Problem Specification.
1. Create Geometry in GAMBIT.
2. Mesh Geometry in GAMBIT.
3. Specify Boundary Types in GAMBIT.
4. Set Up Problem in FLUENT.
5. Solve.
6. Analyze Results.
7. Change the domain size.
8. Unsteady Flow.
Problem Set Citations

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.

	Solver
Solver	Formulation
 Pressure Based Density Based 	 ◆ Implicit ◇ Explicit
Space	Time
◆ 2D ◇ Axisymmetric	 ◇ Steady ◆ Unsteady
Axisymmetric Swirl	Transient Controls
	Non-Iterative Time Advancement Frozen Flux Formulation
Velocity Formulation	Unsteady Formulation
 ◆ Absolute ◇ Relative 	 Explicit 1st-Order Implicit 2nd-Order Implicit
Gradient Option	Porous Formulation
 ◆ Green-Gauss Cell Based ◇ Green-Gauss Node Based ◇ Least Squares Cell Based 	Superficial Velocity Physical Velocity

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.

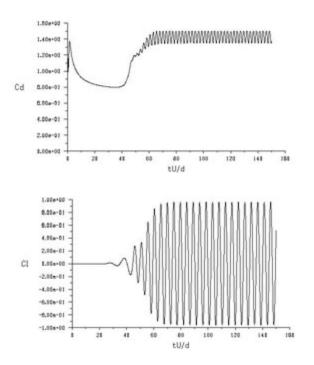
	Force M	onitors	×
Options Print Plot Vrite Per Zone Coefficient	Wall Zones ≣ ≝ cylinder wall	Force Vector X 1 Y 0 Z 0 About Z Axis Y	Plot Window 1 Axes Curves
File Name •ses/ME263/c	ylinder/mesh/	Final_50_50_120)/300Re_cd

4.Iteration

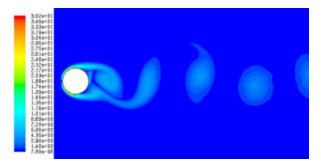
	Iterate	
ime		
Time Ste	ep Size (s) 0.05	
	r of Time Steps 3000	
	epping Method	
Fixe		
♦ Ada		
♦ Variation	iable	
Options	\$	
🗆 Data	a Sampling for Time Statistic	s
teration		
Max Iter	rations per Time Step 20	
	Reporting Interval 1	
UDF P	rofile Update Interval 1	
Iterate	Apply Close H	lelp

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).

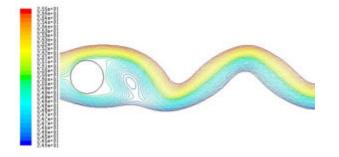
The drag and lift coefficients as functions of nondimensional time, tU/d, for Re=300 are shown below. The mean values of Cd and Cl are taken after the periodic oscillations are established.



Plot the vorticity contour.



Plot streamlines.



5. Making animations

Solve> Animate> Define ...

Active	Name	Every		When		
1	sequence-1	10	÷	Time Step	Y	Define
1	sequence-2	1	A Y	Iteration	y	Define
J	sequence-3	1	A Y	Iteration	y	Define
1	sequence-4	1	A	Iteration	y	Define
i i	sequence-5	1	4	Iteration	V	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...**

Sequence Paramo	ters	Display Type
Storage Type Name In Memory sequence-1 Metafile Window 3 PPM Image Storage Directory		Grid Contours Pathlines Particle Tracks Vectors XY Plot Monitor Monitor Type
		Residuals y

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**:

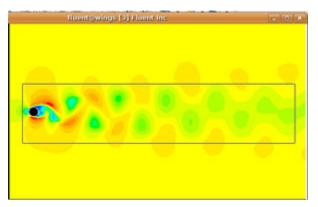
	Contours		8
Options	Contours of		17100
Filled	Velocity		Y
Node Values	Velocity Mag	nitude	Y
Global Range	Min (m/s)	Max (m/s)	
Auto Range	0	1.490337	
L Clip to Range	Surfaces		83
Draw Profiles Draw Grid Levels Setup 20 1	cylinder default-interio inlet outlet wall	or	
Surface Name Patter	n Surface Type:	s	83
Match	axis clip-surf exhaust-fan fan		

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.

Playback	
Playback	Animation Sequences
Playback Mode Play Once	Sequences
Slow Replay Speed Fast	Delete Delete All
Write/Record Format MPEG	Hardcopy Options.

Go to Problem Set.

See and rate the complete learning module.

Go to all FLUENT Learning Modules