Unsteady Flow Past a Cylinder - Numerical Solution

Authors: John Singleton and Rajesh Bhaskaran, Cornell University

Numerical Solution

Convergence Criterion: Turn off Drag, Turn on Lift

Solution > Monitors > Drag > Edit.... Then uncheck Print to Console and uncheck Plot. Click ok.

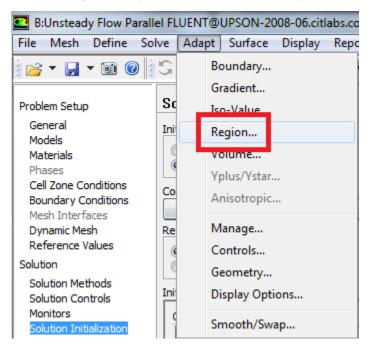
Solution > Monitors > Lift > Edit.... Then check Print to Console, Plot and Write. Click ok. The last option writes the lift coefficient data to a file that is buried in one of the subfolders that FLUENT creates in the working folder. You'll have to dig around to find it.

If you do not see Drag or Lift under Residuals, Statistics and Force Monitors, then click on *Create* and select *Lift* (since we only want to monitor Lift right now). Then check *Print to Console*, *Plot* and *Write*, highlight *cylinderwall* and click *Ok*.

Solution Initialization

First, let's set the initial condition in all of the cells to a velocity of 1 m/s in the X-direction. Solution > Solution Initialization. Set Compute From to farfiel d1.. Click Initialize.

Next, we'll change the velocity in some of the cells to more quickly reach a sinusoidal variation of the lift coefficient. Adapt > Region



Then set *X Min* to 0.5 m, set *X Max* to 32 m, set *Y Min* to 0 m, and set *Y Max* to 32m. (Note: If you instead "patch" the region 0.5 < x < 32 and -5 < y < 5, the periodic vortex shedding starts much quicker; so we recommend that).

Region Adaption		
Options	Input Coordinates	
Inside	X Min (m)	X Max (m)
Outside	.5	32
Shapes	Y Min (m)	Y Max (m)
Quad	0	32
Circle	Z Min (m)	Z Max (m)
	0	0
Manage		
Controls	0	
Collect Deinte with Maure		
Select Points with Mouse		
Adapt Mark Close Help		

Click Mark then click Close. This will select the cells bounded by these four points, so we can change the initial condition in them.

Next, click Patch.

Solution Initialization	
Initialization Methods O Hybrid Initialization Standard Initialization	
Compute from	_
	•
Reference Frame	
 Relative to Cell Zone Absolute 	
Initial Values	
Gauge Pressure (pascal)	.
0	
X Velocity (m/s)	
1	
Y Velocity (m/s)	
0	
	Ŧ
Initialize Reset Patch	

Complete the patching menu as shown below. This will change the initial Y component of velocity in the selected region from 0 to 0.2 m/s.

Patch			×
Reference Frame Relative to Cell Zone Absolute Variable Pressure X Velocity Y Velocity	Value (m/s) 0.2 Use Field Function Field Function	Zones to Patch [0/1] flow_domain Registers to Patch [1/1] hexahedron-r0	
	Patch Close Help		

Click Patch, then click close.

View the Patch

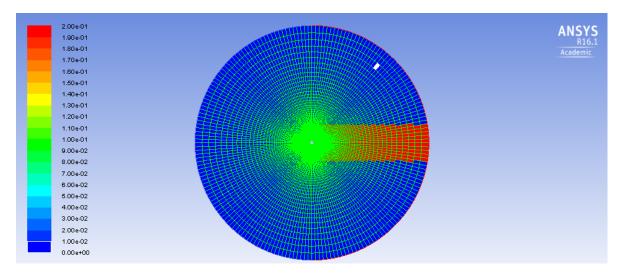
Under Results go to Graphics and go to Contours > Setup.

Complete the set up menu as shown below:

Contours	×	
Options Image: The second s	Contours of Velocity Y Velocity Min O O O Surfaces	
Levels Setup 20 1	cylinderwall farfield1 farfield2 interior-flow_domain surface_body	
Surface Name Pattern Match	New Surface ▼ Surface Types axis clip-surf exhaust-fan fan	
Display Compute Close Help		

Then click Display.

You should see the following image in the graphics window The patched region is in red, where we have changed the Y velocity to be .2 m/s to quickly induce the vortex shedding. (Note the following image is inconsistent with the settings above which perturb the initial guess only above the x axis. Sorry.)

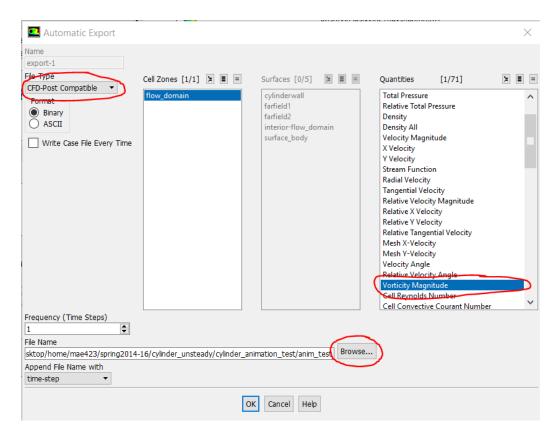


Setting Up Data Export to Create Animation

We would like to create an animation of the vorticity magnitude after the solution has been calculated. To do so, we will need to export data from FLUENT to CFD-Post, the post processor used to view results. To do so, go to Solution > Calculation Activities > Automatic Export > Create > Solution Data Export....

Calculation Activities		
Autosave Every (Time Steps)		
-	Edit	
Automatic Export		
Create Edit Delete		

Next, change File Type to CFD-Post compatible, as this is the program we will use for post processing. Then, select Vorticity Magnitude from the list of variables on the right, so we can make an animation of contours of vorticity. Finally, click Browse, and choose a convenient file location on your local drive to save the data files. **Do NOT choose a folder on a flash drive as that has slow data input/output and can also cause Fluent to crash**. Make note of this location for later use.



Advance Solution in Time

Solution > Run Calculation. Set Time Step Size to 0.2 seconds and set the Number Of Time Steps to 400.

Run Calculation				
Check Case	Preview Mesh Motion			
Time Stepping Method Fixed Settings	Time Step Size (s) 0.2 Number of Time Steps 400 ▼			
Options Extrapolate Variables Data Sampling for Time Statistics Sampling Interval Sampling Options				
Max Iterations/Time Step 20	Reporting Interval			
Profile Update Interval 1 Data File Quantities	Acoustic Signals			
Calculate				

Now, click *Calculate*. (You may have to hit *Calculate* twice.) Now, have a cup of coffee. When complete, close FLUENT to return to the main project window.

Save Project

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules