

# Flow over an Airfoil - Unstructured 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](#)

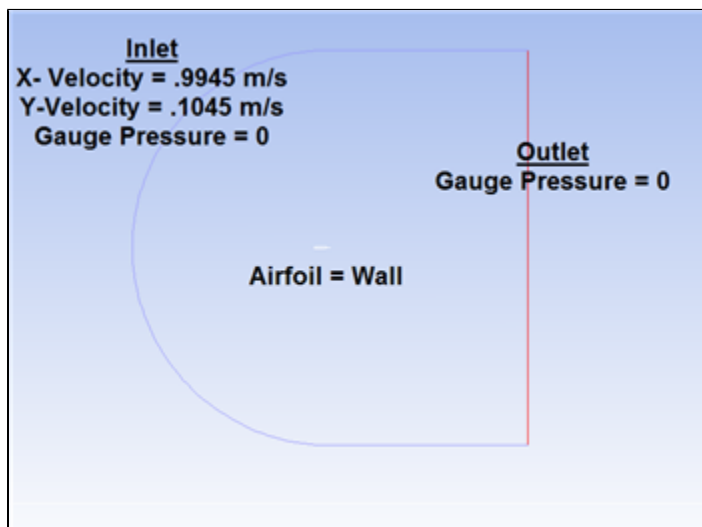
## Mesh

## 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.

### 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](#)