. Holes and isolated faces in the domain are allowed only if they penetrate both the source and destination. Also, the source faces can be disconnected face components while the destination must be a connected face component.

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 find elements of lower quality and 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.

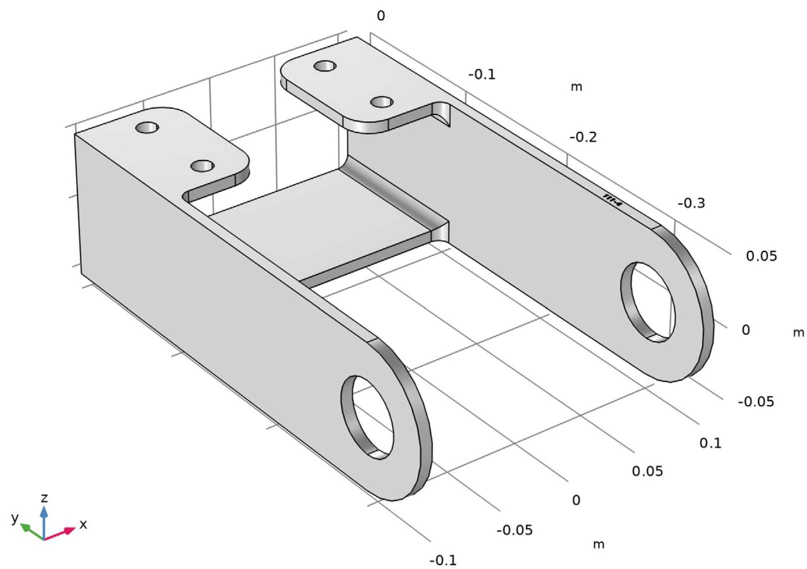


Figure 1: The geometry of the bracket used in this tutorial.

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 tetrahedral mesh for the regions 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 solving 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 **Geometry** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Source** 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 **Mesh** toolbar, click **Swept**.

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

3 From the **Elements** list, choose **Prisms** to create a swept prism mesh. This will make it easier to see in which direction the mesh is swept.

4 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.

5 In the **Error** dialog, click **OK**.

GEOMETRY 1

Partition Domains 1 (pard1)

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

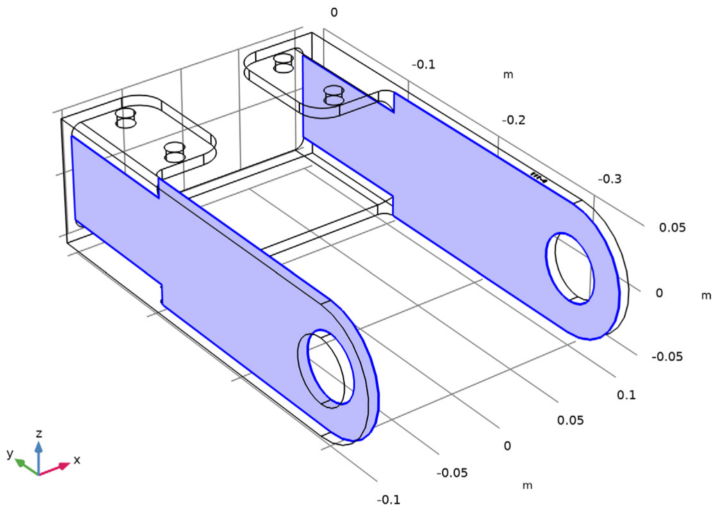
2 On the object **imp1**, select Domain 1 only.

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.

