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'.

This tutorial demonstrates how to do the following:

- 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.

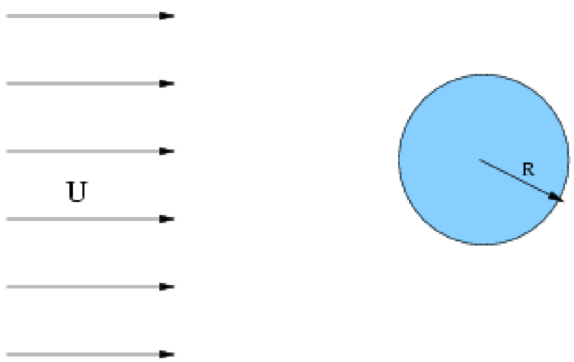


Figure 6.1: Problem Schematic

Preparation

1. Copy the mesh file, `cyl.msh` to your working folder.
2. Start the 2D double precision (2ddp) 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.*

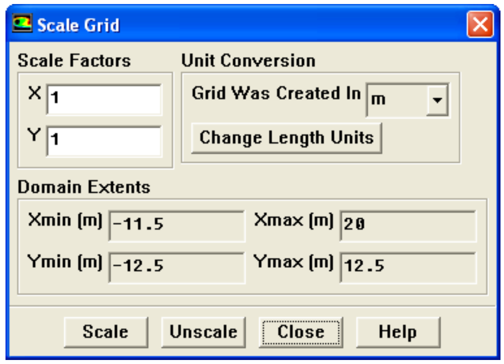
2. Check the grid.

   Grid ⟶ Check

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

3. Check the scale of the grid.

   Grid ⟶ Scale...

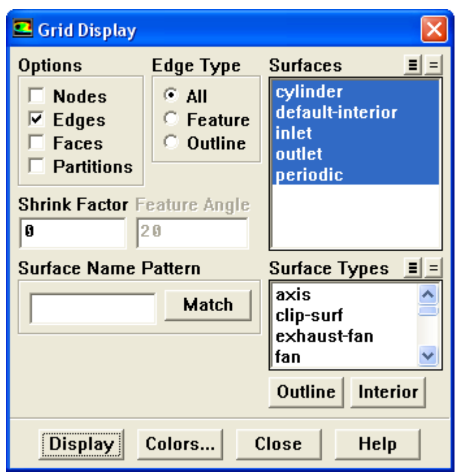


Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units. In this case, do not scale the grid.

(a) Close the Scale Grid panel.

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

Display —→Grid...



(a) Click Display and close the Grid Display panel.

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.

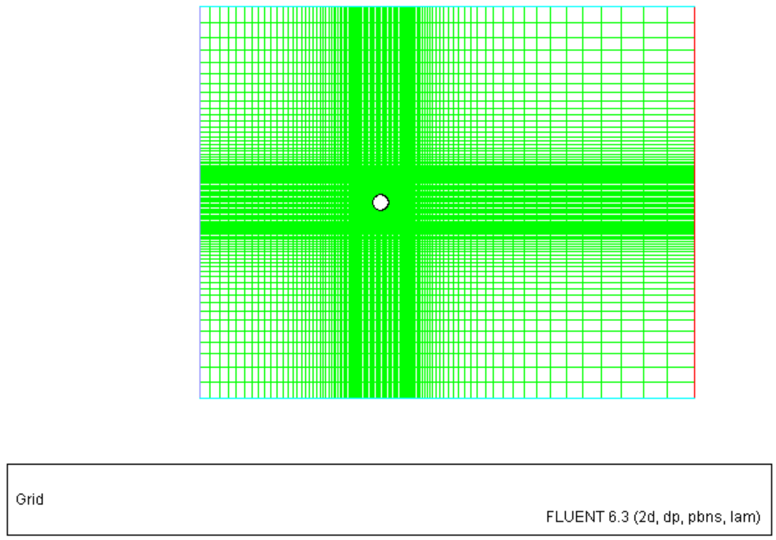


Figure 6.2: Grid Display

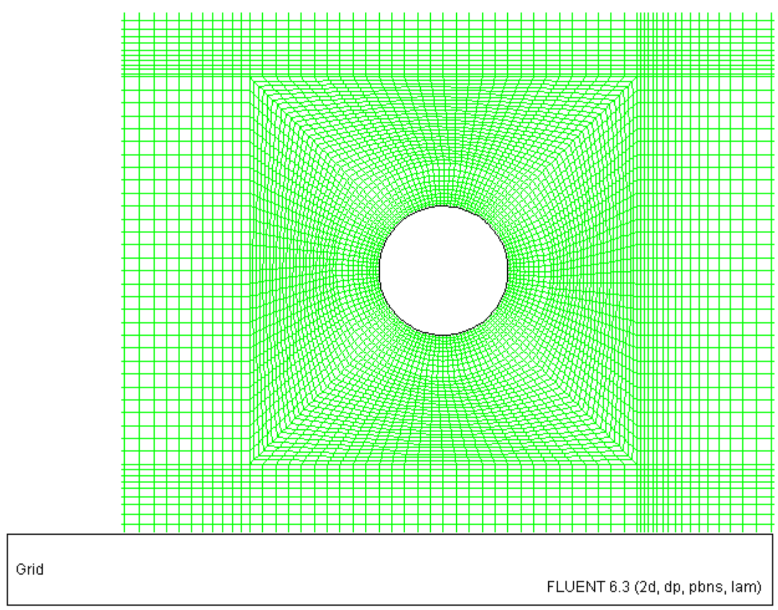


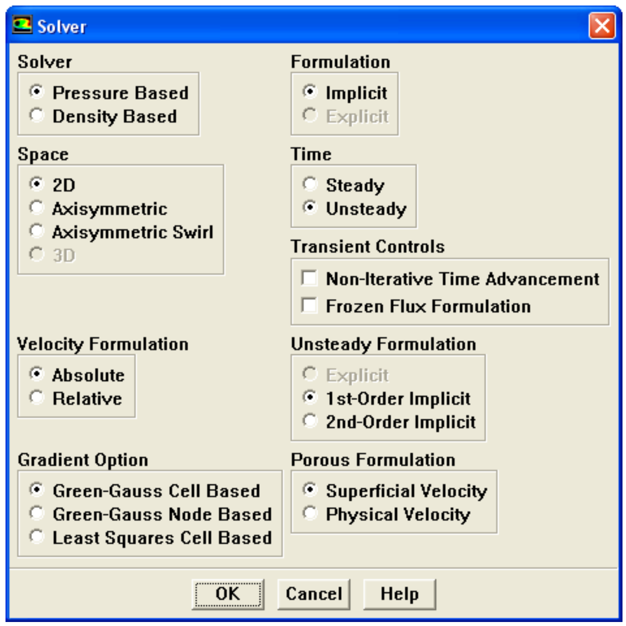
Figure 6.3: Grid Display—Zoom-in View

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. Set up the unsteady solver settings.

   Define ⟶ Models ⟶ Solver...



   (a) Select Unsteady in the Time group box.

   (b) Click OK to close the Solver panel.

Step 3: Materials

1. Change the material properties.

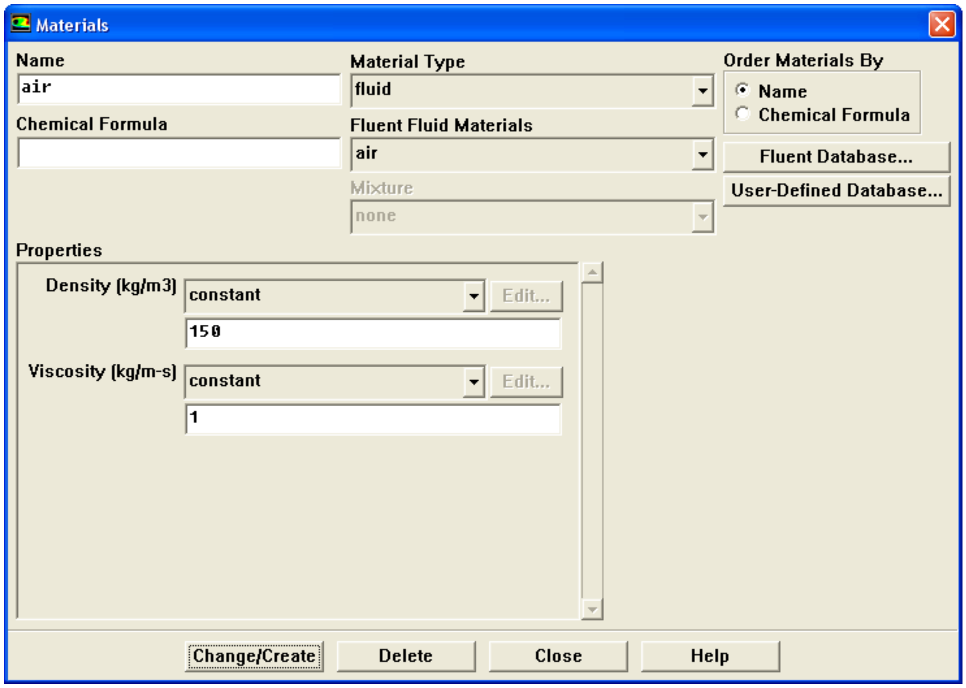
   Define ⟶ Materials...

   (a) Enter 150 for Density and 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 U is unity. Therefore, set the value of density same as the Reynolds number.

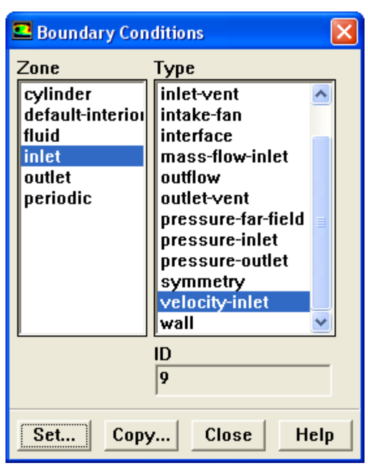


(b) Click Change/Create and close the Materials panel.

Step 4: Boundary Conditions

1. Set the boundary conditions for inlet.

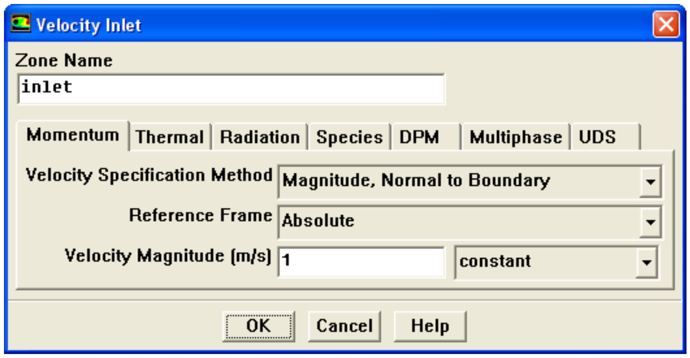
   Define ⟶Boundary Conditions...



2. Select inlet from the Zone selection list.

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

3. Click the Set... button to open the Velocity Inlet panel.



(a) Enter 1 m/s for Velocity Magnitude.

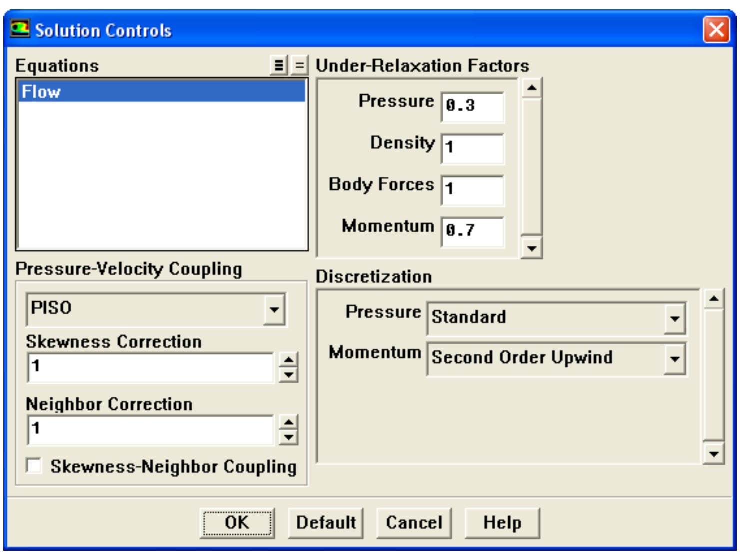
(b) Click OK to close the Velocity Inlet panel.

4. Close the Boundary Conditions panel.

Step 5: Solution

1. Set the solution controls.

   Solve ⟶ Controls ⟶ Solution...



(a) 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 the recommended pressure-velocity coupling for solving transient applications.

(b) Disable Skewness-Neighbor Coupling.

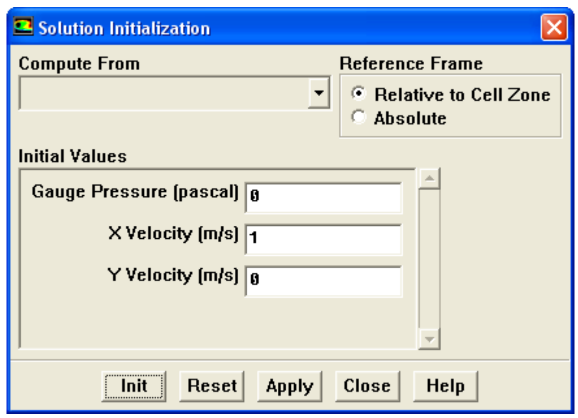
(c) Select Second Order Upwind from the Momentum drop-down list in the Discretization group box.

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

(d) Click OK to close the Solution Controls panel.

2. Initialize the flow.

Solve ⟶ Initialize ⟶ Initialize...

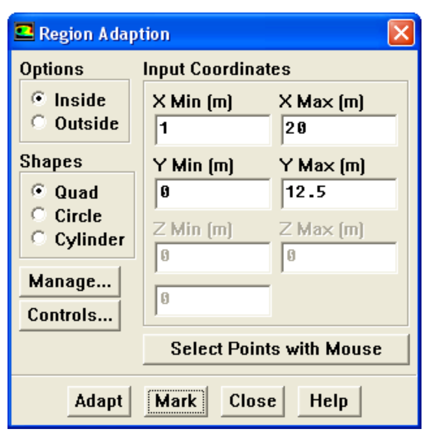


(a) Enter 1 for X Velocity.

(b) Click Init and close the Solution Initialization panel.

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

Adapt ⟶ Region...



(a) Enter 1 m and 20 m for X Min and X Max, respectively.

(b) Enter 0 and 12.5 for Y Min and Y Max, respectively.

(c) Click Mark.

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

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

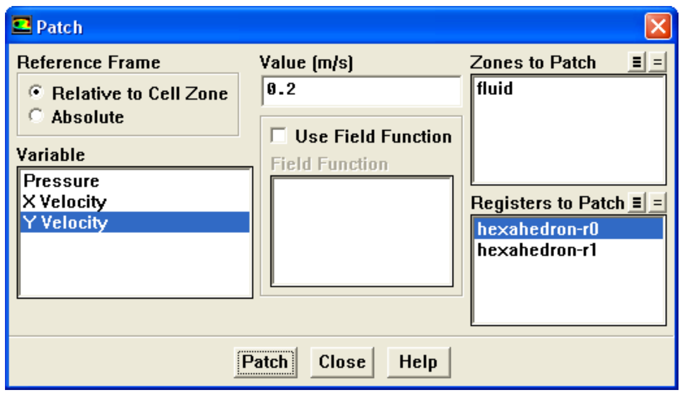
(d) Enter `-12.5` and `0` for Y Min and Y Max, respectively and retain the previous values for X Min and X Max.

(e) Click Mark.

(f) Close the Region Adaption panel.

4. Patch the Y velocity.

Solve ⟶ Initialize ⟶ Patch...



(a) Select `hexahedron-r0` from the Registers to Patch selection list.

(b) Select Y Velocity from the Variable selection list.

(c) Enter `0.2` for Value.

(d) Click Patch.

(e) Deselect `hexahedron-r0` and select `hexahedron-r1` from the Registers to Patch selection list.

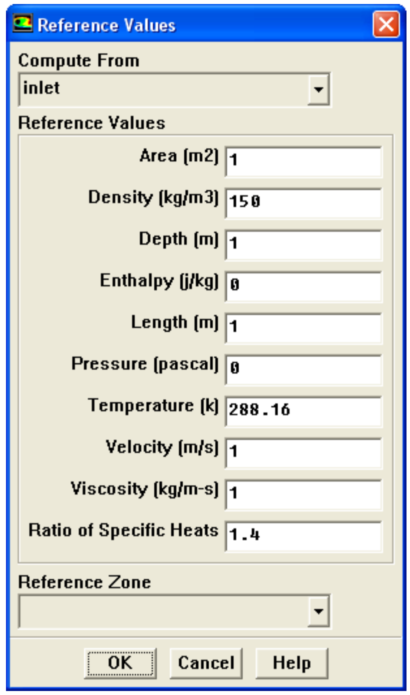
(f) Enter `-0.2` for Value.

(g) Click Patch and close the Patch panel.

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...



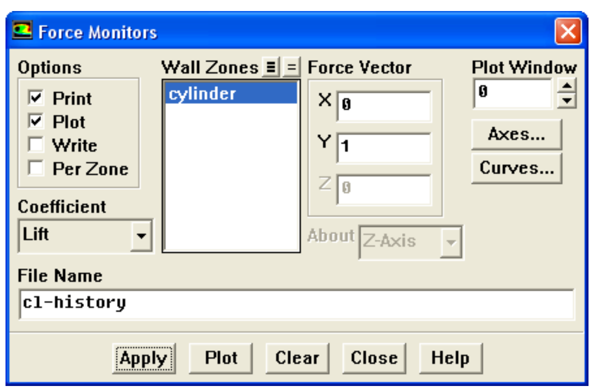
(a) Select inlet from the Compute From drop-down list.

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

(b) Click OK to close the Reference Values panel.

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

Solve ⟶ Monitors ⟶ Force...



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

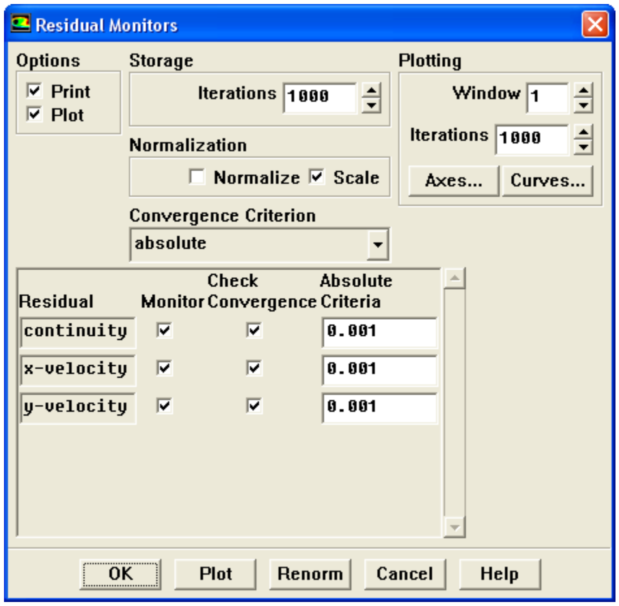
(b) Select cylinder from the Wall Zones selection list.

(c) Enable Print and Plot in the Options group box.

(d) Click Apply and close the Force Monitors panel.

7. Enable plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...

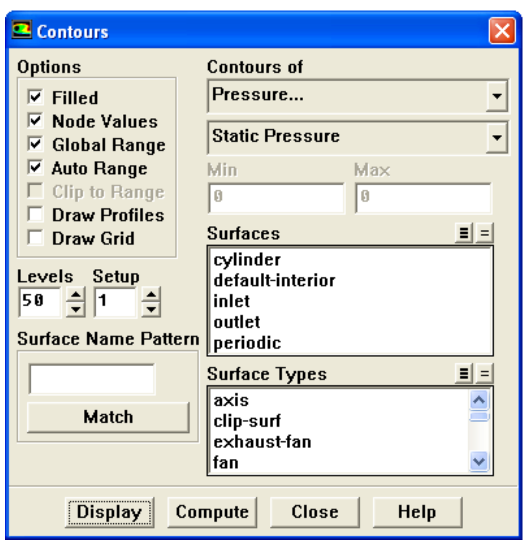


   (a) Enable Plot in the Options group box.

   (b) Click OK to close the Residual Monitors panel.

8. Set animation to visualize vortex shedding.

   Display ⟶ Contours...



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

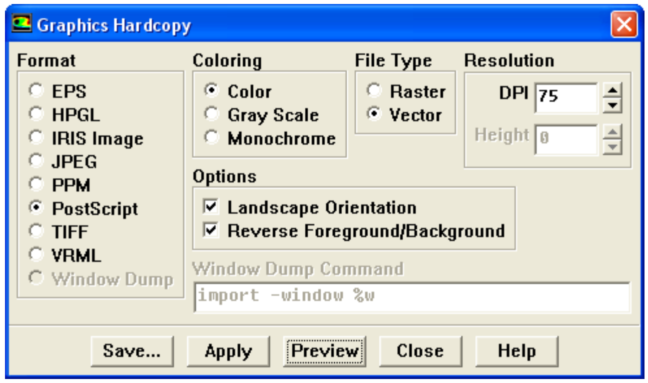
   (b) Enable Filled in the Options group box.

(c) Enter 50 for Levels.

(d) Click Display and close the Contours panel (Figure 6.4).

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

$\boxed{\text{File}}$ $\longrightarrow$ Hardcopy...



i. Select Color from the Coloring group box and Vector from the File Type group box.

ii. Click Preview.

A Question *dialog box appears. Click* No.

iii. Click Apply and close the Graphics Hardcopy panel.

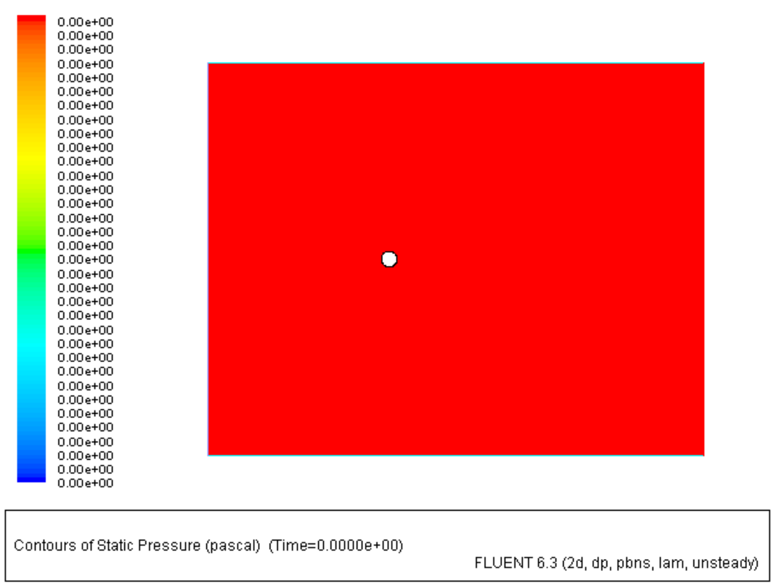
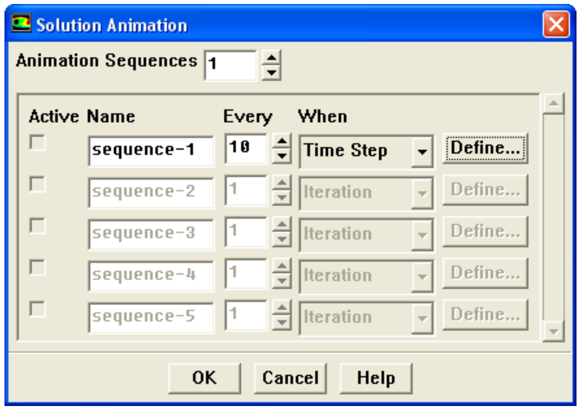


Figure 6.4: Contours of Static Pressure

9. Set the animation controls.

$\boxed{\text{Solve}}$ $\longrightarrow$ $\boxed{\text{Animate}}$ $\longrightarrow$ Define...

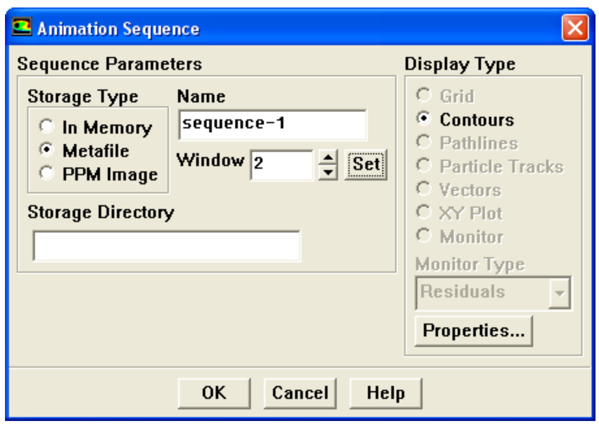


(a) Increase the Animation Sequences to `1`.

(b) Enter `10` for Every.

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

(d) Click Define... for `sequence-1` to open the Animation Sequence panel.



i. Increase Window to `2` and click the Set button to open a graphics window.

ii. Select Contours from the Display Type list to open the Contours panel.

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

iv. Enter `50` for Levels.

v. Click Display and close the Contours panel.

vi. Adjust the view as shown in Figure 6.5.

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.

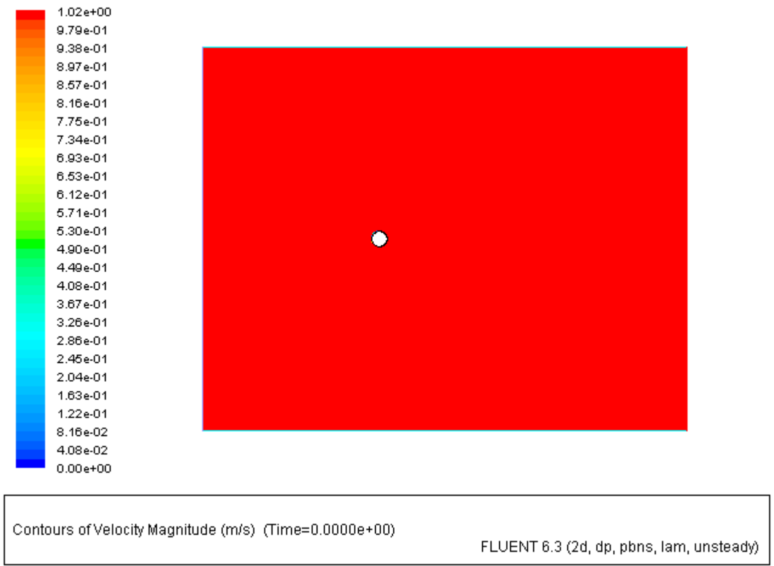
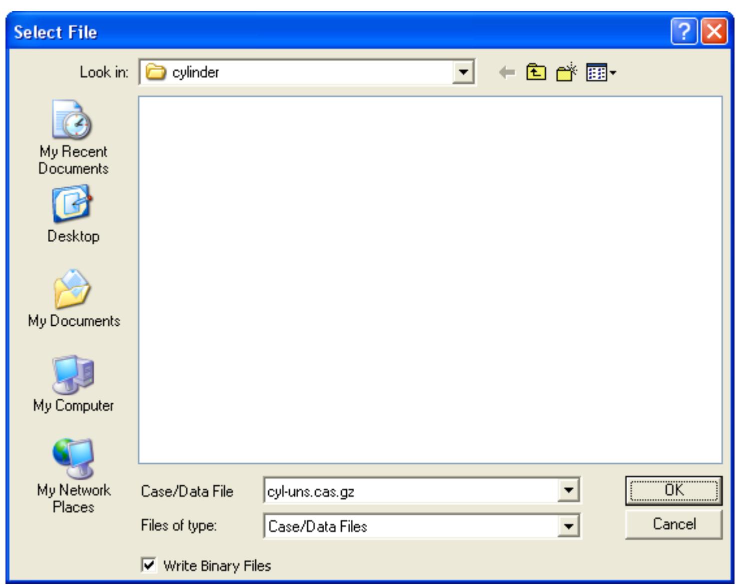


Figure 6.5: Contours of Velocity Magnitude

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

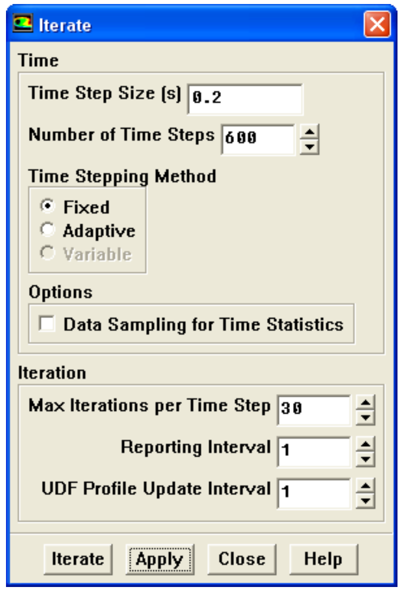
    File ⟶ Write ⟶ Case & Data...



Retain the default enabled 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* Linux/UNIX *platforms.*

11. Iterate the solution.

    Solve ⟶ Iterate...



(a) Enter `0.2` for Time Step Size.

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,

$$D = 1, U = 1 \tag{6.3}$$

Therefore,

$$f = 0.2 \tag{6.4}$$

Cycle time,

$$t = \frac{1}{f} = \frac{1}{0.2} = 5\ sec \tag{6.5}$$

Therefore,

$$time\ step\ size = \frac{5}{25} = 0.2\ sec \tag{6.6}$$

(b) Enter 30 for Max. Iterations per Time Step.

(c) Enter 600 for Number of Time Steps.

(d) Click Apply.

(e) Click Iterate to start the iterations.

(f) Close the Iterate panel.

