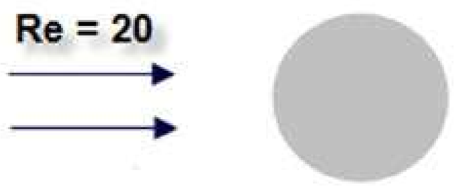
course on ANSYS

FLUENT - Steady Flow Past a Cylinder - Problem Specification

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification



The purpose of this tutorial is to illustrate the setup and solution of a steady flow past a circular cylinder. Flow past a circular cylinder is one of the classical problems of fluid mechanics. For this problem, we will be looking at Reynolds number of 20.
Unable to find DVI conversion log file.

We know D = 2 m. To obtain *Re* = 20, we can arbitrarily set ρ, *V* and μ. For our case, let's set ρ = 1 kg/m^3, *V* = 1 m/s and μ = 0.1 kg/ms.

Preliminary Analysis

For *Re* = 20, we are looking at steady laminar flow. What will be the velocity profile of this flow? What will be the drag coefficient of the cylinder? What will be the pressure coefficient around cylinder? How will the streamlines around cylinder look like?

Let's start the modeling in our quest to find out the answer!

We'll create the geometry and mesh in GAMBIT which is the preprocessor for FLUENT, and then read the mesh into FLUENT and solve for the flow solution.

Go to Step 1: Create Geometry in GAMBIT

See and rate the complete Learning Module

FLUENT - Steady Flow Past a Cylinder - Step 1

Step 1: Create Geometry in GAMBIT

In an external flow such as the flow past a cylinder, we have to define farfield boundaries and mesh the region between the cylinder geometry and the boundaries. Farfield boundaries should be placed well away from the cylinder such that the boundary conditions will not affect the flow near cylinder.

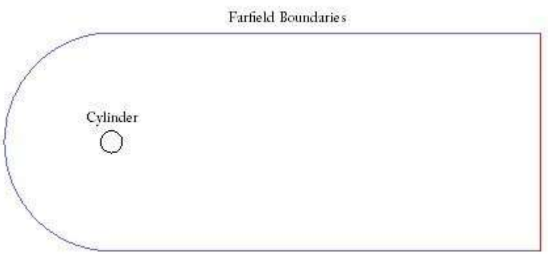
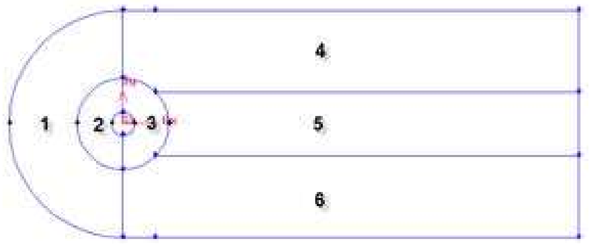


Figure above shows the geometry of such a case.

Strategy for Creating Geometry

To model this flow, we need a cylinder and farfield boundaries. We need finer meshes around the cylinder to capture the active region (call this radius of influence). Downstream of the flow, there will be wake generated by the cylinder, which requires finer mesh to better capture this phenomena. To be able to specify such regions, we split the domain into different faces as shown below.



We set the geometry upstream to be shorter because we have less activity before flow through cylinder. We set the geometry downstream of the cylinder to be relatively longer such that the boundary conditions will not affect the flow near cylinder.

Create a Working Directory

Create a folder called *cylinder* in a convenient location. We'll use this as the working folder in which files created during the session will be stored.

Start GAMBIT

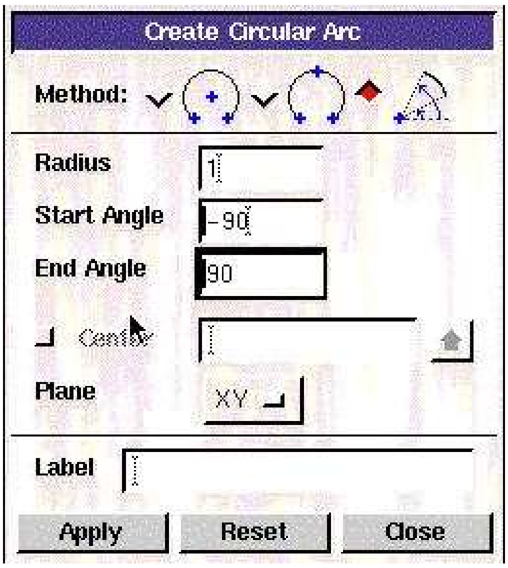
Create a new directory called *cylinder* and start *GAMBIT* from that directory by typing gambit -id cylinder at the command prompt.

Under ***Main Menu***, select **Solver > FLUENT 5/6** since the mesh to be created is to be used in FLUENT 6.0.

Create Cylinder

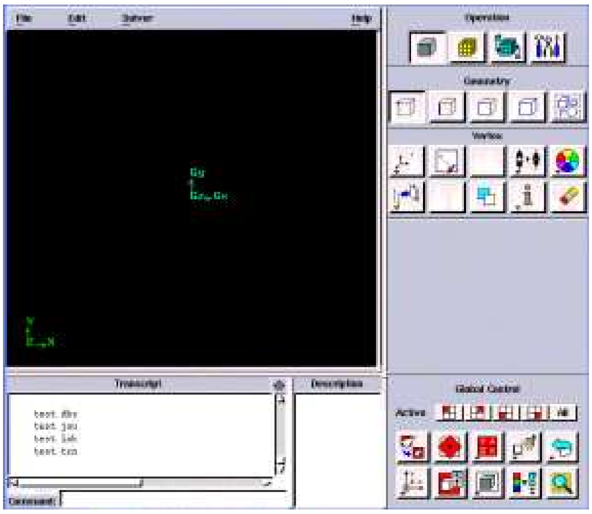
Create the cylinder using two arcs. The cylinder is created with two arc because they are going to be meshed differently. Back arc spans from -90 to 90 deg. Front arc spans from 90 to -90 deg. Both arc with radius 1.

Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge > Arc >



Next to ***Radius***, enter 1. Next to ***Start Angle***, enter -90. Next to ***End Angle***, enter 90. Click ***Apply***. Do the same for front arc but enter different value for angles.

Always make arc in counterclock wise direction

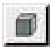
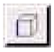


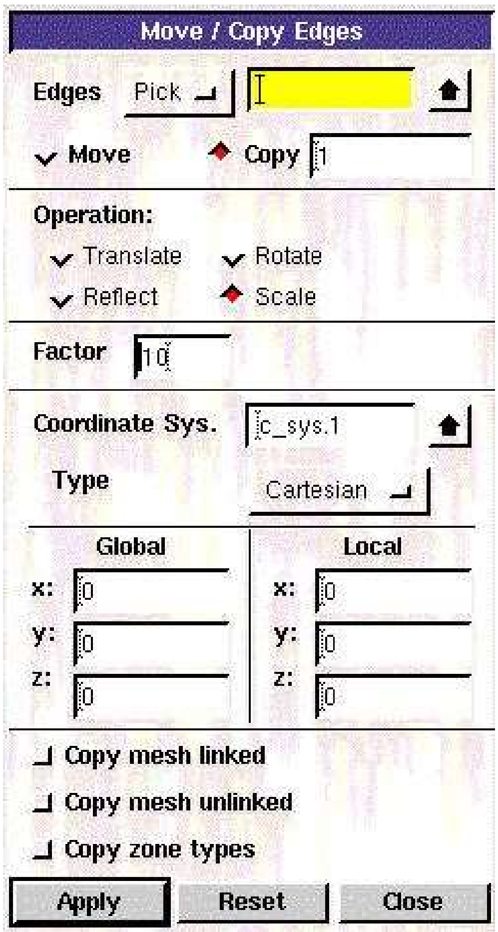
(Click here for animated steps)

Right mouse click on Create Edge icon to see more options.

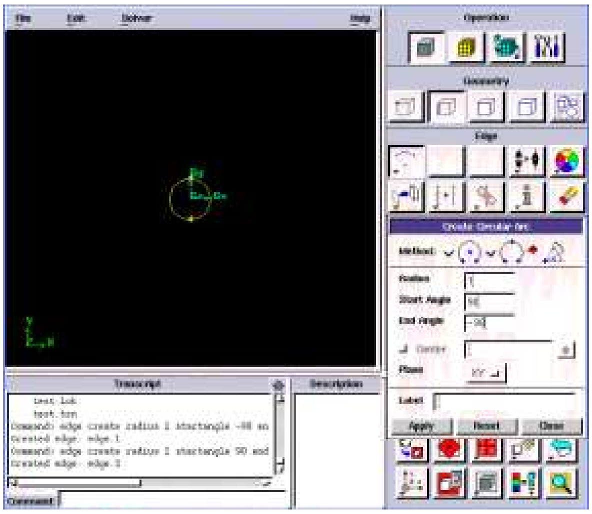
Create Front Outer Boundary

We will create the outer boundary by creating arc 10 times of the cylinder front arc. We can use copy edges and scale by 10 to create the outer boundary.

Operation Toolpad > Geometry Command Button  **> Edge Command Button**  **> Move/Copy Edges** 



Select the front cylinder edge. Make sure that the ***Copy*** is checked. Under ***Operation***, select ***Scale***. Next to ***Factor***, enter a value of 10. This means that the radius of influence we create will be 10 times the size of the cylinder. Click ***Apply***.



(Click here for animated steps)

> ⓘ **Do you know you can also press "Ctrl" + double left click to fit graphic in windows?**

Create Back Outer Boundary

For this part, we will start with bottom up approach where we first create vertices and then using the vertices to create edges. Create the following vertices.

Vertices	X	Y
1	40	10
2	40	-10

Operation Toolpad > Geometry Command Button > Vertex Command Button > Create Vertex

Create the vertices by entering the coordinates under ***Global***.

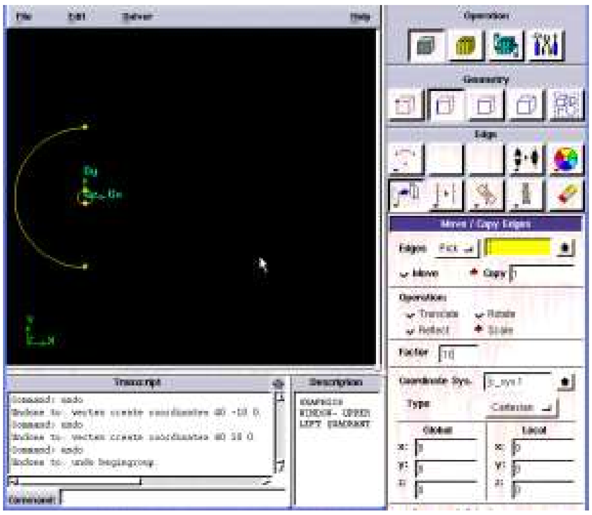
Click the *FIT TO WINDOW* button to scale the display so that you can see all the vertices.

Connect the vertices to create three edges

Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge

"shift" + left click and drag to select

Following animated steps show another method in creating vertices.



"ctrl" + right click on the grid to create a vertex

Do You Know?
Do you know that by clicking and holding right mouse click and move up and down, you can zoom in and out in the graphic window?
Do you know that by clicking middle mouse button, you can move object in graphic window?

Create Radius of Influence

Now we can proceed to create the geometry for radius of influence. Since both the cylinder and radius of influence is of same shape.

Create arc of radius 4 from -45 to 45 deg. Then create another arc from 45 to 45 deg.

Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge > Arc >

Finally split the edge at 90 and -90 deg.

Operation Toolpad > Geometry Command Button > Edge Command Button > Split Edge

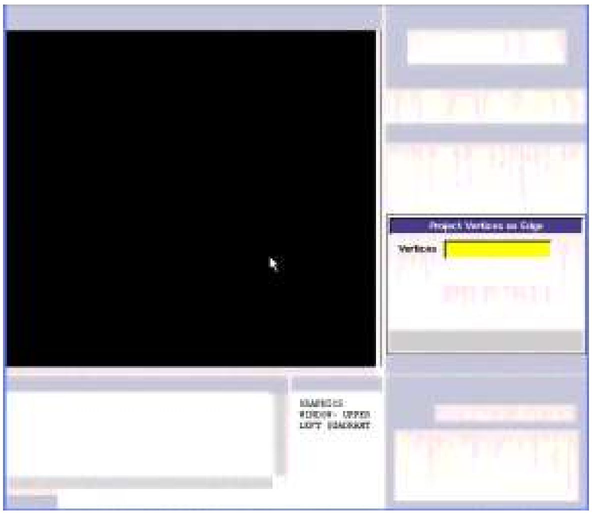
Remember to change the coordinate ***Type*** to *Cylindrical*.

Split Edge at Outer Boundary

For regular mesh, each edge has its opposite edge. Because of this, we can use projection method on the outer boundary to create the edge associated with the Radius of Influence edges.

Operation Toolpad > Geometry Command Button > Vertex Command Button > Project Button

Select the vertex and associated edge. Make sure to select ***Split edge***. At the end of this, you should have 4 new vertices.

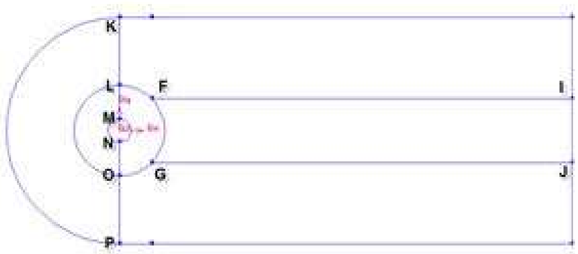


(Animated steps)

Connect all Vertices

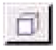
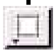
Finally, connect all the remaining vertices **KL, LM, NO, OP, FI** and **GJ**.

The current geometry in Gambit should look like this:



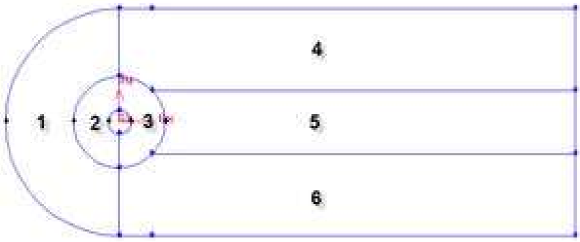
Create Faces

We can now join all the edges to form faces.

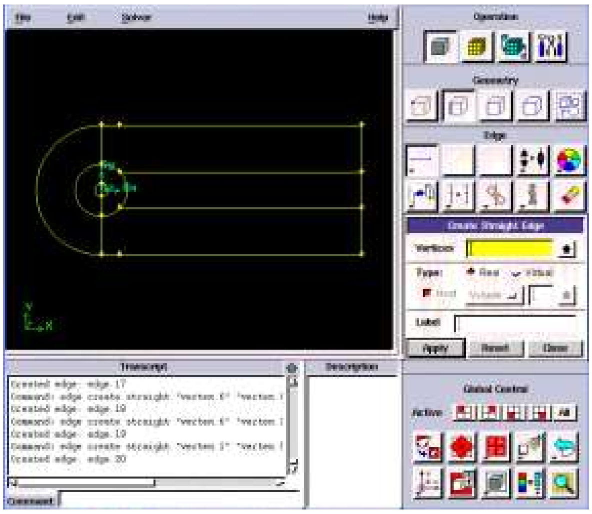
Operation Toolpad > Geometry Command Button  **> Face Command Button**  **> Form Face** 

This brings up the ***Create Face From Wireframe*** menu. Recall that we had selected vertices in order to create edges. Similarly, we will select edges in order to form a face.

There will be total of six faces.



Create all the six faces by connecting appropriate edges.



Animated steps

We are now ready to mesh the geometry.

Go to Step 2: Mesh Geometry in GAMBIT

See and rate the complete Learning Module

FLUENT - Steady Flow Past a Cylinder - Step 2

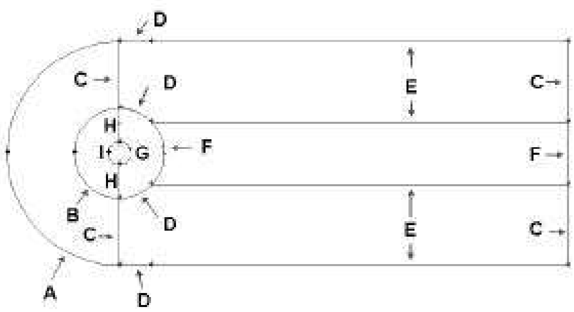
Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Step 2: Mesh Geometry in GAMBIT

Mesh Edges

Before we mesh the faces, we should mesh the edges so that we can better define the mesh.

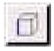
Following figure shows the edges with different mesh properties.



Higher Resolution Image Visit page in new window

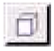
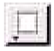
Label	Edge Mesh Properties
A	Interval Count: 36, Double First Length: 0.5
B	Interval Count: 36, Double First Length: 0.2
C	Interval Count: 30, First Length: 0.1
D	Interval Count: 18
E	Interval Count: 90, First Length: 0.1

F	Interval Count: 36
G	Interval Count: 72
H	Interval Count: 30
I	Interval Count: 36, Double First Length: 0.05

Operation Toolpad > Mesh Command Button  **> Edge Command Button**  **> Mesh Edges** 

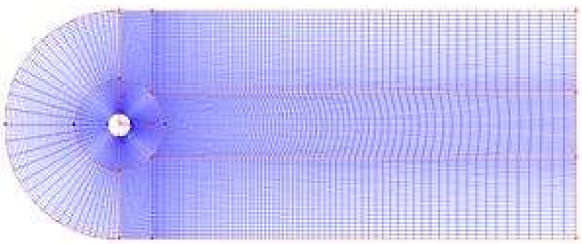
Mesh edge 'A' through 'I' according to the properties shown on the table.

After we have all the edge meshes, we can mesh the faces:

Operation Toolpad > Mesh Command Button  **> Face Command Button**  **> Mesh Faces** 

Select the face one by one. You can use the defaults of ***Quad*** (i.e. quadrilaterals) and ***Map***. Click ***Apply***.

The meshed face should look as follows:



Higher Resolution Image Visit page in new window

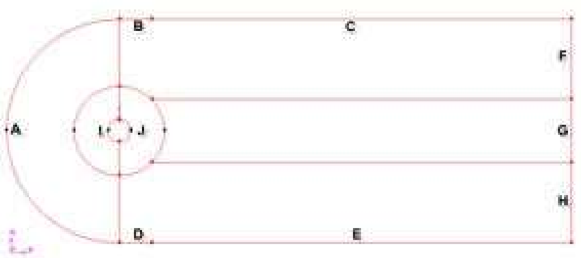
Go to Step 3: Specify Boundary Types in GAMBIT

See and rate the complete Learning Module

FLUENT - Steady Flow Past a Cylinder - Step 3

Step 3: Specify Boundaries in GAMBIT

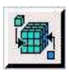
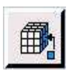
Label the boundaries according to the figure shown below.



Higher Resolution Image Visit page in new window

We will label edge **A** as *farfield1*, edges **B** and **C** as *farfield2*, edges **D** and **E** as *farfield3*, edges **E**,**G** and **H** as *farfield4* and the edges **I** and **J** as *cylinder*.

Edges	Name
A	farfield 1
B,C	farfield 2
D,E	farfield 3
F,G,H	farfield 4
I,J	cylinder

Operation **Toolpad > Zones Command Button**  **> Specify Boundary Types** 

Specify boundary according to the table above. Next to ***Name***, enter the name accordingly. Leave the ***Type*** as ***WALL***. We will specify boundary type using FLUENT.

Save Your Work

Main Menu > File > Save

Export Mesh

Main Menu > File > Export > Mesh...

Save the file as *cylinder.msh*.

Make sure that the ***Export 2d Mesh*** option is selected.

Check to make sure that the file is created.

Go to Step 4: Set Up Problem in FLUENT

See and rate the complete Learning Module

FLUENT - Steady Flow Past a Cylinder - Step 4

Step 4: Set Up Problem in FLUENT

Launch Fluent

Lab Apps > FLUENT 6.3.26

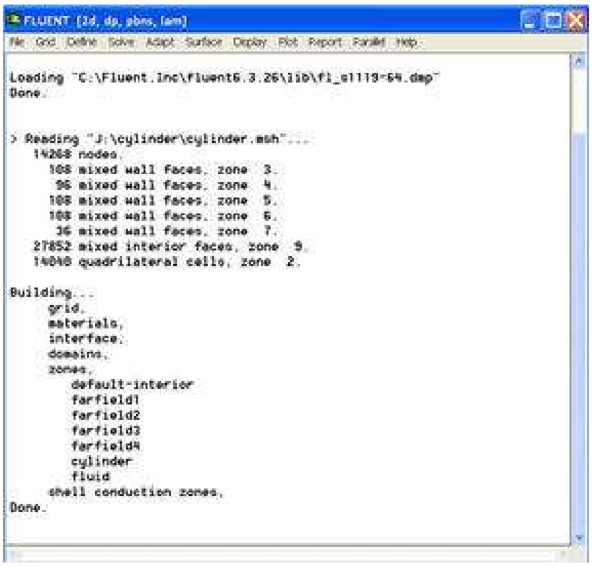
Select ***2ddp*** from the list of options and click ***Run***.

The "2ddp" option is used to select the 2-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the cylinder.msh file. This is the mesh file that was created using the preprocessor *GAMBIT* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:



Higher Resolution Image Visit page in new window

Also, take a look under zones. We can see the five zones ***farfield1***, ***farfield2***, ***farfield3***,***farfield4***, and ***cylinder*** that we defined in *GAMBIT*.

Check and Display Grid

First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:

Main Menu > Grid > Info > Size

The following info should appear (your number of cells might be slightly different because of slight different mesh criteria used):

```
Grid Size

Level    Cells    Faces    Nodes    Partitions
    0    14040    28308    14268             1

 1 cell zone, 6 face zones.
```

Display the grid:

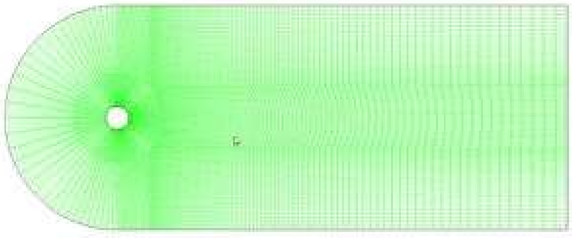
Main Menu > Display > Grid...

Make sure all 6 items under ***Surfaces*** is selected. Then click ***Display***. The graphics window opens and the grid is displayed in it. You can now click ***Close*** in the *Grid Display* menu to get back some desktop space. The graphics window will remain.

> **Graphics Window Operation**
> **Translation:** The grid can be translated in any direction by holding down the ***Left Mouse Button*** and then moving the mouse in the desired direction.
> **Zoom In:** Hold down the ***Middle Mouse Button*** and drag a box from the ***Upper Left Hand Corner*** to the ***Lower Right Hand Corner*** over the area you want to zoom in on.
> **Zoom Out:** Hold down the ***Middle Mouse Button*** and drag a box anywhere from the ***Lower Right Hand Corner*** to the ***Upper Left Hand Corner***.

Use these operations to zoom into the grid to obtain the view shown below.

> **The zooming operations can only be performed with a middle mouse button.**



Higher Resolution ImageVisit page in new window

White Background on Graphics Window
To get white background go to:
Main Menu > File > Hardcopy
Make sure that ***Reverse Foreground/Background*** is checked and select ***Color*** in ***Coloring*** section. Click ***Preview***. Click ***No*** when prompted "*Reset graphics window?*"

You can also look at specific parts of the grid by choosing the boundaries you wish to view under ***Surfaces*** (click to select and click again to deselect a specific boundary). Click ***Display*** again when you have selected your boundaries.

Define Solver Properties

Main Menu > Define > Models > Solver

Use the default setting. Click ***Cancel***.

Main Menu > Define > Models > Viscous

Laminar flow is the default. So we don't need to change anything in this menu. Click ***Cancel***.

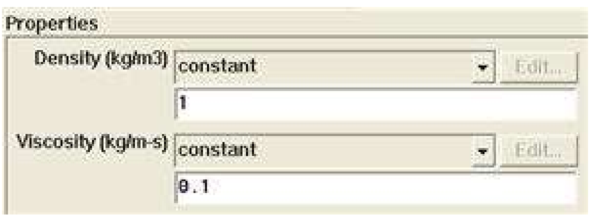
Main Menu > Define > Models > Energy

For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the ***Energy Equation*** unselected and click ***Cancel*** to exit the menu.

Define Material Properties

Main Menu > Define > Materials...

Change ***Density*** to `1.0` and ***Viscosity*** to `0.1`. These are the values that we specified under Problem Specification.

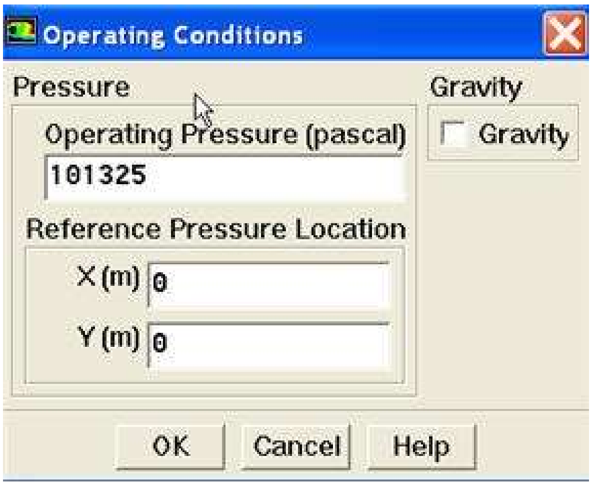


Click ***Change/Create***. Close the window.

Define Operating Conditions

Main Menu > Define > Operating Conditions...

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the ***Operating Pressure***.



Click ***Cancel*** to leave the default in place.

Define Boundary Conditions

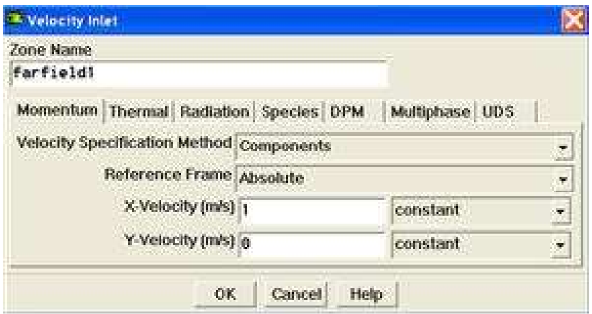
We'll now set the value of the velocity at the inlet and pressure at the outlet.

Use the following table to set boundary type of each zone.