SELECTION LIST

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

2 In the **Model Builder** window, click **Partition Domains 1 (pard1)**.

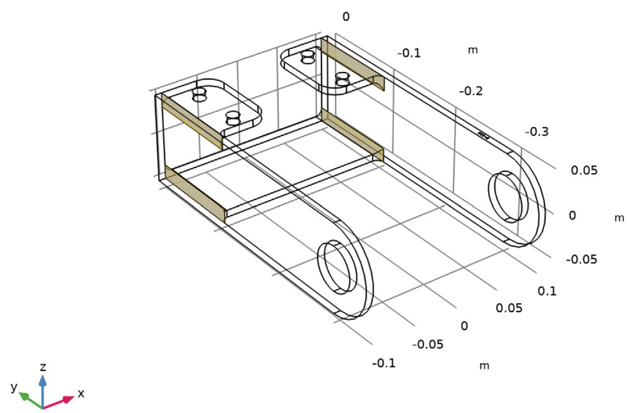
3 In the **Graphics** window toolbar, click ▾ next to **Select Boundaries**, then choose **Select Domains**.

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. Domain 2 can be swept in the directions of the negative z-axis, positive y-axis, and along the x-axis. The remaining two domains can only be swept along the x-axis, domain 1 in the negative and domain 5 in the positive direction. A short summary of the requirements on a geometry for the sweep to work can be found in the *Introduction* and the full list is found in the documentation for the Swept operation.

MESH 1

1 In the **Model Builder** window, click **Mesh 1**.

2 In the **Geometry Cleanup** dialog that opens, click **Open Wizard** to open the **Cleanup Wizard**.



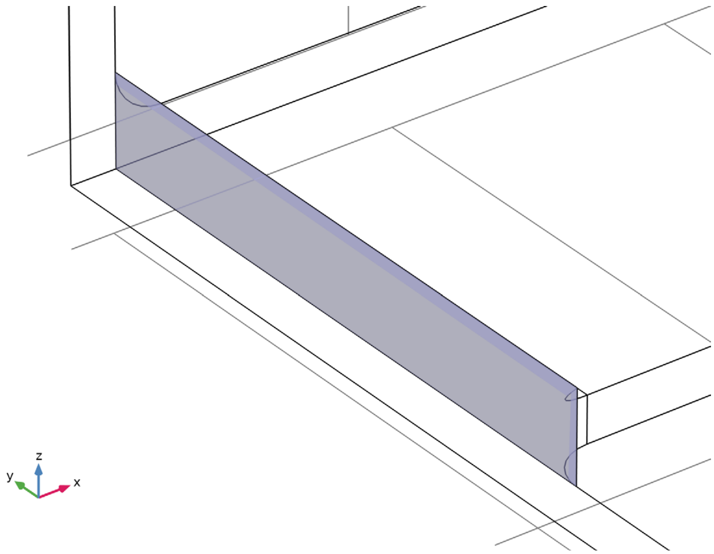
The partitioning created some narrow domain regions that the geometry analysis is detecting.

CLEANUP WIZARD

1 Go to the **Cleanup Wizard** window.

2 In the tree, select **Narrow Domain Regions (4)** > **Region 1 (0–5.7E-4 m)**.

3 Click **Zoom to Selection**.



These narrow domain regions are not ideal in a geometry, but we will continue with the meshing to see how they are affecting the quality of the swept mesh before we decide on something else to do.

Keep the Cleanup Wizard window open for now, until we have decided how to partition the geometry in a better way.

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**.

This time the swept mesher succeeds on all domains but domain 5. The reason are the numbers on boundary 50. We can get rid of the additional 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 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

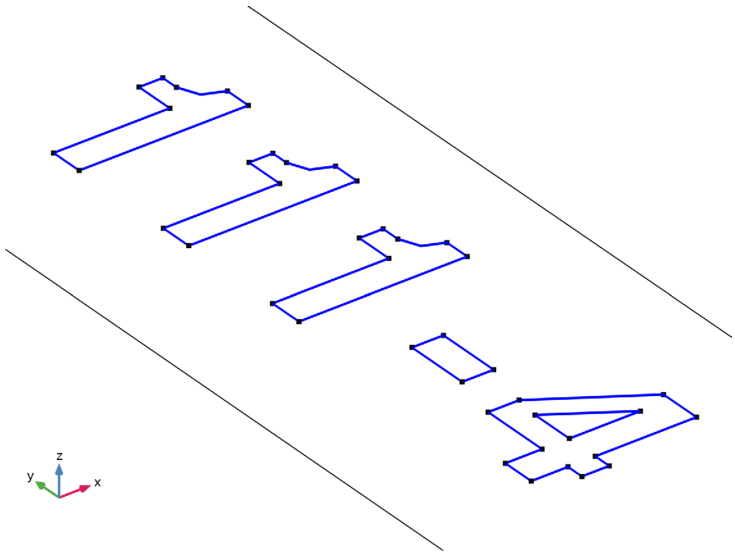
3 Click the **Paste Selection** button for **Edges to ignore**.

