Flow over an Airfoil - Physics Setup

Author: Benjamin Mullen, Cornell University

Physics Setup

Launch the Solver

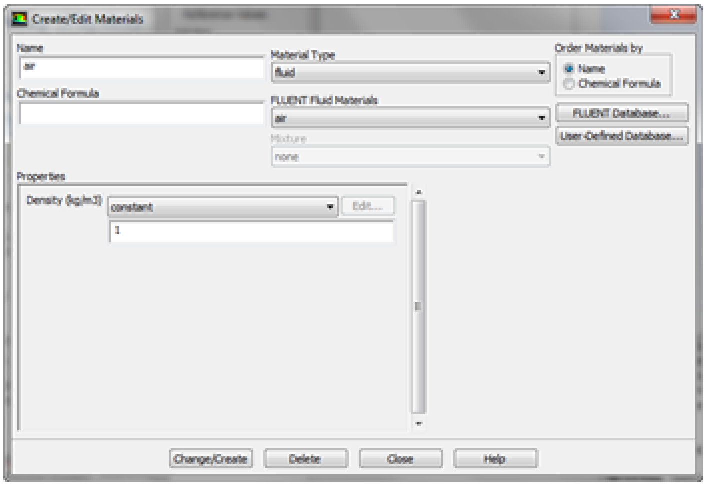
In this step, we will open fluent and define the boundary conditions of the problem. If you haven't already, close the meshing window to return to the *Project Outline* window. Now, click Update Project. This will load the mesh into FLUENT. Now, double click ***Setup***. The *Fluent Launcher* Window should open. Check the box marked ***Double Precision***. To make the solver run a little quicker, under *Processing Options* we will select ***Parallel*** and change the ***Number of Processes*** to 2. This will allow users with a double core processor to utilize both.

Select the Solver

Click ***OK*** to launch Fluent. The first thing we will do once Fluent launches is define the solver we are going to use. Select ***Problem Setup > General***. Under *Solver*, select ***Density-Based***.

Models and Materials

Next, we will define the model we are going to use. We do this by going ***Problem Setup > Models > Viscous-Laminar***. Then press ***Edit...*** This will open the *Viscous Model* Menu Window. Select ***Inviscid*** and press ***OK***. Now, we will specify characteristics of the fluid. Because we specified the fluid as inviscid, we will only have to define the density of the fluid. To make matters even simpler, we are only looking for non-dimensionalized values like pressure coefficient, so we will define the density of our fluid to be 1 kg/m^3. To define the density, click ***Problem Setup > Materials > (double click) Air***. This will launch the *Create/Edit Materials* window.

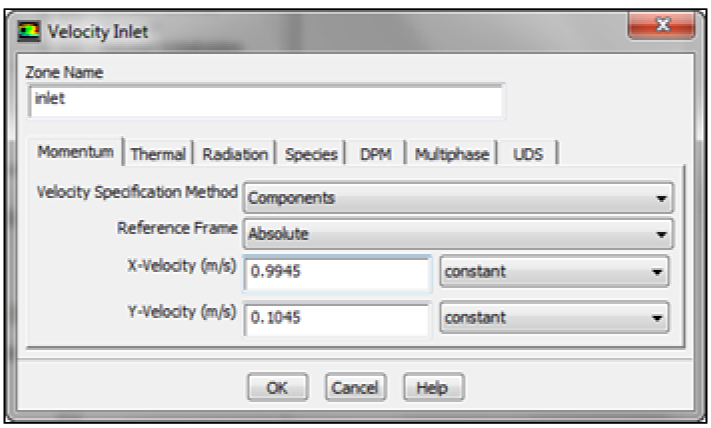


Under *Properties*, ensure that density is set to ***Constant*** and enter `1 kg/m^3` as the density. Click ***Change/Create*** to set the density.

Boundary Conditions

Inlet

Now that the fluid has been described, we are ready to set the boundary conditions of the simulation. Bring up the boundary conditions menu by selecting ***Problem Setup > Boundary Conditions***. In the *Boundary Conditions* window, look under *Zones*. First, let's set the boundary conditions for the inlet. Select ***Inlet*** to see the details of the boundary condition. The boundary condition type should have defaulted to ***velocity-inlet***: If it didn't, select it. Now, click ***Edit*** to bring up the *Velocity-Inlet* Window. We need to specify the magnitude and direction of the velocity. Select ***Velocity Specification Method > Components***. Remember, we want the flow to enter the inlet at an angle of 6 degrees since the angle of attack of the airfoil is 6 degrees; thus, the x velocity will be $\cos(6)$, and the y velocity will be $\sin(6)$. Specify ***X-Velocity*** as `0.9945 m/s` and ***Y-Velocity*** as `0.1045 m/s`. When you have finished specifying the velocity as entering the inlet at 6 degrees (the same thing as having an angle of attack of 6 degrees), press ***OK***



Outlet

In the *Boundary Conditions* window, look under *Zones*. Select ***Outlet*** to see the details of the boundary condition. The boundary condition type should have defaulted to ***pressure-outlet***: if it didn't, select it. Click ***Edit***, and ensure that the ***Gauge Pressure*** is defaulted to `0`. If it is, you may close this window.

Airfoil

In the *Boundary Conditions* window, look under *Zones* and select ***airfoil***. Select ***Type > Wall*** if it hasn't been defaulted.

Reference Values

The final thing to do before we move on to solution is to acknowledge the reference values. Go to ***Problem Setup > Reference Values***. In the *Reference Values* Window, select ***Compute From > Inlet***. Check the reference values that appear to make sure they are as we have already set them.

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules