# Flow over an Airfoil - Physics Setup

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

## **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 Outline window. Now, click 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 **N umber of Processes** to 2. This will allow users with a double core processor to utilize both.

FLUENT Launcher (Setting Edit Only)	
ANSYS	FLUENT Launcher
Dimension 3D 2D Display Options Display Mesh After Reading Combed Graphics Windows Workbench Color Scheme Do not show this panel again	Options  Double Precision  Use Job Scheduler  Use Remote Linux Nodes  Processing Options Serial  Parallel (Local Machine) Number of Processes 2  Show More >>
<u>Dk</u>	ncel <u>H</u> elp •

## 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.

me	Material Type	Order Materials by
ir 🦷	fluid	<ul> <li>Name</li> </ul>
remical Formula	RUENT Red Materials	Chemical Formula
	air	FLUENT Database
	Moture	User-Defined Database.
	none	w l
and the factor of the	A	
Density (kg/m3) constant	• Edt	
1		
	1	

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 *P* roblem Setup > Boundary Conditions. In the Boundary Conditions window, look under Zones. First, let's set the boundary conditions for the inlet. Select *I* nlet 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 beUnable to find DVI conversion log file., and the y velocity will beUnable to find DVI conversion log file. Specify 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

Zone Name			
inlet			
Momentum Thermal Radia	tion Species C	PM   Multiphase   UDS	
Velocity Specification Method	Components 👻		
Reference Frame	Absolute		•
X-Velocity (m/s)	0.9945	constant	•
Y-Velocity (m/s)	0.1045	constant	•

#### 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