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***.

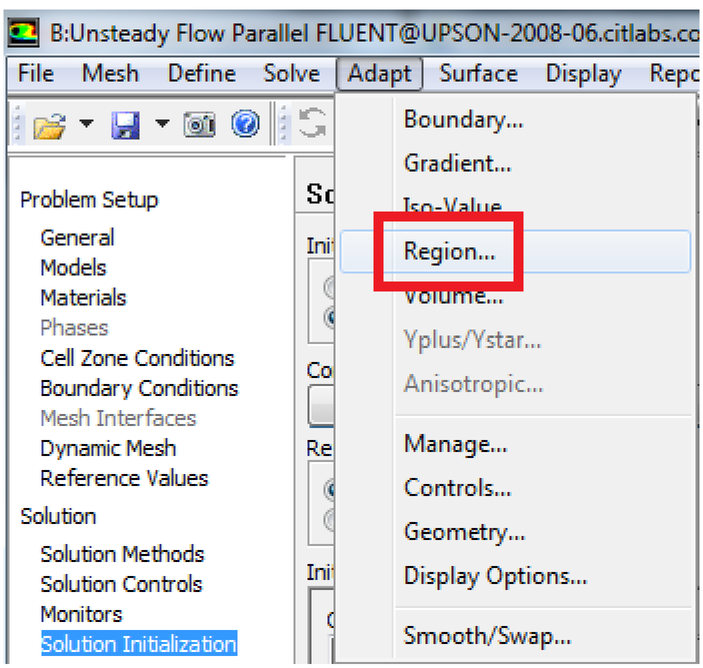
Solution > Monitors > Lift > Edit.... Then check ***Print to Console***, ***Plot*** and ***Write***. Click ***ok***. The last option writes the lift coefficient data to a file that is buried in one of the subfolders that FLUENT creates in the working folder. You'll have to dig around to find it.

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***.

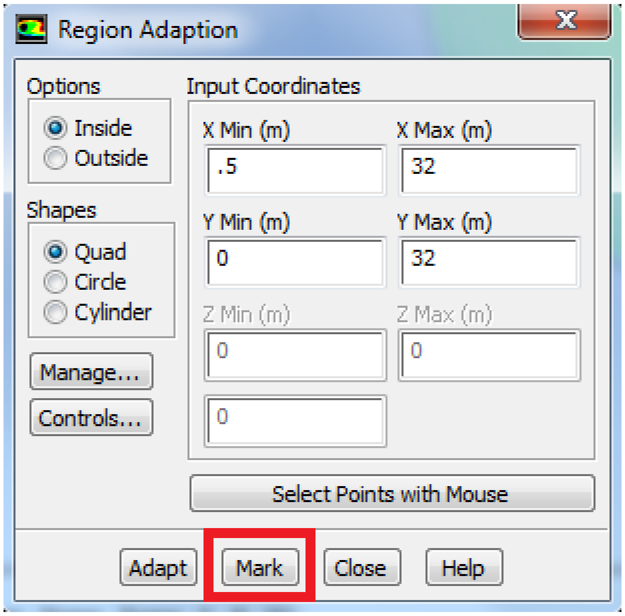
Solution Initialization

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 ***farfield1.***. 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...***.

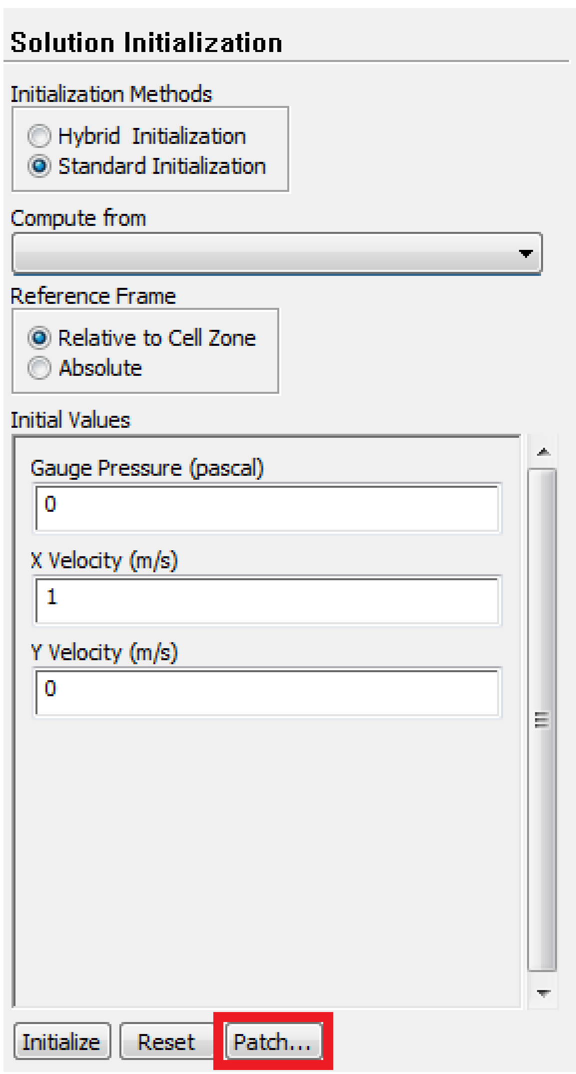


Then set ***X Min*** to 0.5 m, set ***X Max*** to 32 m, set ***Y Min*** to 0 m, and set ***Y Max*** to 32m. (Note: If you instead "patch" the region 0.5 < x < 32 and -5 < y < 5, the periodic vortex shedding starts much quicker; so we recommend that).

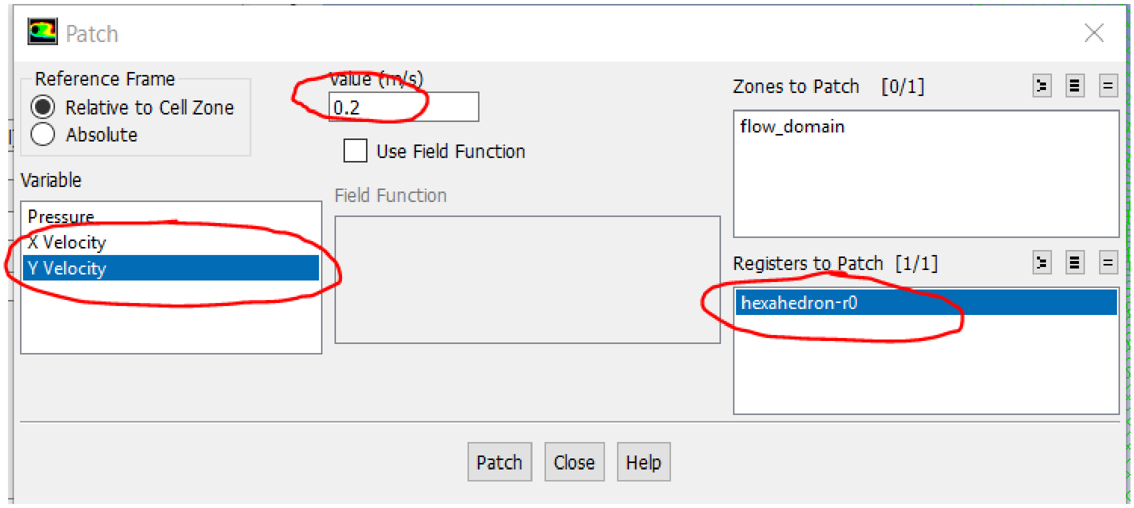


Click ***Mark*** then click ***Close***. This will select the cells bounded by these four points, so we can change the initial condition in them.

Next, click ***Patch***.



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.

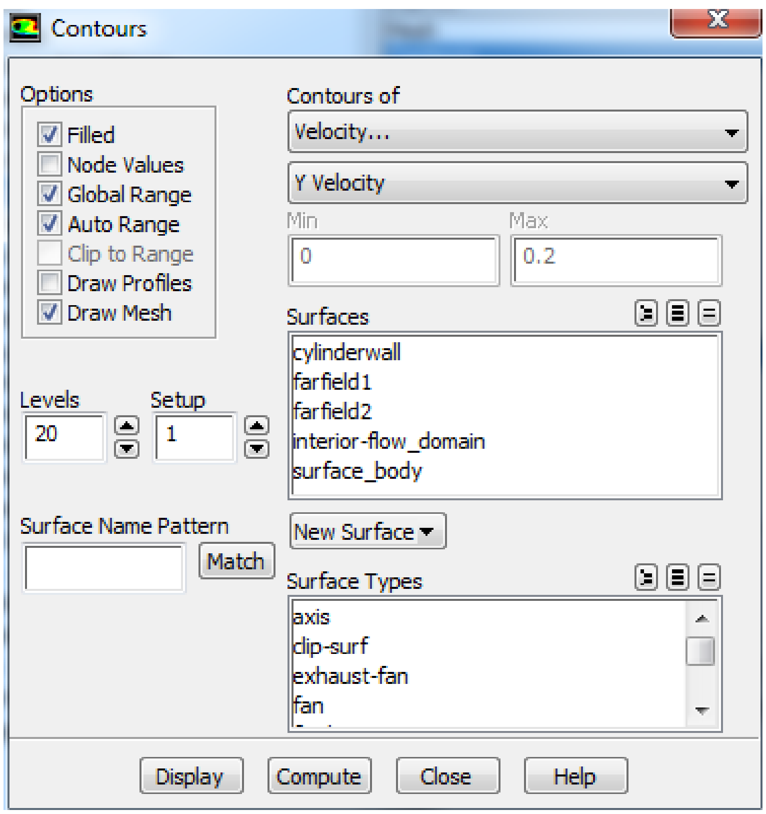


Click ***Patch***,then click ***close***.

View the Patch

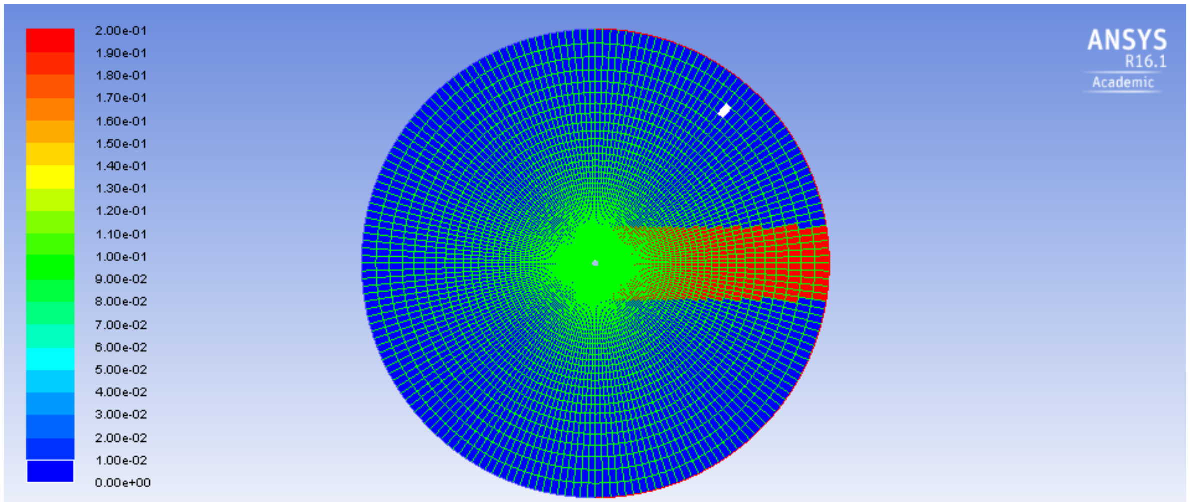
Under ***Results*** go to ***Graphics*** and go to ***Contours > Setup.***

Complete the set up menu as shown below:



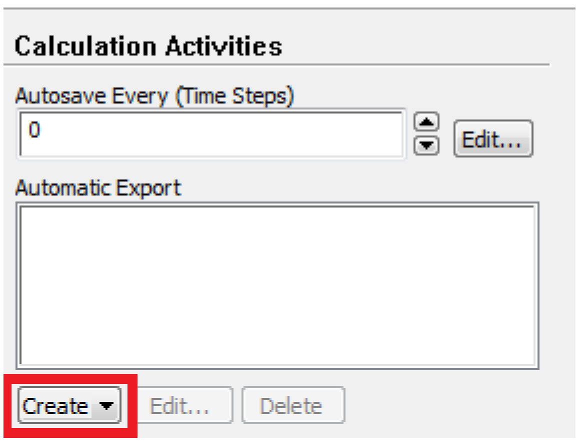
Then click ***Display.***

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.)

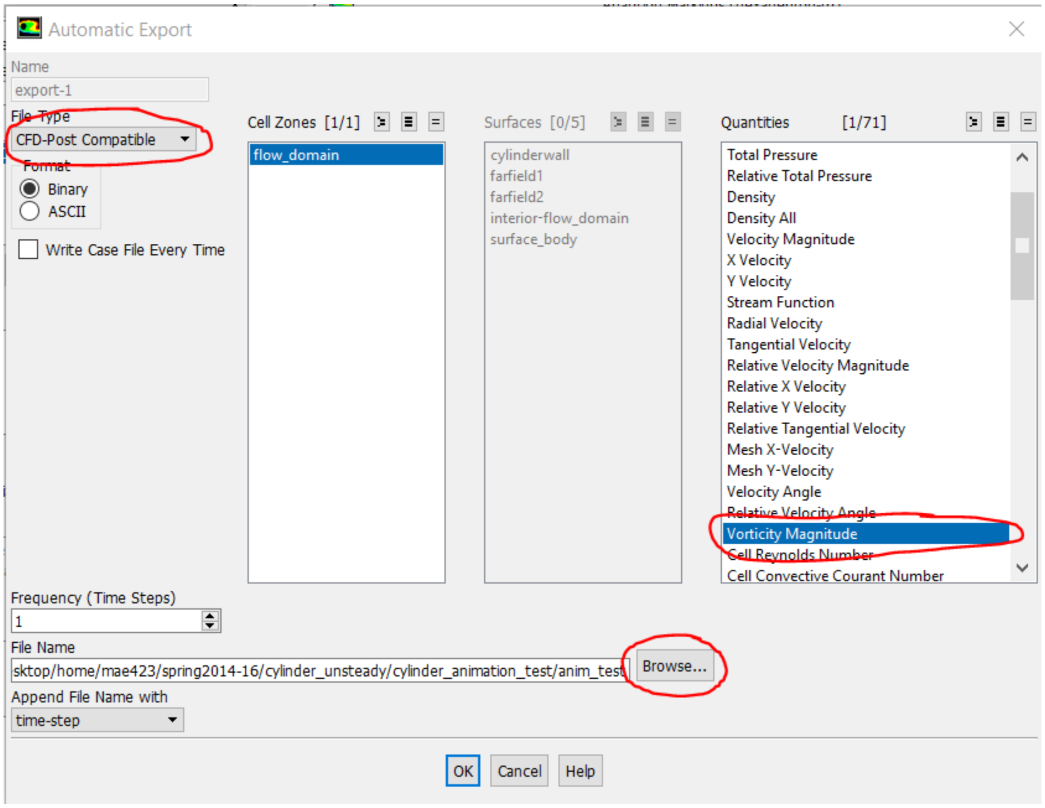


Setting Up Data Export to Create Animation

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...**.

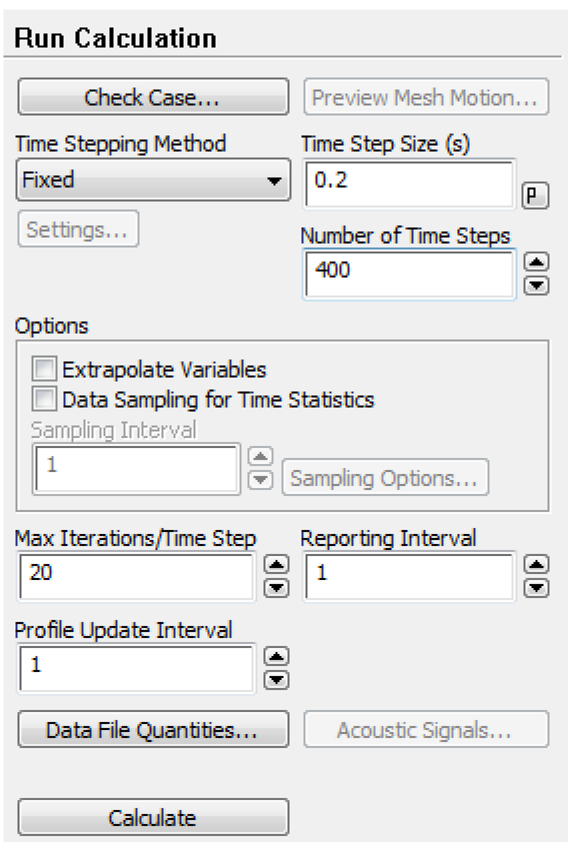


Next, change File Type to CFD-Post compatible, as this is the program we will use for post processing. Then, select Vorticity Magnitude from the list of variables on the right, so we can make an animation of contours of vorticity. Finally, click Browse, and choose a convenient file location on your local 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.



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