CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys
2021 R2
Engineering What's Ahead.

Ansys Meshing Tutorial Guide

Ansys

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

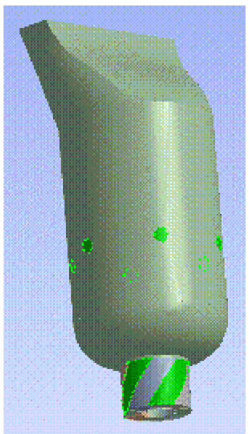
Release 2021 R2
July 2021

Published in the U.S.A.

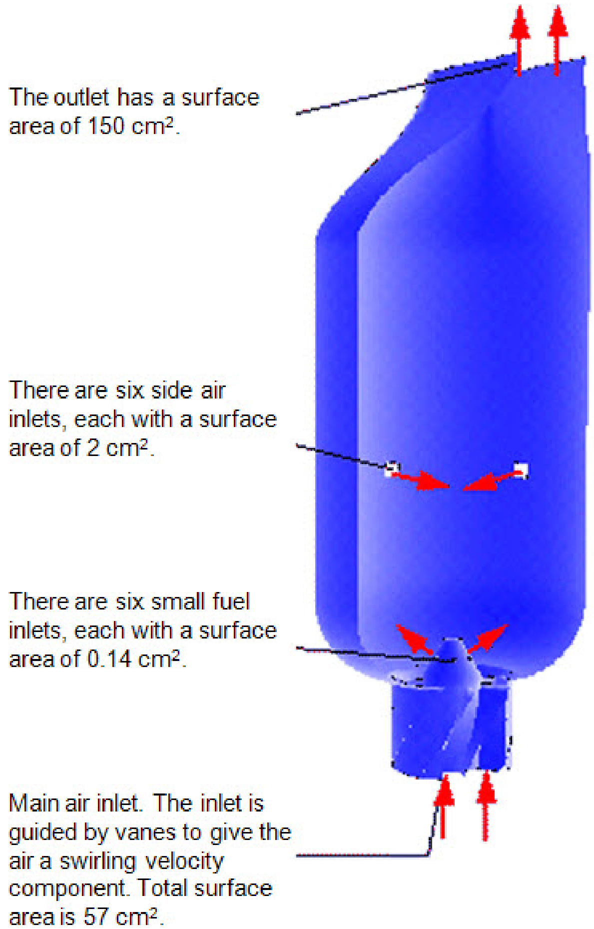
Table of Contents

Can Combustor

This tutorial creates a mesh for a can combustor, which can be found in gas turbine engines. The geometry is complex and consists of five separate solid bodies. It will be imported as a complete geometry from a Parasolid file.



The diagram below shows the geometry schematically with part of the outer wall cut away.



The following geometry and meshing features are illustrated:

- Parasolid import
- Multibody part formation
- Named Selection creation
- Program Controlled inflation

Preparation

This tutorial requires you to have a copy of the Parasolid file `Combustor.x_t`.

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Download the `cancombustor.zip` file here.
3. Unzip the `cancombustor.zip` file you have downloaded to your working folder.

You can proceed to Tutorial Setup (p. 3).

Tutorial Setup

The following sections describe the steps for creating the project and importing the geometry:

Creating the Project

Importing the Geometry

Creating the Project

1. Open Ansys Workbench and add a standalone Mesh system to the Project Schematic. Save the project as `Combustor.wbpj`.

2. Now add geometry to the project.

   On the Project Schematic, right-click the Geometry cell in the Mesh system and select **New DesignModeler Geometry...** to open the DesignModeler application.

3. In the main menu of the DesignModeler application, choose **Units>Centimeter**.

Importing the Geometry

The geometry is imported complete, from a Parasolid file.

1. Import the geometry.

   a. Select **File > Import External Geometry File...** from the main menu.

   b. In the file browser that opens, locate and open the file `Combustor.x_t`.

   c. Click **Generate** to import the combustor.

2. Create a multibody part.

   The Tree Outline should now show that you have **5 Parts, 5 Bodies**. To produce a single mesh that contains all of the bodies rather than one mesh per body, the parts must be combined into a multibody part.

   a. On the toolbar at the top of the window, click **Selection Filter: Bodies** . This means that you can select only solid bodies in the next operation, which helps to make the selection process easier.

   b. Click **Select Mode** and select **Box Select** from the drop-down menu.

   c. In the **Geometry** window, select all five bodies by holding down the left mouse button and dragging a box from left to right across the whole geometry to select all five bodies. To be selected, all of the entities must lie completely within the box that you have drawn. When you release the mouse button, the status bar located along the bottom of the window should change to show that **5 Bodies** are selected.

      When using **Box Select**, the direction that you drag the mouse from the starting point determines which items are selected. Dragging to the right to form the box selects entities that are completely enclosed by the box, while dragging to the left to form the box selects all entities that intersect, or touch, the box.

d. Right-click in the **Geometry** window and select **Form New Part**.

The Tree Outline should now show that you have **1 Part, 5 Bodies**.

The geometry does not need further modifications. It is now complete. From the DesignModeler application's main menu, select **File > Save Project** to save the project and then **File > Close DesignModeler** to return to the Project Schematic. Notice the Geometry cell appears in an up-to-date state ✓.

Now that the geometry is complete, you can proceed to Generating the Mesh (p. 4).

Generating the Mesh

The following sections describe the steps for generating the mesh:

Launching the Meshing Application

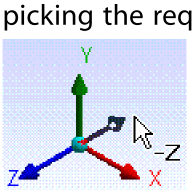
On the Project Schematic, right-click the Mesh cell in the Mesh system and select **Edit...** to launch the Meshing application.

Creating Named Selections

You will create five Named Selections in this tutorial. Detailed instructions are provided for creating the first Named Selection. Less detailed instructions are provided for creating the subsequent Named Selections, but you should create them in a similar fashion, using additional zoom and/or rotation options from the toolbar as needed.

1. To create a Named Selection for the fuel inlet, select the six tiny faces on the cone near the bottom of the combustor. The easiest way to select them is as follows:

   a. Click over the axes in the bottom right corner of the **Geometry** window in the position shown in the figure below. As you move the cursor into this position, the black "-Z"-axis will appear (it is not shown by default). This will put the geometry into a good position for picking the required faces.

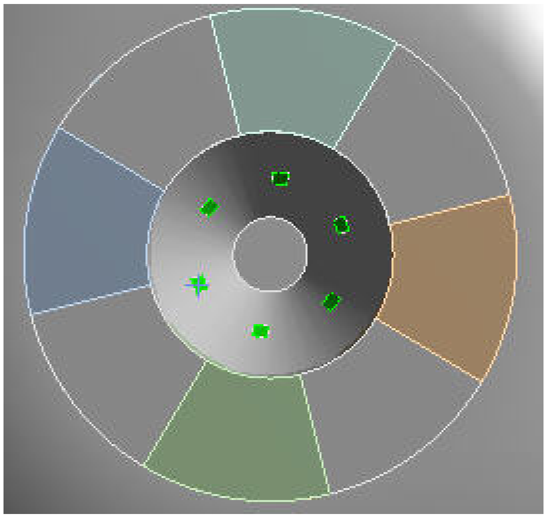
   

   b. On the toolbar, click **Box Zoom** .

c. In the **Geometry** window, zoom the geometry by holding down the left mouse button and dragging a box across the area where the six tiny faces are located. Then release the mouse button.

d. On the toolbar, click **Face** .

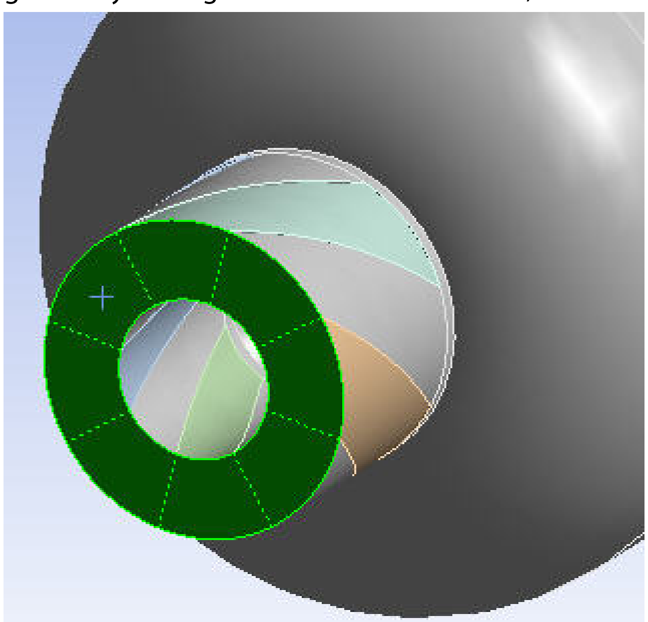
e. Press and hold the **Ctrl** key while picking the six faces, which are shown in green in the figure below (the colors in your geometry may differ from those shown in this tutorial).



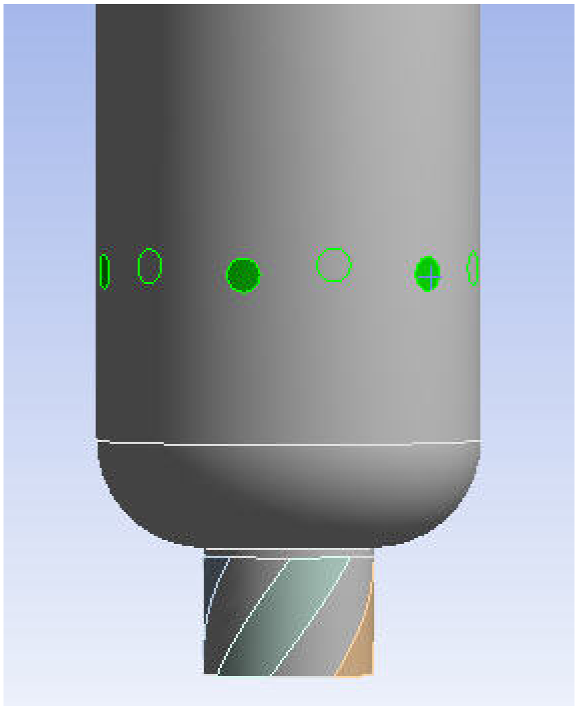
f. After selecting all six faces, release the **Ctrl** key. Right-click in the **Geometry** window and select **Create Named Selection** from the menu.

g. In the **Selection Name** dialog box, type **fuel_inlet** and click **OK**.

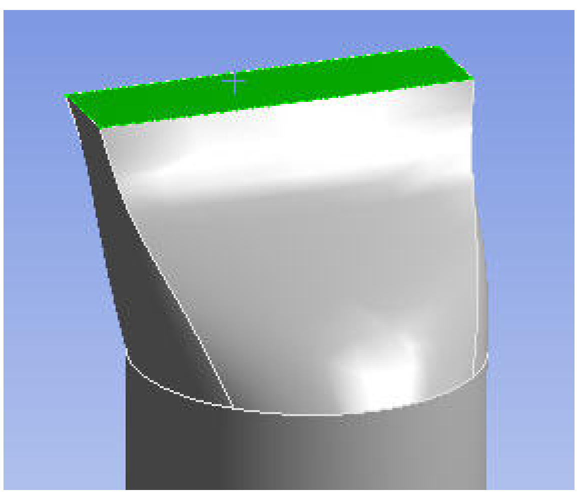
2. To create a Named Selection for the air inlet, select the eight faces at the very bottom of the geometry having the lowest Z-coordinate, as shown below. Name this Named Selection **air_inlet**.



3. To create a Named Selection for the secondary air inlet, select the six small circular faces on the main body of the combustor, as shown below. These introduce extra air to aid combustion. Name this Named Selection **secondary_air_inlet**.



4. To create a Named Selection for the outlet, select the rectangular face with the highest Z-coordinate. Name this Named Selection **outlet**.

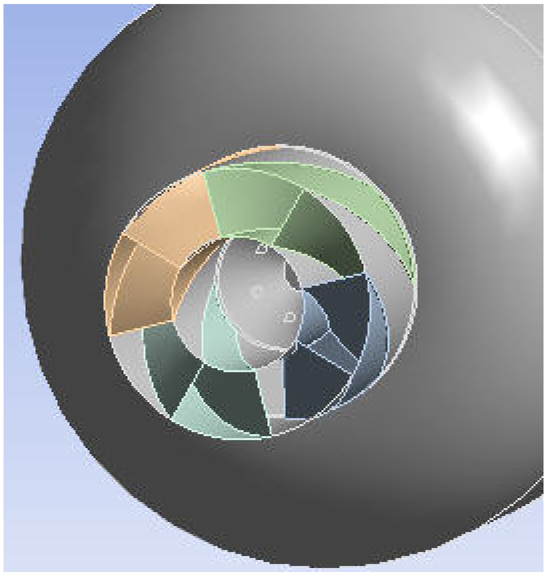


5. Create the last Named Selection, **internal**.

   The faces that you need to select for this Named Selection are not easily seen. The next several steps help to make the selection process easier.

   a. In the Tree Outline, click the Named Selection called **air_inlet**. In the Details View, change the value of **Visible** to **No**.

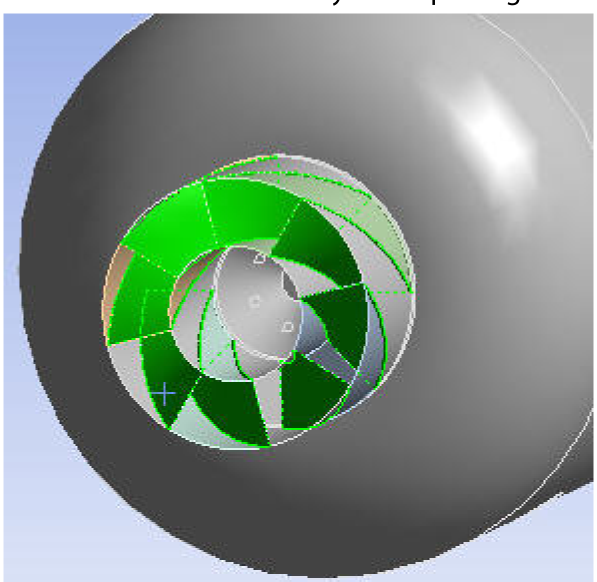
      Look into the combustor inlet. You should see eight curved vanes surrounding the fuel inlet, as shown below. Rotate the view slightly and note that every other vane passage is blocked by faces.



b. From the main menu, select **Tools> Options**. In the left pane of the **Options** dialog box, click the plus sign to expand the Mechanical options. Highlight **Graphics**, and then in the right pane, set **Highlight Selection** to **Both Sides** and click **OK**.

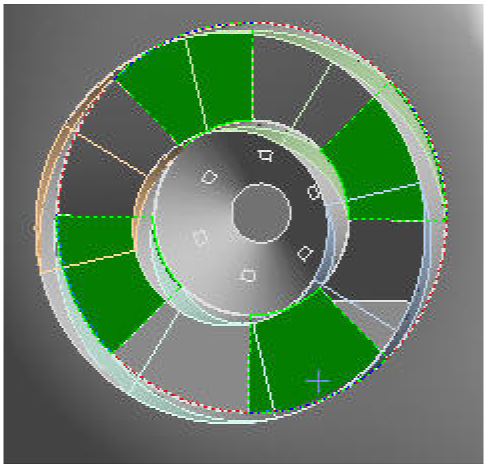
c. On the toolbar, click **Face** .

d. Press and hold the **Ctrl** key while picking the eight faces of the vanes, as shown below.



e. After selecting all eight faces, release the **Ctrl** key. Right-click in the **Geometry** window and select **Hide Face(s)** from the menu.

f. To create the last Named Selection, select the four faces that block the vane passages, as shown below. Name this Named Selection **internal**.



Note:

You are done creating Named Selections. The next step toggles visibility of all faces back on.

6. Restore the visibility of all faces.

   a. In the Tree Outline, click the Named Selection called **air_inlet**. In the Details View, change the value of **Visible** to **Yes**.

   b. Right-click in the **Geometry** window and select **Show Hidden Face(s)** from the menu.

Setting Up the Mesh

This is a complex geometry which will be used to run a simulation with complex physics. To keep the computational time down for the purposes of the tutorial, the default sizing settings will be retained and a very coarse mesh will be generated. If you wanted to get accurate results for the geometry, a much finer mesh and a much longer solution time would be required.

1. In the Tree Outline, click the **Mesh** object.

2. In the Details View, set **Physics Preference** to **CFD** and **Solver Preference** to **CFX**.

3. In the Details View, click to expand the **Sizing** group of controls and notice the default sizing settings.

Setting Up Inflation

It is a good idea to put inflation on the walls.

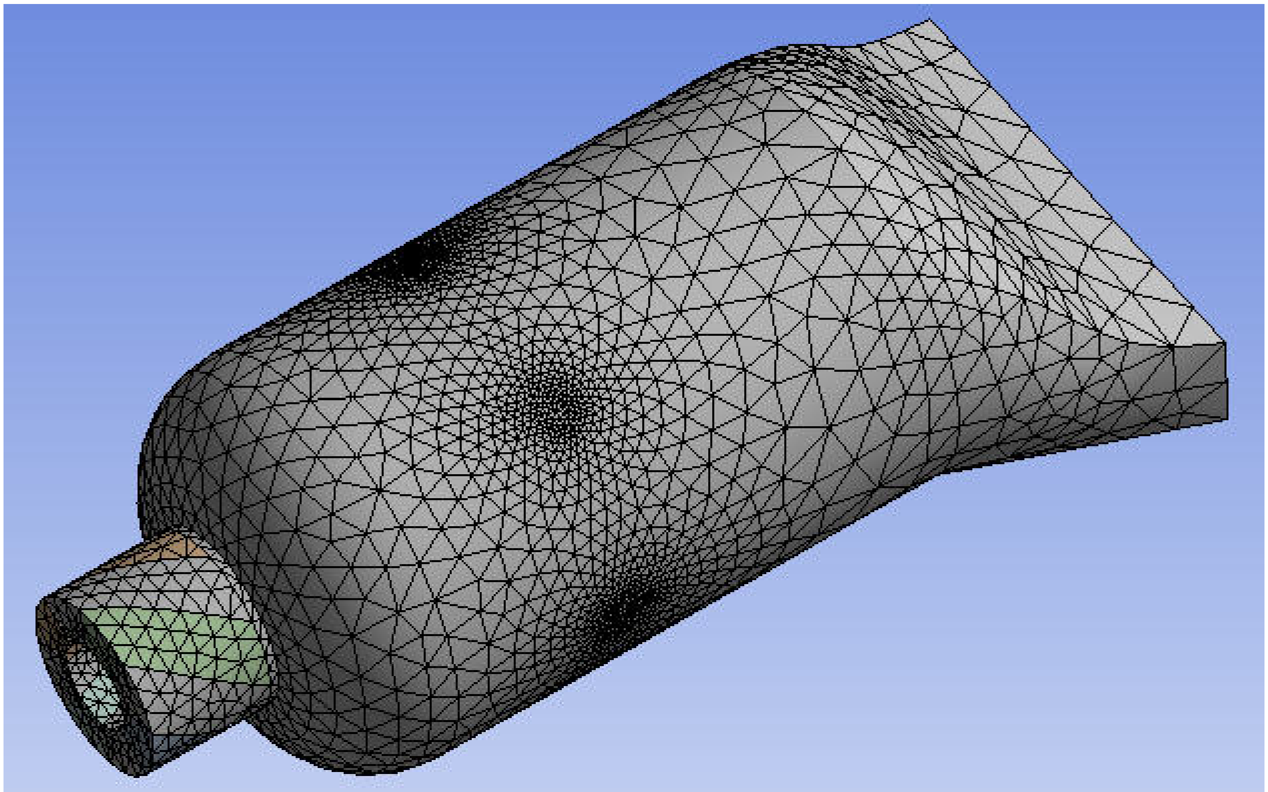
1. In the Details View, click to expand the **Inflation** group of controls.

2. Set **Use Automatic Inflation** to **Program Controlled**.

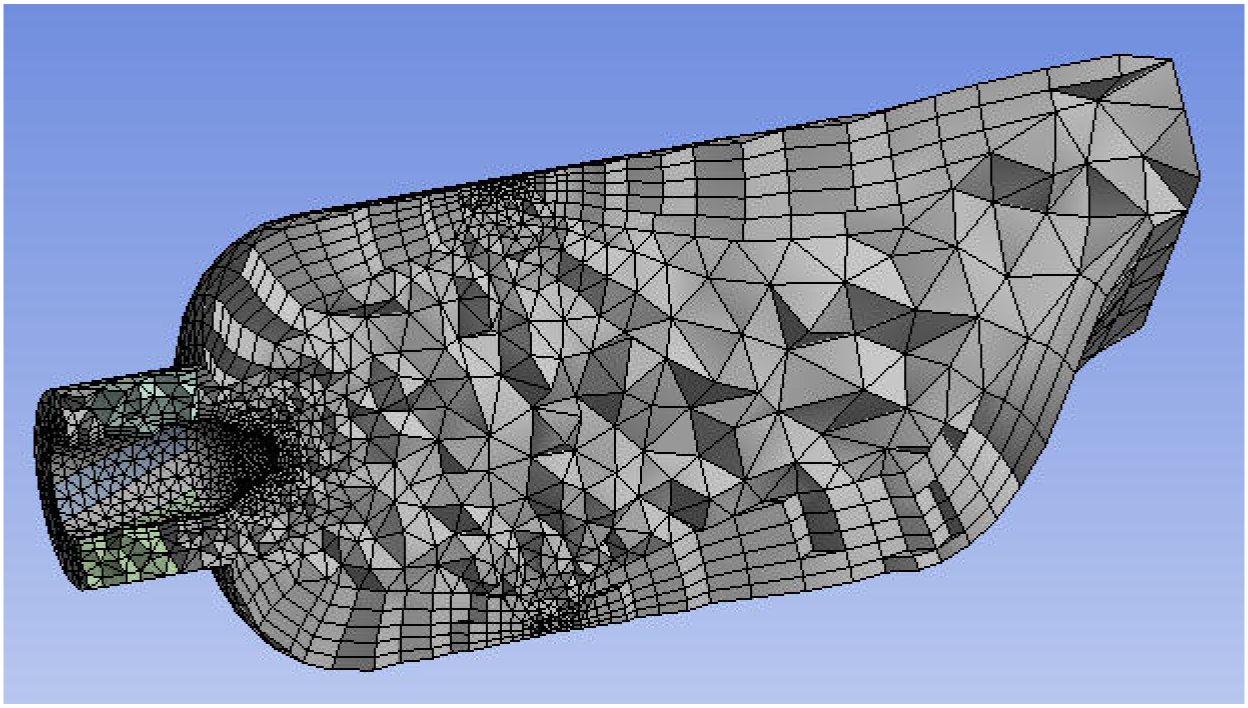
As a result of this setting, all faces in the model are selected to be inflation boundaries, with a few exceptions. For the purposes of this tutorial, the important exception is Named Selections—the faces in Named Selections will not be selected to be inflation boundaries.

Generating the Mesh

1. Generate the mesh by right-clicking **Mesh** in the Tree Outline and selecting **Generate Mesh**. After a few moments, the meshed model appears in the **Geometry** window, as shown below.



2. Activate a section plane to view a section cut through the model. Select **Show Whole Elements**  to view the mesh as shown.



This completes the mesh generation. Note that you may have received a warning about a problem with inflation layer generation. This warning is common when using an automated inflation setup with coarse mesh as the inflation layers do not have adequate room for orthogonal inflation layer growth. This warning(s) can generally be ignored unless you are very concerned with near wall physics. Should this be the case, more selective inflation and/or the use of local size functions should resolve the issue.

From the Meshing application's main menu, select **File > Save Project** to save the project and then **File > Close Meshing** to return to the Project Schematic.

You can exit Ansys Workbench by selecting **File > Exit** from the main menu.

Single Body Inflation

This tutorial demonstrates various ways to apply single body inflation. The 3D inflation capability provided by the Meshing application is mainly used in CFD/Fluids meshing. It provides high quality mesh generation close to wall boundaries to resolve changes in physical properties.

Essentially, there are two methods for applying inflation: globally, using Named Selections; and locally, by scoping an inflation method. This tutorial covers using these methods along with various other settings for defining inflation on a single body.

The following topics are covered:

- Comparing two **Collision Avoidance** settings (**Layer Compression** and **Stair Stepping**), which determine the approach that is to be taken in areas of proximity
- Previewing inflation, which can be used to examine proximity handling, determine the quality of inflation layers, and detect potential quality issues
- Creating a new Named Selection, and automatically applying inflation to all the faces in it
- Scoping inflation to a body and selecting a Named Selection as the inflation boundary
- Comparing three **Inflation** settings (**Smooth Transition**, **Total Thickness**, and **Last Aspect Ratio**), which determine the heights of the inflation layers
- Changing **Solver Preference** and how its value affects default inflation behaviors

Preparation

This tutorial requires you to have a copy of the Ansys Workbench project file `newquart.wbpj` and the project folder `newquart_files` and its contents.

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Download the `newquart.zip` file here.
3. Unzip the `newquart.zip` file you have downloaded to your working folder.

You can proceed to Tutorial Setup (p. 11).

Tutorial Setup

1. Open Ansys Workbench.

2. Select **File** > **Open...** from the main menu.

3. In the file browser that opens, locate and open the file `newquart.wbpj`.

Now that the tutorial is set up, you can proceed to Generating the Mesh (p. 12).

Generating the Mesh

The following sections describe the steps for generating the mesh:

Launching the Meshing Application

On the Project Schematic, right-click the Mesh cell in the Mesh system and select **Edit...** to launch the Meshing application.

Setting the Unit System

On the main menu, click **Units** and select **Metric (mm, kg, N, s, mV, mA)**.

Program Controlled Inflation Using the Fluent Solver

This part of the tutorial demonstrates the use of **Program Controlled** inflation with the Fluent solver.

Notice that three Named Selections are defined already: Symmetry, Inlet, and Outlet. You will create a fourth later in this tutorial.

1. In the Tree Outline, click the **Mesh** object.

   In the Details View, notice that **Solver Preference** is set to **Fluent**.

2. Click to expand the **Sizing** group of controls and change **Curvature Normal Angle** to **12**.

3. Click to expand the **Inflation** group of controls.

   Notice that **Program Controlled** and **Smooth Transition** are selected and **Transition Ratio** is set to **0.272** by default.

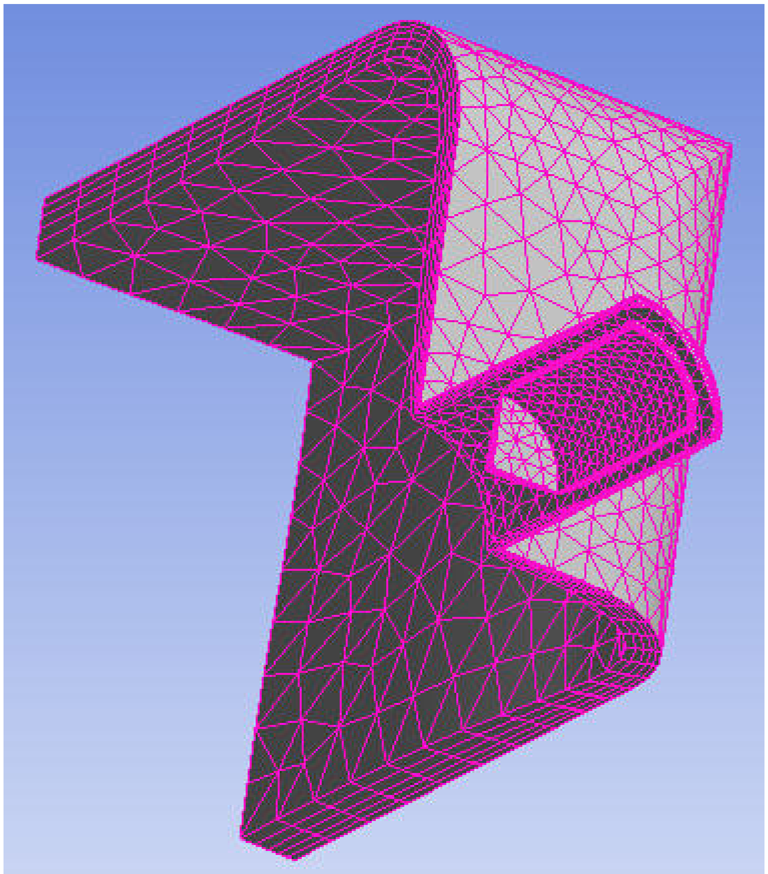
   With **Program Controlled** inflation, inflation will be added to all external faces for which a Named Selection has *not* been defined.

   When **Solver Preference** is **Fluent**, the default **Transition Ratio** is **0.272** because the solver uses a cell-centered scheme. This is in contrast to the **CFX Solver Preference**, which is covered later in this tutorial.

4. Change **Maximum Layers** to **5**.

5. In the Tree Outline, right-click **Mesh** and select **Preview > Inflation**. Previewing inflation helps to identify possible problems with inflation before generating a full mesh.

   After a few moments, a preview of the inflation layers appears in the **Geometry** window, as shown below.

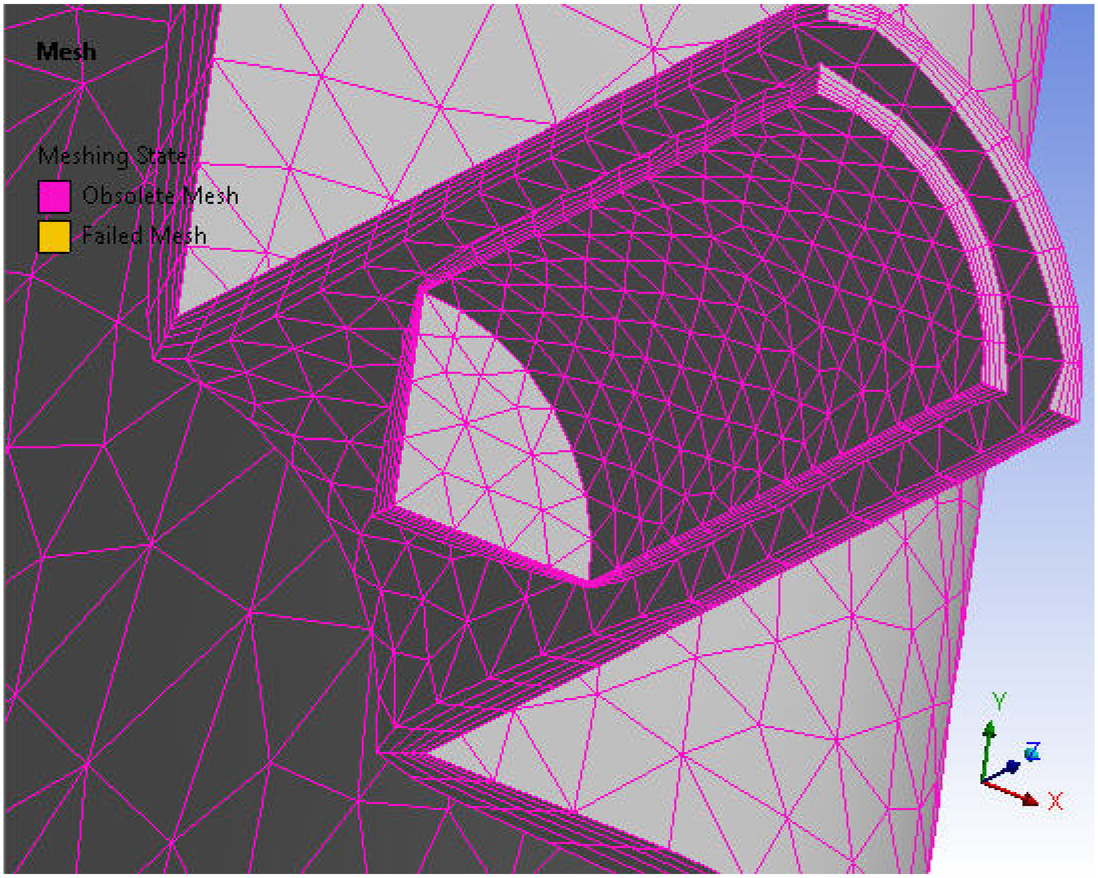
   Because the **Fluent** solver was used, the meshing process used the **Layer Compression** method for **Collision Avoidance** by default.

   Note:

   Notice that the mesh is colored pink. Because the pre-inflation mesh is not a final mesh, it is considered “obsolete", and will remain obsolete until a new mesh is generated. The mesh coloring helps to differentiate between partial and final meshes when you have a fully meshed set of parts and apply pre-inflation or other new boundaries on some of the parts.

   ---

6. Zoom and reposition the model to get a better view of the compressed layers in the area of interest.



Notice the heights of the inflation layers, which are determined by the setting of the **Inflation Option** control. The **Smooth Transition** option, which was used here, uses the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth.

Each triangle that is being inflated will have an initial height that is computed with respect to its area, averaged at the nodes. This means that for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary.

7. On the toolbar, click **Zoom To Fit** .

8. In the Tree Outline, right-click **Mesh** and select **Clear Generated Data**.

9. Click **Yes** to clear the data.

Program Controlled Inflation Using the Ansys CFX-Solver

This part of the tutorial demonstrates the use of **Program Controlled** inflation with the **CFX** solver.

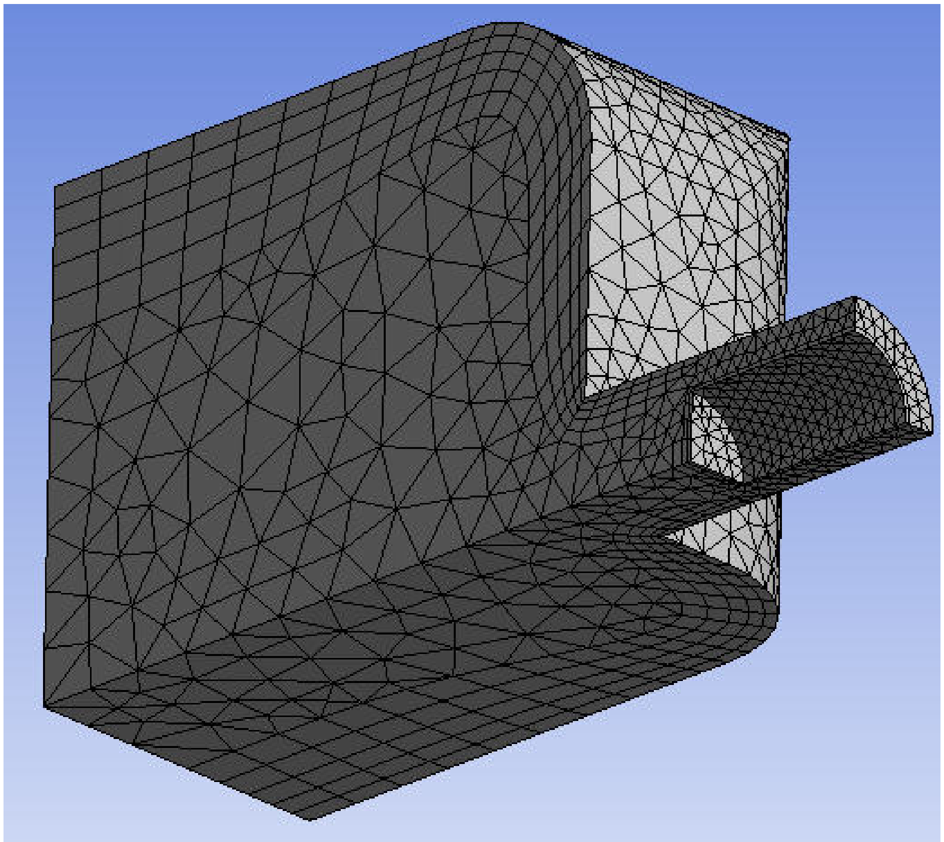
1. In the Details View, change **Solver Preference** to **CFX**.

   Notice the value of **Transition Ratio** has changed from **0.272** to **0.77** automatically.

   When **Solver Preference** is **CFX**, the default **Transition Ratio** is **0.77** because the solver uses a vertex-centered scheme. Increasing the ratio creates a thicker boundary layer.

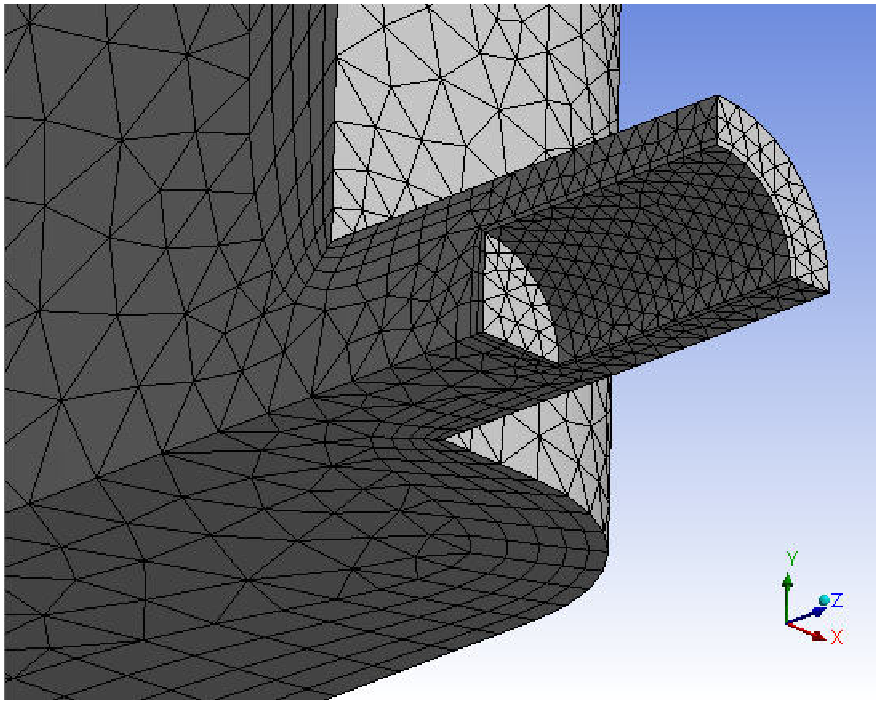
2. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

   After a few moments, the mesh appears in the **Geometry** window, as shown below.



   The inflation layers look different in this mesh because the **CFX** solver uses the **Stair Stepping** method of **Collision Avoidance** by default.

3. Zoom and reposition the model to get a better view of the stair stepped layers in the narrow region.



4. On the toolbar, click **Zoom To Fit** .

5. In the Tree Outline, right-click **Mesh** and select **Clear Generated Data**.

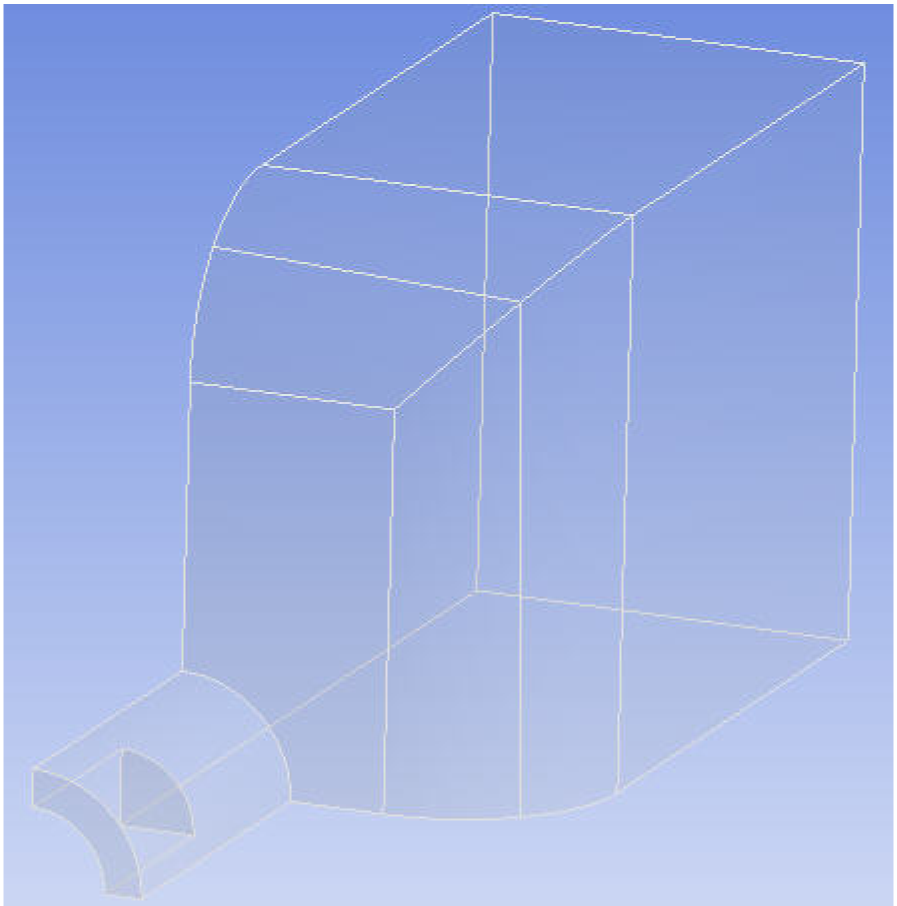
6. Click **Yes** to clear the data.

Program Controlled Inflation Scoped to All Faces in a Named Selection

This part of the tutorial demonstrates the use of **Program Controlled** inflation scoped to all faces in a Named Selection that you create.

1. Create the Named Selection.

   a. Rotate the body so that it is positioned as shown below.



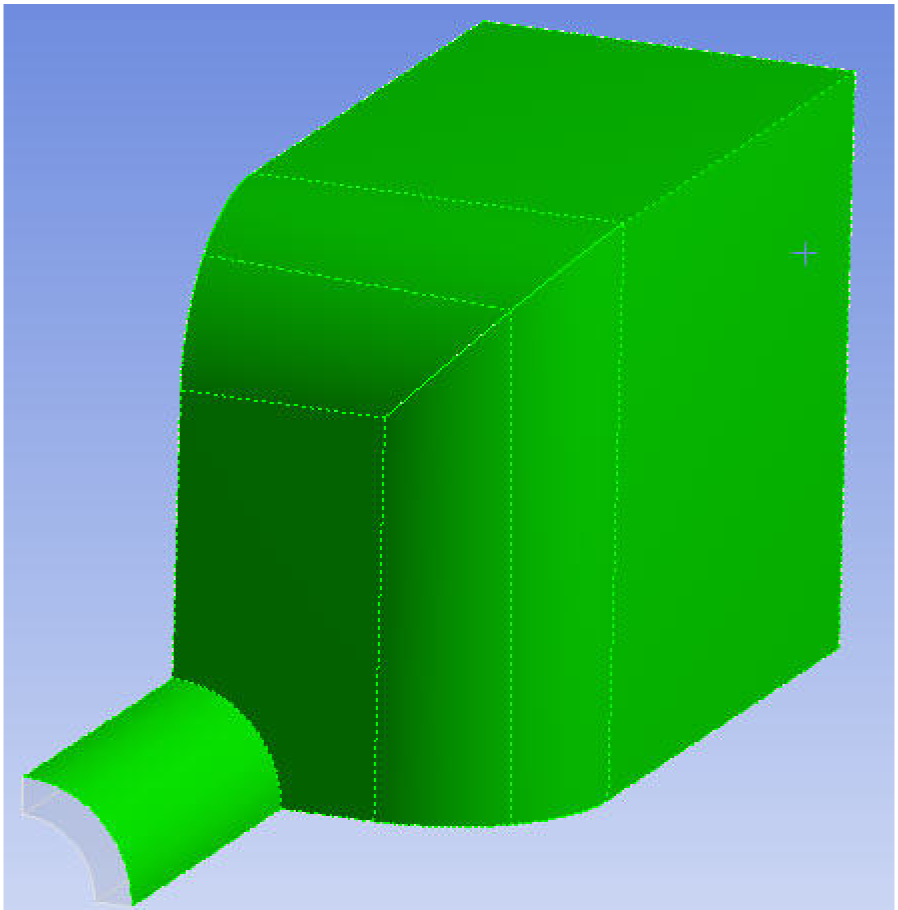
There are 10 faces that you need to select in all.

b. Select the first eight faces, as shown below.

   i. Click **Face** .

   ii. On the keyboard, press and hold **Ctrl**.

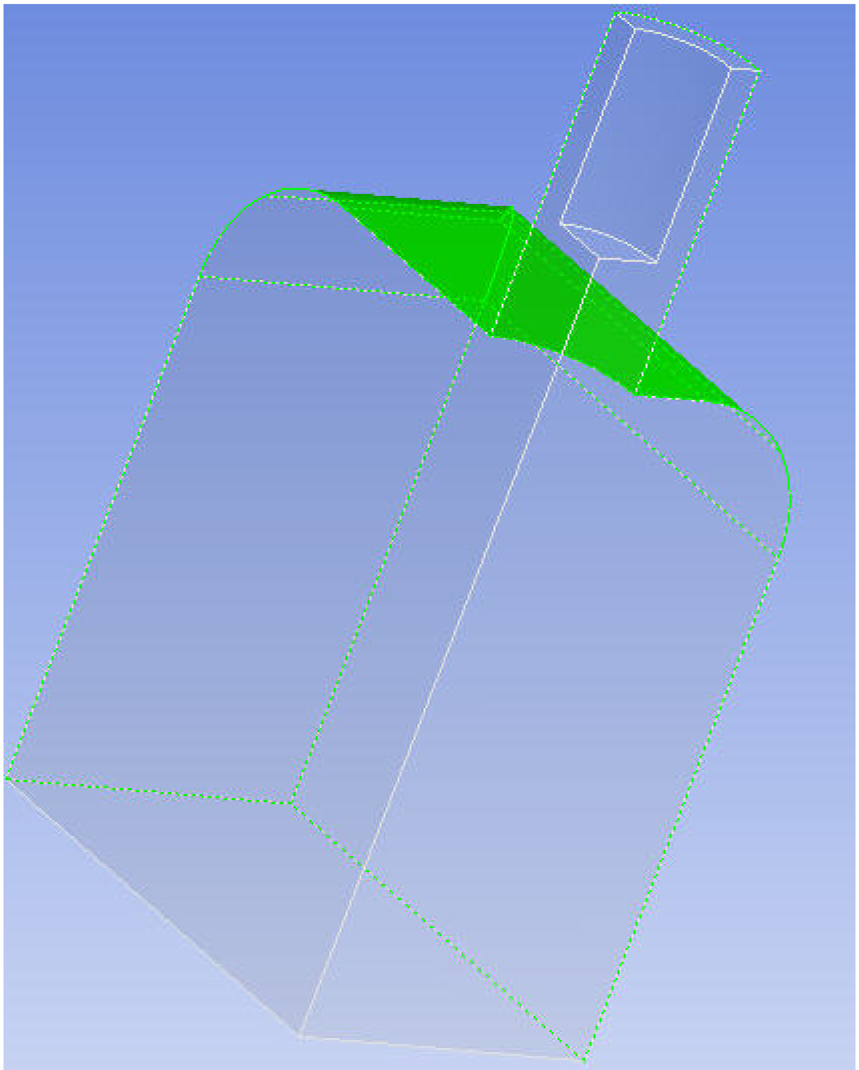
   iii. Select the first eight faces, as shown below.



c. To select the remaining two faces, rotate the body so that it is positioned as shown below.

Note:

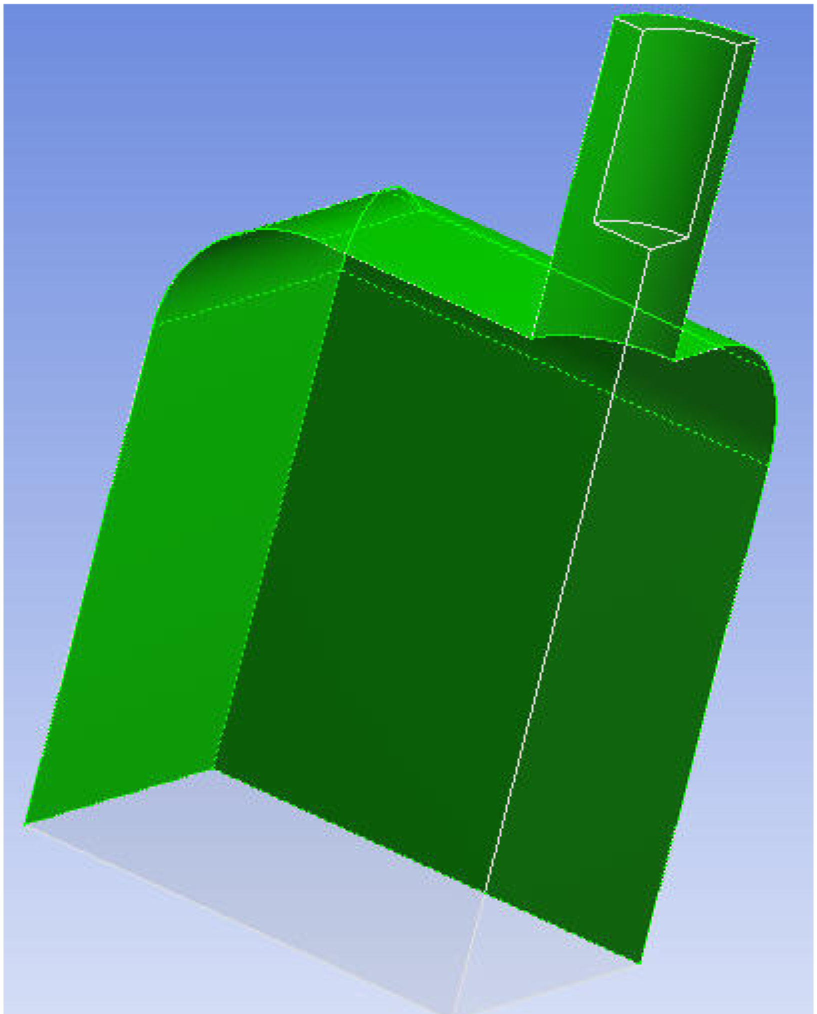
The model you see may look different than what is shown in the image below. The differences are explained in the next steps.



When the image above was captured, the **Draw Face Mode** option was set to **Auto Face Draw** (the default). The **Draw Face Mode** option determines whether face culling is turned on or off:

- **Auto Face Draw** (default) - Turning back-face culling on or off is program controlled.
- **Draw Front Faces** - Face culling is forced to stay on. Back-facing faces are not drawn.
- **Draw Both Faces** - Back-face culling is turned off. Both front-facing and back-facing faces are drawn.

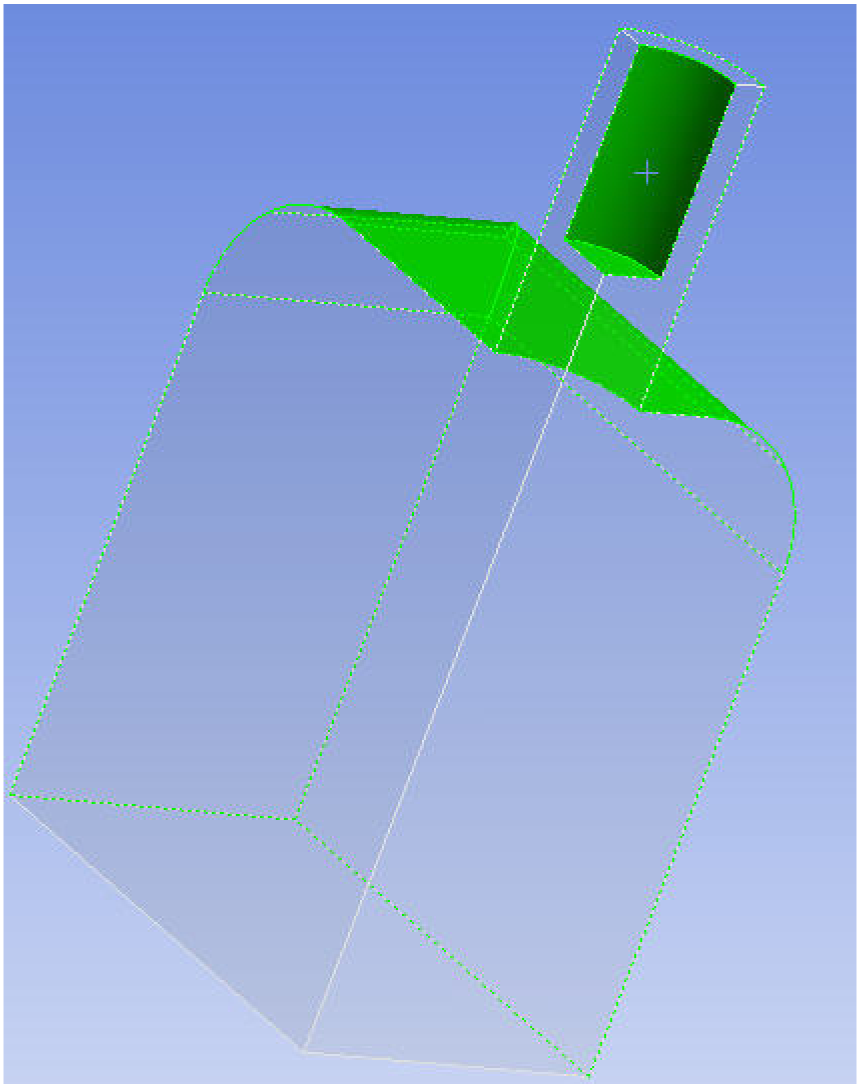
In the image above, the default setting resulted in face culling being forced to stay on, so that back-facing faces were not drawn (that is, it is behaving the same way that the **Draw Front Faces** setting would behave). This makes it easier to identify the faces that still need to be selected. However, because the default setting is program controlled based on other features and options you may have set, you may or may not need to manually select **Draw Front Faces** to proceed with this tutorial. That is, if the model you see looks like this:



i. Click **Face** .

ii. Press and hold **Ctrl**.

iii. Select the last two faces as shown below.



d. Right-click in the **Geometry** window and select **Create Named Selection**.

e. In the **Selection Name** dialog box, type **Wall** and click **OK**.

2. In the Tree Outline, click **Wall** to display the new Named Selection in the **Geometry** window.
3. In the Tree Outline, click **Mesh**.
4. In the Details View, change **Solver Preference** to **Fluent**.
5. In the **Inflation** group, change **Use Automatic Inflation** to **All Faces in Chosen Named Selection**.
6. For **Named Selection**, select **Wall**.
7. Change **Inflation Option** to **Total Thickness**.

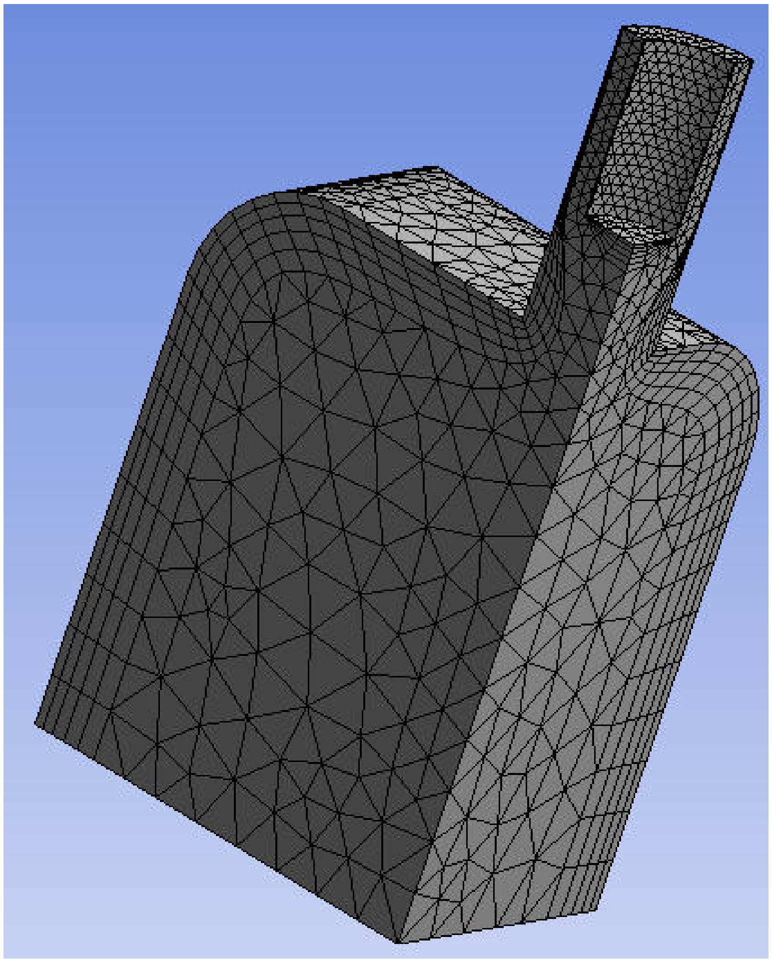
   The **Total Thickness** option creates constant inflation layers using the values of the **Number of Layers** and **Growth Rate** controls to obtain a total thickness as defined by the **Maximum**

Thickness control. Unlike inflation with the **Smooth Transition** option, with **Total Thickness** the thickness of the first inflation layer and each subsequent layer is constant.

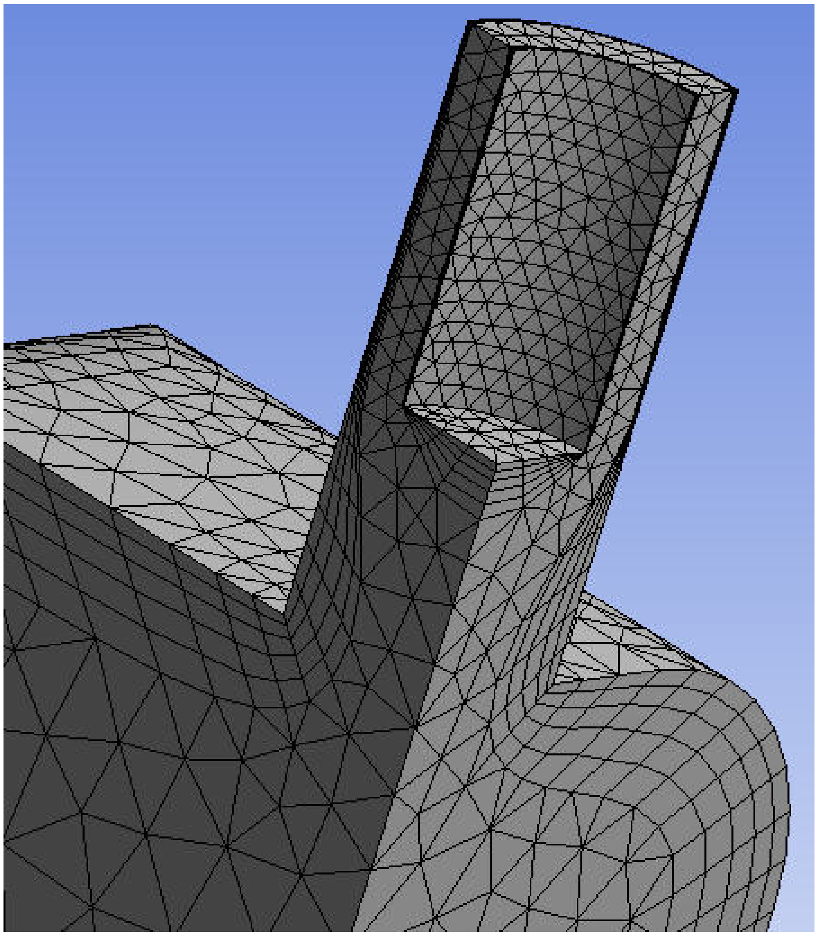
8. Set **Maximum Thickness** to **5**.

9. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

   After a few moments, the mesh appears in the **Geometry** window, as shown below.



10. Zoom and reposition the model to get a better view of the inflation layers in the narrow region.



11. On the toolbar, click **Zoom To Fit** .
12. In the Tree Outline, right-click **Mesh** and select **Clear Generated Data**.
13. Click **Yes** to clear the data.

Scoped Inflation

This section of the tutorial demonstrates scoping inflation to a body and selecting a Named Selection to act as the inflation boundary.

1. In the Tree Outline, click **Mesh**.
2. On the toolbar, click **Body** .
3. Select the body in the **Geometry** window.
4. Right-click in the **Geometry** window and select **Insert > Inflation**.

   In the Details View, notice that inflation will be scoped to **1 Body**.
5. In the Details View, set **Boundary Scoping Method** to **Named Selections**.

6. From the **Boundary** drop-down menu, select **Wall** by highlighting it in the drop-down menu and then pressing **Enter**.

7. Change **Inflation Option** to **Last Aspect Ratio**.

   The **Last Aspect Ratio** option creates inflation layers using the values of the **First Layer Height**, **Maximum Layers**, and **Aspect Ratio (Base/Height)** controls. With this option, the heights of the inflation layers are determined by the aspect ratio of the inflations that are extruded from the inflation base. The aspect ratio is defined as the ratio of the local inflation base size to the inflation layer height.

8. Set **First Layer Height** to **0.5**.

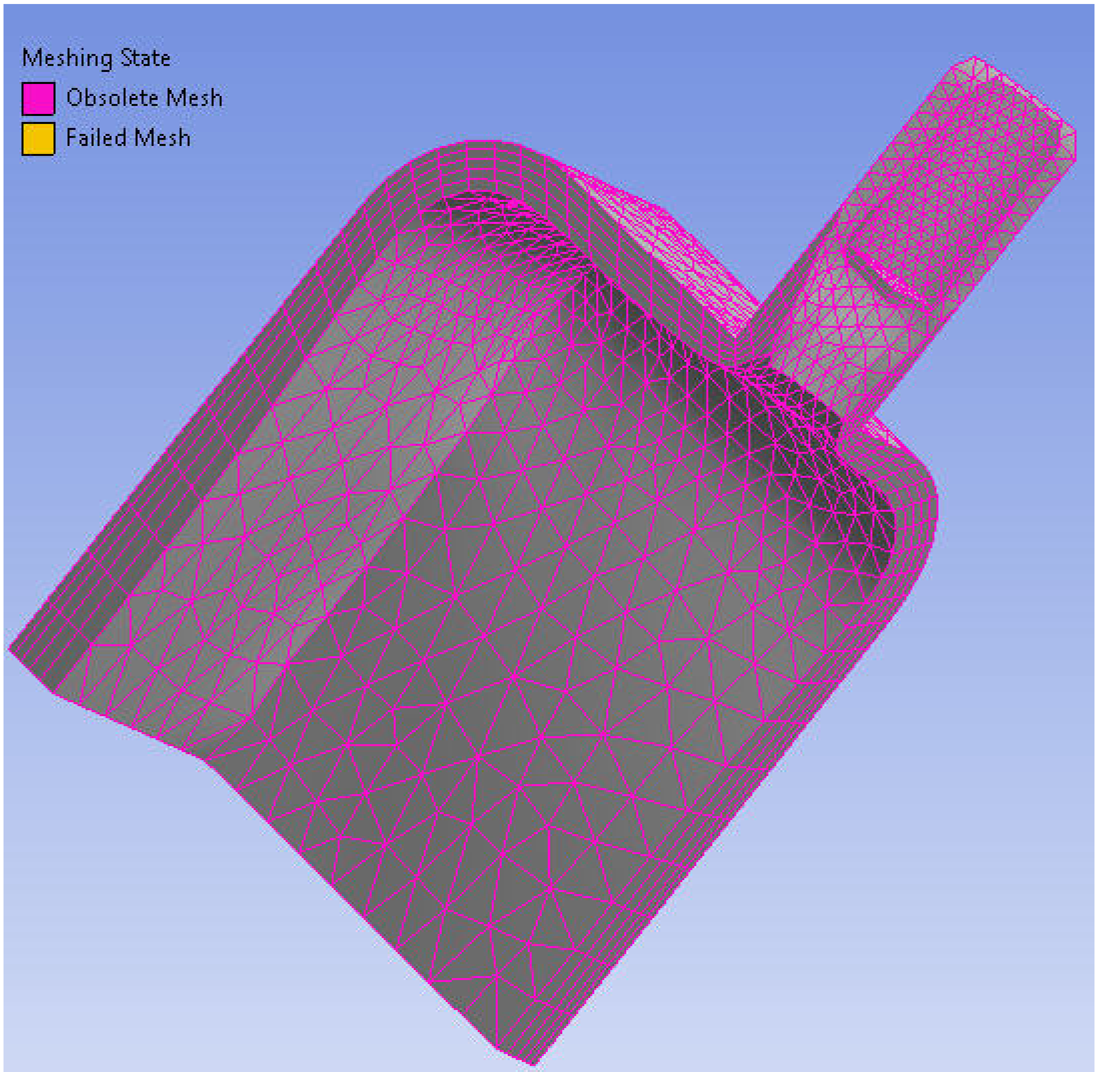
9. You are now finished setting the scoped (local) inflation controls. In the Tree Outline, click **Mesh** to return to the global inflation controls.

10. In the Details View, change **Solver Preference** to **CFX**.

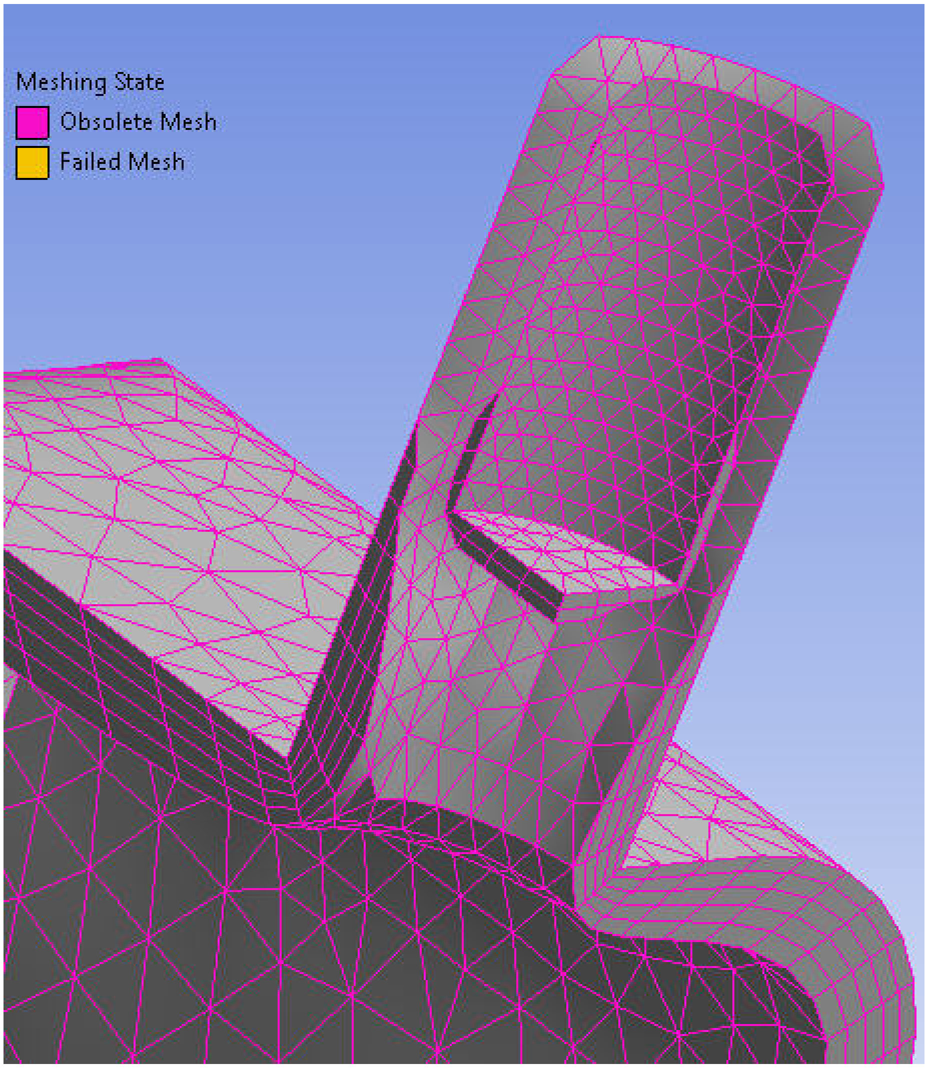
11. Change **Use Automatic Inflation** to **None**.

12. In the Tree Outline, right-click **Mesh** and select **Preview > Inflation**.

    After a few moments, the mesh appears in the **Geometry** window, as shown below.



13. Zoom and reposition the model to get a better view of the inflation layers.



This completes the tutorial. From the Meshing application's main menu, select **File** > **Save Project** to save the project and then **File** > **Close Meshing** to return to the Project Schematic.

You can exit Ansys Workbench by selecting **File** > **Exit** from the main menu.

Mesh Controls and Methods

This tutorial creates a mesh for a piston. The geometry will be imported as a complete geometry from a Parasolid file. The tutorial uses the model of the piston to demonstrate various mesh controls and methods that are available in the Meshing application.

The following topics are covered:

- Parasolid import
- Batch meshing
- Automatic mesh method (Patch Conforming Tetrahedral and Sweep)
- Defeaturing and local (scoped) sizing
- Mapped face meshing
- MultiZone mesh method
- Section planes

Preparation

This tutorial requires you to have a copy of the Parasolid file `PISTON.x_t`.

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Download the `piston.zip` file here.
3. Unzip the `piston.zip` file you have downloaded to your working folder.

You can proceed to Tutorial Setup (p. 27).

Tutorial Setup

The following sections describe the steps for creating the project and importing the geometry:

Creating the Project

1. Open Ansys Workbench and add a standalone Mesh system to the Project Schematic.

The Mesh system contains a Mesh system header and two cells. Notice the system name that is located below the system defaults to **Mesh**.

2. To change the system name, right-click the Mesh system header and select **Rename**. The name that appears below the system is now editable. Type **Piston** and press **Enter** to rename the system **Piston**.

3. Save the project as `Piston.wbpj`.

Importing the Geometry

The geometry is imported complete, from a Parasolid file.

1. Now add geometry to the project. On the Project Schematic, right-click the Geometry cell in the Mesh system and select **Import Geometry > Browse...**.

2. In the file browser that opens, locate and open the file `PISTON.x_t`.

The geometry is complete and does not need modifications. Notice the Geometry cell in the Mesh system has an up-to-date state ✓.

Now that the tutorial is set up, you can proceed to Generating the Mesh (p. 28).

Generating the Mesh

The following sections describe the steps for generating the mesh:

Running the Meshing Application in Batch Mode

Meshing in batch mode requires less RAM.

On the Project Schematic, right-click the Mesh cell in the Mesh system and select **Update** to mesh the geometry in batch mode.

After a short wait, the meshing process is complete. Notice the Mesh cell in the Mesh system has an up-to-date state ✓.

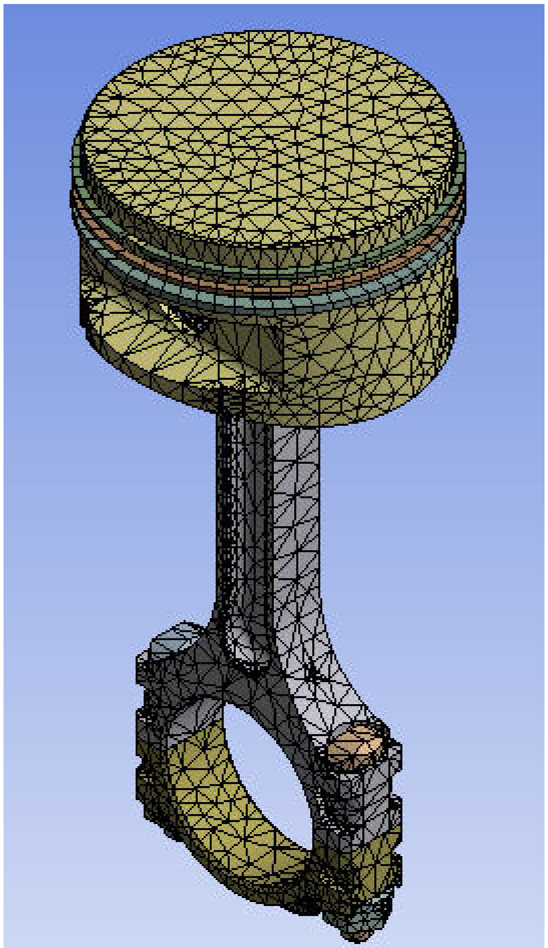
Launching the Meshing Application

Launch the Meshing application to view the mesh and define mesh controls.

1. Right-click the Mesh cell in the Mesh system and select **Edit...**.

2. When the Meshing application opens, click the **Mesh** object in the Tree Outline to view the meshed model in the **Geometry** window as shown below.

   Since no mesh controls have been set, the Automatic mesh method was used by default. When the Automatic method is used, bodies are swept if possible, and the remaining bodies are meshed with the Patch Conforming Tetrahedral mesh method.

Setting the Unit System

On the main menu, click **Units** and select **Metric (m, kg, N, s, V, A)**.

Showing Sweepable Bodies

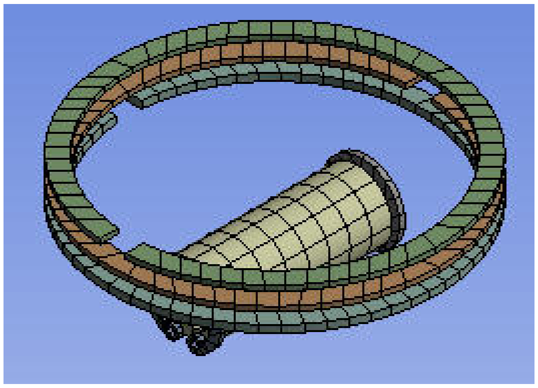
This part of the tutorial demonstrates how to view the bodies that were swept. As an alternative, you can view sweepable bodies prior to meshing.

1. In the Tree Outline, right-click **Mesh** and select **Show > Sweepable Bodies**.

   The sweepable bodies are highlighted in the **Geometry** window.

2. To hide all non-sweepable bodies for a better view of the sweepable bodies, right-click on the **Geometry** window and select **Hide All Other Bodies**.

Only the bodies that were meshed with the sweep method appear in the **Geometry** window, as shown below.



3. Right-click in the **Geometry** window and select **Show All Bodies**.

   All bodies re-appear in the **Geometry** window.

