### 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.

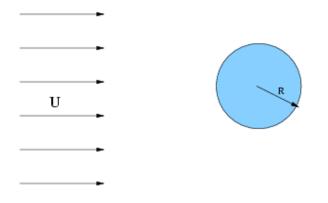


Figure 6.1: Problem Schematic

## 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.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case...}$ 

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

2. Check the grid.

 $Grid \longrightarrow Check$ 

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.

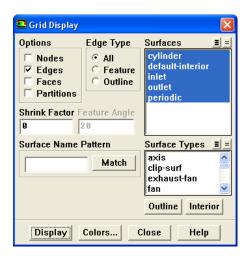
 $Grid \longrightarrow Scale...$ 

Scale Grid			
Scale Factors	Unit Conversion		
× 1	Grid Was Created In m		
Y 1	Change Length Units		
Domain Extents			
Xmin (m) -11.5 Xmax (m) 20			
Ymin (m) -12.5	Ymax (m) 12.5		
Scale	Jnscale Close Help		

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).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 



(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.

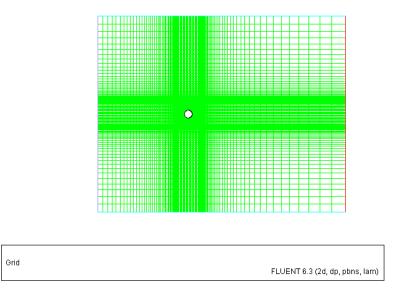


Figure 6.2: Grid Display

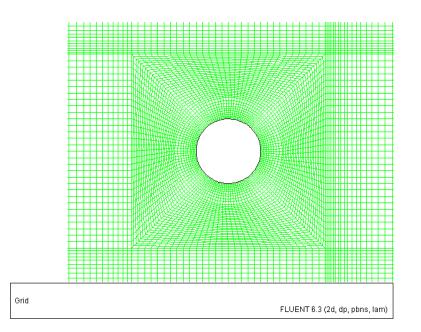
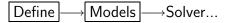


Figure 6.3: Grid Display—Zoom-in View

### 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.



Solver	Formulation
<ul> <li>Pressure Based</li> <li>Density Based</li> </ul>	<ul> <li>Implicit</li> <li>Explicit</li> </ul>
Space	Time
• 2D • Axisymmetric • Axisymmetric Swirl	<ul> <li>C Steady</li> <li>⑦ Unsteady</li> </ul>
Č 3D	Transient Controls
	Non-Iterative Time Advancement Frozen Flux Formulation
Velocity Formulation	Unsteady Formulation
Absolute	C Explicit
○ Relative	<ul> <li>1 st-Order Implicit</li> <li>2nd-Order Implicit</li> </ul>
Gradient Option	Porous Formulation
<ul> <li>Green-Gauss Cell Based</li> <li>Green-Gauss Node Based</li> <li>Least Squares Cell Based</li> </ul>	<ul> <li>Superficial Velocity</li> <li>Physical Velocity</li> </ul>

- (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  $\longrightarrow$  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  $\mu$ ,  $D_i$ , and U is unity. Therefore, set the value of density same as the Reynolds number.

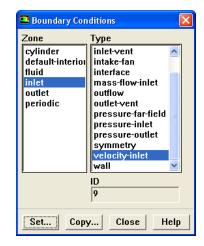
ame	Material Type	Order Materials By
air	fluid	▼ ● Name
hemical Formula	Fluent Fluid Materials	C Chemical Formula
	air	<ul> <li>Fluent Database</li> </ul>
	Mixture	User-Defined Database
	none	
roperties		
Density (kg/m3) consta	t Edit	
150		
Viscosity (kg/m-s)	t Edit	
1		
	-	

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

## **Step 4: Boundary Conditions**

1. Set the boundary conditions for inlet.

Define → Boundary Conditions...



- 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.

Velocity Inlet	×
Zone Name inlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	s
Velocity Specification Method Magnitude, Normal to Boundary	•
Reference Frame Absolute	-
Velocity Magnitude (m/s) 1 constant	<u> </u>
OK Cancel Help	

- (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.

|--|

Solution Controls	Σ
Equations	J = Under-Relaxation Factors
Flow	Pressure 0.3
	Density 1
	Body Forces 1
	Momentum 0.7
Pressure-Velocity Coupling	Discretization
PISO .	Pressure Standard
Skewness Correction 1	Momentum Second Order Upwind -
Neighbor Correction	
Skewness-Neighbor Coupli	
ОК	Default Cancel Help

(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	$\longrightarrow$	Initialize	—→Initialize…
Solve	$\rightarrow$	IIIIIIaiize	—→IIIItiaIIZe…

Solution Initialization	X
Compute From	Reference Frame
J	Relative to Cell Zone     Absolute
Initial Values	
Gauge Pressure (pascal) 🔋	
X Velocity (m/s) 1	
Y Velocity (m/s) 👩	
	v.
Init Reset App	oly Close Help

- (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.  $\boxed{\mathsf{Adapt}} \longrightarrow \mathsf{Region}...$

Region Adaption		X
Options	Input Coordinat	es
<ul> <li>Inside</li> <li>Outside</li> </ul>	X Min (m) 1	X Max (m) 20
Shapes	Y Min (m)	Y Max (m)
• Quad	0	12.5
Circle Cylinder	Z Min (m)	Z Max (m)
Manage	0	
Controls	1	
	Select Point	s with Mouse
Adapt	Mark Clos	e Help

- (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 $\longrightarrow$ Initialize	$\longrightarrow$ Patch
------------------------------------	-------------------------

Patch		X
Reference Frame  Relative to Cell Zone Absolute Variable Pressure X Velocity Y Velocity	Value (m/s) Ø.2 Use Field Function Field Function	Zones to Patch = = fluid Registers to Patch = = hexahedron-r0 hexahedron-r1
	Patch Close Help	

- (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  $\longrightarrow$  Reference Values...

Reference Values
Compute From
inlet 🔹
Reference Values
Area (m2) 1
Density (kg/m3) 150
Depth (m) 1
Enthalpy (j/kg) 👔
Length (m) 1
Pressure (pascal)
Temperature (k) 288.16
Velocity (m/s) 1
Viscosity (kg/m-s) 1
Ratio of Specific Heats 1.4
Reference Zone
<u> </u>
OK Cancel Help

(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	$\longrightarrow$	Monitors	—→Force…
-			

Force Monitor	S	X
Options Print Plot Vrite Per Zone Coefficient Lift File Name	Cylinder     Force Vector       Y     1       Z     0       About     Z-Axis	Plot Window 9 Axes Curves
cl-history		
Арр		lelp

- (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  $\longrightarrow$  Monitors  $\longrightarrow$  Residual...

💶 Residual M	onitors	×			
Options	Storage	Plotting			
<ul><li>✓ Print</li><li>✓ Plot</li></ul>	Iterations 1000				
	Normalization	Iterations 1000			
	🗌 Normalize 🗹	Scale Axes Curves			
	Convergence Criterion				
	absolute	•			
Residual	Check 4 Monitor Convergence (	Absolute 🔶 Criteria			
continuity		0.001			
x-velocity		0.001			
y-velocity		0.001			
		<b>v</b>			
0	OK Plot Renorm Cancel Help				

- (a) Enable Plot in the Options group box.
- (b) Click  $\mathsf{OK}$  to close the Residual Monitors panel.
- 8. Set animation to visualize vortex shedding.