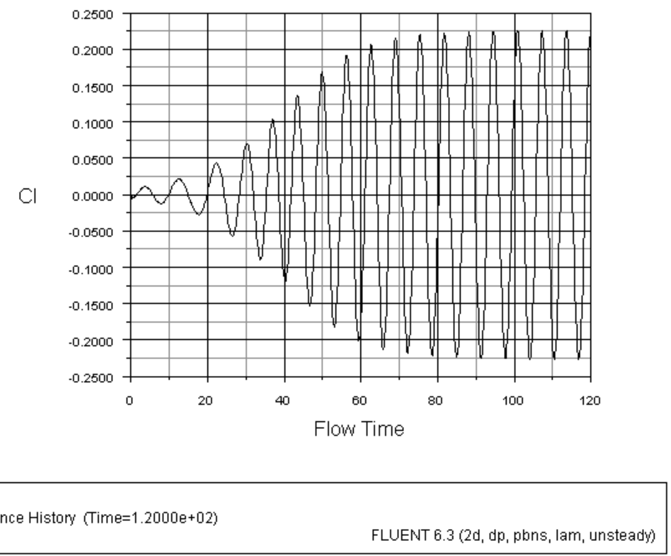


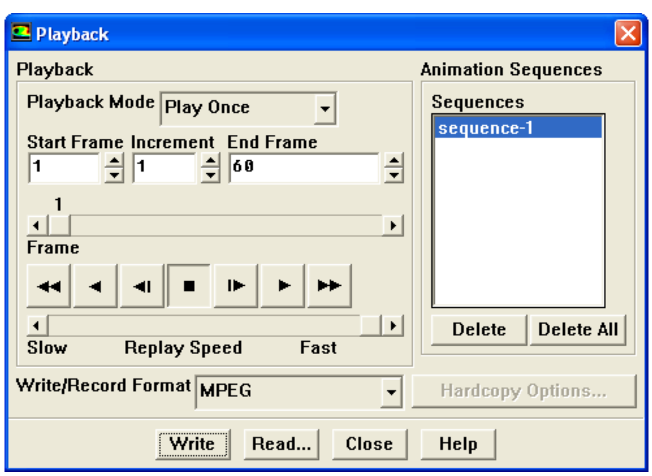
Figure 6.6: Lift Coefficient Plot

Figure 6.6 shows a clear sinusoidal pattern, which 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 from the Write/Record Format drop-down list.

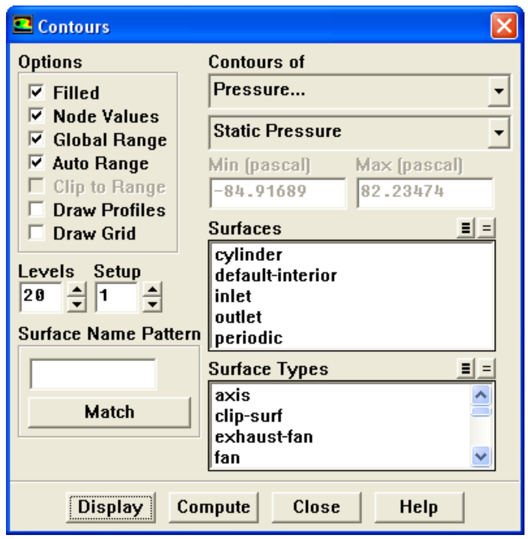
(b) Click Write and close the Playback panel.

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

Step 6: Postprocessing

1. Display the pressure contours (Figure 6.7).

   Display ⟶Contours...



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

   (b) Set the number of Levels to 20.

   (c) Click Display (Figure 6.7).

   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).

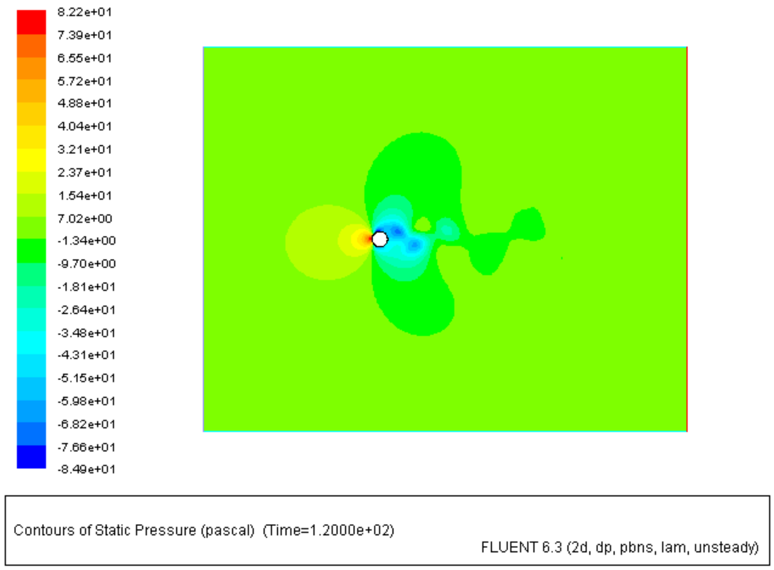
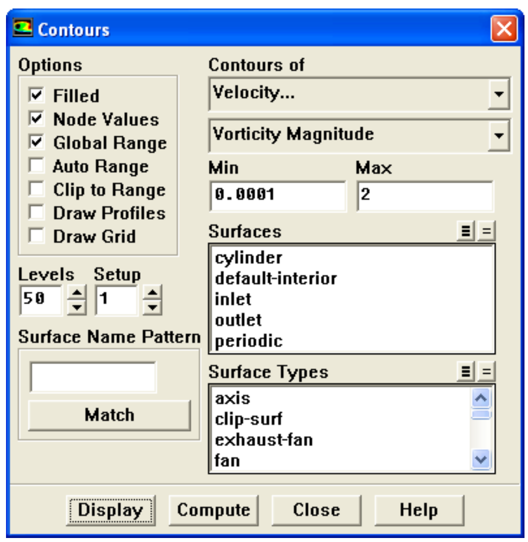


Figure 6.7: Contours of Static Pressure



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

(b) Disable Auto Range and Clip to Range from the Options group box.

(c) Enter `0.0001` and 2 for Min and Max, respectively.

(d) Set Levels to 50.

(e) Click Display (Figure 6.8).

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

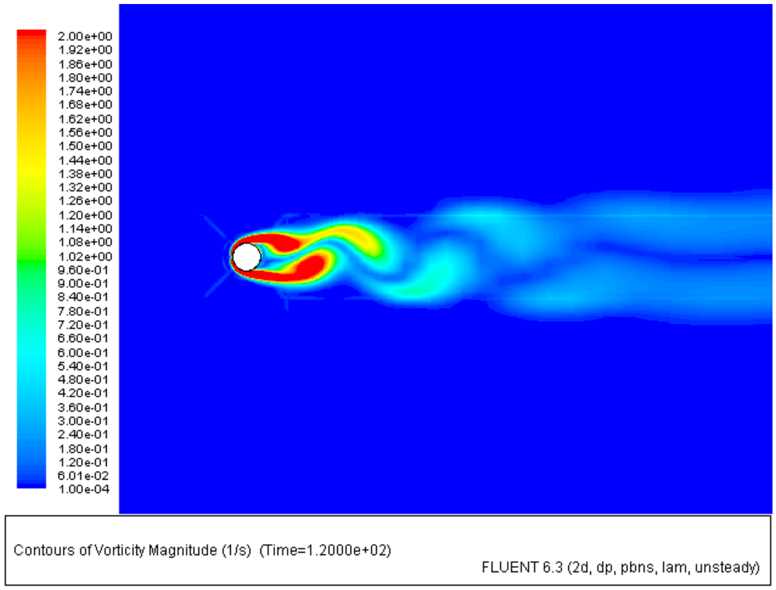
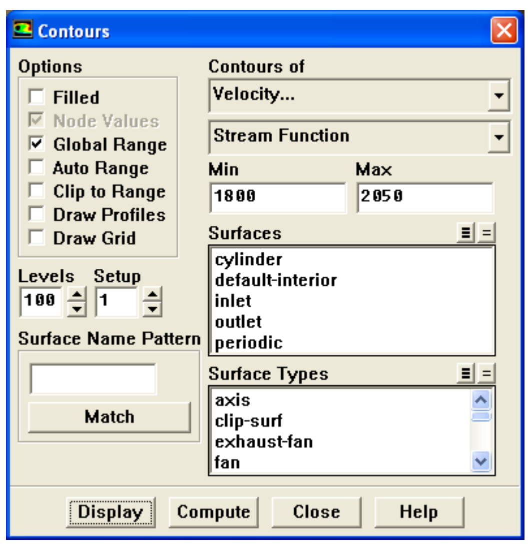


Figure 6.8: Contours of Vorticity Magnitude

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



(a) Select Velocity... and Stream Function from the Contours of drop-down lists.

(b) Disable Filled, Auto Range and Clip to Range from the Options group box.

(c) Enter 1800 and 2050 for Min and Max, respectively.

(d) Set Levels to 100.

(e) Click Display (Figure 6.9).

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.

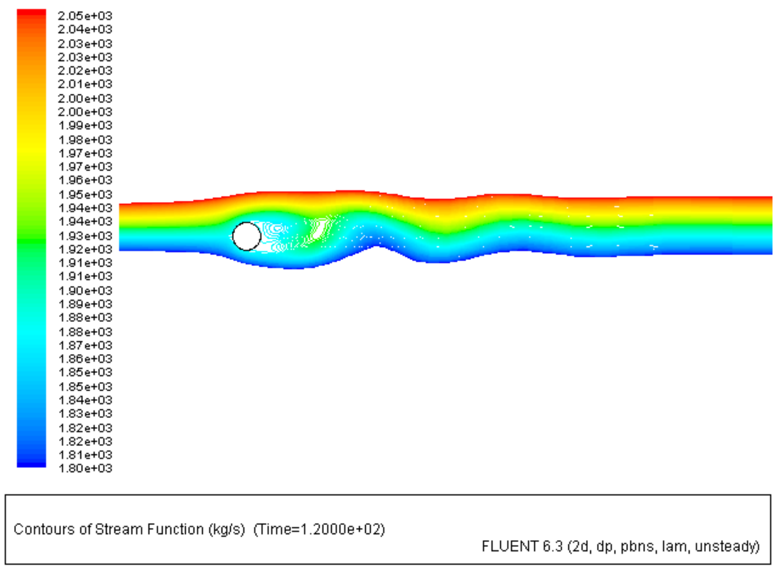


Figure 6.9: Contours of Stream Function

4. Close the Contours panel.

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://mec-mail.eng.monash.edu.au/ mct/pubs/pdfs/ReHoTh05_jfm.pdf
- http://www.mate.tue.nl/mate/pdfs/889.pdf