Adjusting the Element Size for the Unstructured Mesh Generator

Introduction
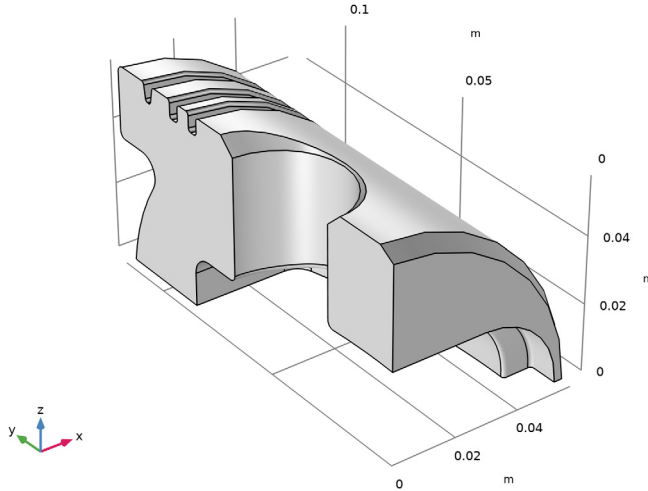
The tet mesher creates an unstructured tetrahedral mesh. It is the most general meshing technique and does not pose any constraints on the structure of the geometry. Hence you can use it to mesh any object. There are nine predefined parameter sets for the mesher, ranging from "extremely fine" to "extremely coarse." These settings result in a good mesh for most geometries and simulation problems. In addition, you can tune the mesh parameters individually, as demonstrated in this tutorial.

Model Definition

Create a tetrahedral mesh for the geometry of an engine piston as shown in the following figure.

As you can see, the geometry contains small details such as fillets and chamfers. To better resolve these details with the mesh, you will work with the following mesh parameters:

- Minimum element size
- Curvature factor
- Resolution of narrow regions
- Maximum element growth rate

You will also learn how to use the tools for assessing the mesh quality.

Application Library path: COMSOL_Multiphysics/Meshing_Tutorials/
piston_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**.
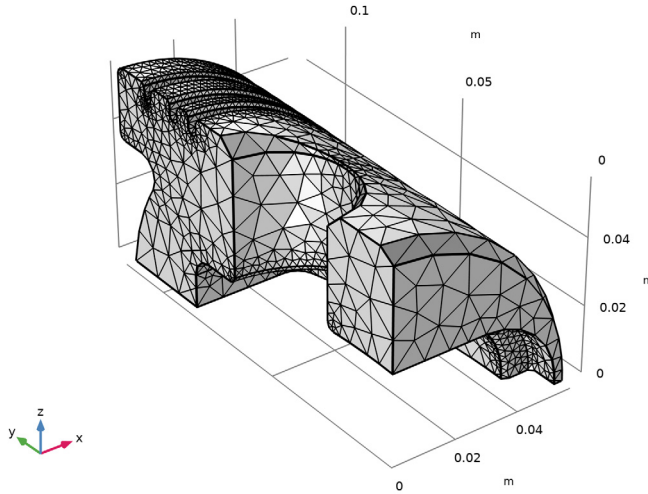
GEOMETRY 1

Import 1 (imp1)

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
piston_quarter.mphbin.

5 Click ⭲ **Import**.

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

The **Messages** window indicates that there are roughly 22,000 tetrahedral elements in
this mesh.

Assume that the current mesh does not resolve details such as fillets and chamfers
sufficiently for your simulation needs and that a finer parameter setting is required. This
would be the case for a stress analysis of the part.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

This mesh consists of approximately 322,000 elements. Many of the finer details of the geometry are adequately resolved, but there is a significant increase in the total number of elements compared to the Normal mesh setting.

In the following you will test how to tune the individual mesh parameters to refine the mesh only on selected boundaries.

MESH STATISTICS

Continue with examining the quality of the mesh.

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Statistics**.

The **Statistics** window contains details about the mesh, including the number and type of elements, and a histogram of element quality.

The default quality measure, *Skewness*, is suitable for most types of meshes, and it is based on the equiangular skew that penalizes elements with large or small angles as compared to the angles in an ideal element. This quality measure is also used when reporting low element quality during mesh generation.

The element quality has a value between 0 and 1, where 1 describes a perfectly symmetric element and 0 describes a degenerated, or completely flat, element. For 3D meshes in general, a minimum quality of about 0.1 means a satisfactory mesh. However, this depends on the type of geometry and physics application. Note also that the quality

number is calculated based on the linear elements. Use the Curved skewness quality measure for information about the quality of the curved elements.

Meshing with predefined parameter sets usually results in a mesh with quite good quality. According to the information under the section **Domain element statistics**, the present mesh has an average quality of 0.65 with a minimum quality of 0.2.

The histogram reveals the element quality distribution. In this case, the elements with low quality, represented by the tail on the left of the distribution plot, represent a very small fraction of the mesh.

Before adjusting individual mesh parameters, start by restoring the mesh to the normal size settings.

2 Right-click **Component 1 (comp1)>Mesh 1** and choose **Settings**.

3 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

4 From the **Element size** list, choose **Normal**.

5 Click ▦ **Build All**.

MESHING SEQUENCE

Size

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

You can now access and modify the default meshing sequence that appears under the **Mesh 1** node.

The first **Size** feature node in the meshing sequence is a *global attribute node*, since it influences all subsequent *operation nodes* in the meshing sequence. This first **Size** node cannot be deleted from the sequence.

Instead of editing parameters of the global **Size** node add a Size subnode to the **Free Tetrahedral 1** mesh operation.
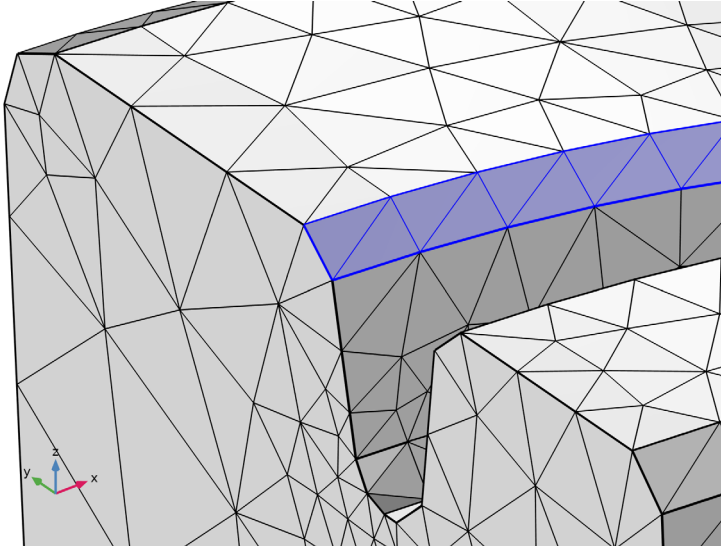
Size 1

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

The **Size 1** node is a *local attribute node* because it only applies to its parent mesh node.

RESOLUTION OF CURVATURE

1 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 Select Boundary 39 only (the boundary highlighted in the figure below).
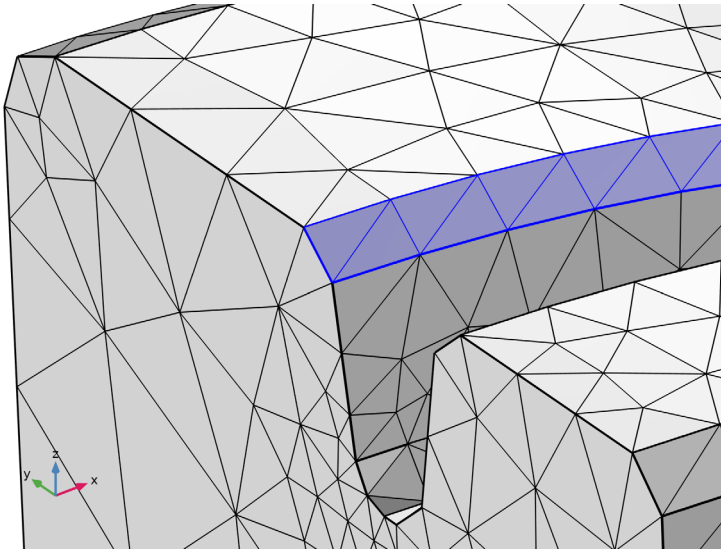

By selecting only one of the fillets you can save time generating the mesh while testing parameter values. You will be able to change the selection of the **Size 1** node to all boundaries after you have found the right set of parameters.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Curvature factor** check box. In the associated text field, type 0.2.

The **Curvature factor** parameter determines the size of boundary elements compared to the curvature of the geometric boundary. The curvature radius multiplied by the curvature factor gives the maximum allowed element size along the boundary. A lower value gives a finer mesh along curved boundaries.

7 Click ▦ **Build All**.

Zoom in on the selected boundary for a closer look at the mesh.

It seems that reducing the resolution of curvature had almost no effect on the number of mesh elements on the fillet. The reason is that another mesh parameter limits the minimum element size allowed in the mesh.
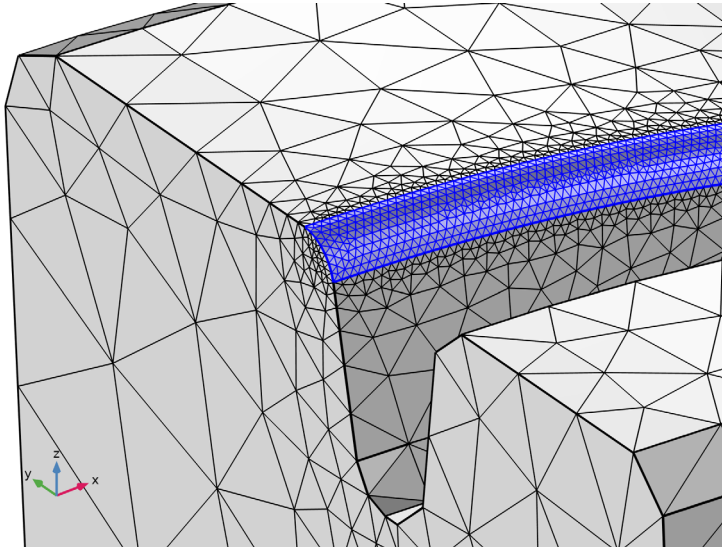
MINIMUM ELEMENT SIZE

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

2 In the **Size** settings window, locate the **Element Size Parameters** section.

3 Select the **Minimum element size** check box. In the associated text field, type 0.0002.

The value in the **Minimum element size** field specifies the minimum allowed element size. You can use this value to, for example, prevent the generation of many elements around small curved parts of the geometry.
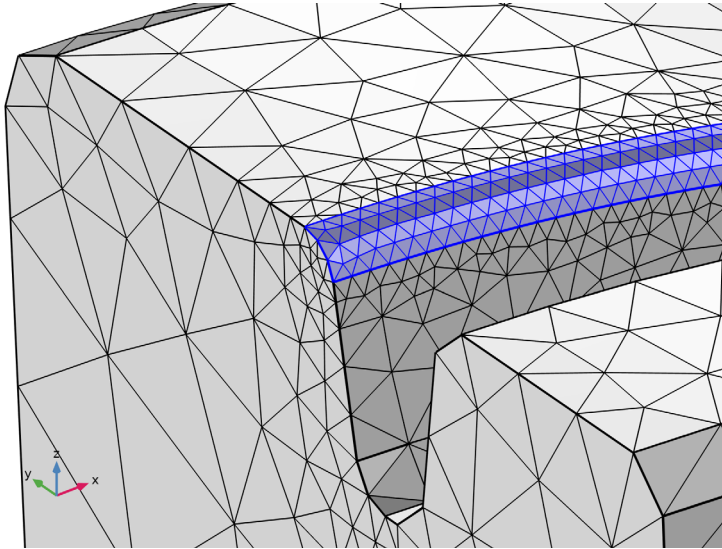
4 Click ⊞ **Build All**.

This time the selected boundary has a much finer mesh. Adjust the mesh again by increasing the resolution of curvature.

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

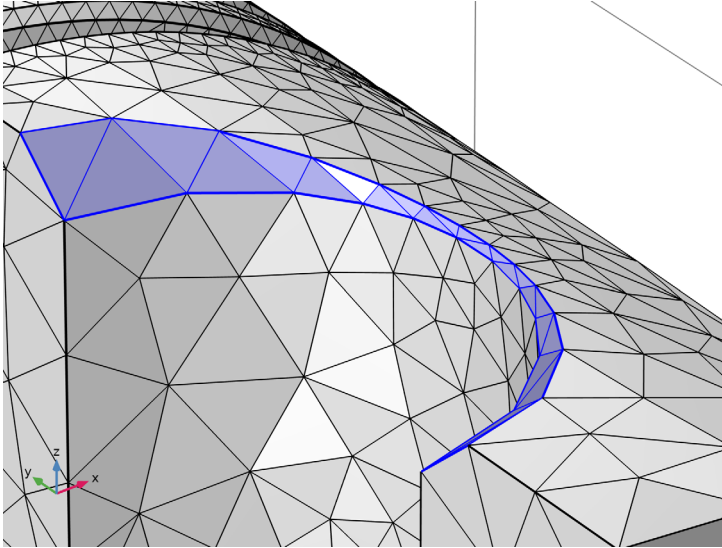**6** In the **Curvature factor** text field, type 0.45.

7 Click ⊞ **Build All**.

Now assume that you also want a better resolution of narrow regions with no curvature
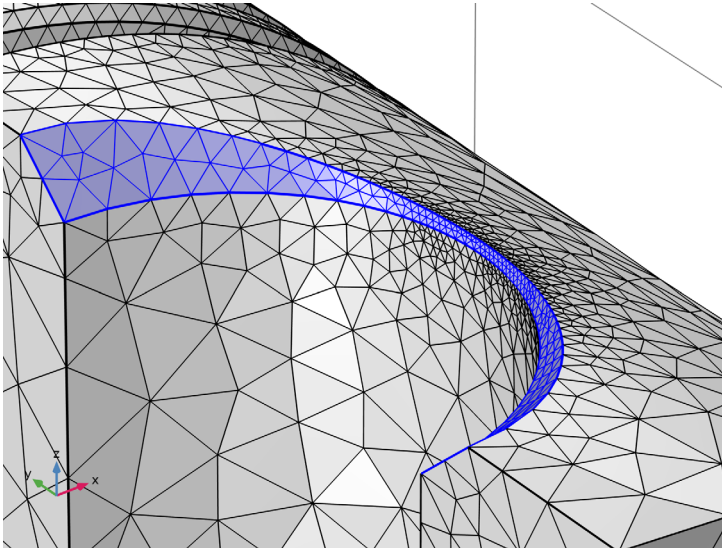such as chamfers.

RESOLUTION OF NARROW REGIONS

1 Add the face highlighted below to the selection. The selection should now contain both boundaries 8 and 39.

2 Select the **Resolution of narrow regions** check box. In the associated text field, type 2.

The **Resolution of narrow regions** mesh parameter controls the number of element layers that are created in narrow regions (approximately). If the value of this parameter is less than one, the mesh generator might create elements that are anisotropic in size in narrow regions.
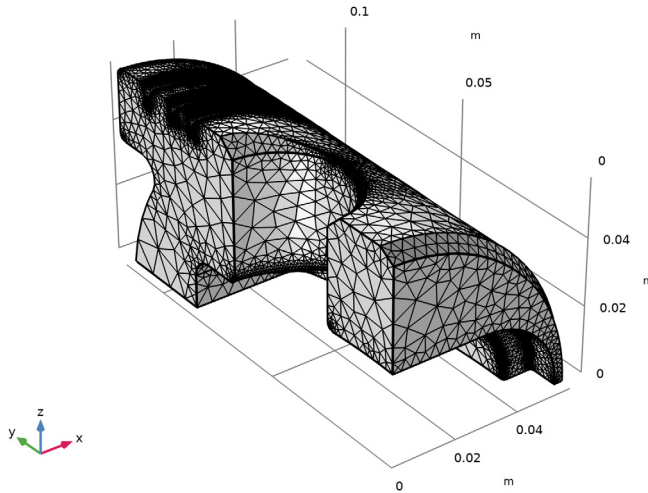
3 Click ▦ **Build All**.

Assume that you are happy with the parameter settings for curved and narrow regions. Now apply these for all boundaries of the geometry.

4 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **All boundaries**.

5 Click ▦ **Build All**.

6 Click the ⤺ **Go to Default View** button in the **Graphics** toolbar to get the view in the figure below.

The fine details of the geometry are resolved satisfactorily with this mesh of approximately 323,000 tetrahedral elements.

Continue with checking the mesh quality.

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

Compared to the mesh with the **Finer** predefined mesh parameter set, the average and minimum element quality are slightly lower. This is expected because the boundaries are finely meshed and the elements are growing toward the inner parts of the geometry according to the **Normal** parameter set specified in the global **Size** node. Allowing even higher element growth will reduce the number of elements further and will result in even lower element quality.

MAXIMUM ELEMENT GROWTH

Reduce the number of mesh elements further by specifying the rate of growth from the small elements on the surface to the larger elements inside.
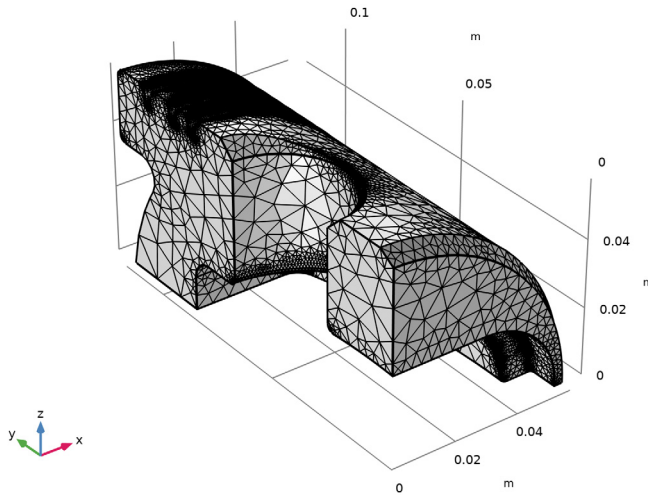
Size

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 growth rate** text field, type 1.8.

The **Maximum element growth rate** parameter determines the maximum rate at which the element size can grow from a region with small elements to a region with larger elements.

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

The mesh now consists of approximately 175,000 elements while keeping the fine mesh on curved and narrow boundaries.

6 Right-click **Component 1 (comp1)>Mesh 1** and choose **Statistics**.

As expected the increase of the growth rate parameter results in an even lower quality. The average quality is now 0.61, and the minimum quality is 0.17, which are both acceptable numbers.
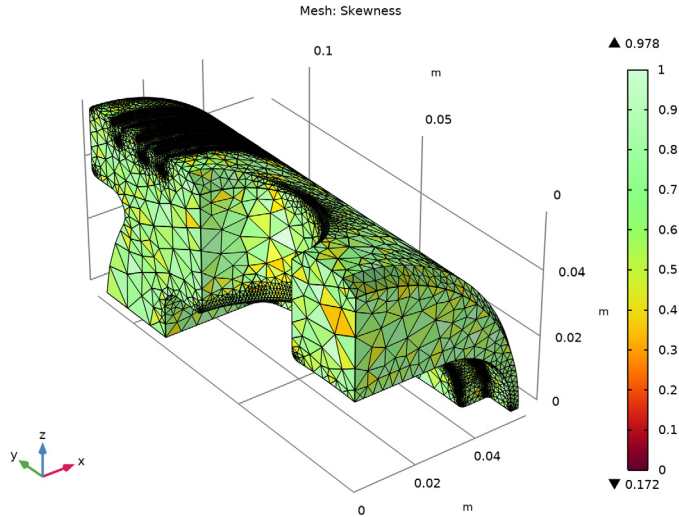
A mesh plot can help with localizing the worst quality elements.
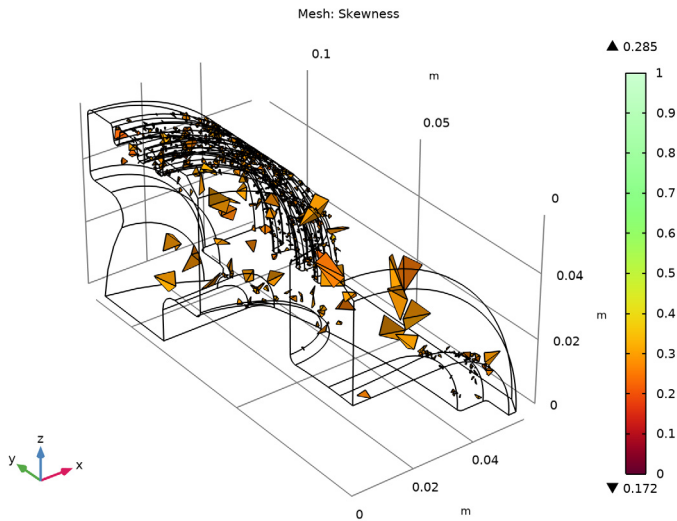
7 In the **Mesh** toolbar, click ▲ **Plot**.

Mesh 1

The **Mesh 1** plot is added to the **3D Plot Group 1** under the **Results** section of the **Model Builder** window. The default mesh plot that appears in the **Graphics** window contains the volume elements colored according to quality.

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

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

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

4 In the **Fraction** text field, type 0.005.

5 In the **Mesh Plot I** toolbar, click ▣ **Plot**.

Mesh: Skewness

You can now see 0.5% of the tetrahedral elements with the worst quality. These are mostly located in the regions where the elements are growing from the surfaces toward the inside of the geometry.