FLUENT - Flow Past a Cylinder - Step 5

Solve

Solve > Control > Solution

Under **Discretization**, set **Momentum** to Second-Order Upwind.

Solve > Initialize > Initialize...

Select inlet under **Compute From**. We'll set these values to be equal to those at the inlet.

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.

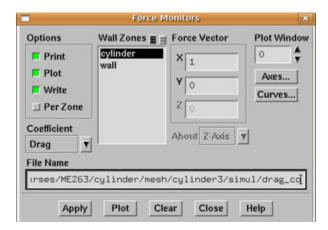Select **Print** and **Plot** under Options.

Click **OK**.

Solve > Monitors > Force...

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

Set the **Force Vector** components for the drag. The drag is the force in the direction of the freestream. So to get the drag coefficient, set **X** to 1 and **Y** to 0.

Record the histories of Cd. Under **Options**, select **Write**. Fill in the name in the box under **File Name**, then the text file containing drag coefficients at each iteration will be stored in the file.

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

Similarly, set the Force Monitor options for the Lift force. The lift is defined as the force component perpendicular to the direction of the freestream. So under **Force Vector**, set **X** to 0 and **Y** to 1.

Click **Apply**.

Report > Reference Values

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

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?

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