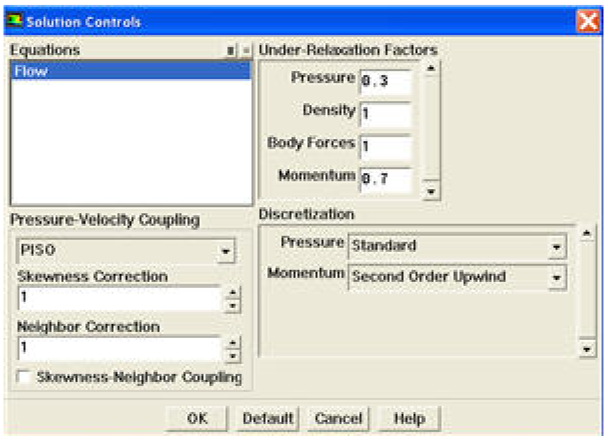
FLUENT - Unsteady Flow Past a Cylinder - Step 5

Step 5: Solve!

Set Solution Controls

Main Menu > Solve > Controls > Solution...



Select ***PISO*** from the ***Pressure-Velocity Coupling*** drop-down list.

> PISO allows the use of higher time step size without affecting the stability of the solution. Hence it is recommended pressure-velocity coupling for solving transient applications.

Uncheck ***Skewnes-Neighbor Coupling***.

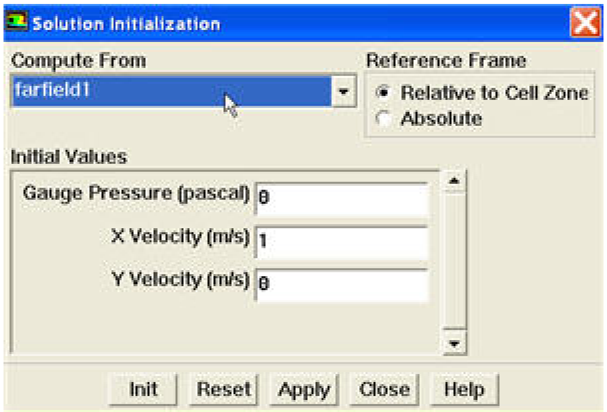
Select ***Second Order Upwind*** from the Momentum drop-down list in the ***Discretization*** group box. Click ***OK*** to close the ***Solution Controls*** panel.

Set Initial Guess

Initialize the flow field to the values at the inlet:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* menu that comes up, choose ***farfield1*** under ***Compute From***. The ***X Velocity*** for all cells will be set to 1 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the ***farfield1*** boundary condition.



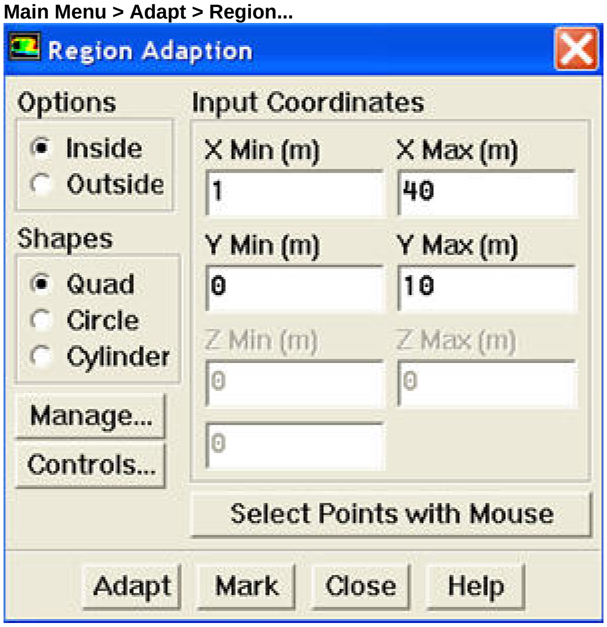
Click ***Init***. This completes the initialization. ***Close*** the window.

Patch Region

We will patch the upper region downstream of the flow to create asymmetry so that we can obtain stable oscillation of vortex shedding faster.

To do this, we will create a register to patch the Y velocity in downstream of cylinder.

Main Menu > Adapt > Region...

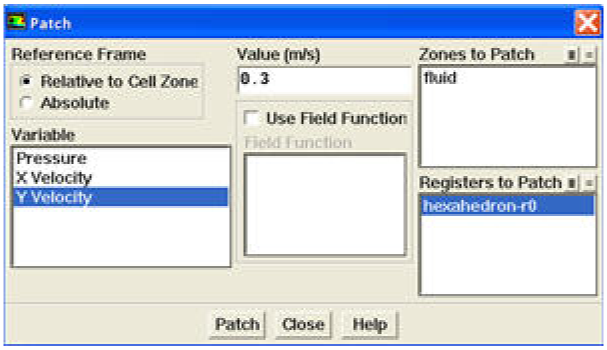


Enter `1` and `40` for ***X Min*** and ***X Max***. Enter `0` and `10` for ***Y Min*** and ***Y Max***. Click ***Mark***. FLUENT will print the following message in the console window:
`5416 cells marked for refinement, 0 cells marked for coarsening`

Close the *Region Adaption* panel.

We will now patch Y velocity in the registered region.

Main Menu > Solve > Initializate > Patch...

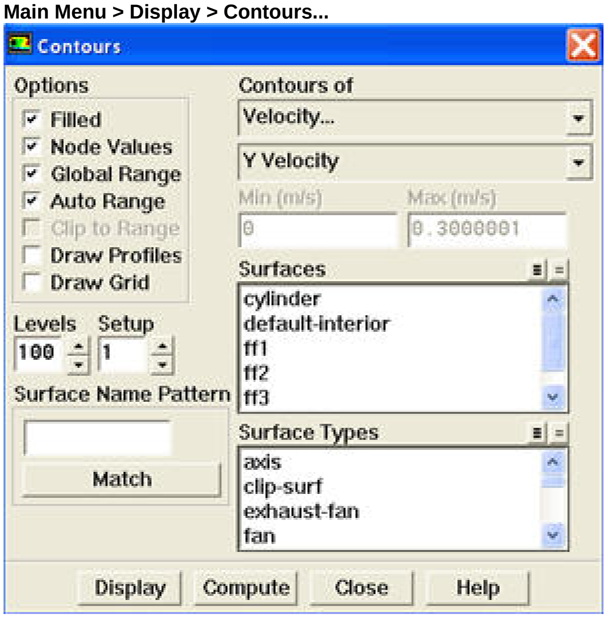


Higher Resolution Image

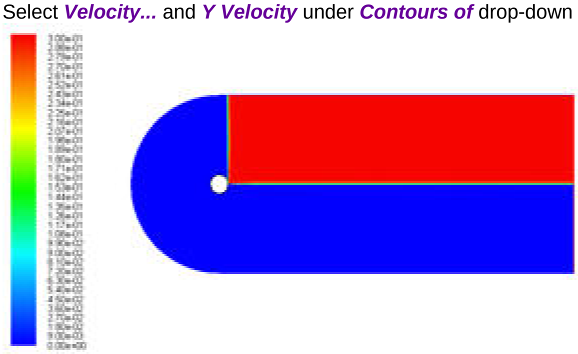
Select ***hexahedron-r0*** from the ***Registers to Patch***. Select ***Y Velocity*** from the ***Variable*** selection list. Enter `0.3` for ***Value***. Click ***Patch***.

To check whether you have patch the region, plot contour of velocity in the y direction.

Main Menu > Display > Contours...



Select ***Velocity...*** and ***Y Velocity*** under ***Contours of*** drop-down list. Make sure to check the ***Filled*** under ***Options***. Click ***Display.***



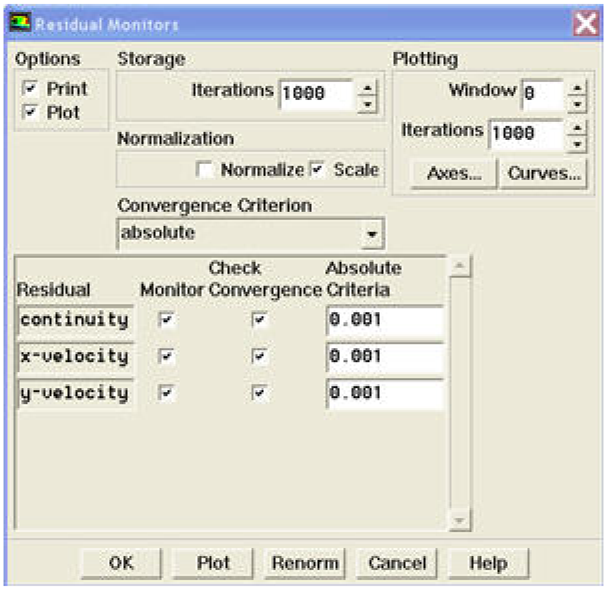
Higher Resolution Image

As can be seen, the Y Velocity is zero everywhere except for the patched region, we have Y Velocity of 0.3 m/s.

Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-3.

Main Menu > Solve > Monitors > Residual...



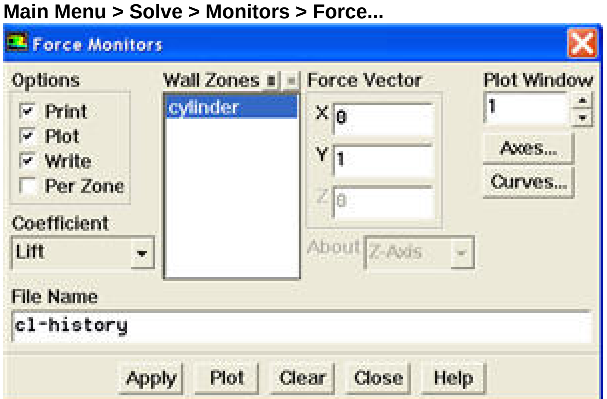
Default value for ***Convergence Criterion*** for ***continuity***, ***x-velocity***, and ***y-velocity*** is `1e-3`.

Also, under ***Options***, select ***Plot*** and ***Print***. This will plot the residuals in the graphics window as they are calculated.

Click ***OK***.

Monitor also the lift coefficient on the cylinder.

Main Menu > Solve > Monitors > Force...

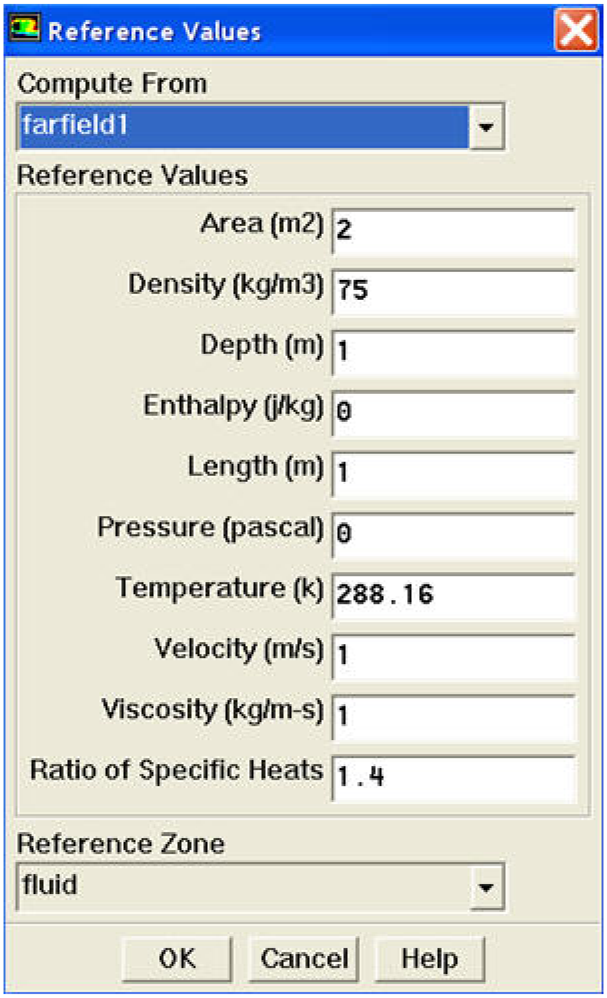


Under ***Coefficient***, select ***Lift***. Select ***cylinder*** under ***Wall Zones***. Under ***Options***, select ***Print, Plot*** and ***Write***. Note that ***Plot Window*** is 1. Click ***Apply***.

Set Reference Values

The reference values are used to non-dimensionalize the forces and moments action on the wall surface.

Main Menu > Reference Values...



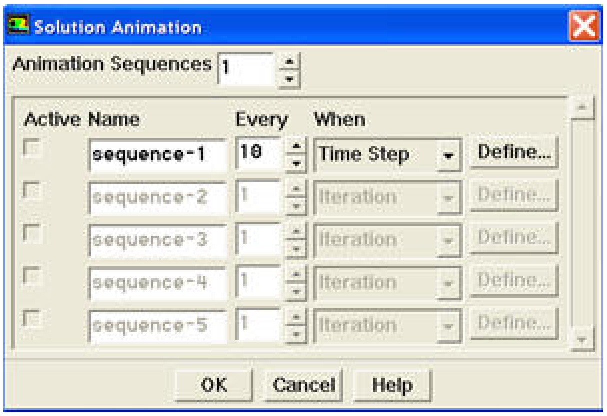
Select ***farfield1*** from the ***Compute From*** drop-down list. FLUENT will update the reference values based on the boundary conditions at ***farfield1***. Change ***Area*** to *2*. Click ***OK***.

> Note that cross sectional area for a 2D cylinder is the diameter of the cylinder. Setting the right area is important for getting correct drag coefficient.

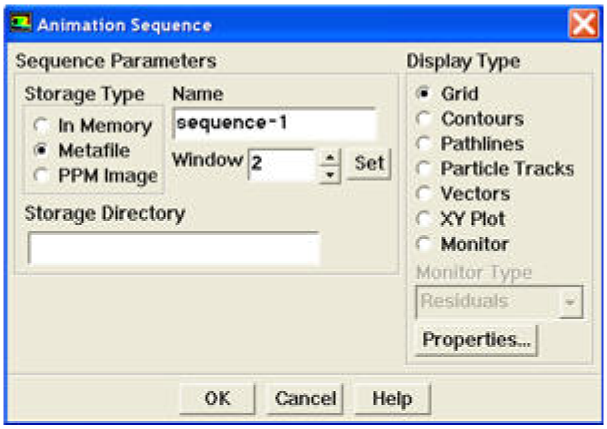
Set Animation Control (Optional)

Let's set the animation to observe the vorticity magnitude.

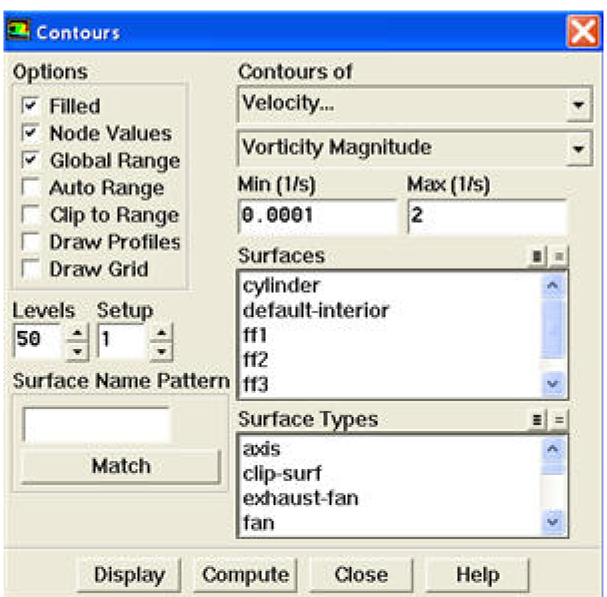
Main Menu > Solve > Animate > Define...



Increase the ***Animation Sequences*** to `1`. Enter `10` for ***Every***. Select ***Time Step*** from ***When*** drop-down list. Click ***Define...*** for ***sequence-1*** to open the Animation Sequence panel.



Increase ***Window*** to 2 and click the ***Set*** button to open a graphics window. Select ***Contours*** from the ***Display Type*** list to open the Contours panel. Select ***Velocity...*** and ***Vorticity Magnitude*** from the ***Contours of*** drop-down lists.