 $\mathsf{Display} \longrightarrow \mathsf{Contours}...$ 

Contours	X
Options	Contours of
Filled	Pressure
✓ Node Values ✓ Global Range	Static Pressure 🗸
🗹 Auto Range	Min Max
Clip to Range	0
Draw Grid	Surfaces 📃 📃
Levels Setup 50 1 1	cylinder default-interior inlet outlet
	periodic Surface Types I =
Match	axis clip-surf exhaust-fan gan
Display Co	mpute Close Help

- (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.
  - $\mathsf{File} \longrightarrow \mathsf{Hardcopy...}$

🗳 Graphics Hardcopy 🛛 🛛 🔀					
Format	Coloring	File Type	Resolution		
C EPS	Color	C Raster	DPI 75		
○ HPGL ○ IRIS Image	<ul> <li>Gray Scale</li> <li>Monochrome</li> </ul>	Vector	Height 0		
O JPEG	Options	J			
○ PPM ● PostScript	Landscape Or	ientation			
C TIFF	Reverse Fore	ground/Backg	round		
C VRML C Window Dump	Window Dump Co	mmand			
import -window %w					
Save	Apply Previe	w Close	Help		

- 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.

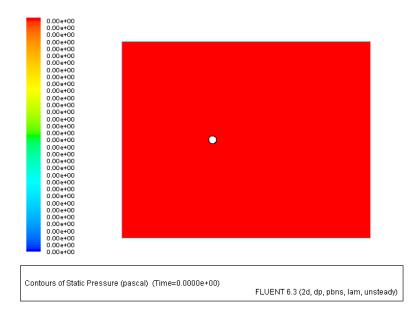


Figure 6.4: Contours of Static Pressure

9. Set the animation controls.

$$\boxed{\mathsf{Solve}} \longrightarrow \boxed{\mathsf{Animate}} \longrightarrow \boxed{\mathsf{Define}} \dots$$

Solution Animation						×
Animati	Animation Sequences 1					
Active	Name	Ever	ry When			
	sequence-1	10	Time Step	•	Define	
	sequence-2	1	tteration	-	Define	
	sequence-3	1	tteration	-	Define	
	sequence-4	1	▲ Iteration	-	Define	
	sequence-5	1	tteration	-	Define	-
OK Cancel Help						
OK Cancel Help						

- (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.

Animation Seque	ence	×
Sequence Parame	eters	Display Type
Storage Type C In Memory Metafile O PPM Image Storage Director	Name sequence-1 Window 2 2 Set	<ul> <li>○ Grid</li> <li>○ Contours</li> <li>○ Pathlines</li> <li>○ Particle Tracks</li> <li>○ Vectors</li> <li>○ XY Plot</li> <li>○ Monitor</li> <li>Monitor Type</li> <li>Residuals</li> <li>▼</li> <li>Properties</li> </ul>
OK Cancel Help		

- 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  $50\ {\rm for}\ Levels.$
- 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.

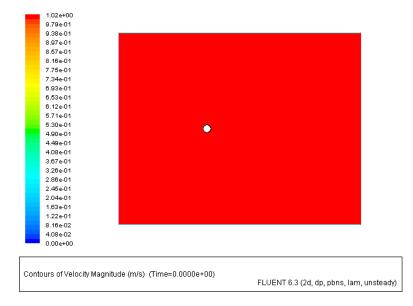


Figure 6.5: Contours of Velocity Magnitude

10. Save the case and data files (cyl-uns.cas.gz and cyl-uns.dat.gz).
File → Write → Case & Data...

Select File					? 🛛
Look in:	🗀 cylinder		•	+ 🗈 💣 🖩	]-
My Recent Documents					
Desktop					
My Documents					
My Computer					
My Network Places	Case/Data File Files of type:	cyl-uns.cas.gz Case/Data Files		- -	(OK) Cancel
	Write Binary Fil	,		_	

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.

 $Solve \longrightarrow Iterate...$ 

🖴 Iterate 🛛 🔀		
Time		
Time Step Size (s) 0.2		
Number of Time Steps 688		
Time Stepping Method		
• Fixed		
C Adaptive C Variable		
Options		
☐ Data Sampling for Time Statistics		
Iteration		
Max Iterations per Time Step 30		
Reporting Interval 1		
UDF Profile Update Interval		
Iterate Apply Close Help		

#### (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,

$$f = 0.2$$
 (6.4)

Cycle time,

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

Therefore,

time step size 
$$=$$
  $\frac{5}{25} = 0.2 \ sec$  (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.

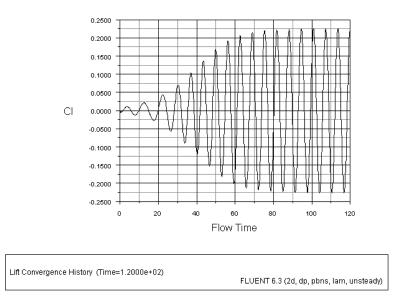


Figure 6.6: Lift Coefficient Plot

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/(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.

 $\fbox{Solve} \longrightarrow \fbox{Animate} \longrightarrow \r{Playback}...$ 

Search Playback	
Playback	Animation Sequences
Playback Mode Play Once 👻	Sequences
Start Frame Increment End Frame	sequence-1
1 Frame	
Slow Replay Speed Fast	Delete Delete All
Slow Replay Speed Fast	L]
Write/Record Format MPEG	Hardcopy Options
Write Read Close	Help

(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  $\longrightarrow$  Contours...

Contours	X			
Options	Contours of			
✓ Filled	Pressure 👻			
✓ Node Values ✓ Global Range	Static Pressure			
🗹 Auto Range	Min (pascal) Max (pascal)			
Clip to Range	-84.91689 82.23474			
Draw Profiles Draw Grid	Surfaces 📃 =			
Levels Setup 20 1 1 Surface Name Pattern	cylinder default-interior inlet outlet periodic			
	Surface Types 📃 📃			
Match	axis A clip-surf exhaust-fan fan V			
Display Compute Close Help				

- (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).

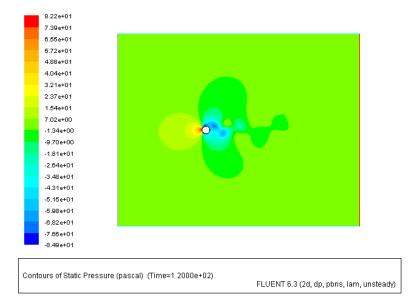


Figure 6.7: Contours of Static Pressure

Contours		×	
Options	Contours of		
🗹 Filled	Velocity	<b>-</b>	
✓ Node Values ✓ Global Range	Vorticity Magnitude		
Auto Range	Min	Max	
Clip to Range	0.0001	2	
Draw Grid	Surfaces	E	
Levels Setup	cylinder default-interior		
50 1 4	inlet		
Surface Name Patteri	outlet		
	periodic		
	Surface Types	<u>= =</u>	
	axis	~	
Match	clip-surf		
	exhaust-fan fan	~	
	Iran		
Display Co	mpute Close	Help	

- (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.

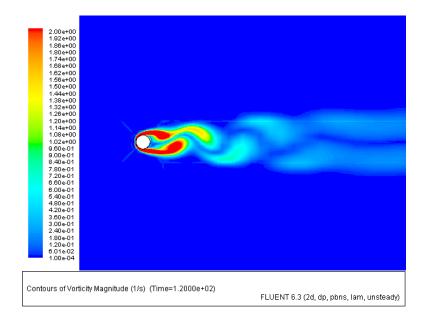


Figure 6.8: Contours of Vorticity Magnitude

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

Contours		×
Contours Options Filled Global Range Auto Range Clip to Range Draw Profiles Draw Grid Levels Setup 100 1 1 Surface Name Pattern Match	Contours of Velocity Stream Function Min 1800 Surfaces Cylinder default-interior inlet outlet periodic Surface Types axis clip-surf	X Max 2050 = = =
	exhaust-fan fan	<b>~</b>
Display Compute Close Help		

- (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.

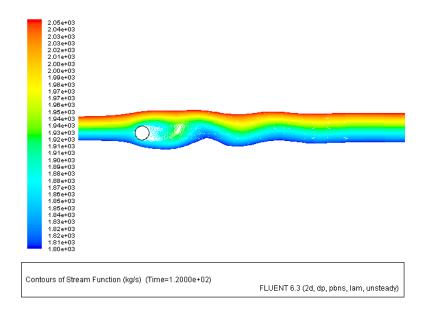


Figure 6.9: Contours of Stream Function

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