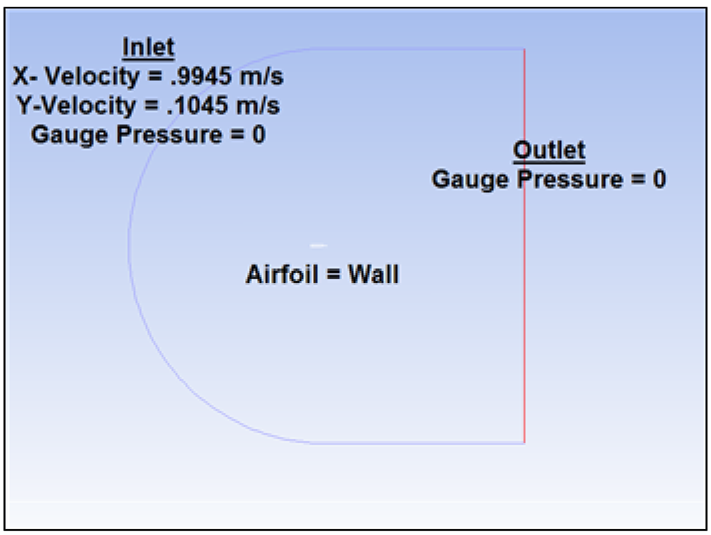
Flow over an Airfoil - Pre-Analysis & Start-Up

Author: Benjamin Mullen, Cornell University

Pre-Analysis & Start-Up

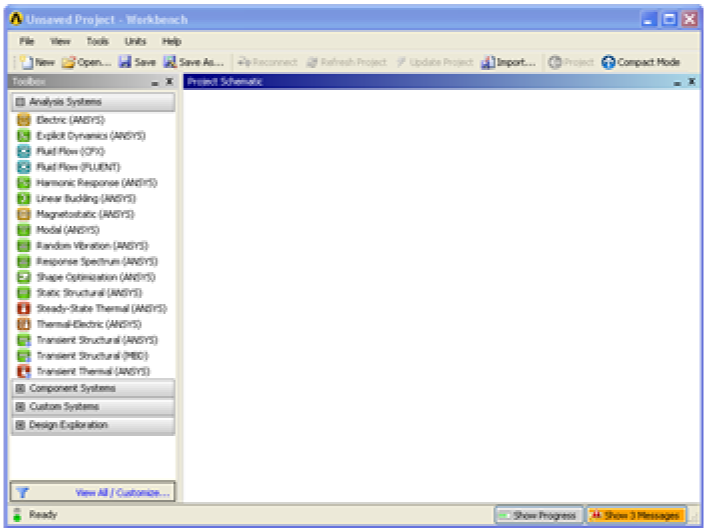
Boundary Conditions

One of the simple things we can think about before we set up the simulation is begin planning the boundary conditions of the set up. One of the popular meshes for simulating an airfoil in a stream is a C-Mesh, and that is what we will be using. At the inlet of the system, we will define the velocity as entering at a 6 degree angle of attack (as per the problem statement), and at a total magnitude of 1. We will also define the gauge pressure at the inlet to be 0. As for the outlet, the only thing we can assume is that the gauge pressure is 0. As for the airfoil itself, we will treat it like a wall. Together, these boundary conditions form the picture below:

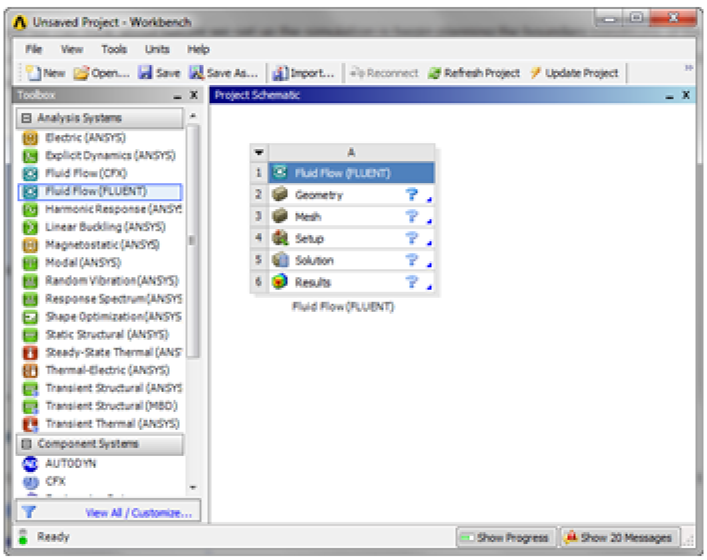


Open ANSYS Workbench

Now that we have the pre-calculations, we are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen as seen below



To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We will be using FLUENT to complete the simulation. Load the ***Fluid Flow (FLUENT)*** box by dragging and dropping it into the Project Schematic.



Once you have loaded FLUENT into the project schematic, you are ready to create the geometry for the simulation.

Go to Step 2: Geometry

Go to all FLUENT Learning Modules