4 In the **Paste Selection** dialog, type 123-164 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

7 Click the **Zoom to Selection** button for **Edges to ignore**.



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

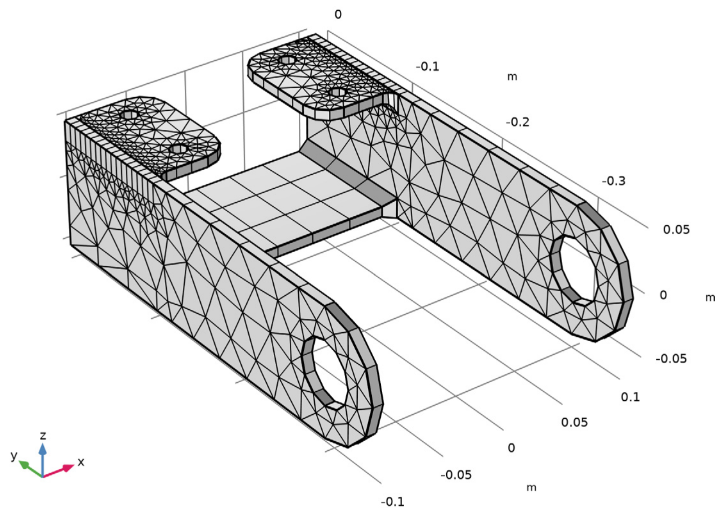
The selected edges are now hidden from the mesher, so that it will be possible to create a swept mesh for the domain.

MESH 1

1 Right-click **Mesh 1** and choose **Build All**.

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

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



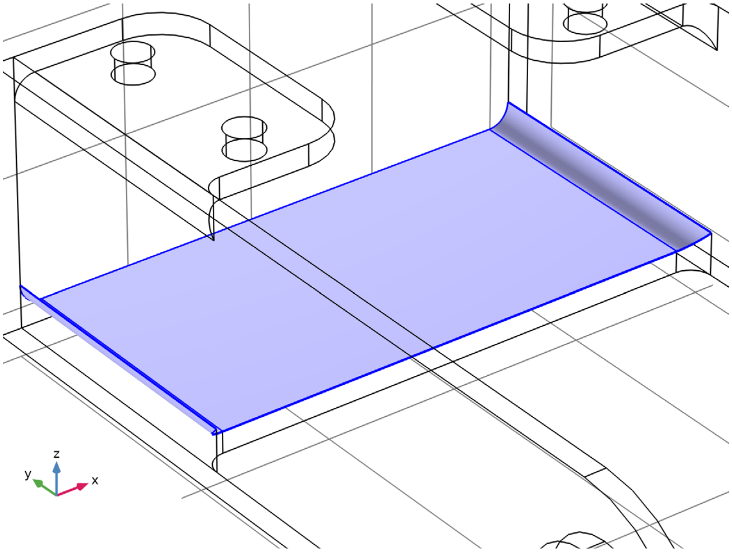
Swept 1

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 predefined element size for a better fit of the feature size of the geometry.

1 In the **Model Builder** window, click **Size**.

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

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

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `6[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, right-click **Swept 1** and choose **Distribution**.

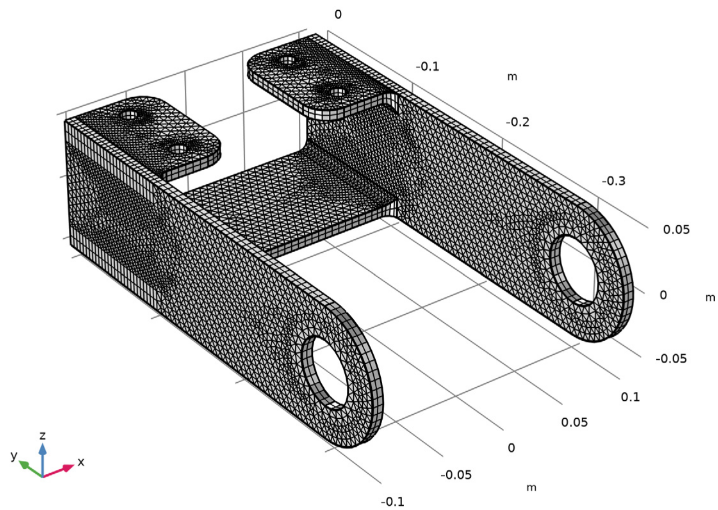
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

4 Click **Build All**.

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

6 In the **Model Builder** window, click **Mesh 1**.



The information in the **Messages** window shows that the mesh has close to 17,000 elements. Open the mesh **Statistics** to get a more detailed information about the quality of the whole mesh.

7 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 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 element of lower quality.

Generate a mesh plot to find the locations the mesh elements with quality lower than a certain threshold.

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

RESULTS

Mesh 1

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

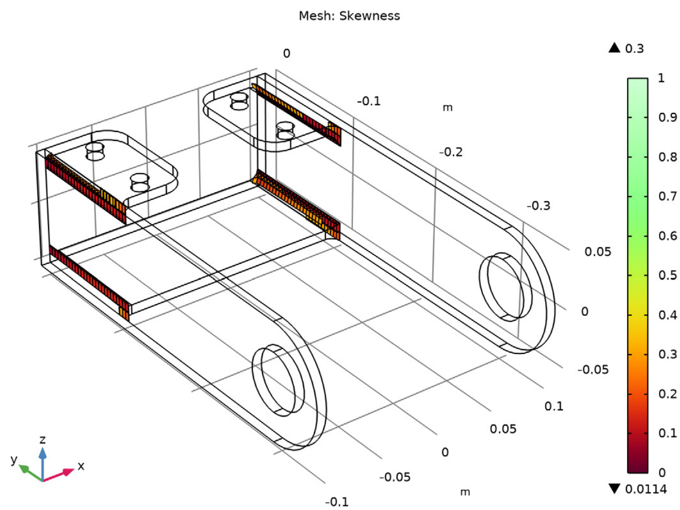
2 Select the **Enable filter** checkbox.

3 Click the  button. From the menu, choose **Mesh > qualskewness - Element quality (Skewness) - 1**.

4 In the **Expression** text field, type `qualskewness<0.3`.

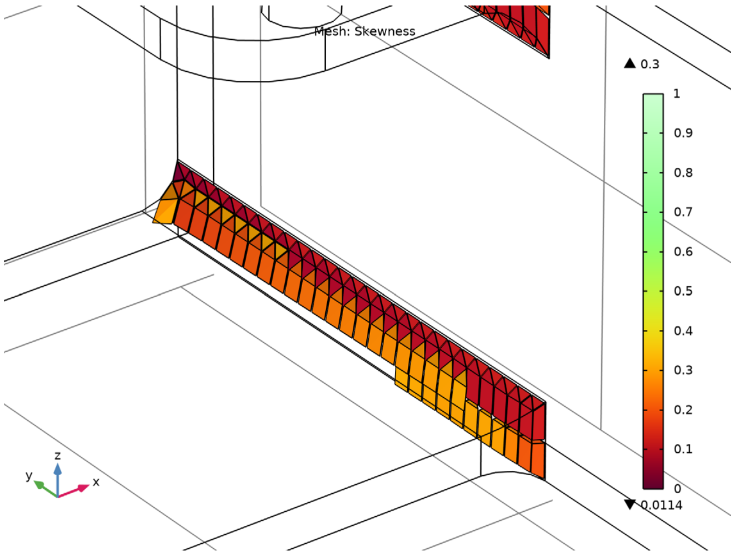
5 Click to expand the **Shrink Elements** section. In the **Element scale factor** text field, type `0.9`.

6 In the **Mesh Plot 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.

7 Click the **Zoom Box** button in 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.



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

GEOMETRY 1

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.

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.

Import 1 (imp1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Import 1 (imp1)**.

2 In the **Settings** window for **Import**, click **Build Selected**.

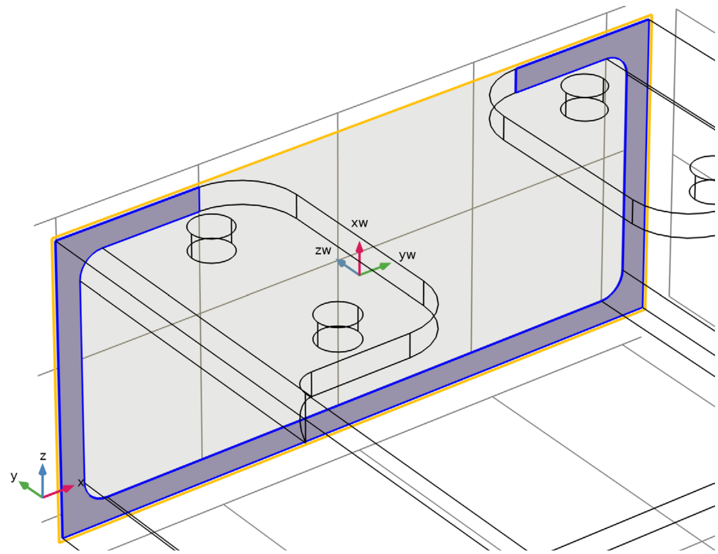
Work Plane 1 (wp1)

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 type** list, choose **Face parallel**.

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



5 In the **Offset in normal direction** text field, type `0.02[m]`.

6 Click **Go to Plane Geometry**.

Work Plane 1 (wp1) > 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 `0.033[m]`.

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

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

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

Work Plane 1 (wp1) > Array 1 (arr1)

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

2 Select the object **r1** 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 `0.087[m]`.

7 In the **yw** text field, type `-0.196[m]`.

8 Click **Build Selected**.

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

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** 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 (m)
0.14

4 Select the **Reverse direction** checkbox.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

6 From the **Show in physics** list, choose **Off**.

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.

7 Click **Build Selected**.

Partition Domains 1 (pard1)

Using the extruded solids, you can now partition the bracket using the **Partition Domains** operation used earlier.

1 In the **Model Builder** window, click **Partition Domains 1 (pard1)**.

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

3 From the **Partition with** list, choose **Objects**.

4 Click to select the **Activate Selection** toggle button for **Objects**.

5 From the **Objects** list, choose **Extrude 1**.

6 Clear the **Keep objects** checkbox.

7 Click **Build All Objects**.

Mesh Control Faces 1 (mcf1)

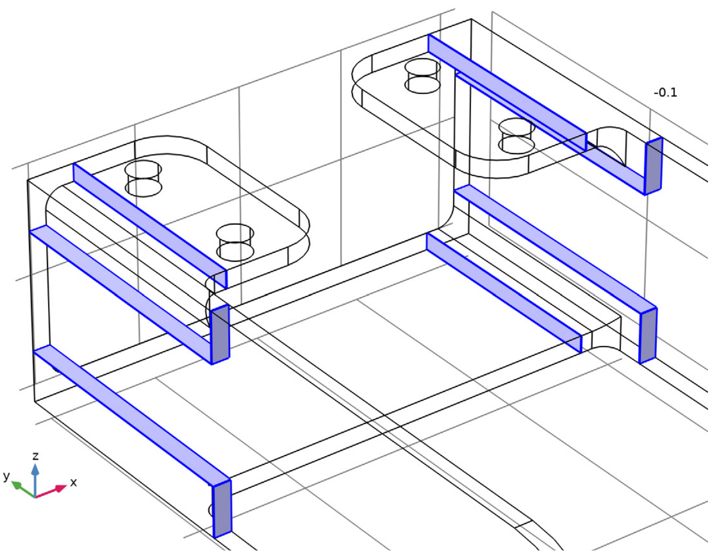
While it is advantageous to be able to partition the geometry for meshing, the additional domains often require extra steps during the simulation setup, for example to define material properties. To avoid this, use the Mesh Control Faces feature to remove the interior faces that separate the domains.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



4 Click **Build Selected**.

CLEANUP WIZARD

1 Go to the **Cleanup Wizard** window.

2 Click the **Refresh** button in the window toolbar to make sure that the analysis is not detecting anything new in the geometry.

3 Close the **Cleanup Wizard** window as the **Remaining Details** list is now empty.

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.

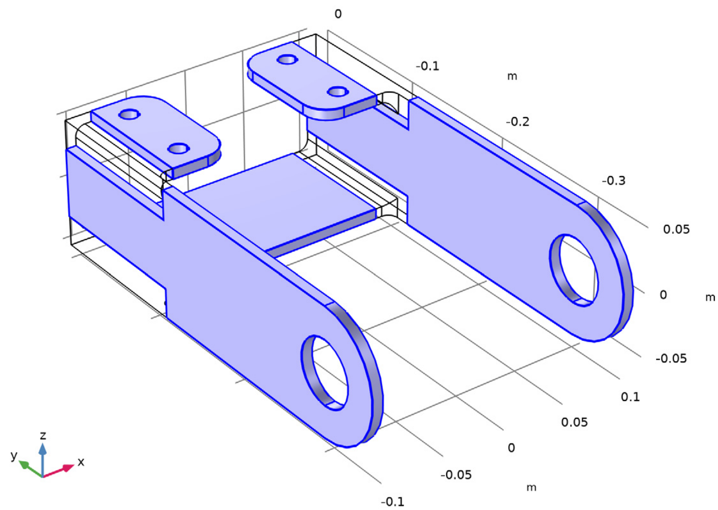
Swept 1

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.



To lower the number of elements, generate hexahedral elements by meshing the source faces with a quadrilateral mesh.

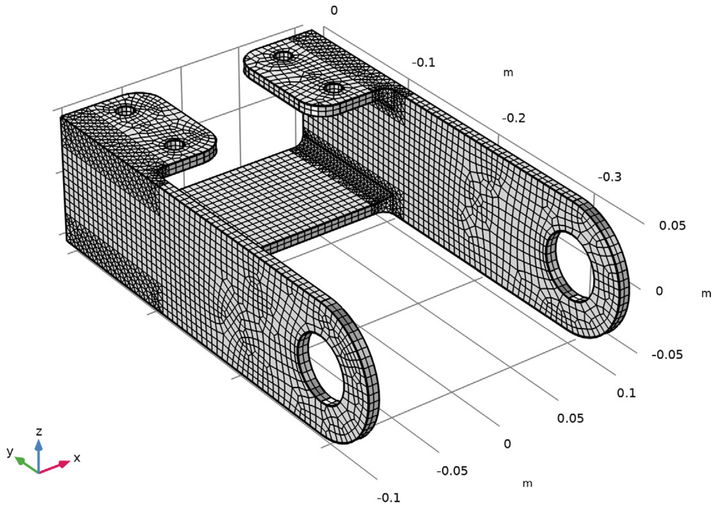
5 Locate the **Mesh Generation** section. From the **Elements** list, choose **Hexahedra**.

6 Click **Build All**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **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.

3 In the **Model Builder** window, 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.

4 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. You can also find this information in the Information section in the Settings window of the Swept and Free Tetrahedral nodes.

Size 1

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. Here, we choose to only override the Minimum element size and the Curvature factor with smaller values.

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**.
4 Click the **Custom** button.
5 Locate the **Element Size Parameters** section. Select the **Minimum element size** checkbox.
6 Select the **Curvature factor** checkbox.

Assume that some of the fillet faces require an even finer mesh. Add another Size node that we apply to those boundaries.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 11 and 36 only.
5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
   Only the minimum element size and the curvature factor need to be changed.
6 Click the **Custom** button.
7 Locate the **Element Size Parameters** section. Select the **Minimum element size** checkbox.
8 Select the **Curvature factor** checkbox.
9 Click **Build Selected**.

Size

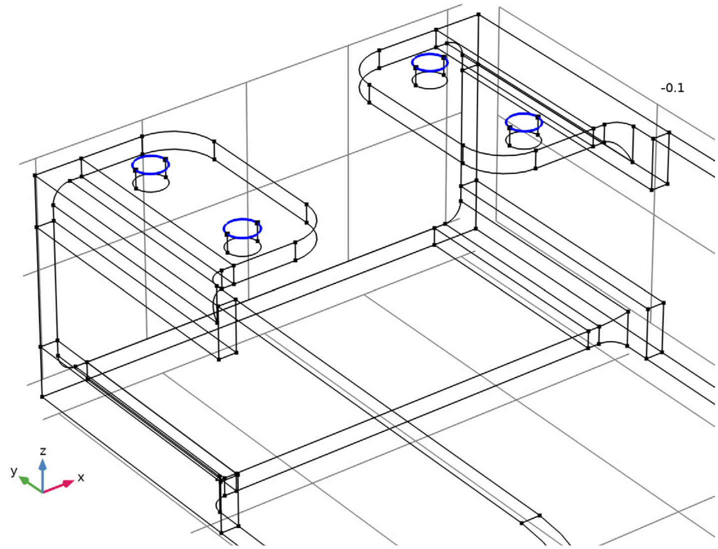
We will also make sure to resolve the curvature around the bolt holes better by first meshing the top edges of the cylindrical holes with a specified distribution. This must be done before the mesh is swept in the adjacent domains, so we start by building the top **Size** node as this will make sure the operation is added directly after this node.

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

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Edges 42, 43, 47, 48, 78, 79, 83, and 84 only.



Distribution 1

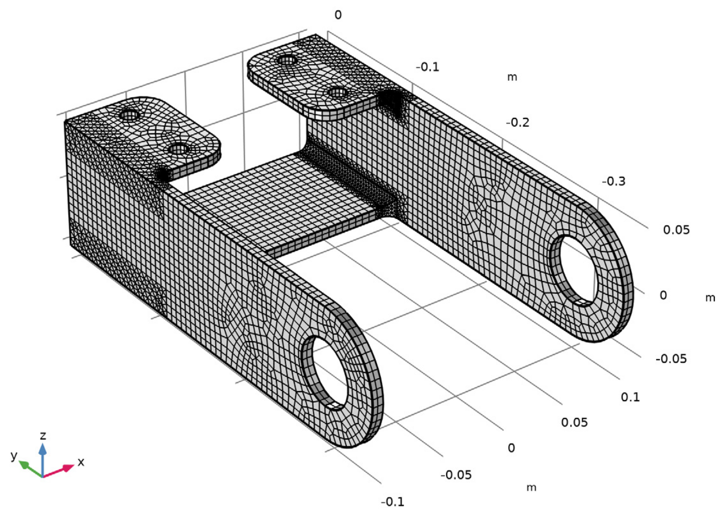
1 Right-click **Edge 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 8.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** right-click **Free Tetrahedral 1** and choose **Build All**.



In this final mesh, which has approximately 23,000 domain elements, the fillet regions are better resolved by the mesh.

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

A look in the **Statistics** window reveals that the minimum element quality is slightly lower 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** > **Mesh Plot 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** checkbox.

5 Locate the **Coloring and Style** section. From the **Element color** list, choose **Magenta**.

Filter 1

1 Right-click **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 **Mesh Plot 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 **Coloring and Style** section. From the **Element color** list, choose **Gray**.

5 In the **Mesh Plot 1** toolbar, click **Plot**.

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