4. In the Tree Outline, right-click **Mesh** and select **Show** > **Sweepable Bodies** again.

5. Right-click the **Geometry** window and select **Hide Body**.

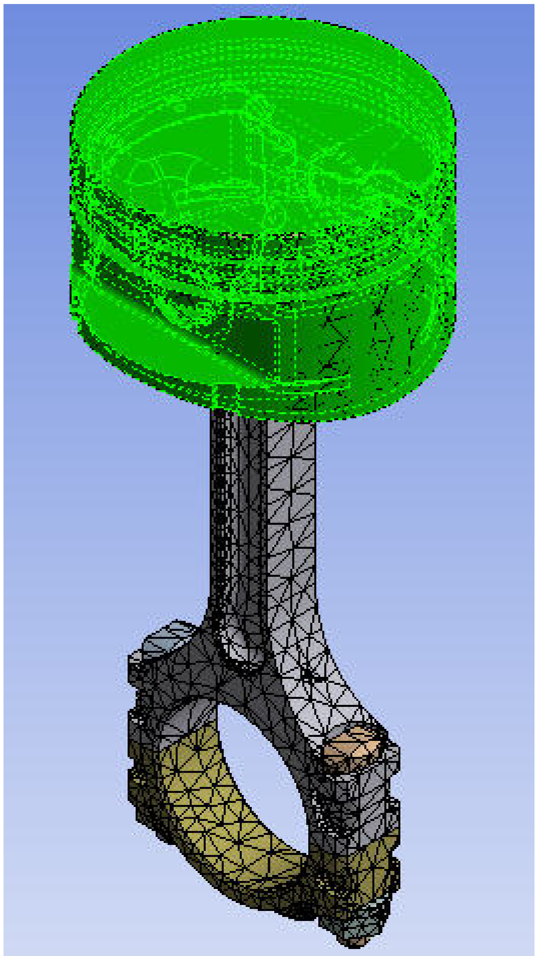
   Now only the sweepable bodies are hidden. Those bodies that remain in the **Geometry** window were meshed with the Patch Conforming Tetrahedral method.

Using Defeaturing and Local Sizing

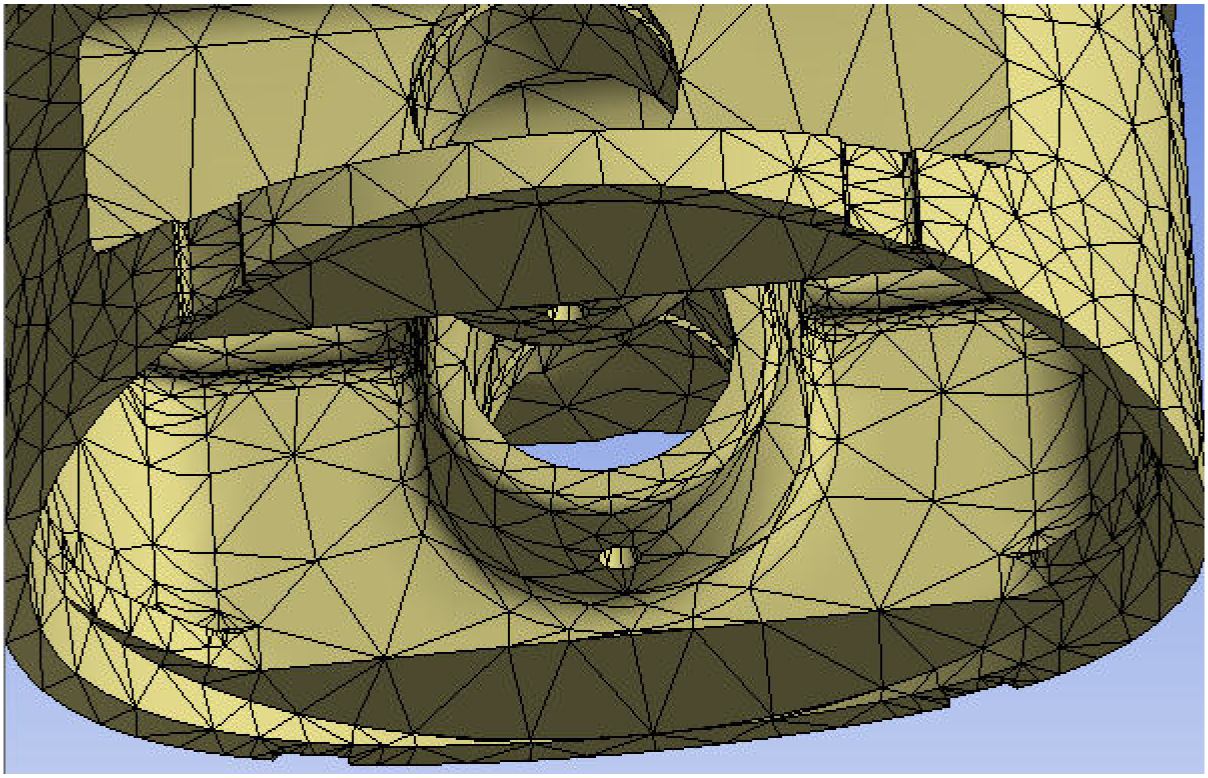
Thin regions of the model can create poor quality mesh and/or increase the mesh size. This part of the tutorial demonstrates how to use local sizing and defeaturing to get a better quality mesh on regions of interest.

1. On the toolbar, click **Body** .

2. In the **Geometry** window, select the top of the piston as shown below.



3. Right-click in the **Geometry** window and select **Hide All Other Bodies**.
4. Rotate and zoom in on the underside of the geometry so that the model is positioned as shown below. Notice the thin regions at the bottom of the model and how the mesh conforms to the small patches..



5. Use local sizing with defeaturing to generate better quality elements.

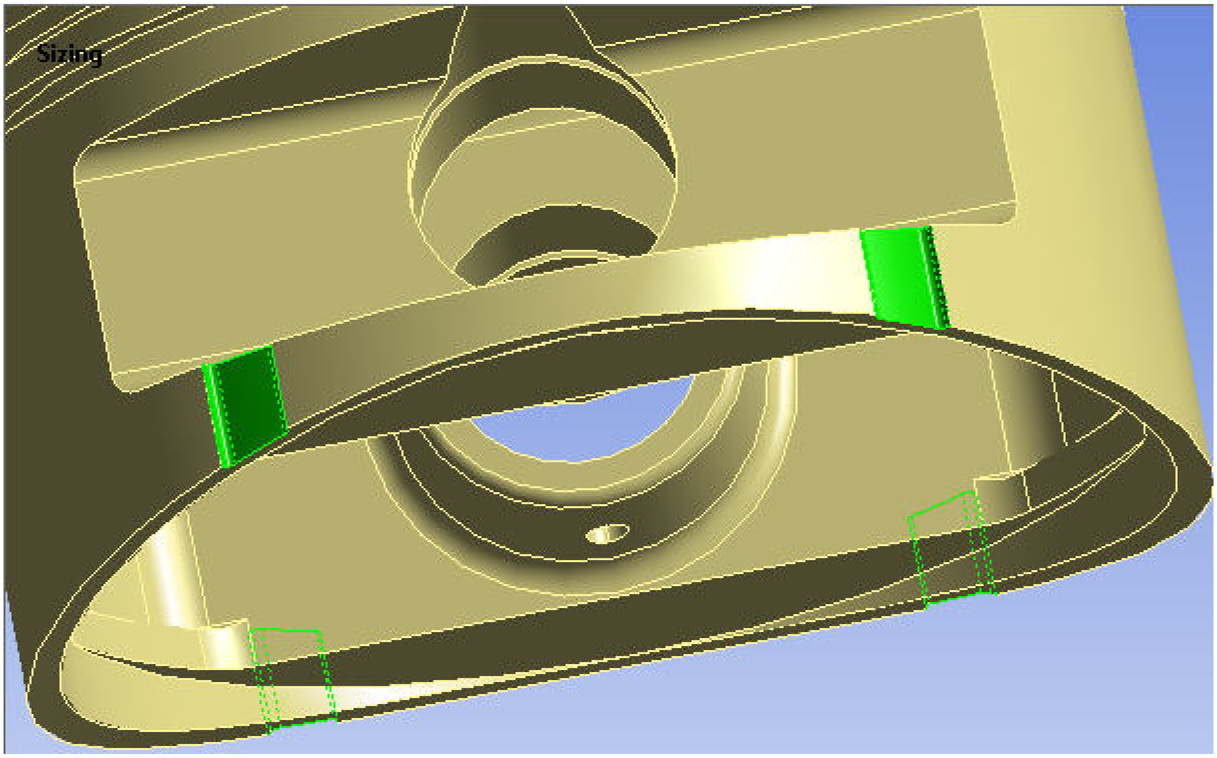
   a. In the Tree Outline, right-click **Mesh** and select **Insert > Sizing**.

   b. On the toolbar, click **Face** .

   c. On the keyboard, press and hold **Ctrl**.

   d. In the **Geometry** window, select the faces as shown below.

      Note that double-clicking a face will select that face and the attached ones up to the angle. Rotate the model to select the faces on the other side.



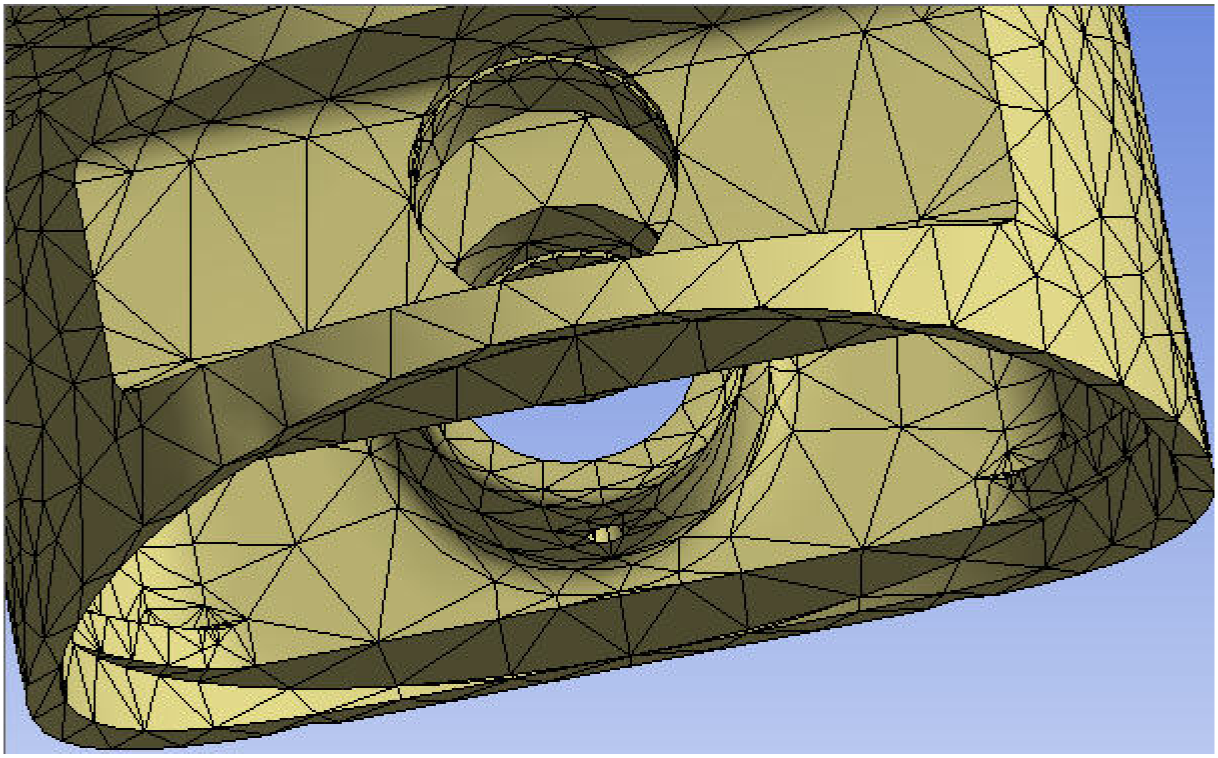
e. In the Details View, click to **Apply** the selection. Verify that the Sizing is scoped to **20 Faces**.

f. Set **Defeature Size** to 0.002 m.

6. In the Tree Outline, right-click **Mesh** and select **Update** .

7. In the Tree Outline, click **Mesh**.

   Notice the defeaturing removed the thin, poor quality mesh, thereby reducing the element count. Also, the narrow faces are defeatured.



8. Right-click the **Geometry** window and select **Show All Bodies**.

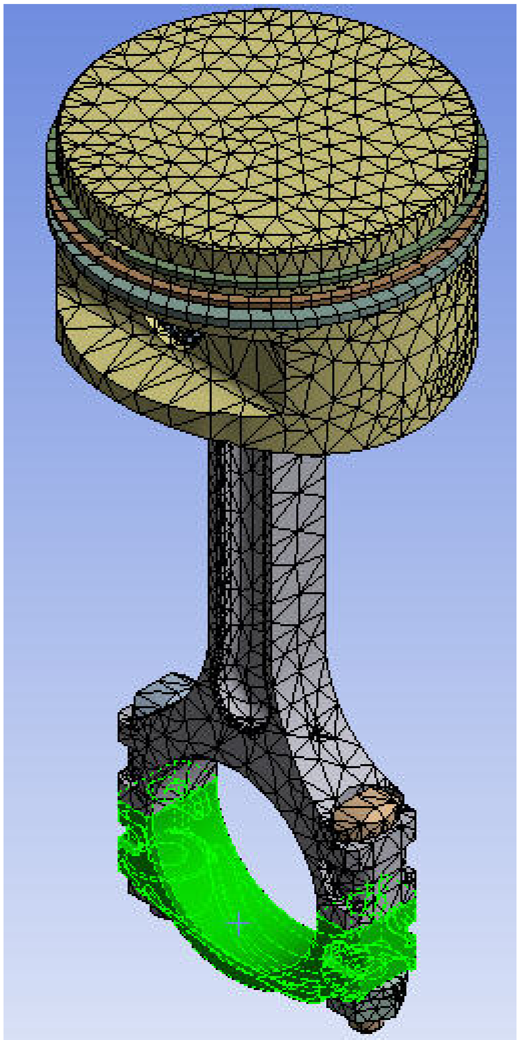
9. On the triad located in the lower right corner of the **Geometry** window, click the sphere to return the view to isometric.

Defining Mapped Face Meshing

This part of the tutorial demonstrates how to use **Face Meshing** controls to create a mapped face meshing. Face Meshing controls for mapped meshing attempt to generate a mapped mesh on selected faces. The Meshing application determines a suitable number of divisions for the edges on the boundary face automatically. If you specify the number of divisions on the edge with a **Sizing** control, the Meshing application attempts to enforce those divisions.

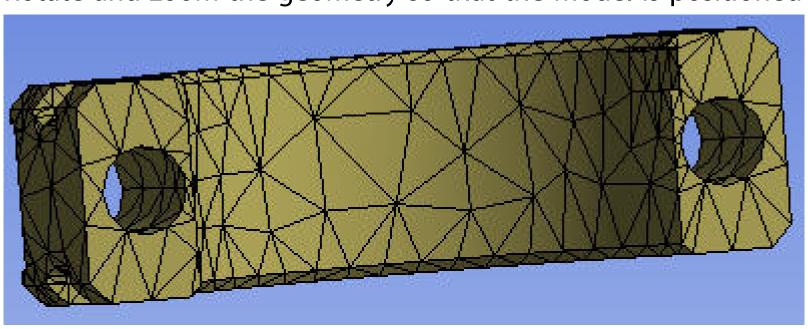
1. On the toolbar, click **Body** .

2. In the **Geometry** window, select the bottom of the piston as shown below.



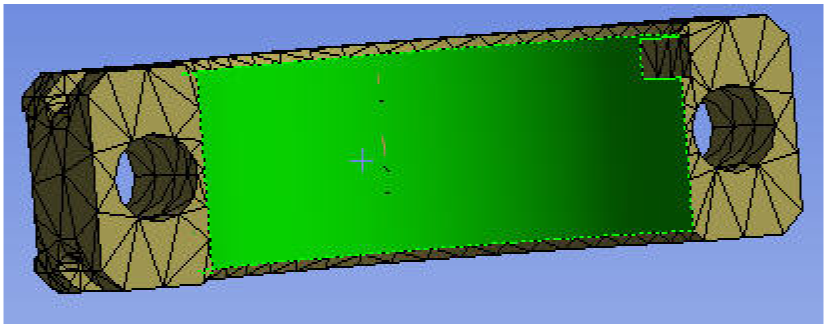
3. Right-click in the **Geometry** window and select **Hide All Other Bodies**.

4. Rotate and zoom the geometry so that the model is positioned as shown below.



5. In the Tree Outline, right-click **Mesh** and select **Insert >Face Meshing**. In the Details View, **Mapped Mesh** is set to **Yes** as the default.

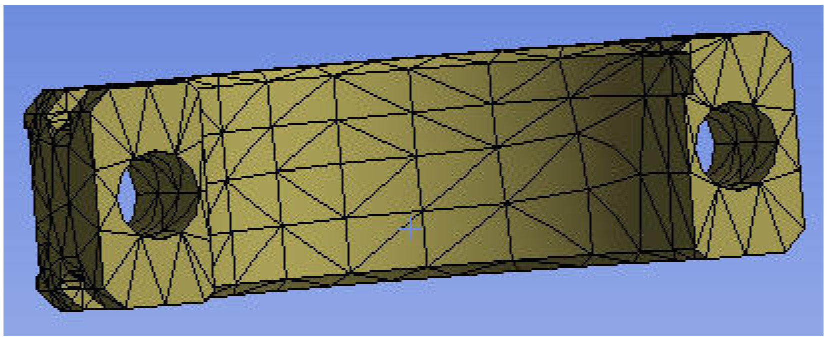
   a. In the **Geometry** window, select the face as shown below.



b. In the Details View, click **Apply**.

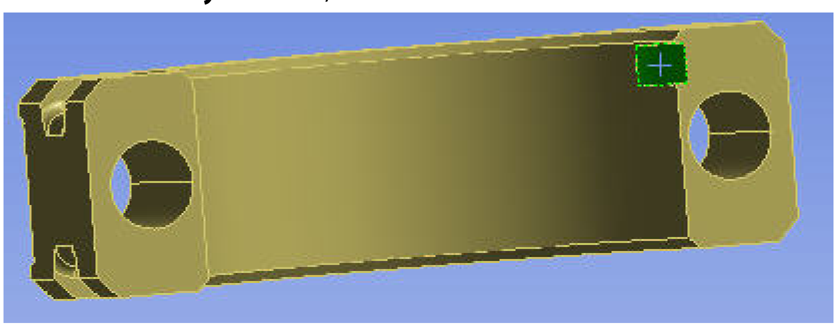
c. In the Tree Outline, right-click **Mesh** and select **Update**.

Notice the changes in the mesh.



6. In the Tree Outline, right-click **Mesh** and select **Insert** > **Face Meshing** again.

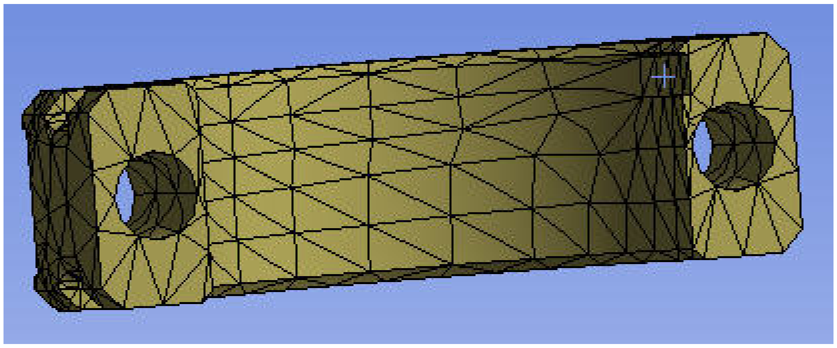
a. In the **Geometry** window, select the small face as shown below.



b. In the Details View, click **Apply**.

c. In the Tree Outline, right-click **Mesh** and select **Update**.

Notice the changes in the mesh.



7. In the **Geometry** window, right-click and select **Show All Bodies**.

8. On the triad located in the lower right corner of the **Geometry** window, click the sphere to return the view to isometric.

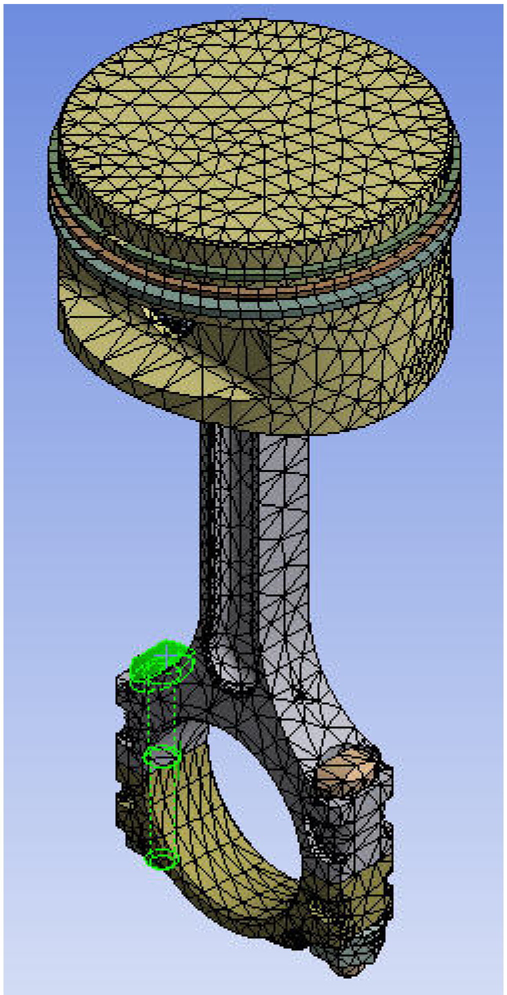
Using the MultiZone Mesh Method

This part of the tutorial demonstrates how to obtain a MultiZone mesh for one of the bodies in the piston.

1. Select the bolt on the bottom left.

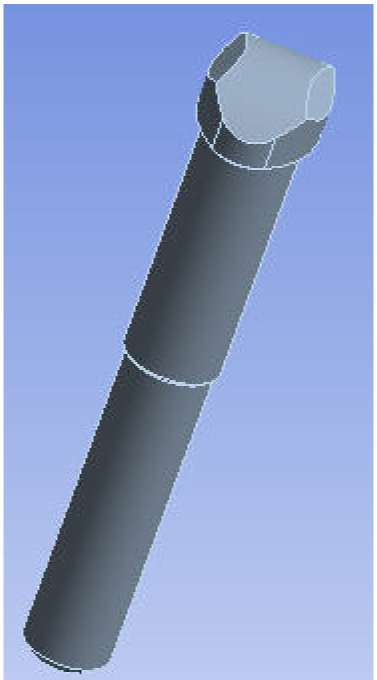
   a. On the toolbar, click **Body** .

   b. In the **Geometry** window, select the bolt as shown below.



c. Right-click in the **Geometry** window and select **Hide All Other Bodies**.

d. Rotate and zoom the geometry so that the model is positioned as shown below.



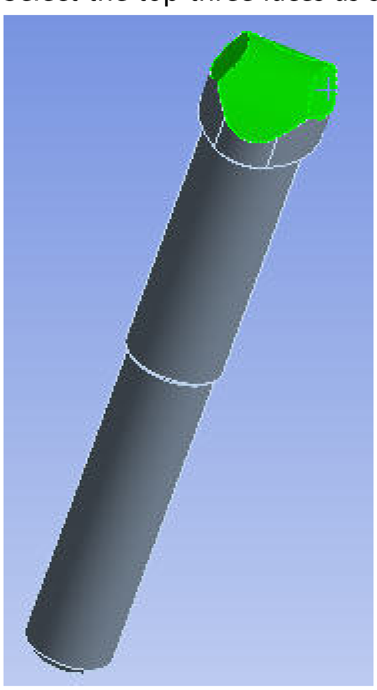
2. Insert virtual cells.

   a. In the Tree Outline, right-click **Model (A3)** and select **Insert > Virtual Topology**.

   b. Click **Face** .

   c. On the keyboard, press and hold **Ctrl**.

   d. Select the top three faces as shown below.



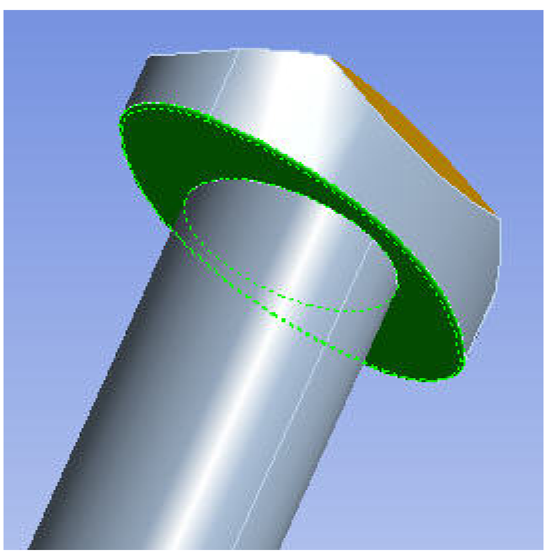
   e. Right-click in the **Geometry** window and select **Insert > Virtual Cell**.

f. Rotate the bolt so that it is positioned as shown below.

g. Click **Face** .

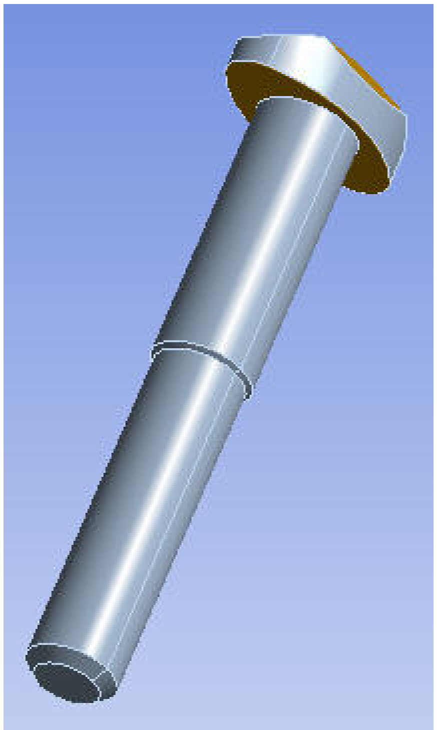
h. On the keyboard, press and hold **Ctrl**.

i. Select the bottom and bottom ridge of the top of the bolt as shown below. There are five faces to select—one for the bottom and four that make up the ridge.



j. Right-click in the **Geometry** window and select **Insert > Virtual Cell**.

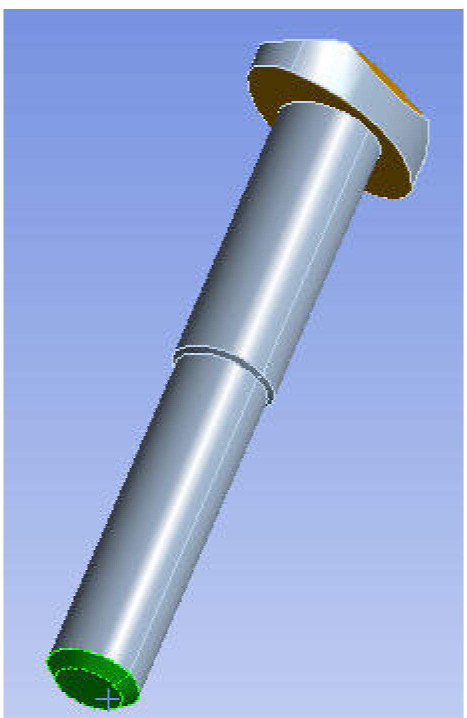
k. Position the bolt as shown below.



l. Click **Face** .

m. On the keyboard, press and hold **Ctrl**.

n. Select the bottom three faces of the bolt as shown below.



o. Right-click in the **Geometry** window and select **Insert** > **Virtual Cell**.

3. Set up the MultiZone method for the bolt

    a. In the Tree Outline, right-click **Mesh** and select **Insert > Method**.

    b. In the **Geometry** window, select the bolt.

    c. In the Details View, click **Apply**.

    d. In the Details View, change **Method** to **MultiZone**.

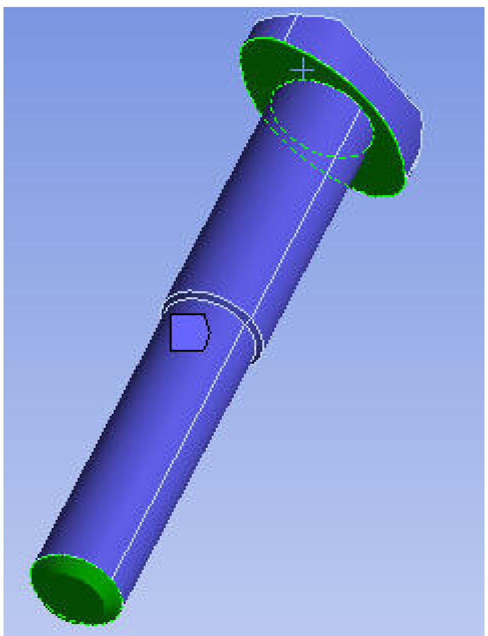
    e. Change **Src/Trg Selection** to **Manual Source**.

    There are three source faces to select.

        i. Click **Face** .

        ii. On the keyboard, press and hold **Ctrl**.

        iii. Select the first two faces as shown below.

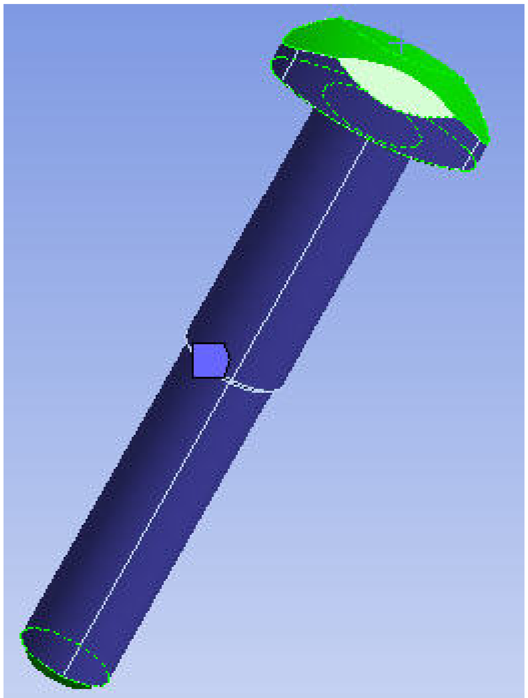


        iv. Reposition the bolt so you will be able to pick the topmost face.

        v. Click **Face** .

        vi. On the keyboard, press and hold **Ctrl**.

        vii. Select the third face as shown below.



viii. In the Details View, click to **Apply** your selections.

4. On the toolbar, click **Update** .

5. In the Tree Outline, click **Mesh**.

   Notice the changes to the mesh. By inserting the MultiZone method for the bolt, a mesh composed of mostly hexahedral elements was obtained. The mesh obtained is rather coarse. Next, define a local sizing control for a finer mesh and increased solution accuracy.

6. Define local sizing.

   a. In the Tree Outline, right-click **Mesh** and select **Insert** > **Sizing**.

   b. On the toolbar, click **Body** .

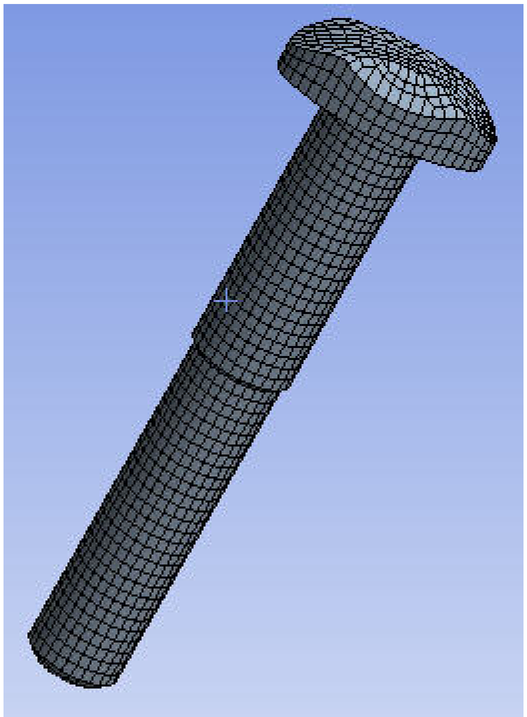
   c. In the **Geometry** window, select the bolt.

   d. In the Details View, click **Apply**.

   e. Set **Element Size** to **0.001**.

   f. In the Tree Outline, right-click **Mesh** and select **Update**.

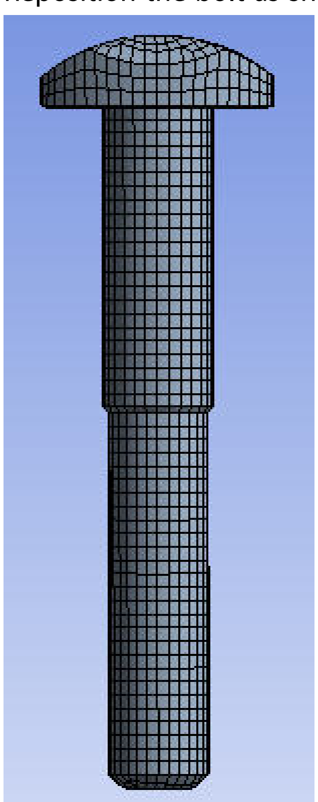
      Notice the changes to the mesh.

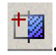


Defining a Section Plane

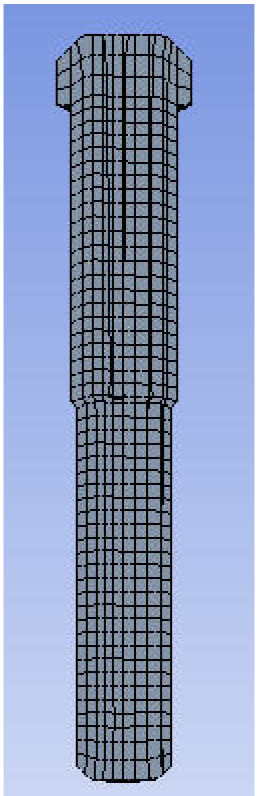
This part of the tutorial demonstrates how to activate a section plane to view a section cut through the bolt.

1. Reposition the bolt as shown below.



2. On the toolbar, click **New Section Plane** .

3. Press the left mouse button and drag a line down the center of the bolt to slice it in half. Rotate the geometry to view the section cut.



This completes the tutorial. From the Meshing application's main menu, select **File > Save Project** to save the project and then **File > Close Meshing** to return to the Project Schematic.

You can exit Ansys Workbench by selecting **File > Exit** from the main menu.

Sizing Options

This tutorial illustrates the use of the Meshing application's Sizing Options. This feature allows for greater control over global sizing options, including the following properties:

- Angles between normals for adjacent mesh elements (Curvature-type sizing)
- Number of mesh elements employed in the gaps between two geometric entities (Proximity-type sizing)
- Gradation between minimum and maximum sizes based on a specified growth rate

During the tutorial, the influence of each of the following sizing options on a mesh will be demonstrated:

- Uniform
- Proximity
- Curvature
- Proximity and Curvature

Preparation

This tutorial requires you to have a copy of the Ansys Workbench project file `nacelle.wbpj` and the project folder `nacelle_files` and its contents.

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Download the `nacelle.zip` file here.
3. Unzip the `nacelle.zip` file you have downloaded to your working folder.

You can proceed to Tutorial Setup (p. 47).

Tutorial Setup

1. Open Ansys Workbench.
2. Select **File** > **Open...** from the main menu.
3. In the file browser that opens, locate and open the file `nacelle.wbpj`.

Now that the tutorial is set up, you can proceed to Generating the Mesh (p. 48).

Generating the Mesh

The following sections describe the steps for generating the mesh:

Launching the Meshing Application

On the Project Schematic, right-click the Mesh cell in the Mesh system and select **Edit...** to launch the Meshing application.

Setting the Unit System

On the main menu, click **Units** and select **Metric (m, kg, N, s, V, A)**.

Expanding the Sizing Controls

1. In the Tree Outline, click **Mesh**.

2. In the Details View, click to expand the **Sizing** group of controls. Notice that **Use Adaptive Sizing** is set to **No** and **Capture Curvature** is enabled. **Capture Curvature** is the default when **Physics Preference** is set to **CFD** and **Solver Preference** is set to **Fluent**.

Adaptive Sizing

When **Use Adaptive Sizing** is set to **Yes**, the mesher uses the value of the element size to determine a starting point for the mesh size. The value of the element size can either be computed by the mesher automatically or be user-defined.

It is not recommended to use **Adaptive** sizing as the resulting mesh quality is not as good as mesh generated using other sizing options.

Using Uniform Sizing

In this part of the tutorial, you will use **Uniform** sizing. With this setting, the following factors contribute to the final mesh distribution:

- **Max Size**
- **Growth Rate**

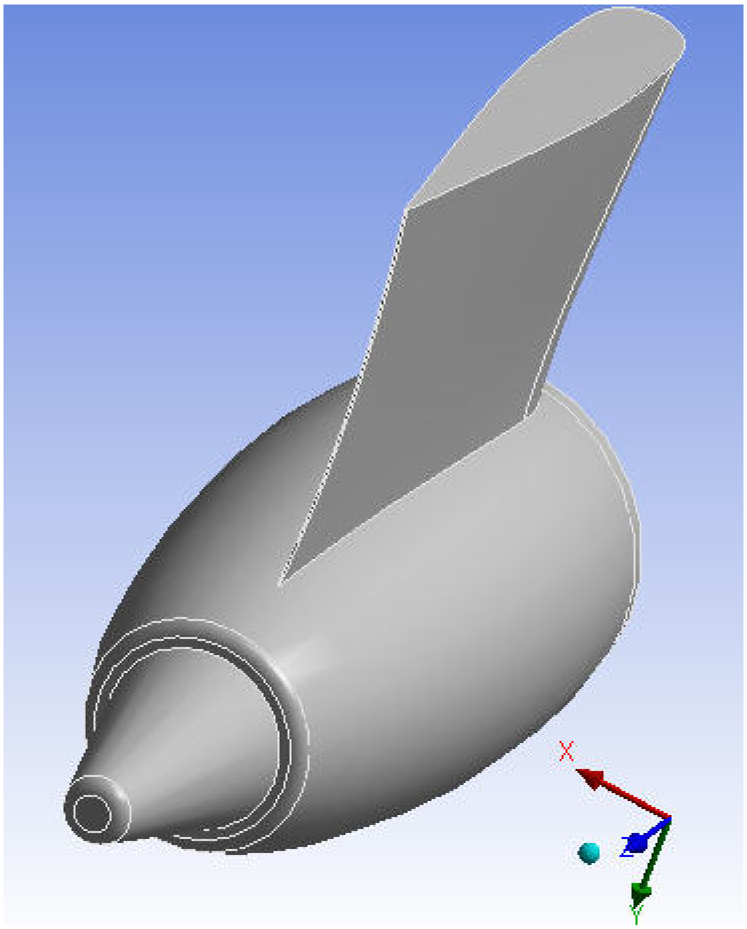
1. In the Details View, set **Capture Curvature** to **No**.

   Ensure that **Capture Proximity** is also set to **No**.

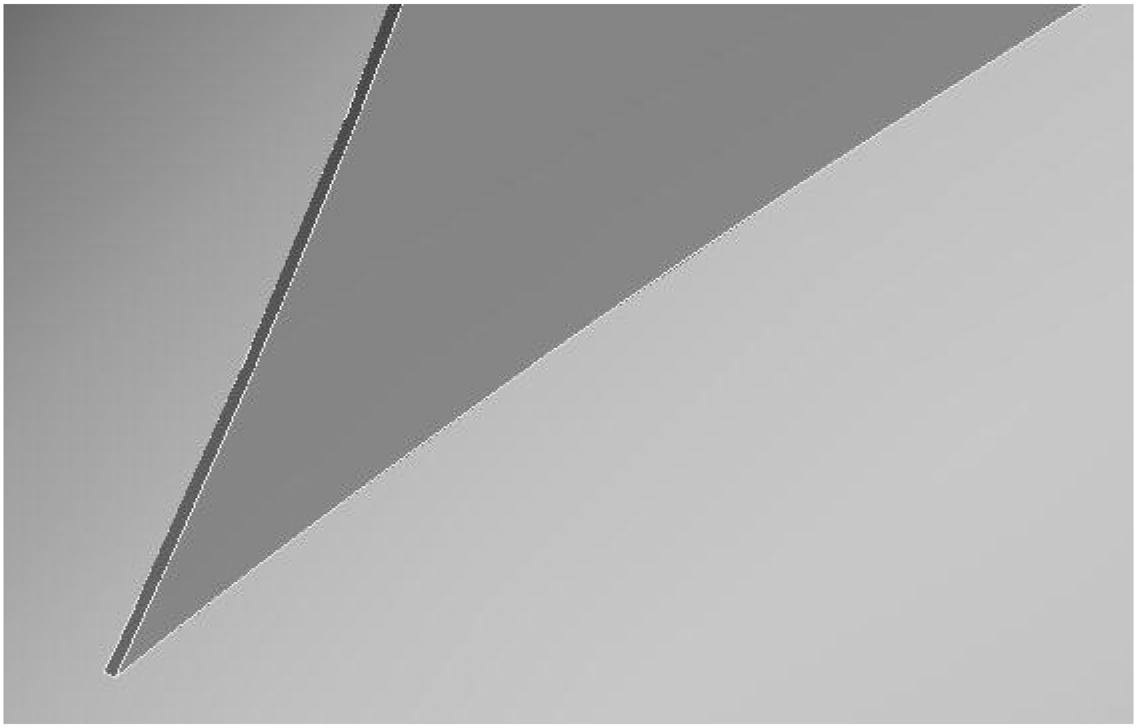
2. Insert a sizing control.

   a. In the Tree Outline, right-click **Mesh** and select **Insert > Sizing**.

   b. Rotate the geometry so that it is positioned as shown below.

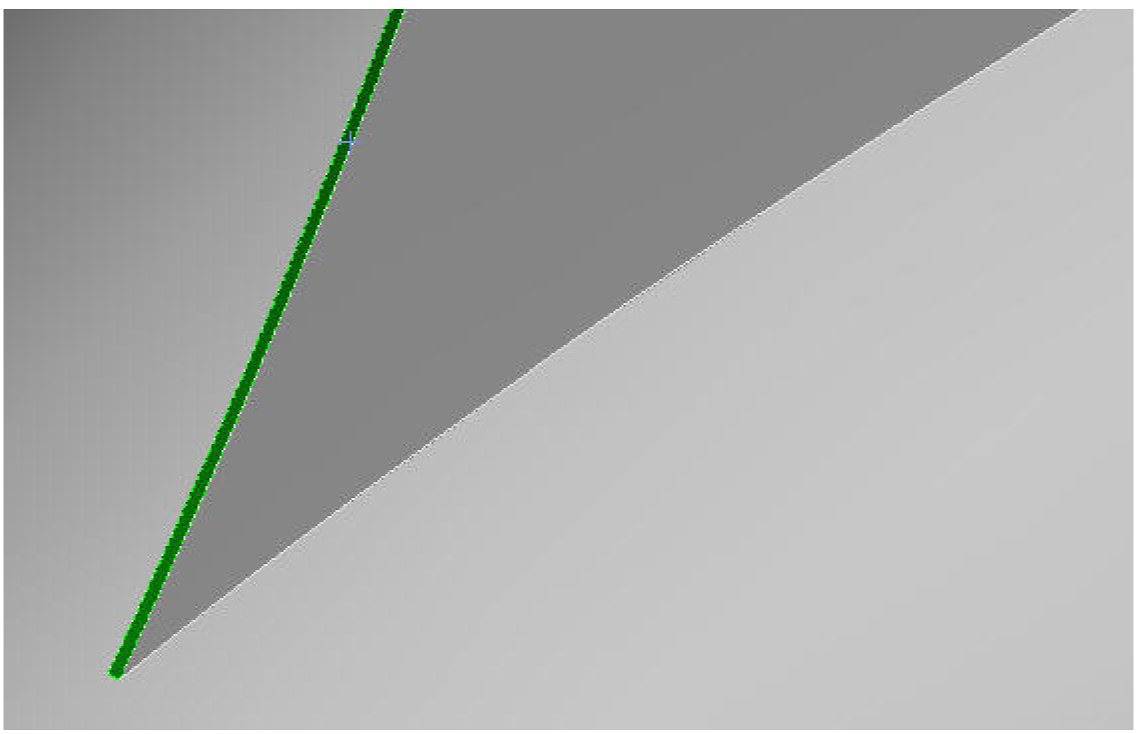
   

   c. Zoom the geometry as shown below.



d. On the toolbar, click **Face** .

e. Select the narrow face as shown below.



f. In the Details View, click **Apply** to complete the selection.

g. Set **Element Size** to 0.01.

h. Set **Behavior** to **Hard**.

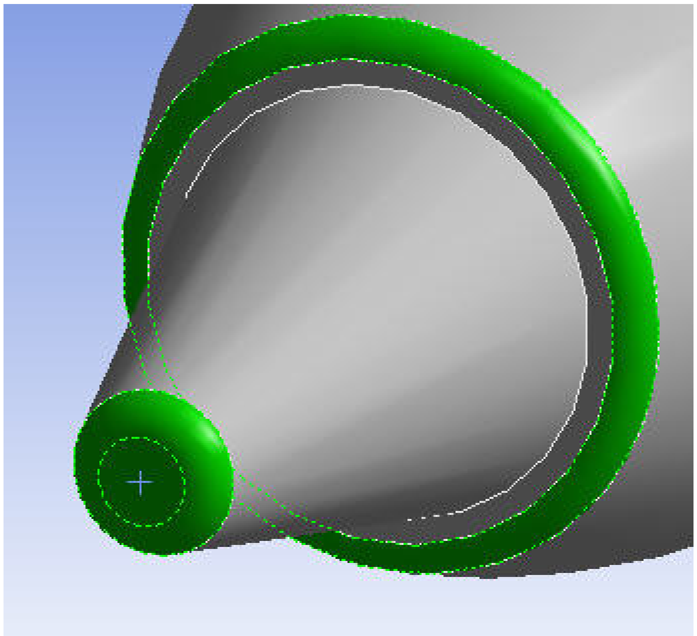
3. Insert a second sizing control. In the Tree Outline, right-click **Mesh** and select **Insert** > **Sizing**.

   a. Reposition the geometry so that it is positioned as shown below.

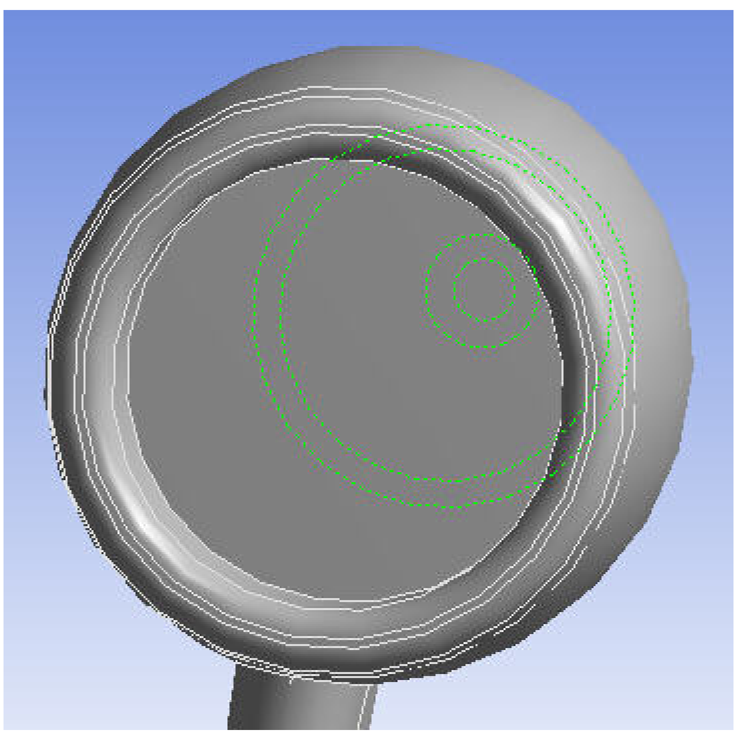
   b. Click **Face** .

   c. On the keyboard, press and hold **Ctrl**.

   d. There are eight faces that you need to select. Click the first three faces, as shown below.

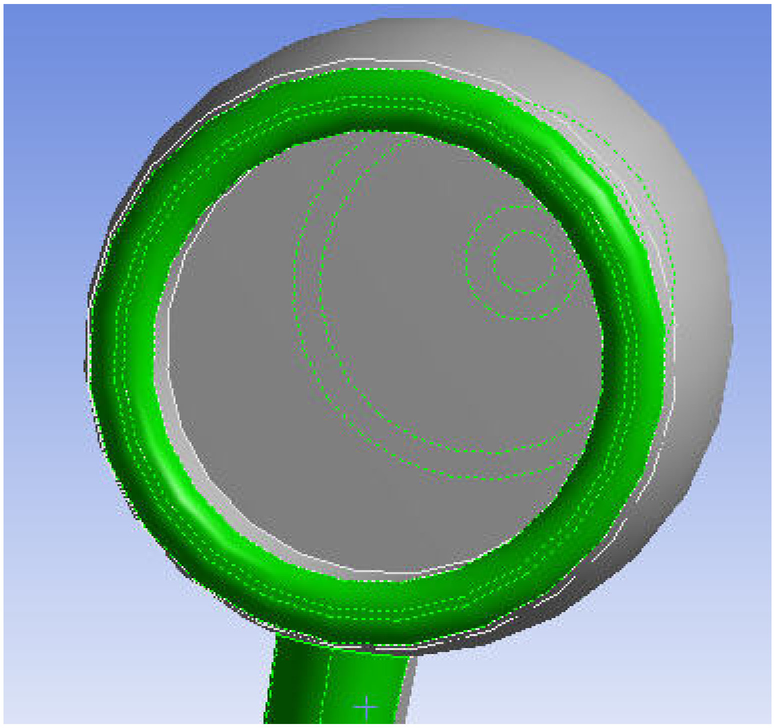
   e. To select the remaining five faces, rotate the geometry so that it is positioned as shown below.

f. Click **Face** .

g. Press and hold **Ctrl**.

h. Select the remaining five faces as shown below.



i. In the Details View, click **Apply** to complete the selection.

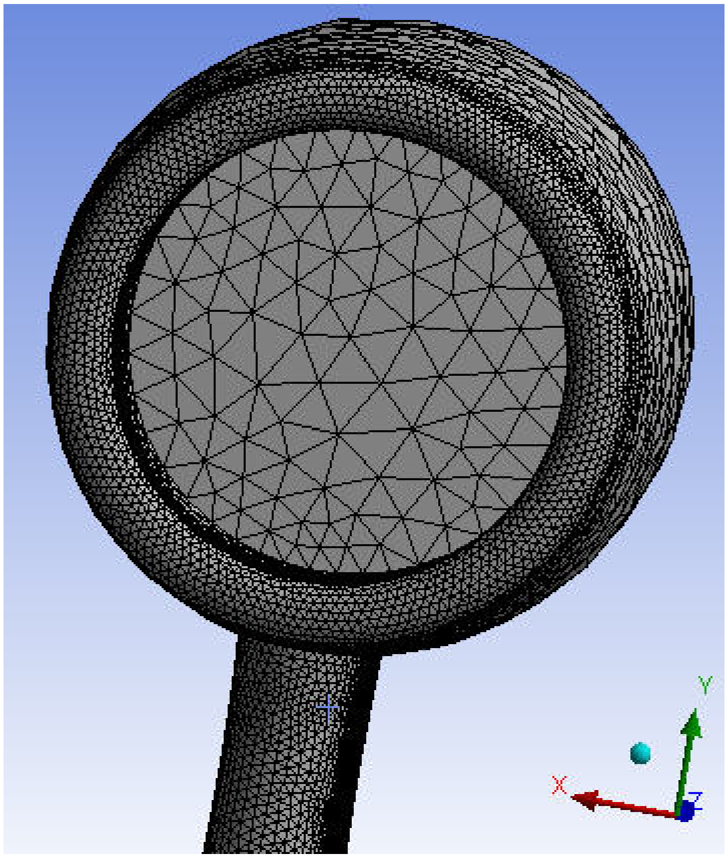
In the Details View, the **Geometry** field should contain the text **8 Faces**.

j. Set **Element Size** to 0.05.

k. Set **Growth Rate** to 1.2.

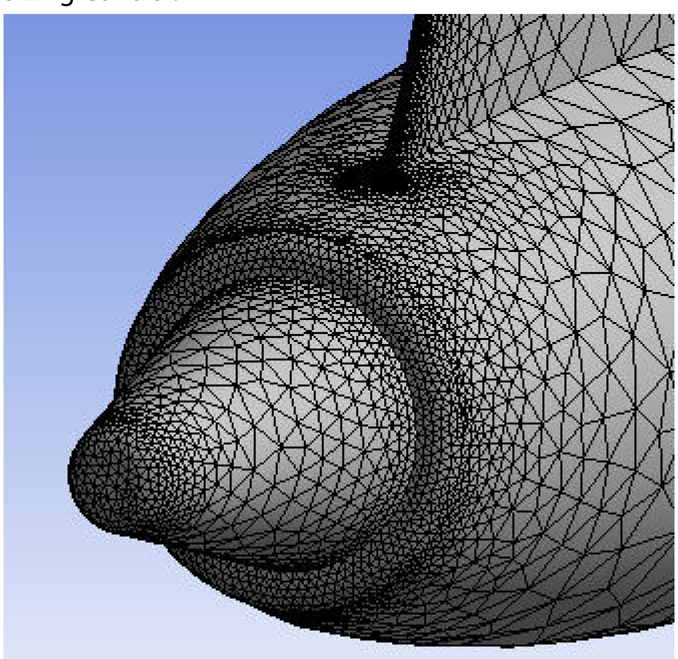
4. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

After a few moments, the mesh appears in the **Geometry** window, as shown below.



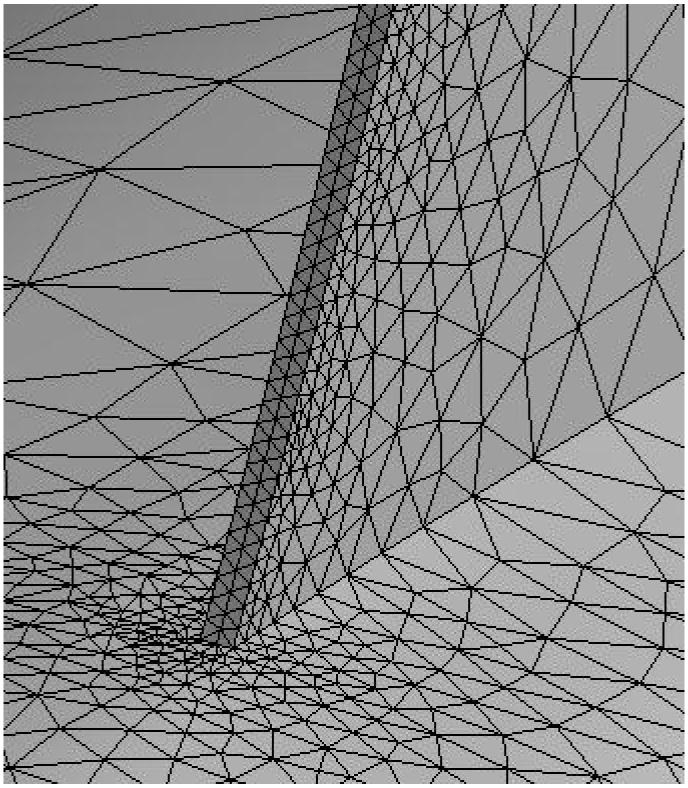
5. Rotate the geometry so that it is positioned as shown below.

   Notice how the element size is relatively uniform on the faces that were selected for the second sizing control.



6. Click **Box Zoom** and zoom the geometry as shown below.

Notice how the element size is relatively uniform on the faces that were selected for the first sizing control.



7. On the triad located in the lower right corner of the **Geometry** window, click the sphere to return the view to isometric.

8. To get ready for the next part of the tutorial, you need to delete the sizing controls.

   In the Tree Outline, right-click **Face Sizing** and select **Delete**. Answer **Yes** when prompted.

   Repeat for **Face Sizing 2**.

Using Proximity-Based Sizing

In this part of the tutorial, **Capture Proximity** is set to **Yes**. This setting lets you specify the minimum number of element layers created in regions that constitute gaps in the model, where a gap is defined in one of two ways:

- The internal volumetric region between two faces
- The area between two opposing boundary edges of a face

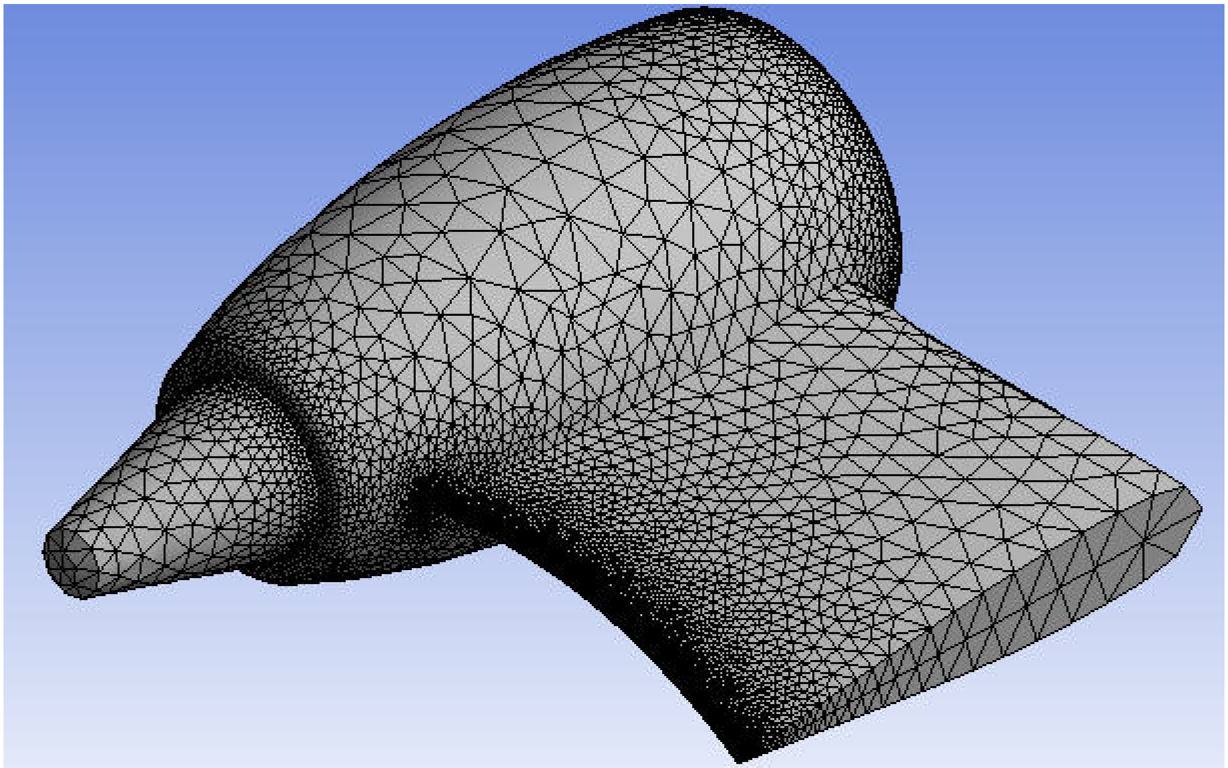
1. In the Details View, set **Capture Proximity** to **Yes**.

2. Set **Num Cells Across Gap** to 2.

3. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

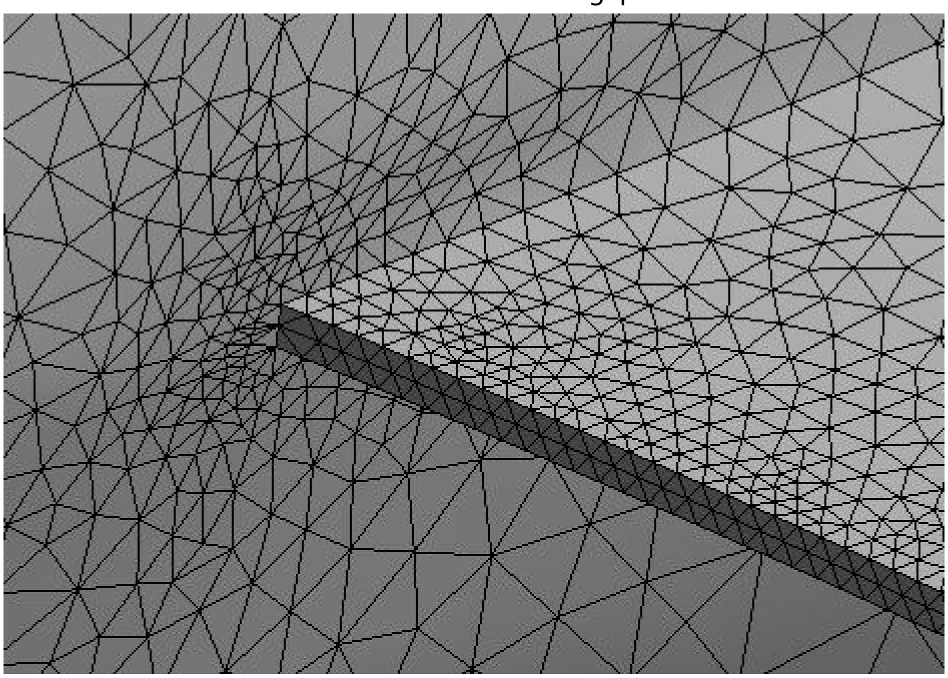
   After a few moments, the mesh appears in the **Geometry** window.

4. Rotate the geometry so that it is positioned as shown below.



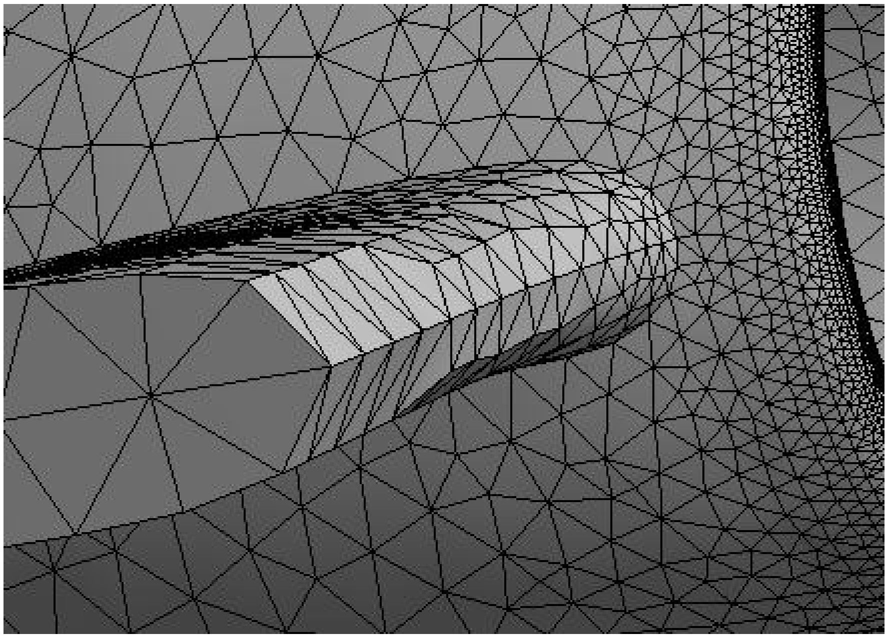
5. Click **Box Zoom** and zoom the geometry as shown below.

   Notice that there are two elements across the gap.



6. Rotate the geometry so that it is positioned as shown below.

   Notice how the curved surface is meshed.



7. Do not reposition the geometry. Continue with the next part of the tutorial.

Using Curvature-Based Sizing

In this part of the tutorial, **Capture Curvature** is set to **Yes**. The mesher examines curvature on edges and faces and computes element sizes on these entities such that the size will not violate the maximum size or the curvature normal angle, which are either computed by the mesher automatically or user-defined.

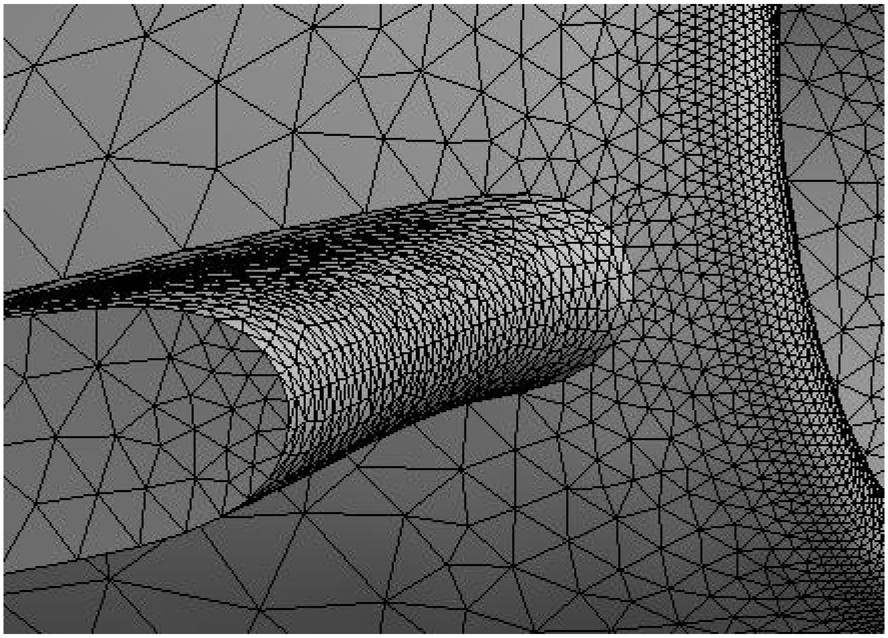
1. In the Details View, set **Capture Curvature** to **Yes**.

2. Set **Capture Proximity** to **No**.

3. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

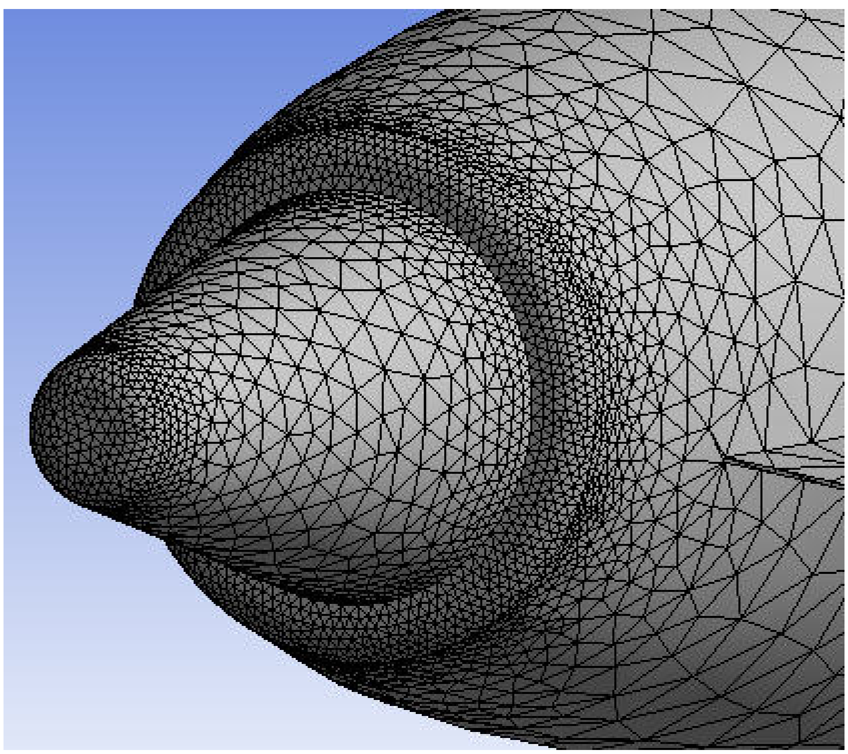
   After a few moments, the mesh appears in the **Geometry** window, as shown below.

   Notice that the curvature is well resolved now.



4. Rotate the geometry so that it is positioned as shown below.

   Notice that the curvature is well resolved here as well.



5. On the triad located in the lower right corner of the **Geometry** window, click the sphere to return the view to isometric.

Using Both Proximity and Curvature-Based Sizing

In this part of the tutorial, both **Capture Curvature** and **Capture Proximity** are set to **Yes**. With these settings you can obtain the combined effect of both the proximity and curvature sizing. You can use all of the proximity and curvature sizing parameters to define the setting.

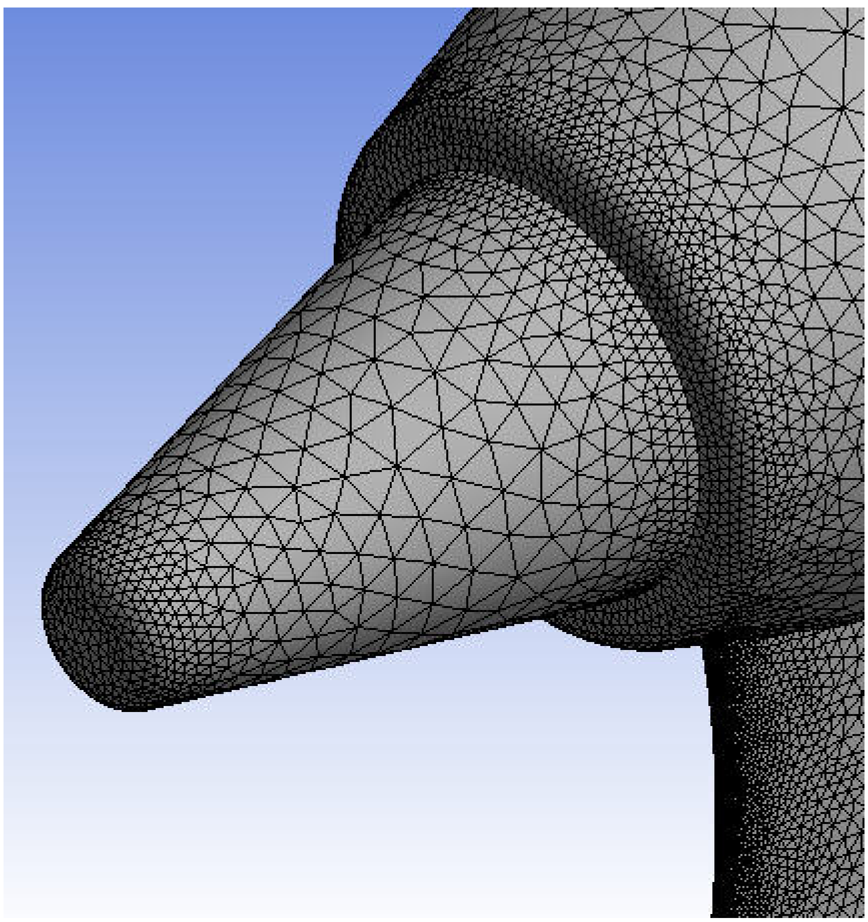
1. In the Details View, set both **Capture Curvature** and **Capture Proximity** to **Yes**.

2. In the Tree Outline, right-click **Mesh** and select **Generate Mesh**.

   After a few moments, the mesh appears in the **Geometry** window.

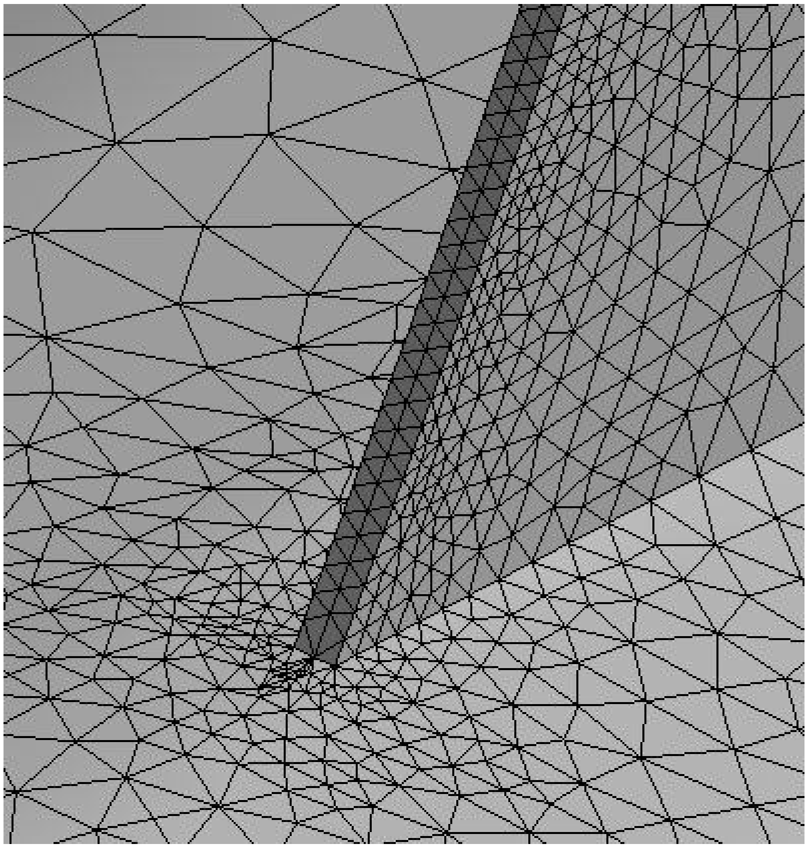
3. Click **Box Zoom**  and zoom the geometry as shown below.

   Notice that the curvature is well resolved.

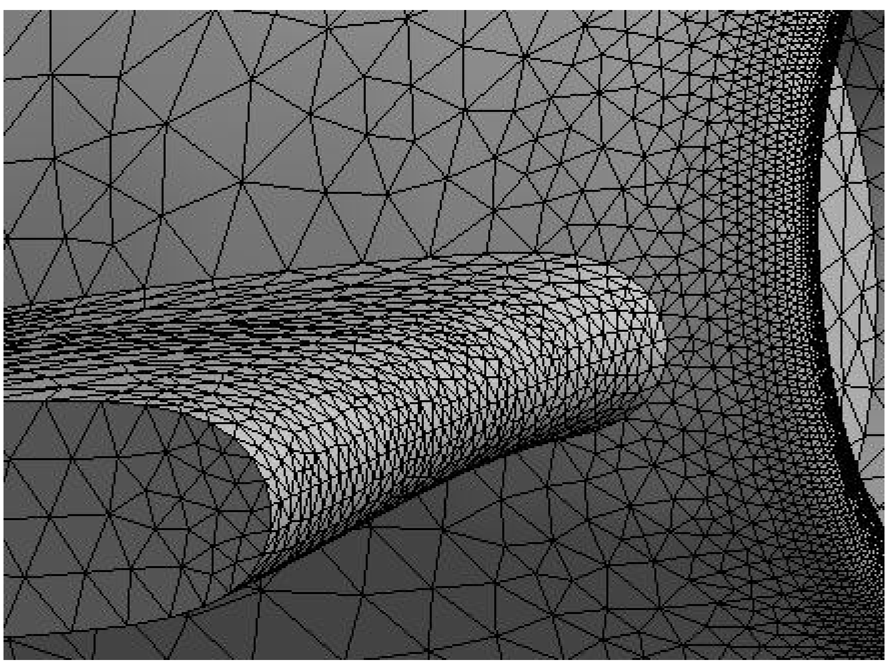
4. Rotate the geometry so that it is positioned as shown below.

   Notice that there are two elements across the gap.



5. Rotate the geometry so that it is positioned as shown below.

   Notice that the curvature is well resolved.



This completes the tutorial. From the Meshing application's main menu, select **File > Save Project** to save the project and then **File > Close Meshing** to return to the Project Schematic.

You can exit Ansys Workbench by selecting **File > Exit** from the main menu.