Boundary Layer Meshing — Adding Boundary Layers to an Imported Mesh

Introduction

A boundary layer mesh is a type of structured anisotropic mesh where mesh elements have high aspect ratios. For example, in CFD applications close to no-slip wall boundaries, the flow is highly anisotropic, and a much finer mesh is needed to resolve high gradients.

With physics-controlled meshing, a boundary-layer mesh is automatically generated for fluid flow applications, and can easily be generated for other applications. In all cases, the boundary-layer mesh is inserted after the interior mesh, which means the interior mesh is deformed to accommodate the boundary-layer mesh. This tutorial series focuses on the generation and setup of boundary-layer meshes. Throughout the series, we will also explore key aspects of boundary-layer meshes, such as the height of the boundary layers, the growth rate of elements from the boundary to the interior, and the various techniques for corner handling.

This tutorial focuses on creating a boundary-layer mesh for an imported mesh. We will also explore how to apply the Refine operation to increase the number of elements in the imported mesh without generating a new mesh. You will learn how to configure the number and distribution of elements in the boundary layer and how to control the deformation of the interior mesh.

The other tutorial in the series, Boundary Layer Meshing — Exploring the Settings, details how to modify the settings for a boundary-layer mesh generated by a physics-controlled meshing sequence. You will discover how to control the number of layers, define the layer thicknesses, and create a boundary-layer mesh with different properties for different boundaries. The tutorial also demonstrates the available methods for adding boundary layers around corners and the setup of mesh plots for assessing the element quality.

Model Definition

MODEL GEOMETRY AND IMPORTED MESH

The imported mesh in Figure 1 represents the 2D cross-section of a ball valve in a pipe geometry. The valve is rotated by 20 degrees to a partially open position.
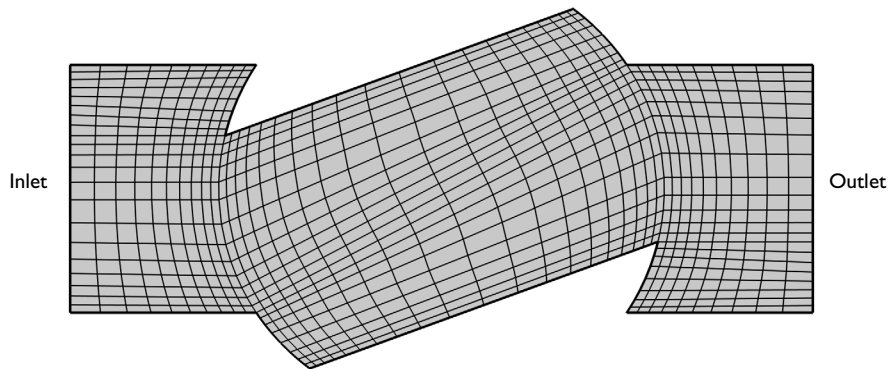

Figure 1: Model geometry showing the imported structured mesh and the boundary conditions.

The structured mesh used as a starting point in this tutorial is imported from a COMSOL native MPHBIN-file

Figure 1 also shows the inlet and outlet boundaries for the valve geometry. All other boundaries are assumed to be wall boundaries for which a boundary-layer mesh will be inserted.

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/` `valve_boundary_layers_import`

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 🔵 **2D**.
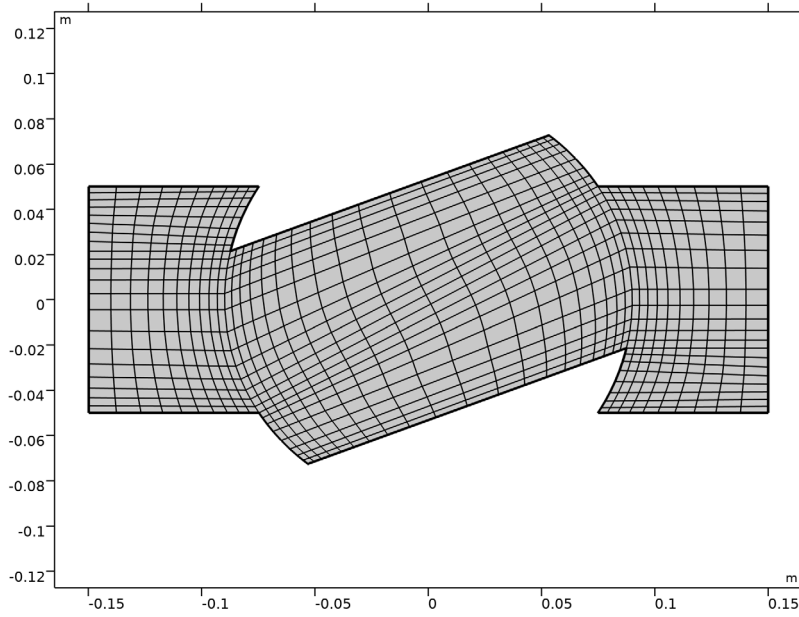
2 Click ☑ **Done**.

MESH 1

Import 1

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

Instead of creating or importing a geometry, we are here importing a mesh, so the software assumes that the **Mesh 1** sequence will define the geometric model on which the physics will be set up. This is indicated by the frame around the icon for the **Mesh 1** node, which is now also decorated with a compass similar to that of the icon of the **Geometry 1** node. Meshing sequences that define the geometric model are always user-controlled, which means that physics-controlled meshing is not available, and you need to create a mesh suitable for the simulation.

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 valve_mesh.mphbin.
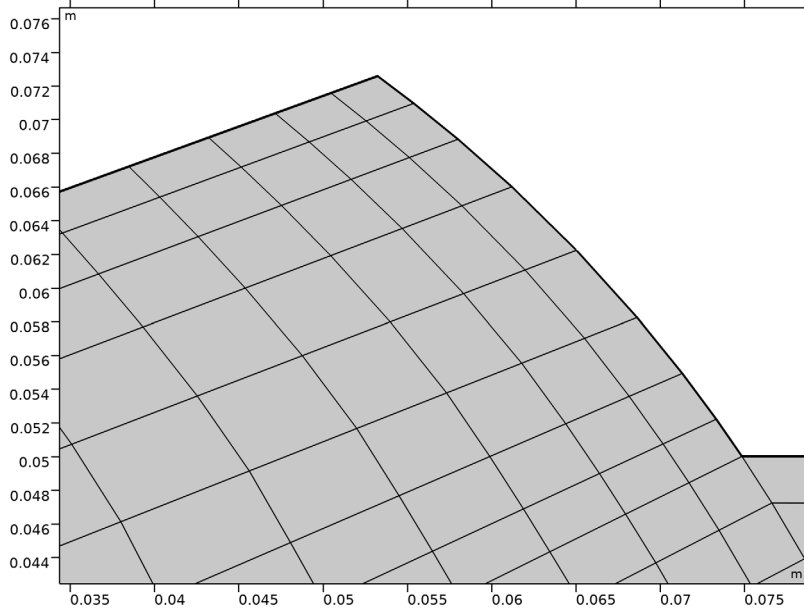
5 Click ⊞ **Import**.

After the mesh is imported, we can see the list of imported selections in the **Boundary Selections** subsection. You can click on the selections' names to see them highlighted in

the **Graphics** window. In this case, we have a selection for the wall boundaries that we will use later while creating the mesh.

We have imported a structured mesh consisting of quad elements. It was built in such a way that there are more elements closer to the wall boundaries, and larger mesh elements are placed in the interior of the domain. In the following, we will modify this mesh by refining the elements, followed by adding a boundary layer mesh to better resolve the region close to the wall boundaries.

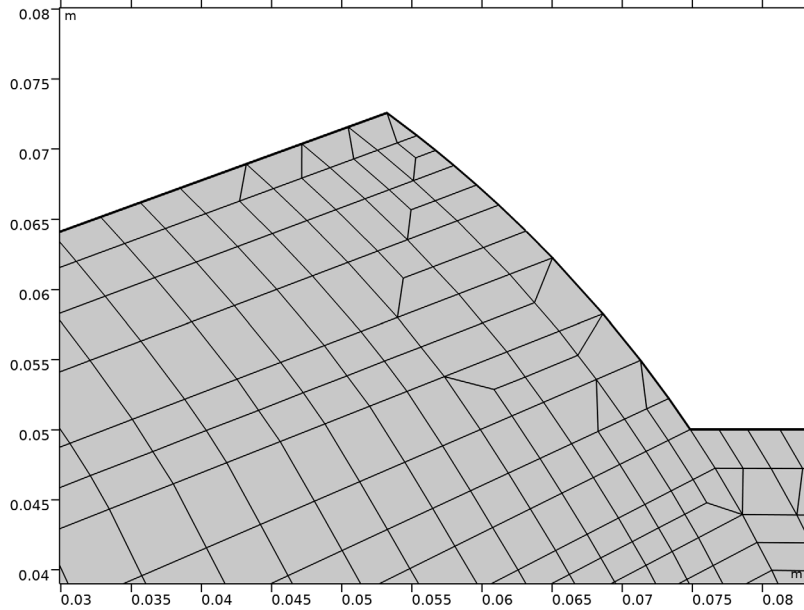**6** Zoom in around the valve to inspect the mesh.

Assume that we would like to increase the number of elements in the imported mesh without generating a new mesh. We can easily do this using the Refine operation.

Refine 1

The refine operation splits elements a specified number of times by inserting new mesh vertices and edges in the existing elements. It places new and moved mesh vertices on a higher-order representation of the faces and edges, which means that a refined mesh will resolve curved boundaries.

1 In the **Mesh** toolbar, click ![icon] **Modify** and choose **Refine**.

2 In the **Settings** window for **Refine**, locate the **Refine Options** section.

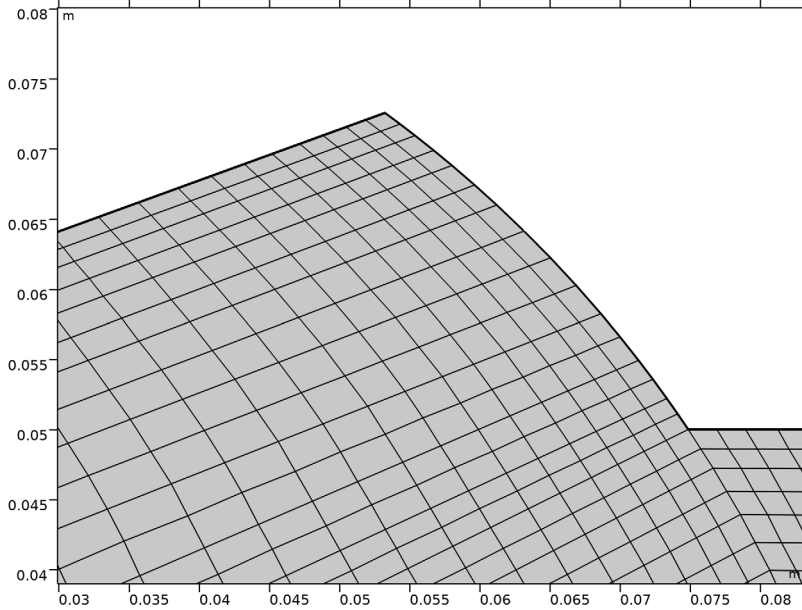**3** From the **Refinement method** list, choose **Split longest side**.

4 Click ⬛ **Build Selected**.

The longest side refinement method bisects the longest edge of each element. For a regular quad element, as some elements are in the current mesh, the result is a triangle and a somewhat skewed quad element. Thus, this method is generally unsuitable for nonsimplex meshes because the refinement destroys the structured nature of such meshes.

5 From the **Refinement method** list, choose **Regular refinement**.

6 Click ⬛ **Build Selected**.

The regular refinement method bisects each edge of an element. This results in 2^{dim} new elements of the same type, where dim is the element dimension. In this case, each two dimensional element is split into 4 elements.

You can also see that the refined mesh preserves the curvature of the boundary.

Boundary Layers 1

We will now continue with inserting a boundary layer mesh. Boundary layer meshes are important in some applications, for example, along no-slip boundaries when solving fluid flow problems. Since the physics-controlled mesh set-up is not applicable for imported meshes, we will manually configure the boundary layer mesh generation.

In the **Mesh** toolbar, click ▨ **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

We use the Boundary layer properties attribute node to set up the thickness of the boundary layer, as well as the number of layers and their distribution.
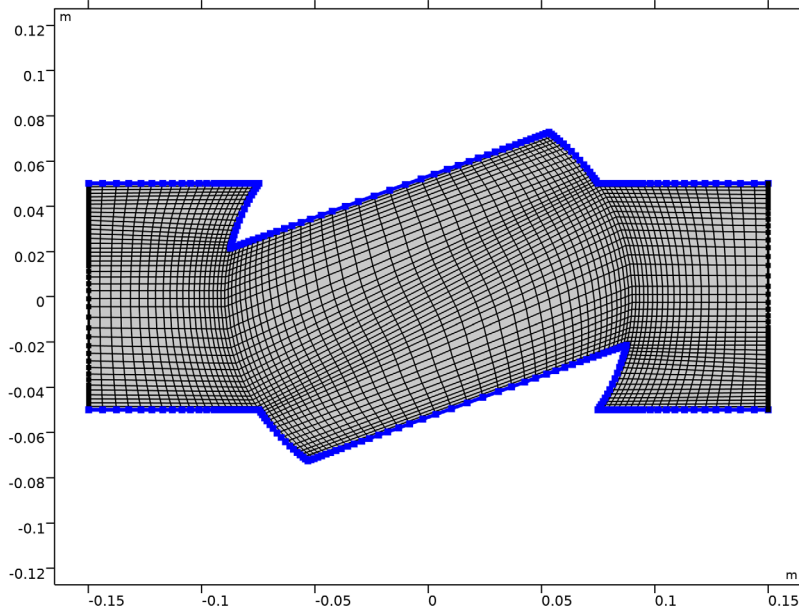
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

When you add a new Boundary Layer node, the Selection list in the Boundary Layer section is empty, and you need to select the boundaries where the boundary layer mesh will be inserted.

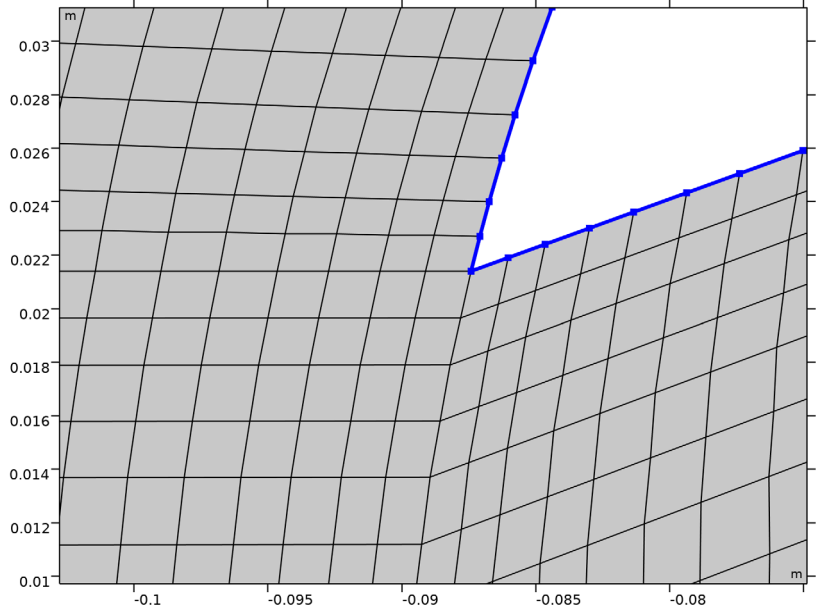**3** From the **Selection** list, choose **Wall Boundaries**.

By using the named selection imported from the mesh file, you can ensure that the selected boundaries are retained if the mesh file is re-imported. If the new mesh file also contains a selection with the same name, the settings will automatically apply to the boundaries in the selection.

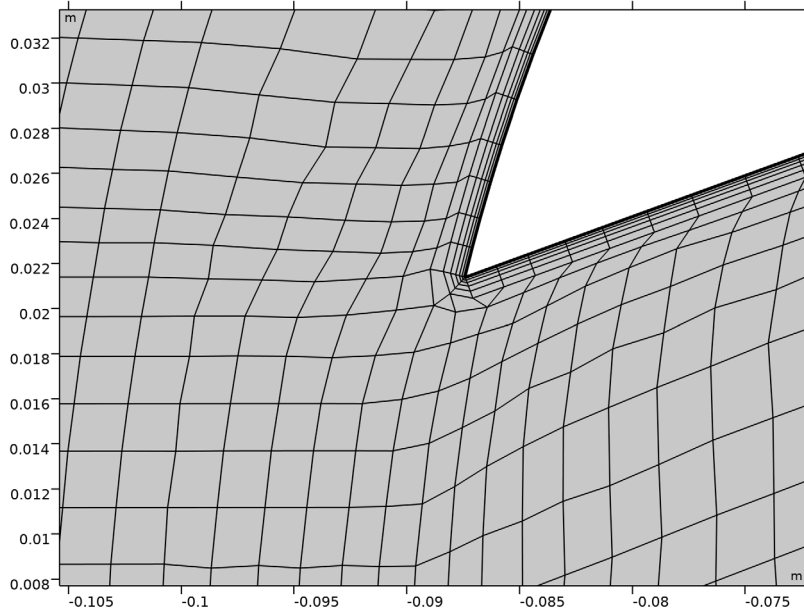**4** Click the ⊹ **Zoom Extents** button in the **Graphics** toolbar.

5 Locate the **Layers** section. In the **Number of layers** text field, type 6 to specify the number of layers in the boundary layers mesh.

6 In the **Stretching factor** text field, type 1.4 to specify the increase in thickness between two consecutive boundary layers.

7 From the **Thickness specification** list, choose **All layers**.

8 In the **Total thickness** text field, type 1[mm] to precisely control the thickness of the boundary layers mesh.

9 Zoom around the sharp corner near the valve entrance.
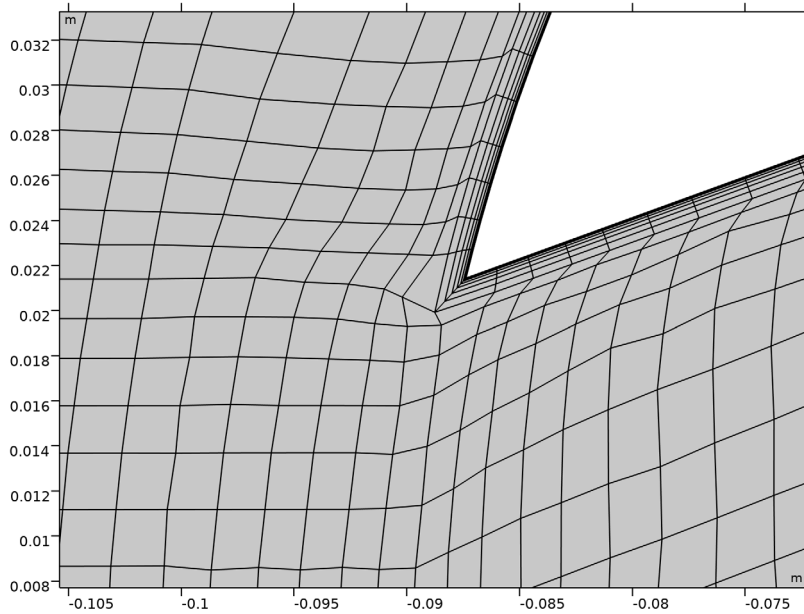
10 Click ⊞ **Build All**.

Six layers have been added at the selected wall boundaries, where the thickness of the layers increases from the first to the last layer with the specified stretching factor. Notice that the elements around the corner are split, which is the default method for handling sharp corners. While this method provides excellent resolution around the corner, it also introduces triangular elements into the otherwise quad-only mesh. To obtain a quad-only structured mesh, we can use the No special handling method for sharp corners.

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.

2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.

3 From the **Handling of sharp corners** list, choose **No special handling**.

With this option, the corners with large angles will not be treated differently from the surrounding boundaries, meaning that the boundary layer mesh will be wrapped around the sharp corners.
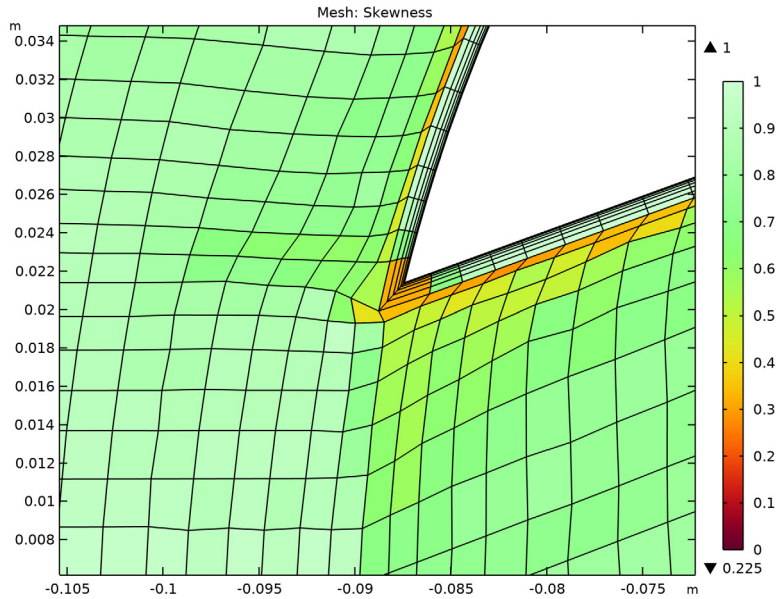
4 Click **Build All**.

Notice that by inserting the boundary layer mesh, the quad mesh is deformed, and the deformed elements propagate from the boundaries into the interior of the domain. This is because, by default, the transition in element size between the boundary layer mesh and the interior mesh is smoothed. We can inspect the element quality by creating a Mesh plot.

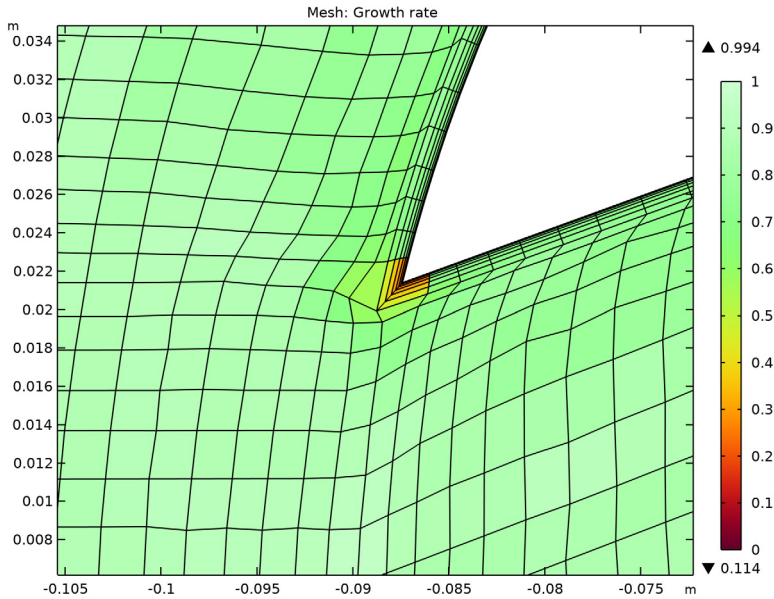**5** In the **Mesh** toolbar, click **Plot**.

Mesh 1

You should see a plot similar to the one above, showing mesh element quality based on skewness. The majority of elements are of high quality, except for some elements around the sharp corner that have somewhat lower but still acceptable quality. Another way to consider the element quality is by using the growth rate of elements as a measure. This is particularly important for CFD applications, where we usually want to ensure that the difference in the size of neighboring elements is not too large. A plot of the growth rate quality measure helps with finding suitable values for the stretching factor that controls the thickness of the boundary layers, as well as the parameters controlling the transition from the boundary layer to the interior mesh.

1 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

2 From the **Quality measure** list, choose **Growth rate**.

3 In the **Mesh Plot 1** toolbar, click ▣ **Plot**.

Mesh: Growth rate

Here, most of the mesh elements have very high quality. The growth rate between the boundary layers and the interior mesh is low (high quality in the plot), since the default transition settings ensure a smooth size transition. A side effect of this smoothing is that some quad elements in the interior mesh can become slightly distorted. In case this becomes important for your application, you can control how deep into the interior mesh the deformation is propagated, or even turn off this option.
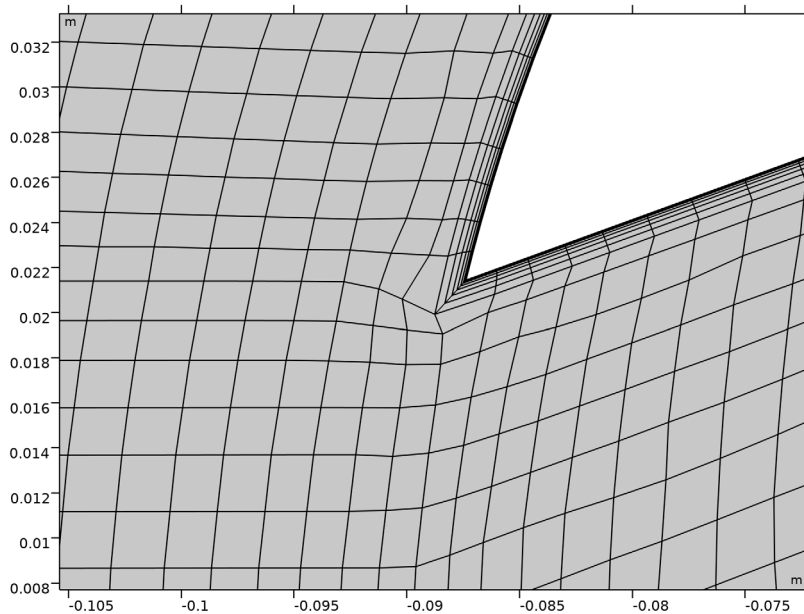
MESH 1

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

Here you can modify the settings for smoothing the transition in element size from the boundary layer mesh to the interior mesh. Increasing the number of smoothing iterations can yield better results, but it will also increase the time it takes to generate the boundary layer mesh. In the Maximum element depth to process field, you can specify the maximum element depth from the boundary layer interface for the mesh points to be smoothed. We can try to reduce the default value.

3 In the **Maximum element depth to process** text field, type 2.

4 Click ⊞ **Build All**.

Now only the first two layers of quad elements next to the boundary layer mesh are deformed by the size smoothing.
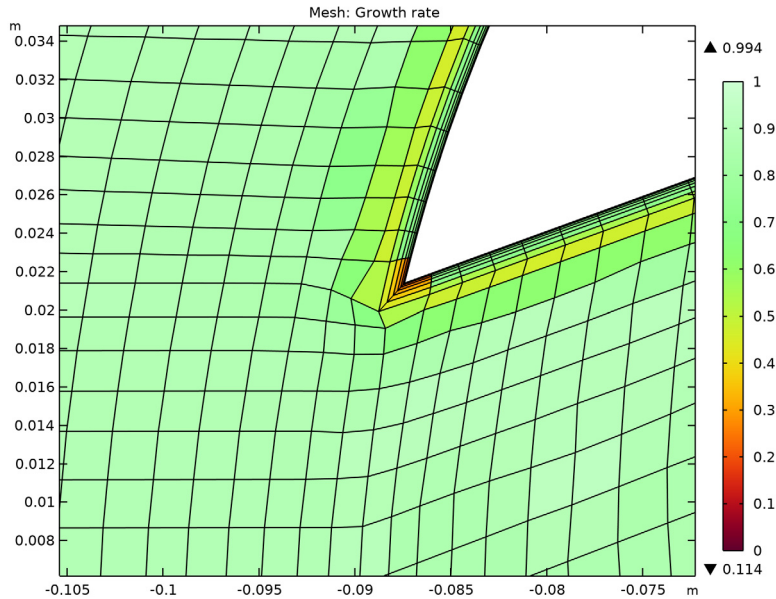
If skewness becomes a problem in your structured mesh when smoothing the transition in element size, you may want to turn this option off, and instead generate the structured mesh in a way that helps with a good size transition.

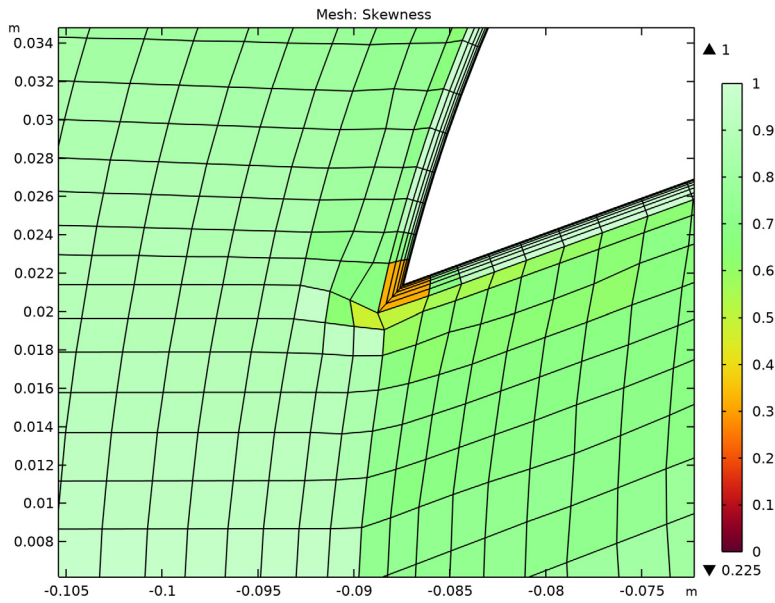We can take a look at the mesh quality after this latest change.

Mesh 1

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

Reducing the depth of the smoothing had only a minor effect on the growth rate in this case. This is because the structured mesh was already generated such that the elements were thinner and closer to the wall boundaries.

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Quality measure** list, choose **Skewness**.

4 In the **Mesh Plot 1** toolbar, click [icon] **Plot**.

Mesh: Skewness

Looking at the skewness plot, we can see some improvement since the smoothing of the element size deforms fewer quad elements compared to the previous setting.