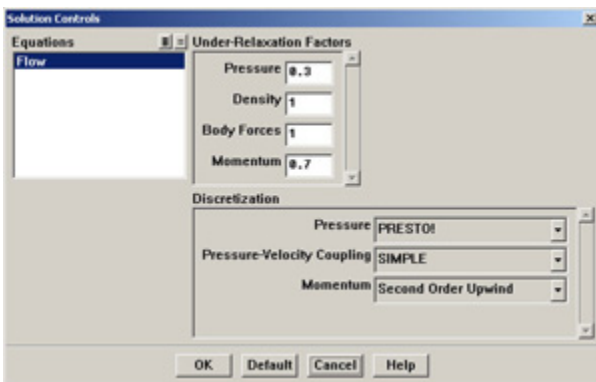# FLUENT - Flow over an Airfoil- Step 5

Step 5: Solve!

Solve > Control > Solution

Take a look at the options available.

Under *Discretization*, set *Pressure* to *PRESTO!* and *Momentum* to *Second-Order Upwind*.
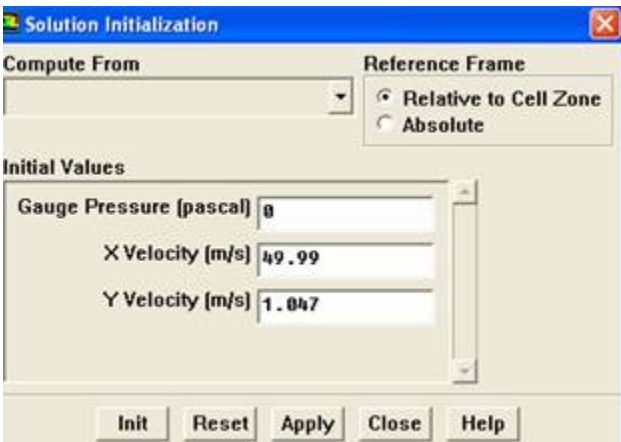

(click picture for larger image)

Click *OK*.

Solve > Initialize > Initialize...

As you may recall from the previous tutorials, this is where we set the initial guess values (the base case) for the iterative solution. Once again, we'll set these values to be equal to those at the inlet (to review why we did this look back to the tutorial about CFG programs) . Select *farfield1* under *Compute From*.

Click *Init*.

Solve > Monitors > Residual...

Now we will set the residual values (the criteria for a good enough solution). Once again, we'll set this value to 1e-06.
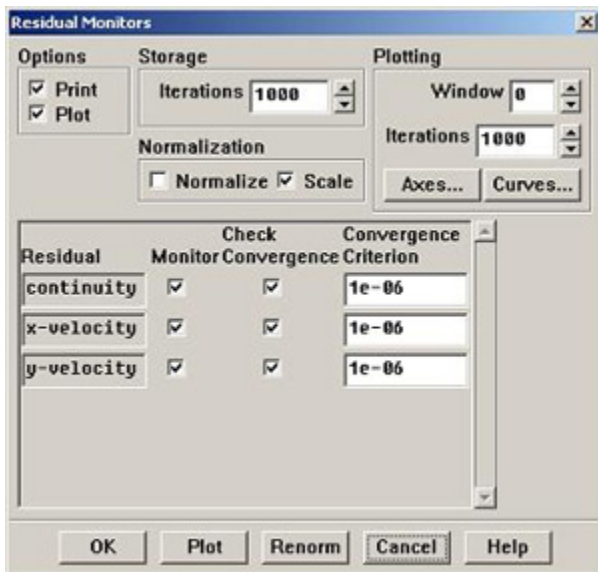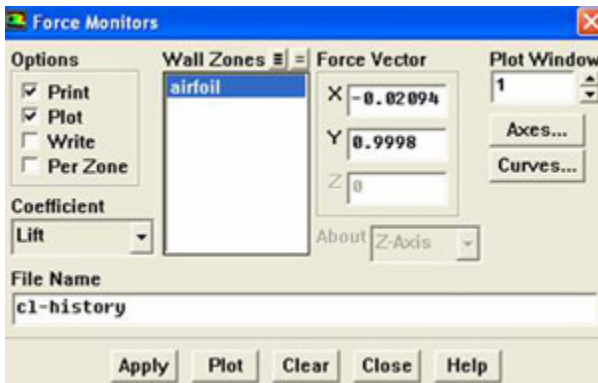
(click picture for larger image)

Click **OK**.

Solve > Monitors > Force...

Under *Coefficient*, choose *Lift*. Under *Options*, select *Print* and *Plot*. Then, Choose *airfoil* under *Wall Zones*.

Lastly, set the *Force Vector* components for the lift. The lift is the force perpendicular to the direction of the freestream. So to get the lift coefficient, set *X* to -sin(1.2°)=-020942 and *Y* to cos(1.2°)=0.9998.
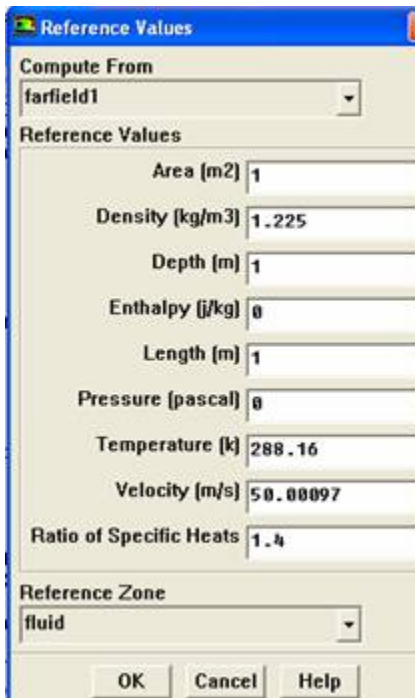

(click picture for larger image)

Click **Apply** for these changes to take effect.

Similarly, set the *Force Monitor* options for the *Drag* force. The drag is defined as the force component in the direction of the freestream. So under *Force Vector*, set *X* to cos(1.2°)=0.9998 and *Y* to sin(1.2°)=0.020942 Turn on only Print for it.

Report > Reference Values

Now, set the reference values to set the base cases for our iteration. Select *farfield1* under *Compute From*.

Click **OK**.

Note that the reference pressure is zero, indicating that we are measuring gage pressure.

Main Menu > File > Write > Case...

Save the case file before you start the iterations.

Solve > Iterate

Make note of your findings, make sure you include data such as;

What does the convergence plot look like?

How many iterations does it take to converge?

How does the Lift coefficient compared with the experimental data?

Main Menu > File > Write > Case & Data...

Save case and data after you have obtained a converged solution.

Go to Step 6: Analyze Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules