

# Flow over an Airfoil - Mesh

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

## Mesh

## Creating an Unstructured Mesh

Watch the following video for a demonstration on how to create an unstructured mesh for the airfoil using "inflation" and "sphere of influence" features in the ANSYS mesher. The advantage of this meshing strategy is that it can be adapted to complex geometries.


Skip to 'Named Selections' section below if you followed the video.

## \*\*ALTERNATIVE APPROACH\*\*


**NOTE:** You can either create an unstructured mesh by following the above video or a structured mesh by following the steps below.

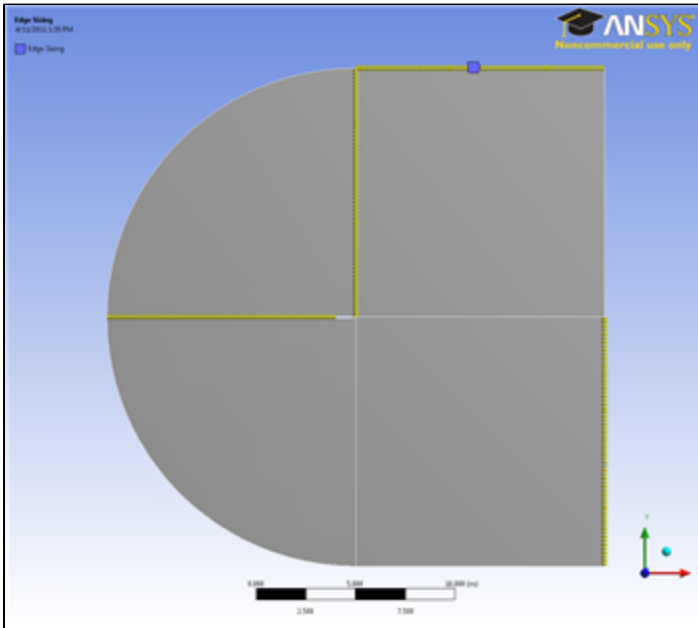
### Create a Structured Mesh

#### 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 over all of the quadrants of the geometry. When all of the faces are highlighted green, in the *Details view* Window select **Geometry > Apply**. Next, select

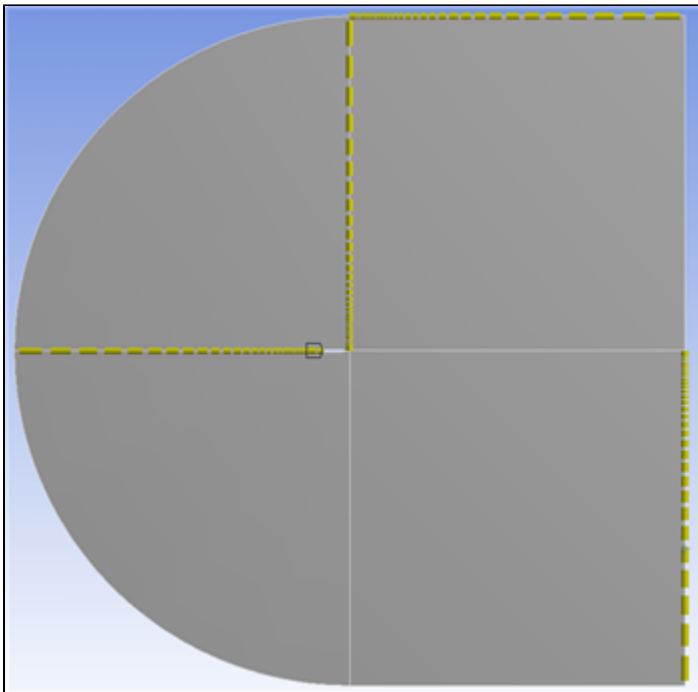
#### Edge Sizing

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 by holding Ctrl and using the left mouse button:

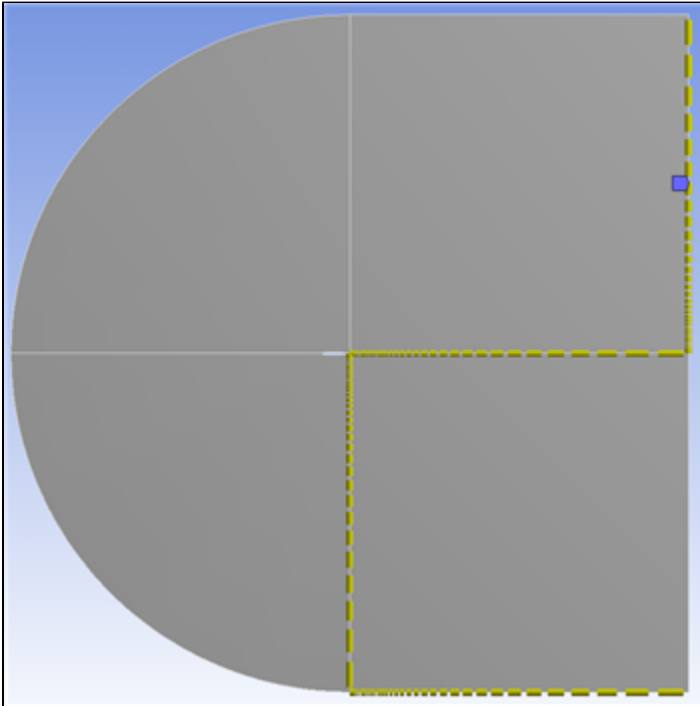


[Click here to enlarge](#)

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:



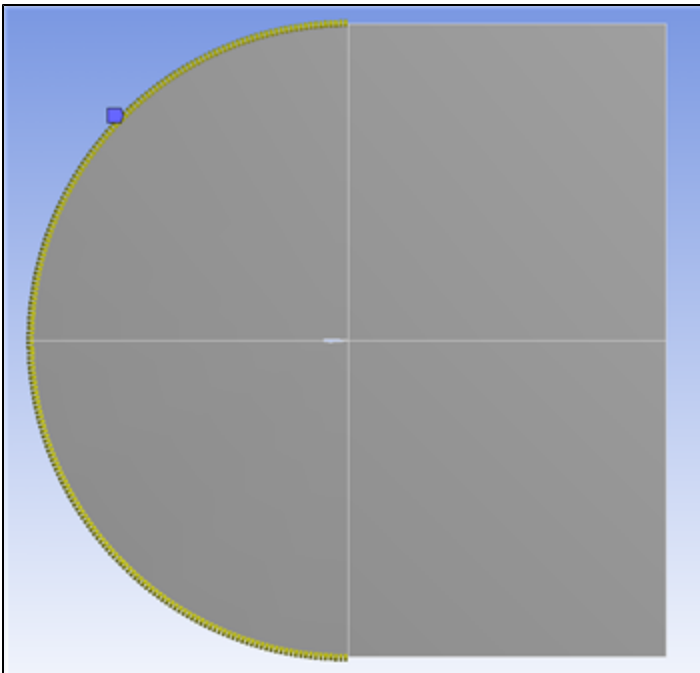
Notice that the element sizes get smaller towards the airfoil. This will give us a better resolution around the airfoil where the flow gets more complicated. Create a new edge sizing with the same parameters, but choose the 4 remaining straight edges (see figure below). The number of divisions will still 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



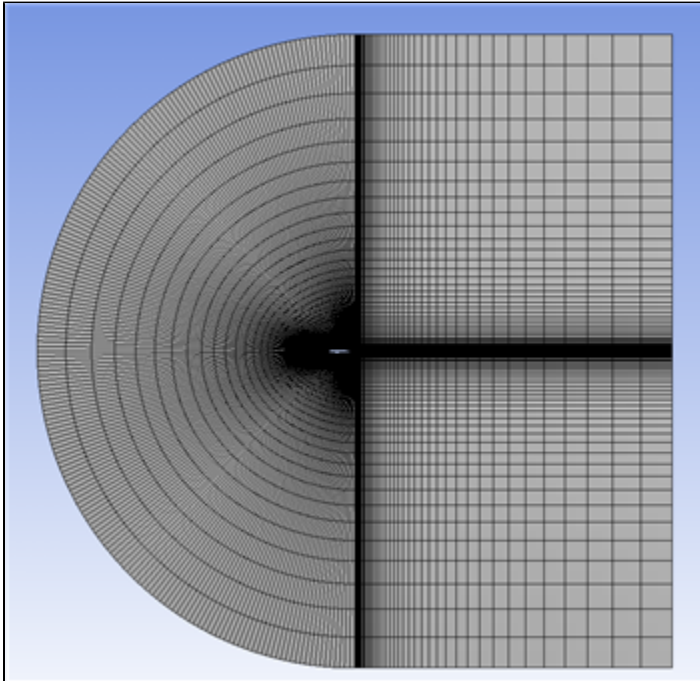
#### Edge Bias

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.

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.



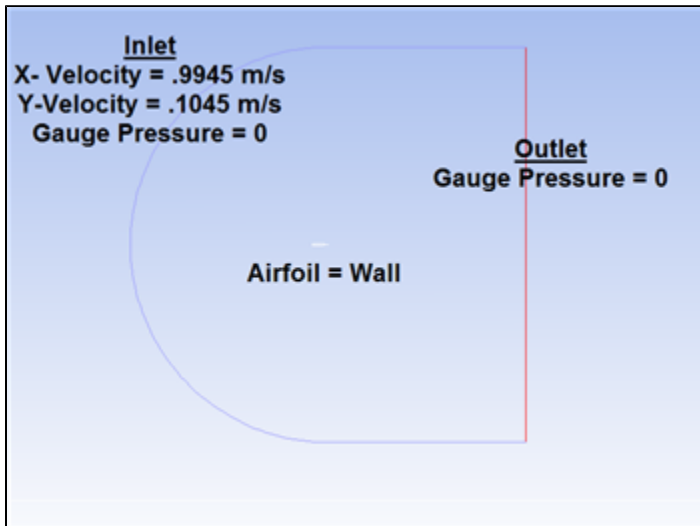
Now, select **Mesh > Generate** to generate the mesh. It should look like this.




[Click here to enlarge](#)

## 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](#)