Tutorial 6. Flow Past a Circular Cylinder

Introduction

The purpose of this tutorial is to illustrate the setup and solution of an unsteady flow past a circular cylinder and to study the vortex shedding process.

Flow past a circular cylinder is one of the classical problems of fluid mechanics. The geometry suggests a steady and symmetric flow pattern. For lower value of Reynolds number, the flow is steady and symmetric. Any disturbance introduced at the inlet gets damped by the viscous forces. As the Reynolds number is increased, the disturbance at the upstream flow can not be damped. This leads to a very important periodic phenomenon downstream of the cylinder, known as 'vortex shedding'.

In this tutorial you will learn how to:

- Read an existing mesh file in FLUENT.
- Check the grid for dimensions and quality.
- Solve a time dependent simulation.
- Set the time monitors for lift coefficient and observe vortex shedding.
- Set up an animation to demonstrate the vortex shedding.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

Consider a cylinder of unit diameter (Figure 6.1). The computational domain consists of an upstream of 11.5 times the diameter to downstream of 20 times the diameter of the cylinder and 12.5 times the diameter on each cross-stream direction. The Reynolds number of the flow, based on the cylinder diameter, is 150.

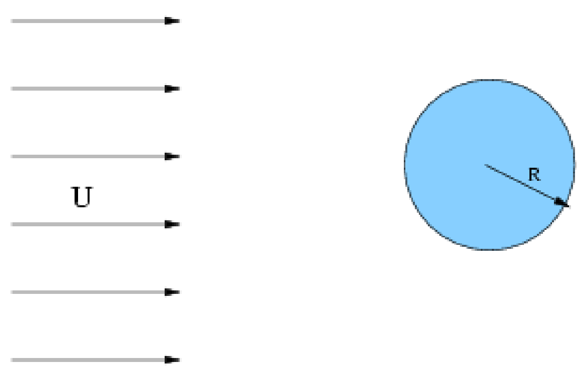


Figure 6.1: Problem Schematic

Preparation

1. Copy the mesh file, `cyl.msh` to your working directory.

2. Start the 2D double precision solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file, `cyl.msh`.

   File ⟶ Read ⟶ Case...

   FLUENT *will read the mesh file and report the progress in the console window.*

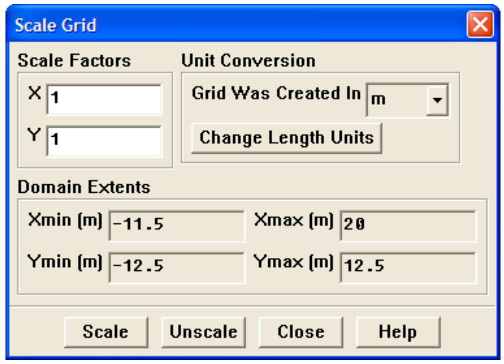
2. Check the grid.

   Grid ⟶ Check

   This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

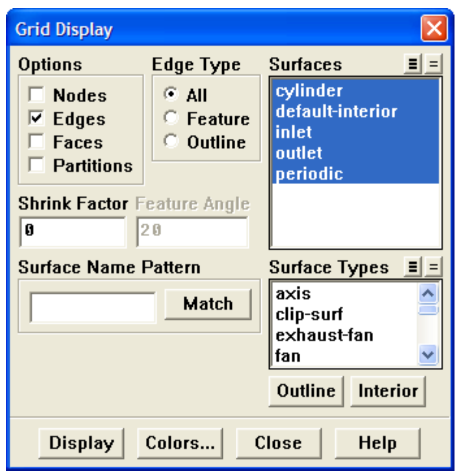
   Grid ⟶ Scale...



Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units. In this case, do not scale the grid.

4. Display the grid (Figures 6.2 and 6.3).

   Display ⟶Grid...



Zoom-in using the middle mouse button to see the mesh around the cylinder (Figure 6.3). The boundary layer is resolved around the cylinder. A submap mesh is used in the block containing the cylinder.

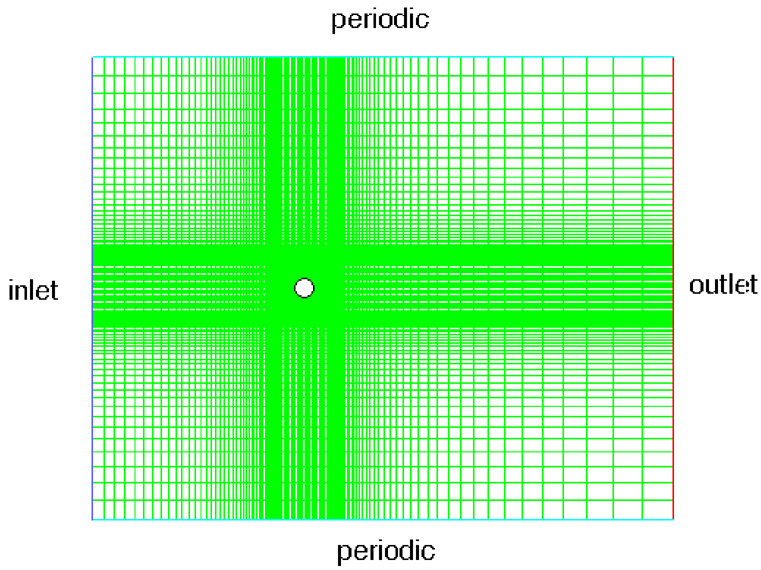


Figure 6.2: Grid Display

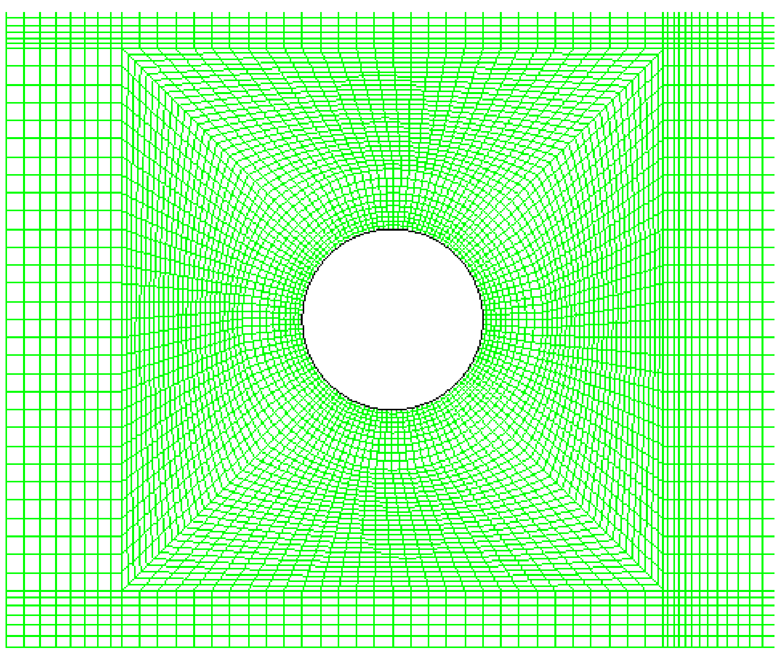


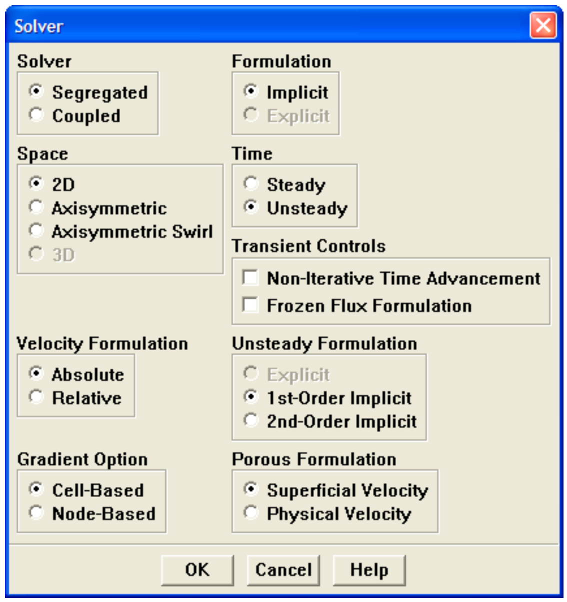
Figure 6.3: Grid Display—Zoom-in View

Step 2: Models

This is an unsteady problem in a symmetric geometry. In experiments, uncontrollable disturbances in the inlet flow cause the start of the vortex shedding. Similarly, in the computational model, the numerical error accumulates and the vortex shedding starts.

1. Switch to an unsteady solver settings.

   Define ⟶ Models ⟶ Solver...



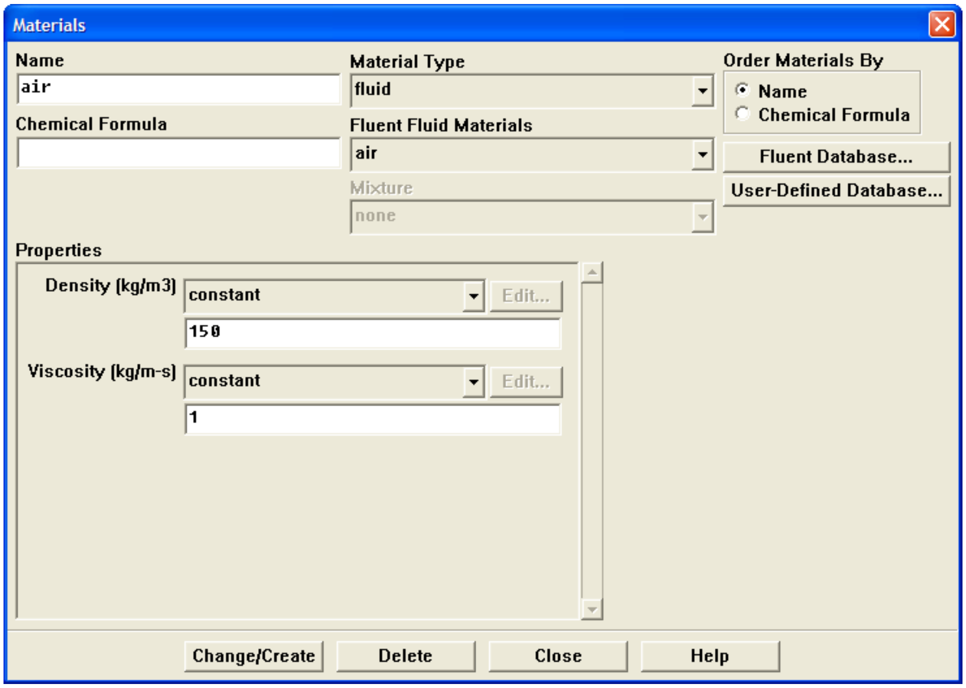
   (a) Under Time, enable Unsteady.

   (b) Click OK.

Step 3: Materials

1. Change the material properties.

   Define ⟶Materials...



   (a) Specify a value of 150 for Density and a value of 1 for Viscosity.

   The Reynolds number is defined as:

$$R_e = \frac{U \times D_i \times \rho}{\mu} \tag{6.1}$$

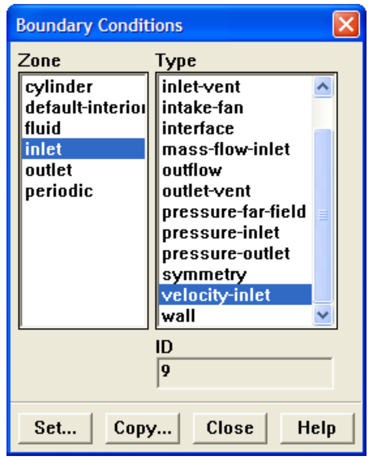
   The value of μ, D_i, and ρ is unity. Therefore, set the value of density same as the Reynolds number.

   (b) Click Change/Create and close the panel.

Step 4: Boundary Conditions

1. Set the boundary conditions for inlet.

   Define ⟶Boundary Conditions...

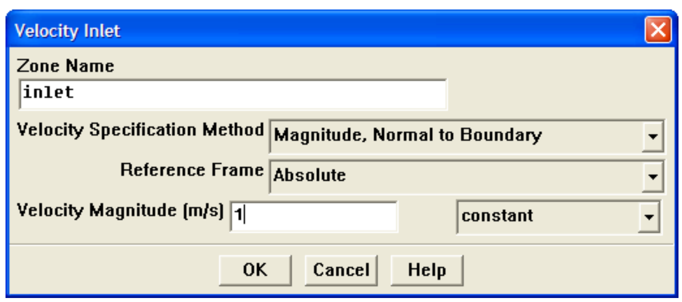


2. Under Zone, select inlet.

   The Type *will be reported as* velocity-inlet.

3. Click Set....

   The Velocity Inlet *panel opens.*



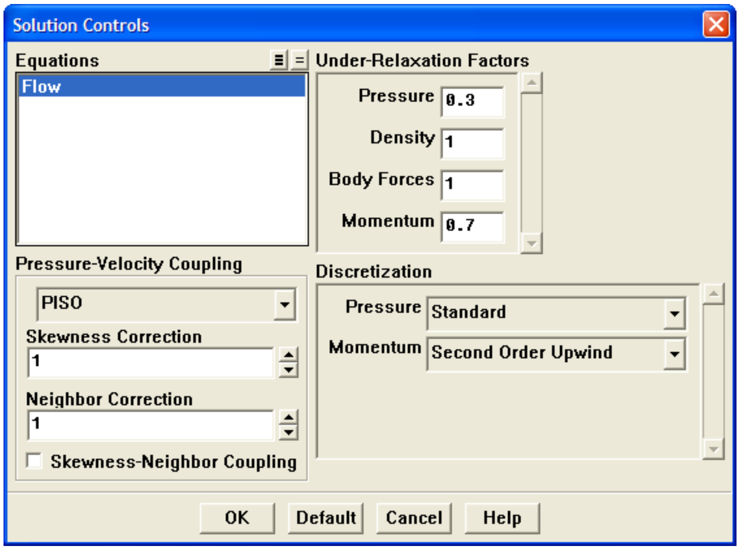
   (a) Specify a value of 1 for Velocity Magnitude (m/s).

   (b) Click OK to accept the settings and close the panel.

Step 5: Solution

1. Set the solution controls.

   Solve ⟶ Controls ⟶Solution...



   (a) Select PISO in 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.

   (b) Disable Skewness-Neighbor Coupling option.

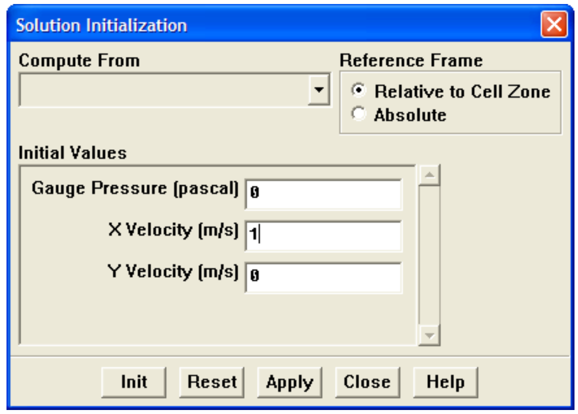
   (c) Select Second Order Upwind scheme for Momentum.

   Since the simulation is transient, start with higher order schemes right from initial conditions.

   (d) Click OK to close the panel.

2. Initialize the flow.

   Solve ⟶ Initialize ⟶Initialize...

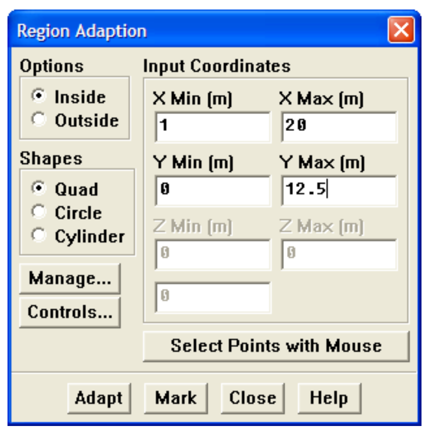


(a) Set X Velocity (m/s) to 1.

(b) Click Init and close the panel.

3. Create registers to patch the Y velocity in down-stream of cylinder.

$\boxed{\text{Adapt}}\longrightarrow$Region...



(a) Set X Min (m) and X Max (m) to 1 and 20 respectively.

(b) Set Y Min (m) and Y Max (m) to 0 and `12.5` respectively.

(c) Click Mark.

FLUENT *will print the following message in the console window:*

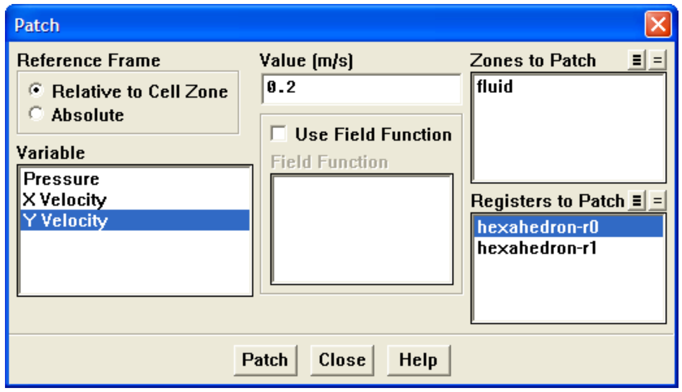
`2928 cells marked for refinement, 0 cells marked for coarsening.`

(d) Change Y Min (m) and Y Max (m) to `-12.5` and 0 respectively.

(e) Click Mark again.

4. Patch the Y velocity.

   Solve ⟶ Initialize ⟶ Patch...



   (a) Under Registers to Patch, select `hexahedron-r0`.

      i. Under Variable, select Y Velocity.

      ii. Set the Value as `0.2`.

      iii. Click Patch.

   (b) Under Registers to Patch, deselect `hexahedron-r0` and select `hexahedron-r1`.

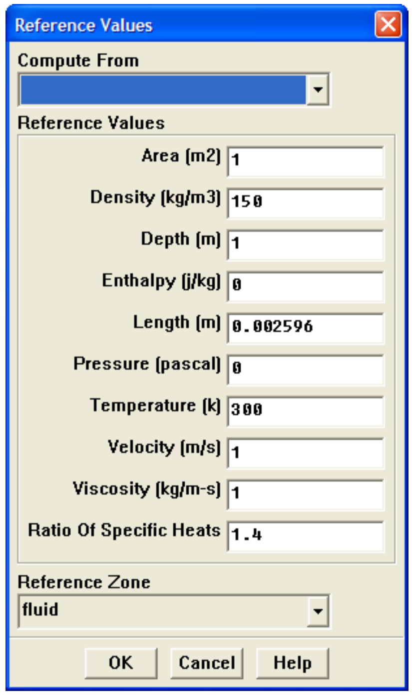
      i. Change the Value to `-0.2`.

      ii. Click Patch again.

5. Set the reference values used to compute the lift, drag, and moment coefficients.

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

   Report ⟶ Reference Values...



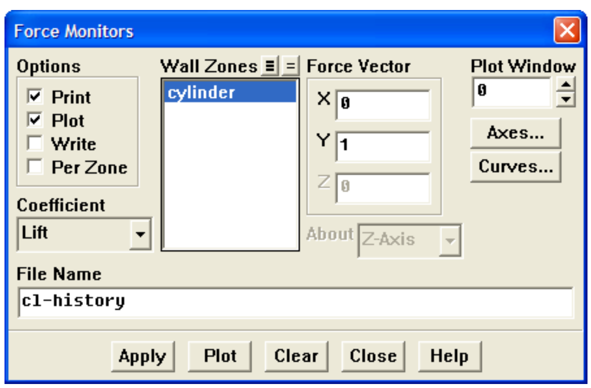
(a) Select inlet in the Compute From drop-down list.

FLUENT *will update the* Reference Values *based on the boundary conditions at the inlet boundary.*

(b) Click OK.

6. Set the monitor for lift coefficient on cylinder wall.

Solve ⟶ Monitors ⟶Force...



(a) Select Lift in the Coefficient drop-down list.

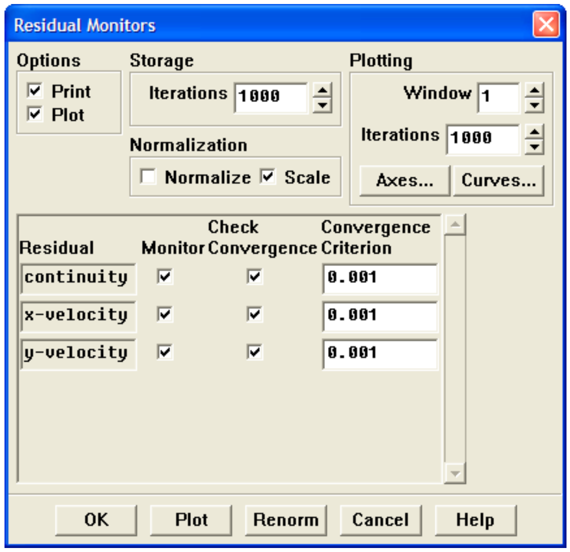
(b) Under Wall Zones, select cylinder.

(c) Under Options, enable Print and Plot.

(d) Click Apply and close the panel.

7. Enable plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶Residuals...

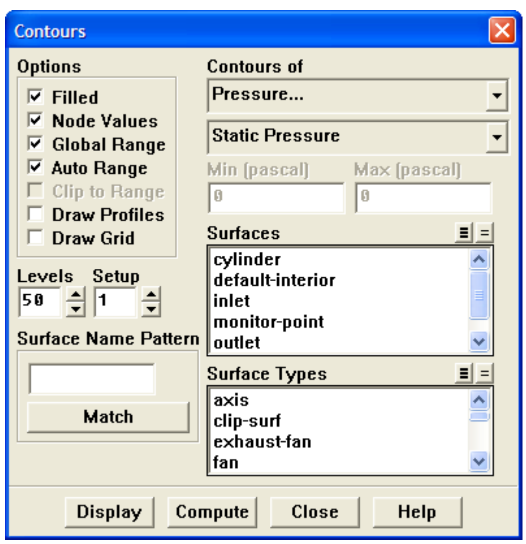


   (a) Under Options, enable Plot.

   (b) Click OK.

8. Set animation to visualize vortex shedding.

   Display ⟶Contours...



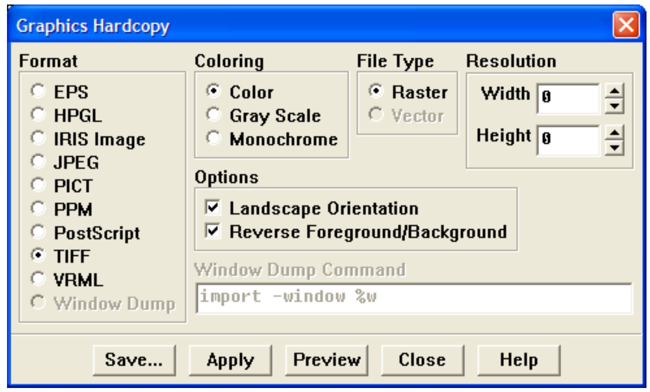
   (a) Select Pressure... and Static Pressure in the Contours of drop-down lists.

   (b) Under Options, enable Filled.

   (c) Click Display (Figure 6.4).

(d) Change the background of graphics window to white.

File ⟶Hardcopy...



(e) Click Preview.

A question dialog box appears.

(f) Click No in the Question dialog box that appears.

(g) Click Apply and close the panel.

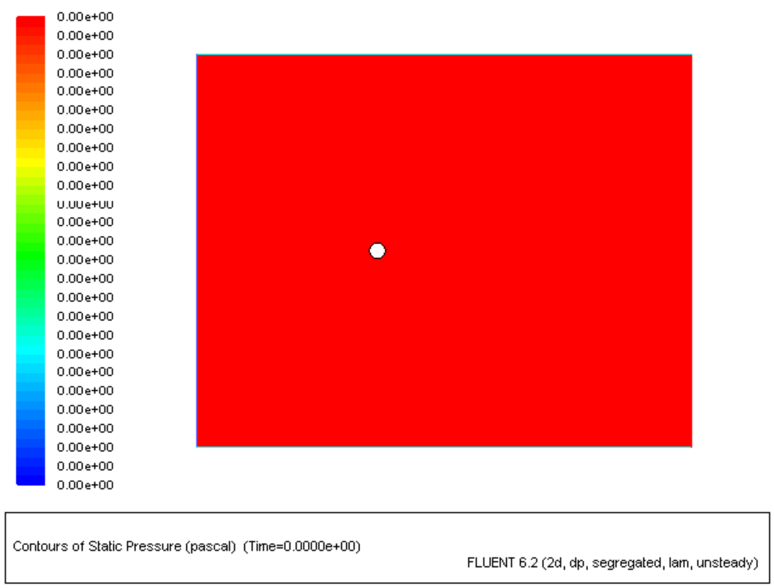
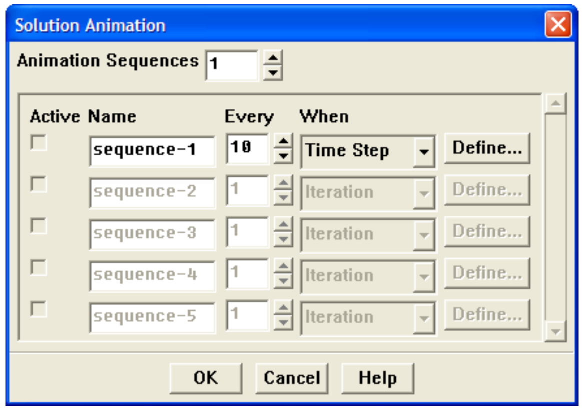


Figure 6.4: Contours of Static Pressure

9. Set the animation controls.

   Solve ⟶ Animate ⟶ Define...



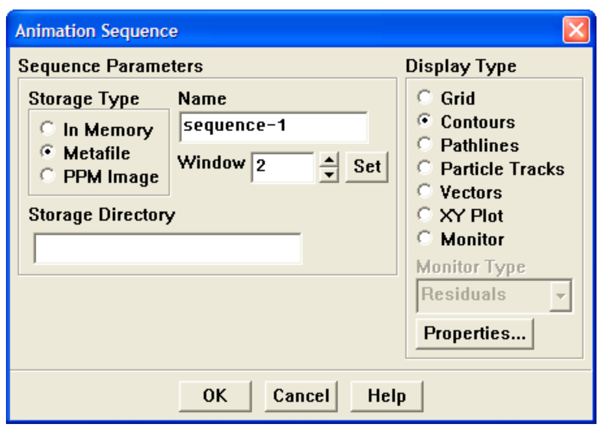
   (a) Increase the Animation Sequences to 1.

   (b) Under Every, increase the value to 10 .

   (c) Select Time Step in the When drop-down list.

   (d) Click Define... next to `sequence-1`.

   This opens Animation Sequence *panel.*



   i. Increase Window to 2 and click Set.

      This opens a graphics window.

   ii. Under Display Type, enable Contours.

      This opens Contours *panel.*

   iii. Select Velocity... and Velocity Magnitude under Contours of drop-down lists.

   iv. Set Levels to 50.

   v. Click Display and close the Contours panel.

vi. Adjust the view as shown in Figure 6.5.

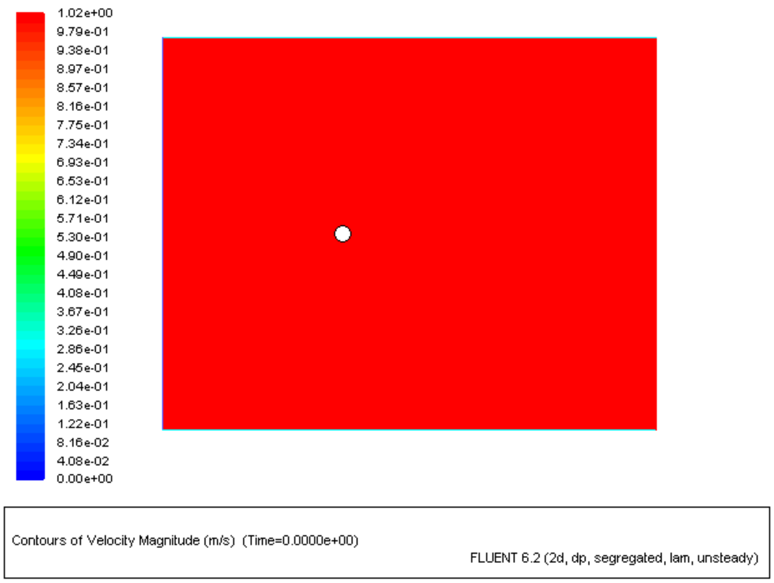


Figure 6.5: Contours of Velocity Magnitude

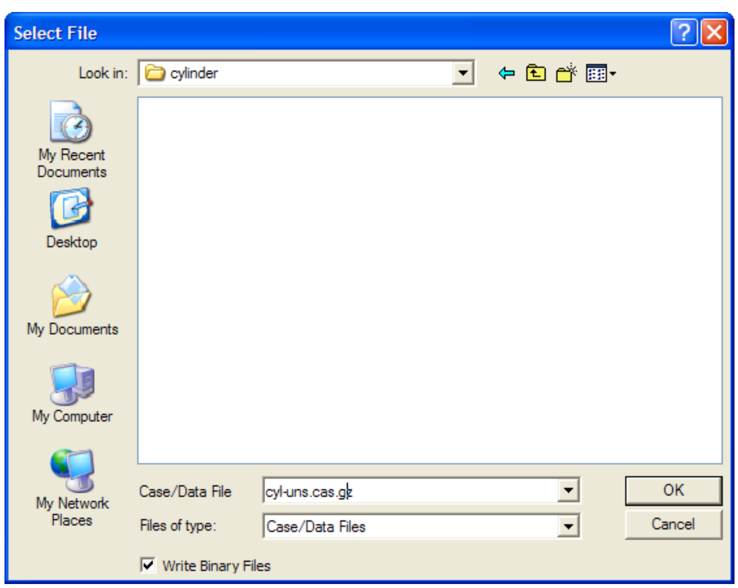
vii. Click OK to close the Animation Sequence panel.

(e) Click OK to close the Solution Animation panel.

This will save `.hmf` *file after every 10 time steps. You can create an animation in the form of movie clip using these files.*

10. Save the case and data files (`cyl-uns.cas.gz` and `cyl-uns.dat.gz`).

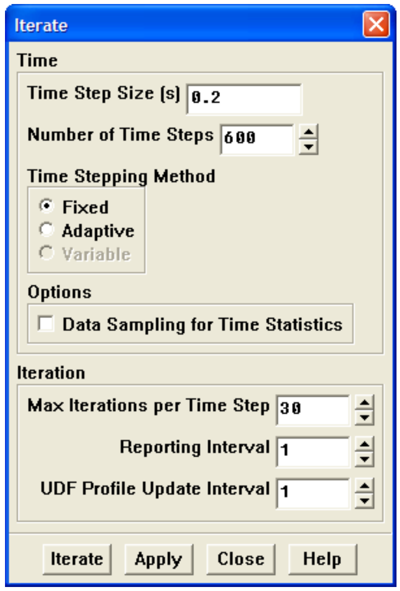
File ⟶ Write ⟶ Case & Data...



Retain the default activated Write Binary Files *option so that you can write a binary file. The* `.gz` *option will save zipped files, this will work on both,* Windows *as well as* **UNIX** *platforms.*

11. Iterate the solution.

    Solve ⟶Iterate...



(a) Set the Time Step Size as 0.2.

The Strouhal number for flow past cylinder is roughly 0.2. In order to capture the shedding correctly, you should have at least 20 to 25 time steps in one shedding cycle.

$$Sr = 0.2 = \frac{f \times D}{U} \tag{6.2}$$

In this case, D=1 and U=1. Therefore, $f = 0.2$.

Cycle time, t = 1 / f = 1 / 0.2 = 5 sec

Therefore, time step size = 5 / 25 = 0.2 sec.

(b) Set the Maximum Iterations per Time Step to 30.

(c) Set the Number of Time Steps to 600.

(d) Click Apply.

(e) Click Iterate to start iterations.

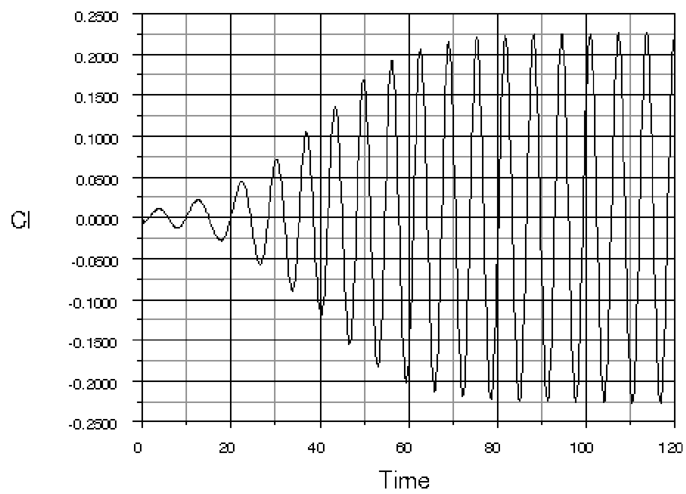


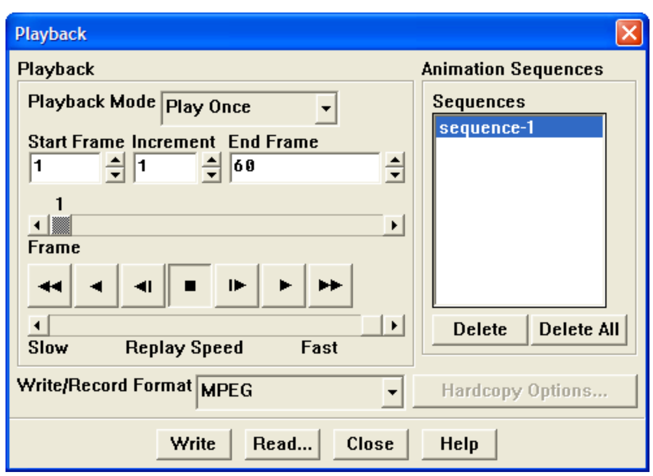
Figure 6.6: Lift Coefficient Plot

Figure 6.6 shows a clear sinusoidal pattern. This is a sign of a sustained vortex shedding process. All the other flow variables also show the asymmetry in the solution. This plot can be used to compute the correct value of Strouhal number. The problem is non-dimensionalized (i.e., $D = U = 1$) and $Sr = f = 1/(\text{shedding cycle time}) = 1/6.32 = 0.158$.

The results matches fairly well with the value (0.183) as reported in the literature [3].

12. Create an animation using the `.hmf` files.

    Solve ⟶ Animate ⟶ Playback...



(a) Select MPEG, in Write/Record Format drop-down list.

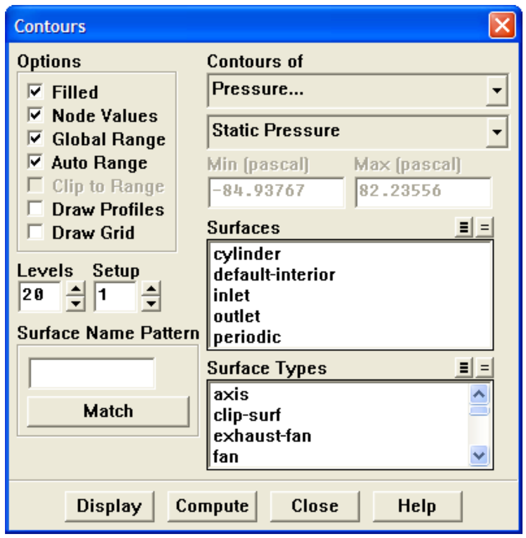
(b) Click Write.

This creates a movie file in the working directory, which can be viewed using Windows Media Player.

Step 6: Postprocessing

1. Display the pressure contours (Figure 6.7).

   Display ⟶Contours...



   (a) Under Options, enable Filled.

   (b) Set the number of Levels to 20.

   (c) Click Display.

   The contour shows a clear asymmetric pattern in the flow. The local pressure minima are the center of the vortices.

2. Display the contours of vorticity magnitude (Figure 6.8).

   Display ⟶Contours...

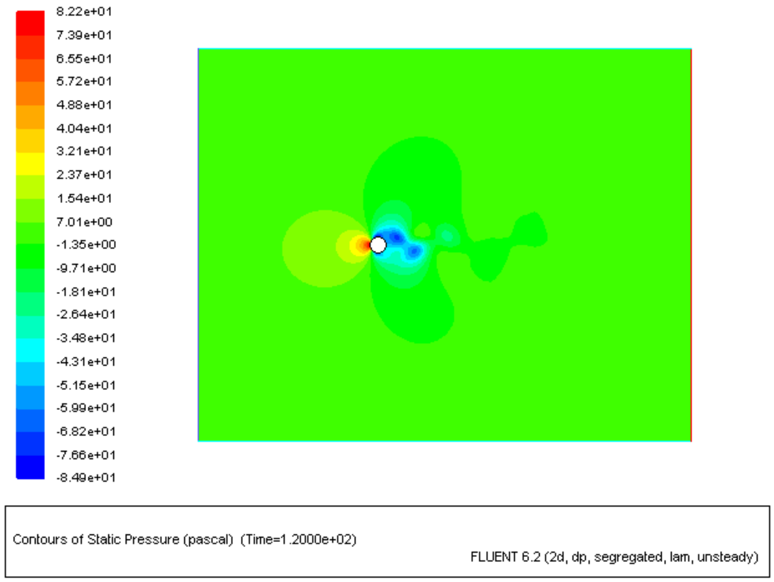
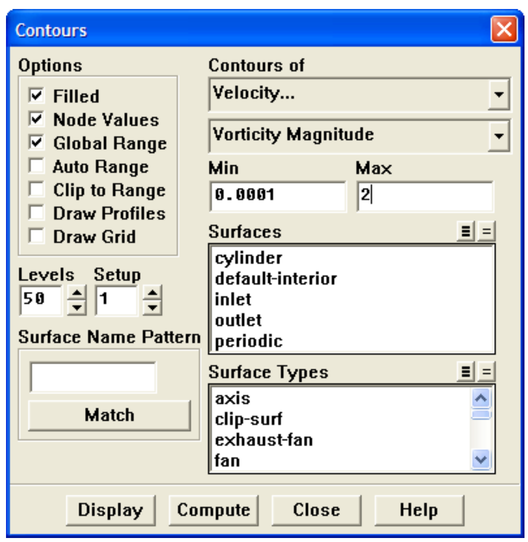


Figure 6.7: Contours of Static Pressure



(a) Select Velocity... and Vorticity Magnitude in the Contours of drop-down list.

(b) Under Options, disable Auto Range and Clip to Range.

(c) Set Min and Max values to `0.0001` and 2 respectively.

(d) Set Levels to 50.

(e) Click Display.

The figure shows clear vortex shedding process. Zoom-in the view around cylinder.

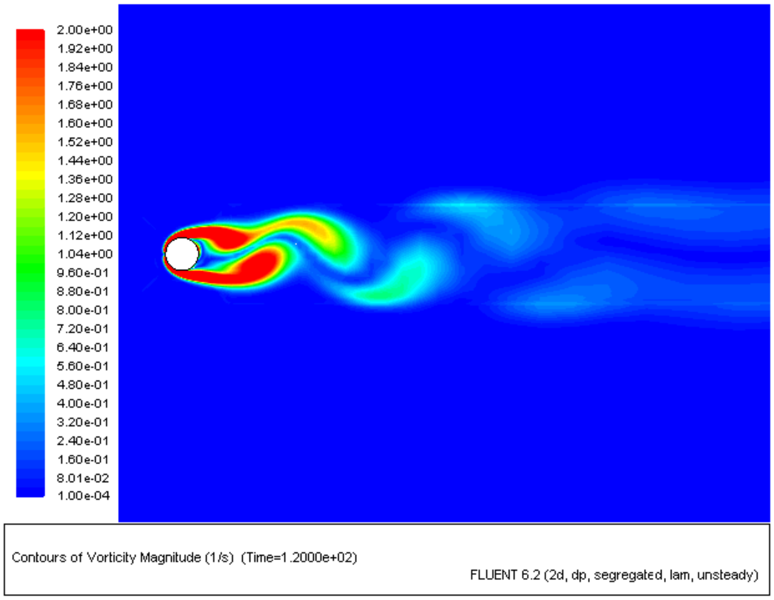
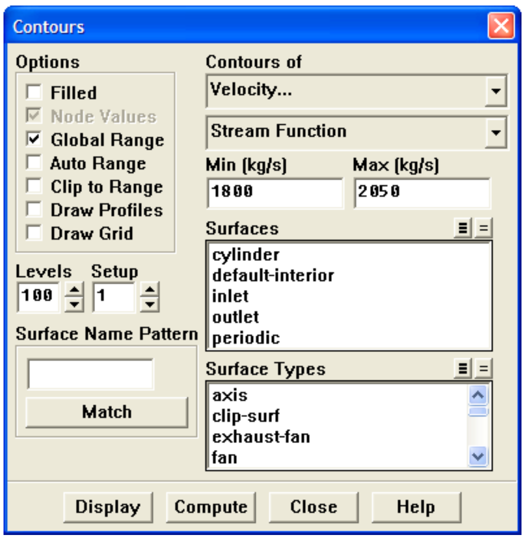


Figure 6.8: Contours of Vorticity Magnitude

3. The instantaneous streamline can be displayed to see the incipient and shed vortex clearly (Figure 6.9).

   Display ⟶Contours...



   (a) Select Velocity... and Stream Function in the Contours of drop-down list.

   (b) Under Option, disable Filled.

   (c) Set Levels to 100.

   (d) Under Options, disable Auto Range and Clip to Range.

(e) Set Min and Max values to 1800 and 2050 respectively.

(f) Click Display.

The contour shows the incipient vortex at the top end and shed vortex at the bottom end in the wake of the cylinder. Zoom in to get a better view of the shedding process.

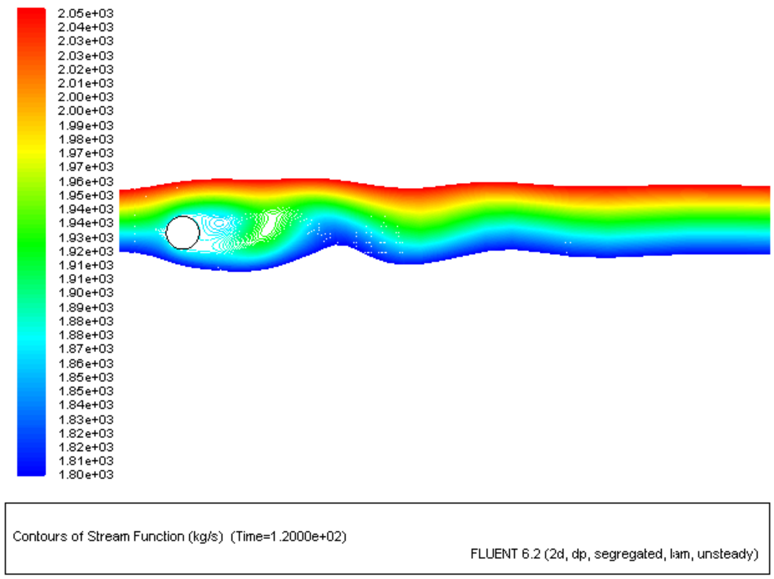


Figure 6.9: Contours of Stream Function

Summary

This tutorial demonstrated a classical problem of flow past the cylinder. Different methods like monitor plots and animations were used to track the vortex shedding phenomenon. Additional aspects like choosing time step, using PISO for transient simulation and calculating the Strouhal number were also covered.

References

1. J.D. Anderson, *Fundamentals of Aerodynamics*, 2nd Ed., Ch. 3: pp. 229.

2. I. H. Shames, *Mechanics of Fluid*, 3rd Ed", Ch. 13: pp. 669-675.

3. C.H.K. Williamson and G.L. Brown, *A Series in to represent the Strouhal- Reynolds number relationship of the cylinder wake*, J. Fluids Struct. 12,1073 (1998).

Exercises/ Discussions

1. Run the solution at different Reynolds numbers and compare the solutions.
2. Use NITA schemes and record the run time for transient simulation
3. Will the cylinder demonstrate any shedding if the flow modeled as inviscid. Simulate the inviscid flow conditions and compare the pressure coefficient with theory.
4. Is it possible to simulate flow around a square block with unit dimension using the same grid? How can you achieve that?
5. What changes you will need to make in the set up if:
   (a) The cylinder rotates at some constant rotational speed
   (b) The cylinder oscillates about its mean position in vertical direction

Links for Further Reading

- http://www.aoe.vt.edu/ devenpor/aoe3054/manual/expt3/
- http://www.math.ntnu.no/ andreas/fronttrack/gas/cylinders/
- http://tfc.snu.ac.kr/choi/PHF02767.pdf
- http://mec-mail.eng.monash.edu.au/ mct/pubs/pdfs/ReHoTh05_jfm.pdf
- http://www.mate.tue.nl/mate/pdfs/889.pdf