Disable ***Auto Range*** and ***Clip to Range*** from the ***Options*** group box. Enter `0.0001` and `2` for ***Min*** and ***Max***, respectively. Select ***Levels*** to 50. Click ***Displ ay***. Click ***OK*** to close the *Animation Sequence* panel. Click ***OK*** to close the *Solution Animation* panel. This will save .hmf file after every 10 time steps. We can later create an animation in the form of movie clip using these files. Save the case and data files.
Main Menu > File > Write > Case & Data...

Reviewing Animation

We can review the animation created after we are done with iterations.
Solve > Animate > Playback...
To write in MPEG format, go to ***Write/Record Format***, select ***MPEG***. Click ***Write***.

Note that you don't necessarily get a good format when exporting to MPEG. It is advisable to use the available playback option.

Iterate the Solution

Main Menu > Solve > Iterate...

You will have to input the time step size for iteration. Smaller time step means more accurate result but more computational time. We need to find the balance between accuracy and computational time.

Calculating Time Step Size

The Strouhal number for flow past cylinder is roughly 0.183 as reported by Williamson .
In order to capture the shedding correctly, we should have at least 20 to 25 time steps in one shedding cycle. Let's use 25 for our case.

No Latex value

No preview is available, please have your Latex markup text selected when inserting the Latex macro or click edit on an existing Latex macro when text is present to preview the rendered result.

Example Markup

```
\begin{eqnarray}
    y&=&ax^{2}+bx+c \nonumber\\
    E&=&mc^2 \nonumber\\
    {\delta y \over \delta x}
        &=& {{a\over b}\over c}
\end{eqnarray}
```

Example Render

$$\begin{eqnarray} y&=&ax^{2}+bx+c \nonumber\\ E&=&mc^2 \nonumber\\ {\delta y \over \delta x} &=& {{a\over b}\over c} \end{eqnarray}$$

For more information about **Latex**, you can find in the following documentation LaTeX Plugin

For our case, **D = 2, U = 1**
Therefore, shedding frequency **f = 0.0915**
Cycle time,

No Latex value

No preview is available, please have your Latex markup text selected when inserting the Latex macro or click edit on an existing Latex macro when text is present to preview the rendered result.

Example Markup

```
\begin{eqnarray}
    y&=&ax^{2}+bx+c \nonumber\\
    E&=&mc^2 \nonumber\\
    {\delta y \over \delta x}
        &=& {{a\over b}\over c}
\end{eqnarray}
```

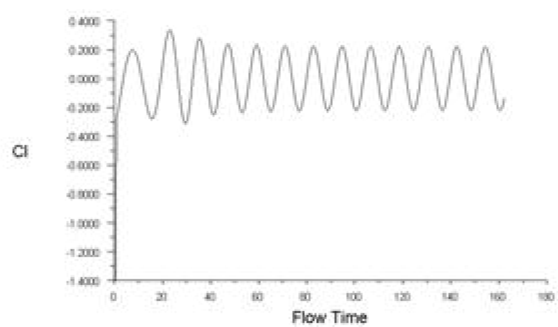
Example Render

$$\begin{eqnarray} y&=&ax^{2}+bx+c \nonumber\\ E&=&mc^2 \nonumber\\ {\delta y \over \delta x} &=& {{a\over b}\over c} \end{eqnarray} \qquad (1)$$

For more information about **Latex**, you can find in the following documentation LaTeX Plugin

Therefore, **Time Step Size = 10.9/25 = 0.436 sec ~ 0.4 sec**

Enter `0.4` for ***Time Step Size (s)***. Enter `30` for ***Max. Iterrations per Time Step***. Enter `800` for ***Number of Time Steps***. Click ***Apply***. Click ***Iterate*** to start the iterations.



Higher Resolution Image

We can see a clear sinusoidal pattern, a sign of sustained vortex shedding process after 40s. Stop the iteration after about 350s.

Save the case and solution.
Main Menu > File > Write > Case & Data...

Use the default name (Mesh's file name "***cylinder***") and click ***OK***.

Go to Step 6: Analyze Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules