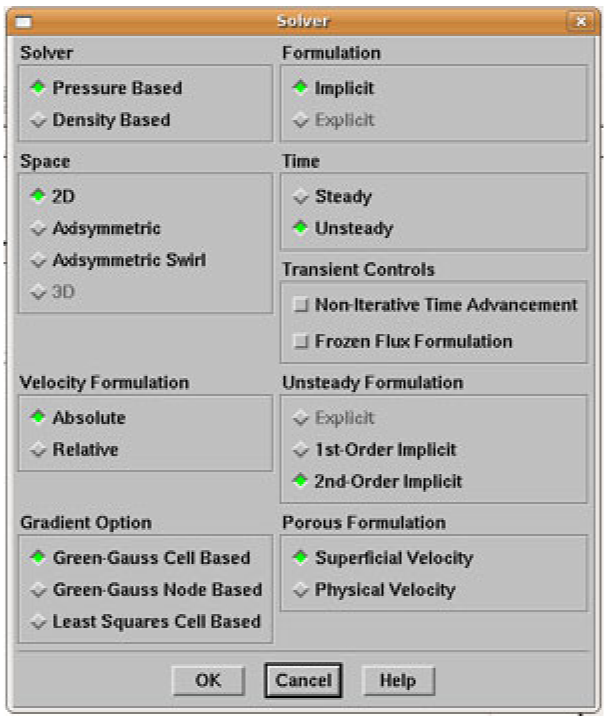
FLUENT - Flow Past a Cylinder - Step 8

Unsteady Flow

For Re > 47, the flow becomes unsteady. We simulate two cases, Re = 300 and 1000, using the large domain and mesh created in Step 7. For unsteady flow we need to change some setups in Fluent.

1. **Define > Models > Solver...**

Select **Unsteady** under **Time**. Choose **2nd-Order Implicit** under **Unsteady Formulation**.



2. **Define>Materials...**

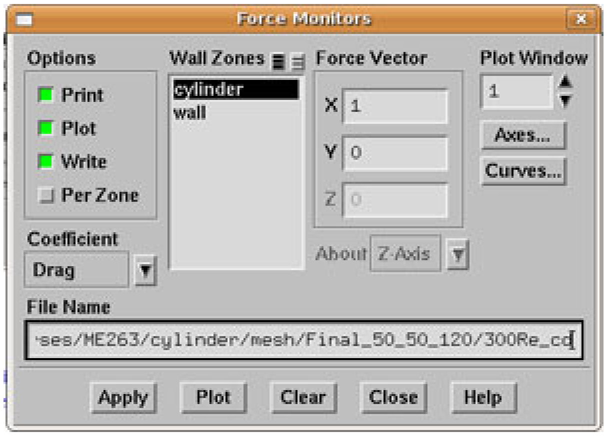
Use 1 for **Density** and proper **Viscosity** so that Re = 300.

For Re = 1000, you will again need to adjust the viscosity.

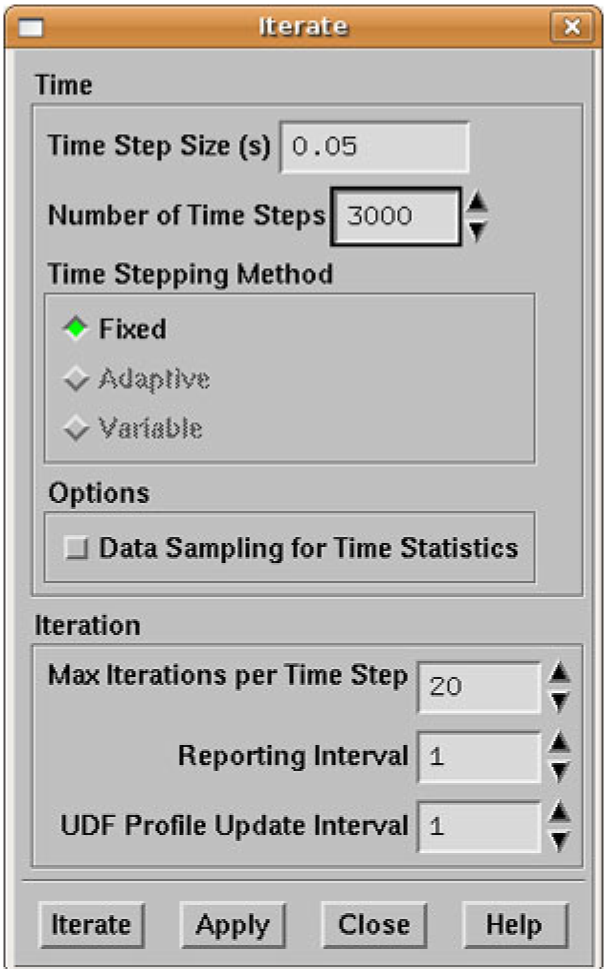
3. Record the histories of Cd and Cl.

Solve> Monitors > Force...

Select **Print**, **Plot** and **Write**. Fill in the name in the box under **File Name**, then the text file containing Cd or Cl each time step will be recorded in the file.

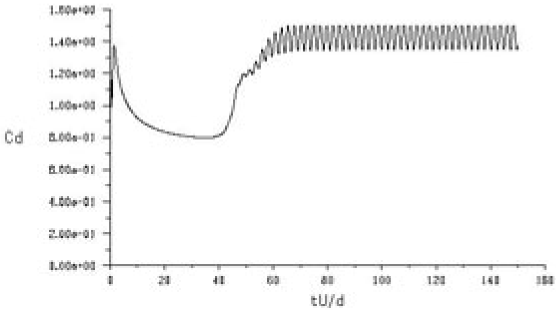


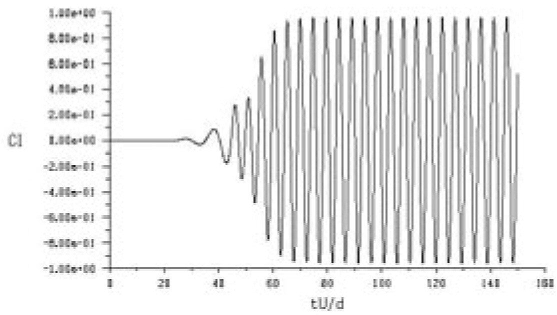
4.**Iteration**



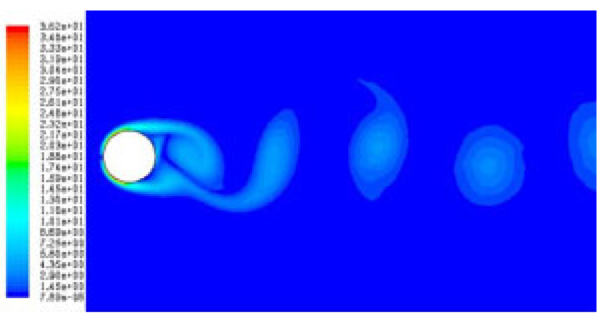
Because the flow is unsteady, we need to define the size of the time step. In this case we use 0.05s as the time step and run the simulation for 3000 time steps. Note that in terms of the dimensionless scale, the time step is 5% of the residence time (d/U).

The drag and lift coefficients as functions of nondimensional time, tU/d, for Re=300 are shown below. The mean values of Cd and Cl are taken after the periodic oscillations are established.

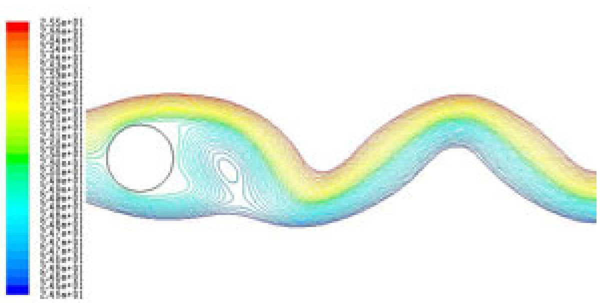




Plot the vorticity contour.

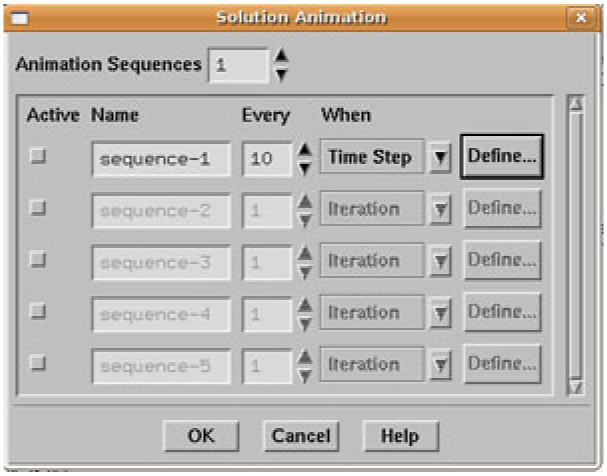


Plot streamlines.

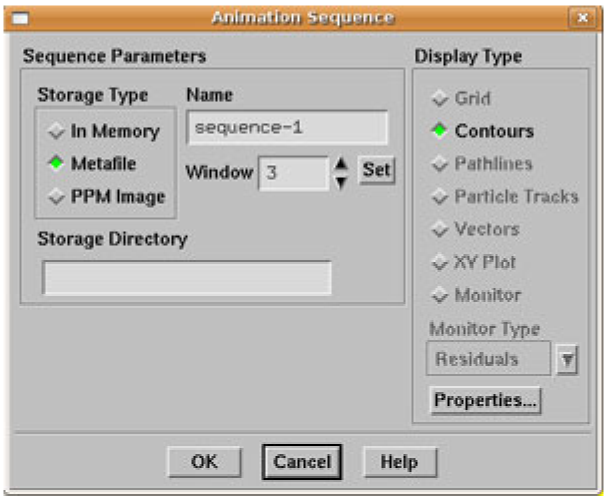


5. Making animations

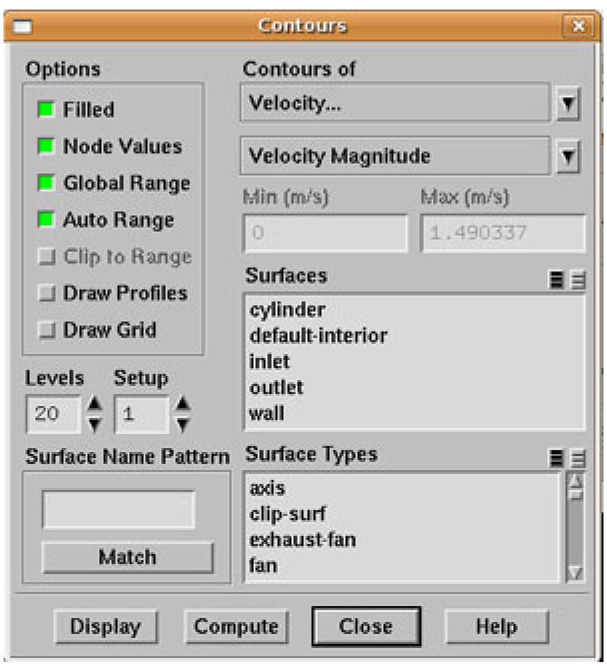
Solve> Animate> Define...



Choose **Time Step** under **When**. Type 10 (or up to 50) in **Every** box so that a frame is recorded every 10 time steps. Note that the more frequently you record a frame, the larger data the code will produce. You may delete the data files generated by Fluent after creating the animation (sequence.xxx.hmf). Click **Define...**



For **Window**, we use number 3, which means that the velocity magnitude window is being recorded (You need to choose proper window number if you have different windows); then click **Set** and the figure window 3 will show up. Under **Display Type**, choose **Contours**:

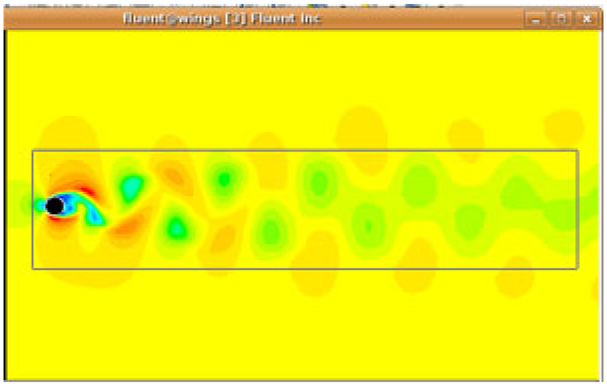


Choose **Velocity Magnitude**. Click **Display** and **Close**.

Click **OK** for **Animation Sequence** panel.

Click **OK** for **Solution Animation** panel.

Adjust the size of the velocity contour for a better view of the flow over the cylinder. Use middle button to choose the view you want: Press the middle button, move the arrow from upper left to lower right to zoom in and from lower right to upper left to zoom out.



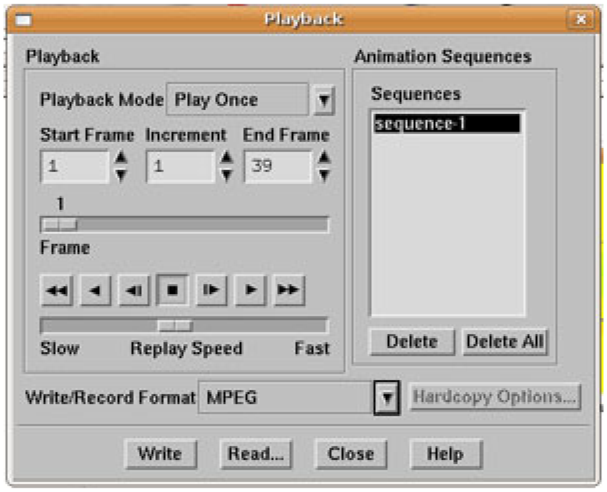
This window will record the velocity contours every a few time steps as specified while the flow is evolving.

After the time-stepping is finished, we can make the animation from the recorded frames.

Solve>Animate>Playback...

Under **Sequences**, select sequence-1. For **Write/Record Format**, choose **MPEG**.

Click **Write** and a MPEG file will export to your work directory. You may also specify the playback speed here.



Go to Problem Set.

See and rate the complete learning module.

Go to all FLUENT Learning Modules