Mesh

The instructions below show you how to generate a structured mesh ("mapped face mesh" in ANSYS mesher lingo) for the airfoil flow domain. This methodology generates a high quality mesh but can be time-consuming and is not applicable to complex geometries. An alternate meshing strategy covered in the Exercises section can be used to generate a high quality unstructured mesh that is applicable to complex geometries.

**Mapped Face Meshing**

First, we will apply a mapped face meshing control to the geometry. In the Outline window, click on Mesh to bring up the Meshing Toolbar. In the Meshing Toolbar, select **Mesh Control > Mapped Face Meshing**. Making sure the face selection filter is selected , select all four faces by holding down the right mouse button and dragging the mouse of all of the quadrants of the geometry. When all of the faces are highlighted green, in the Details view select **Geometry > Apply**. Next, select

**Edge Sizing**

Next, we will apply edge sizing controls to all of the edges of the mesh. To begin, go to Mesh Control > Sizing. Next, click the edge selection filter . Select the following 4 edges buy holding Ctrl and using the left mouse button:
Once the edges are selected, in the Details View Window select Geometry > Apply. Next, select Type > Number of Divisions. Change the Number of Divisions to 50. Select Behavior > Hard. We also want the mesh to have a bias, so select the first bias type: Bias > ----- - - - , and give the edge sizing a Bias Factor of 150. The Edge sizing should now look like this:

![Edge sizing with first bias type](image1)

Notice that the element sizes get smaller towards the airfoil. This will give us a better resolution around the airfoil where the flow gets more complicated. Create a new edge sizing with the same parameters, but choose the 4 remaining straight edges (see figure below). The number of divisions will still be 50, but now will be selecting a different biasing type by selecting the second Bias option: Bias > - - - - - - . Again, set the Bias Factor to 150.

![Edge sizing with second bias type](image2)

**Edge Bias**

It is important to make sure that the edge divisions to this point are biased towards the center of the mesh; otherwise you may run into some problems later. If your mesh does not match the pictures in the tutorial, make sure to change the parameters of the mesh to make sure that they do: this might mean choosing different edges for the different biasing types than those outlined in this tutorial.

---

*Image 1: Edge sizing with first bias type.*

*Image 2: Edge sizing with second bias type.*
Finally, create a third edge sizing, and select the rounded edges as the geometry. Again, select **Type > Number of Divisions**, and change **Number of Divisions** to 100. Select **Behavior > Hard**. This time, we will not bias the edges.

Now, select **Mesh > Generate** to generate the mesh. It should look like this.

**Named Selections**

Now will assign names to some of the edges to make creating boundary conditions for the mesh easier. Let’s recall the boundary conditions we planned in the Pre-Analysis Step:
The edges highlighted blue are the inlet, the edges highlighted red are the outlet, and the airfoil is highlighted white in the middle. Now we are ready to name the sections. In the Outline window, select geometry - this will make seeing the edges a little easier. Again make sure the edge selection tool is selected. Now, select the two vertical edges on the far right side of the mesh. Right click, and select Create Named Selections. Name the edges outlet. Next, select the edges that correspond to the inlet of the flow as defined by the picture above. Again, right click and select Create Named Selections and this time name the selection inlet. Finally, select the two edges making up the airfoil, and name the selection airfoil.

Go to Step 4: Physics Setup
Go to all FLUENT Learning Modules