Zone	**Type**
farfield1	velocity-inlet, V_x = 1 m/s
farfield2	velocity-inlet, V_x = 1 m/s
farfield3	velocity-inlet, V_x = 1 m/s
farfield4	pressure-outlet
cylinder	wall

Main Menu > Define > Boundary Conditions...

Select ***farfield1*** under ***Zone***. Change the ***Type*** of boundary as ***velocity-inlet***. A new window will pop up. Change ***Magnitude***, ***Normal to Boundary to Components*** under ***Velocity Specification Method***. Input value *1* next to ***X-Velocity***. Click OK. Do the same for *farfield2* and *farfield3*.



The (absolute) pressure at the farfield downstream is 1 atm. Since the operating pressure is set to 1 atm, the outlet gauge pressure = outlet absolute pressure - operating pressure = *0*. Choose ***farfield4*** under ***Zone***. The ***Type*** of this boundary is ***pressure-outlet***. Click on ***Set...***. The default value of the ***Gauge Pressure*** is 0. Click ***Cancel*** to leave the default in place.

Lastly, click on ***cylinder*** under ***Zones*** and make sure ***Type*** is set as ***wall***.

Click ***Close*** to close the *Boundary Conditions* menu.

Go to Step 5: Solve!
See and rate the complete Learning Module
Go to all FLUENT Learning Modules

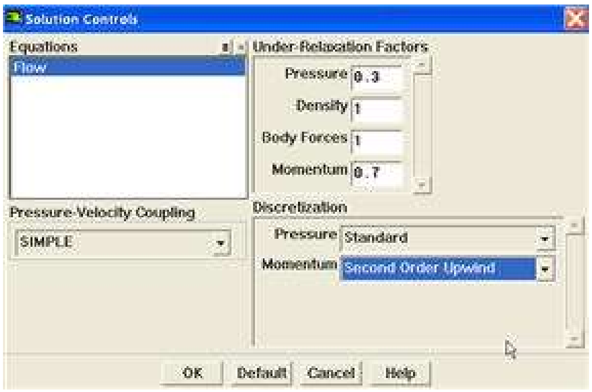
FLUENT - Steady Flow Past a Cylinder - Step 5

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Change ***Momentum*** to ***Second Order Upwind***.



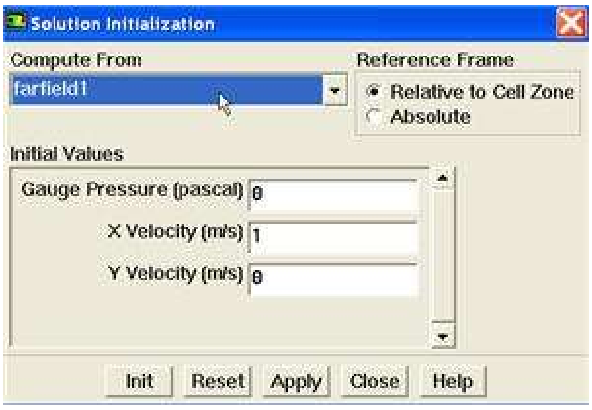
Click ***OK***.

Set Initial Guess

Initialize the flow field to the values at the inlet:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* menu that comes up, choose ***inlet*** under ***Compute From***. The ***X Velocity*** for *all* cells will be set to 1 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the inlet boundary condition.



Click ***Init***. This completes the initialization. ***Close*** the window.

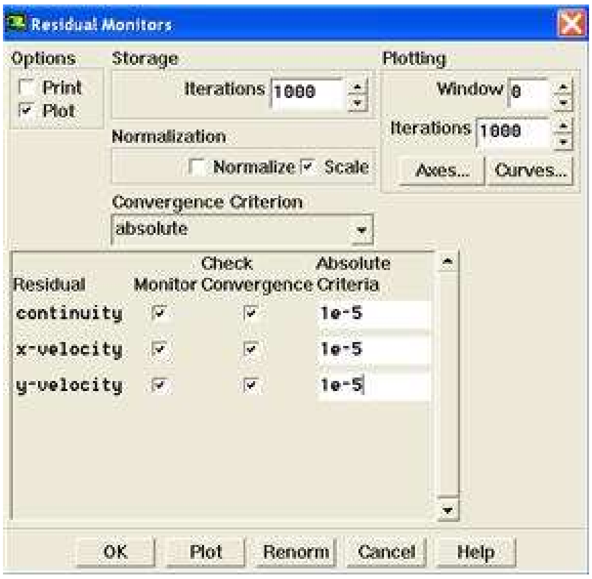
Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below 1e-6.

Main Menu > Solve > Monitors > Residual...

Change the residual under ***Convergence Criterion*** for ***continuity***, ***x-velocity***, and ***y-velocity***, all to 1e-5.

Also, under ***Options***, select ***Plot***. This will plot the residuals in the graphics window as they are calculated.



Click ***OK***.

Monitor also the drag coefficient on the cylinder.

Main Menu > Solve > Monitors > Force...

Select *cylinder* under ***Wall Zones***. Under ***Options***, select ***Plot*** and ***Write***. Note that ***Plot Window*** is 1.

Setting Reference Values

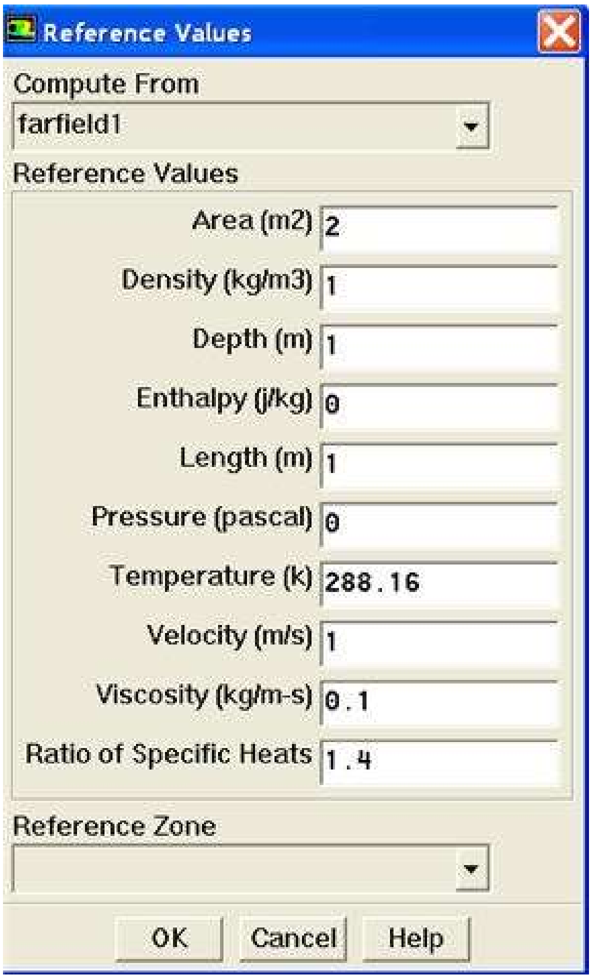
To plot C $_d$, we need to set the reference value.

Unable to find DVI conversion log file.

ⓘ Note that cross sectional area for a 2D cylinder is the diameter of the cylinder.}

Main Menu > Report > Reference Values...

Under ***Reference Values***, change ***Area*** to *2*, ***Density*** to *1*, ***Velocity*** to *1* and ***Viscosity*** to *0.1*.



This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

Type in `cylinder.cas` for ***Case File***. Click ***OK***. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

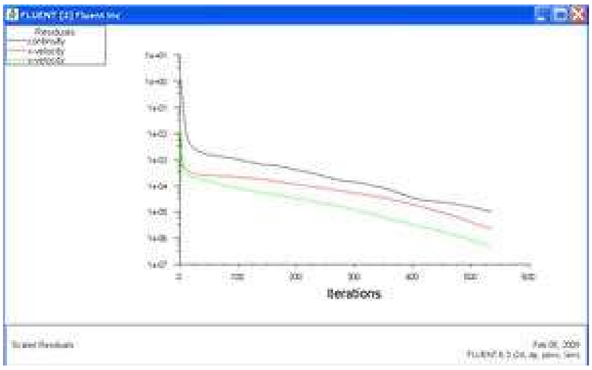
Iterate Until Convergence

Start the calculation by running 1000 iterations:

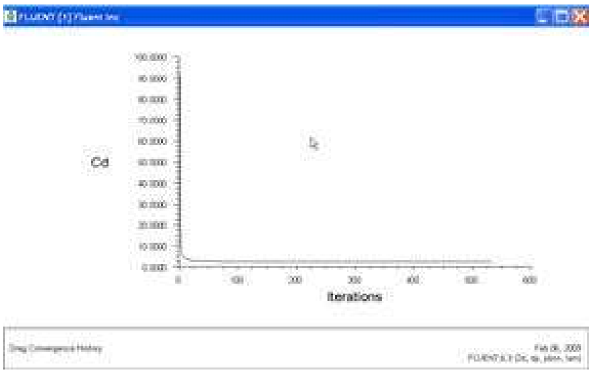
Main Menu > Solve > Iterate...

In the *Iterate Window* that comes up, change the ***Number of Iterations*** to `1000`. Click ***Iterate***.

The residuals and drag coefficient for each iteration are printed out as well as plotted in the graphics window as they are calculated.



Higher Resolution ImageVisit page in new window



Save the solution to a data file:

Main Menu > File > Write > Data...

Enter `cylinder.dat` for ***Data File*** and click ***OK***. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

Go to Step 6: Analyze Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules

FLUENT - Steady Flow Past a Cylinder - Step 6

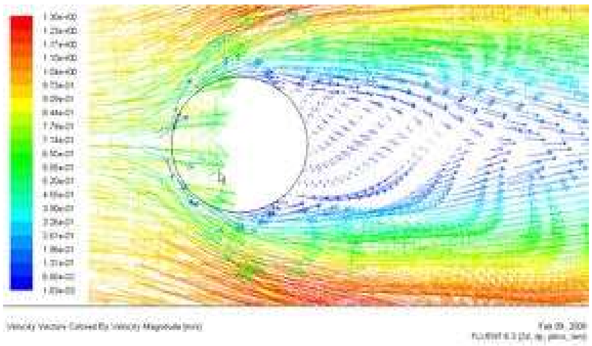
Step 6: Analyze Results

Plot Velocity Vectors

Let's plot the velocity vectors obtained from the FLUENT solution.

Display > Vectors

Set the ***Scale*** to 14 and ***Skip*** to 4. Click ***Display***.



Higher Resolution ImageVisit page in new window

From this figure, we see that there is a region of low velocity and recirculation at the back of cylinder.

Zoom in the cylinder using the middle mouse button.

Pressure Coefficient

Pressure coefficient is a dimensionless parameter defined by the equation $C_p = \frac{(p - p_{\text{ref}})}{q_{\text{ref}}}$ where *p* is the static pressure, p_{ref} is the reference pressure, and q_{ref} is the reference dynamic pressure defined by

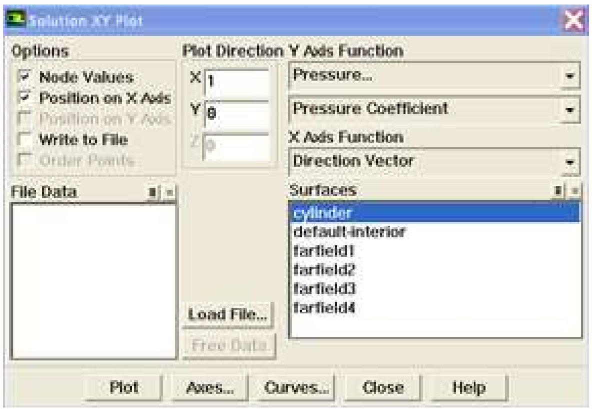
Unable to find DVI conversion log file.

The reference pressure, density, and velocity are defined in the **Reference Values** panel in Step 5.

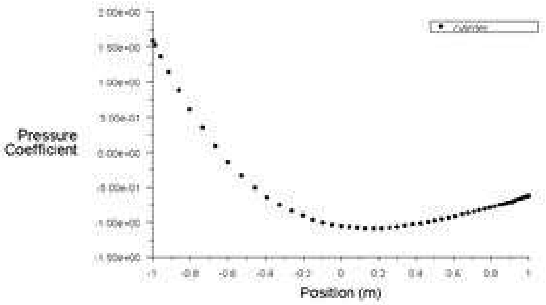
Let's plot pressure coefficient vs x-direction along the cylinder.

Plot > XY Plot...

Change the ***Y Axis Function*** to ***Pressure***..., followed by ***Pressure Coefficient***. Then, select ***cylinder*** under ***Surfaces***.



Click ***Plot***.



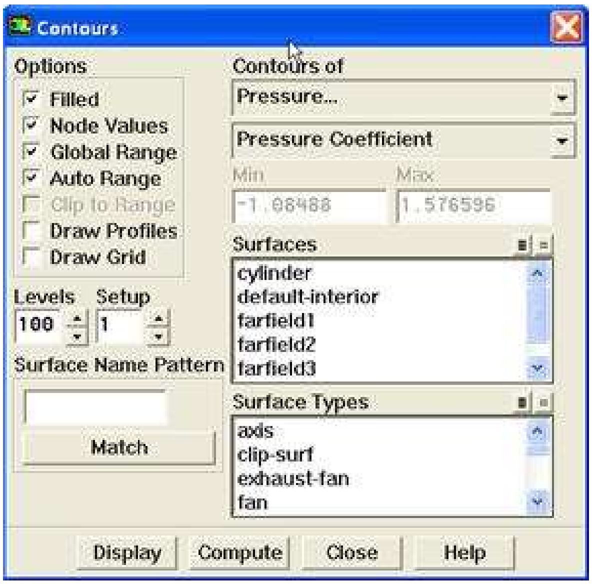
Higher Resolution ImageVisit page in new window

As can be seen, the pressure coefficient at the back is lower than the pressure coefficient at the front of the cylinder. The irrecoverable pressure is due to the separation at the back of cylinder and the frictional loss.

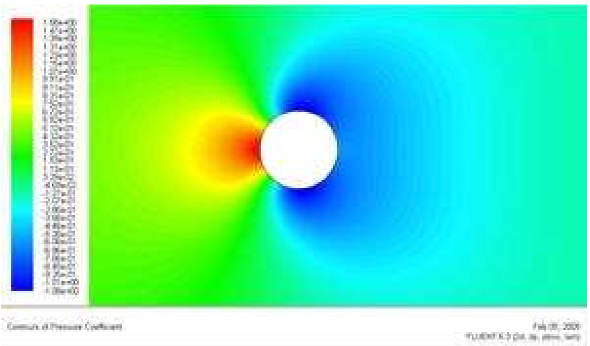
Now, let's take a look at the Contour of Pressure Coefficient variation around the cylinder.

Display > Contours

Under ***Contours of***, choose ***Pressure..*** and ***Pressure Coefficient***. Select the ***Filled*** option. Increase the number of contour levels plotted: set ***Levels*** to `100`.



Click ***Display***.



Higher Resolution Image Visit page in new window

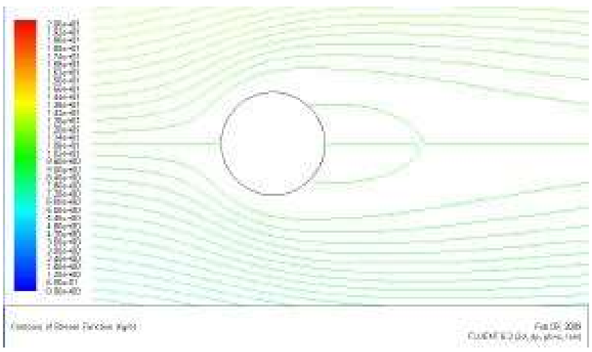
Because the cylinder is symmetry in shape, we see that the pressure coefficient profile is symmetry between the top and bottom of cylinder.

Plot Stream Function

Now, let's take a look at the Stream Function.

Display > Contours

Under ***Contours of***, choose ***Velocity..*** and ***Stream Function***. Deselect the ***Filled*** option. Click ***Display***.



Higher Resolution Image Visit page in new window

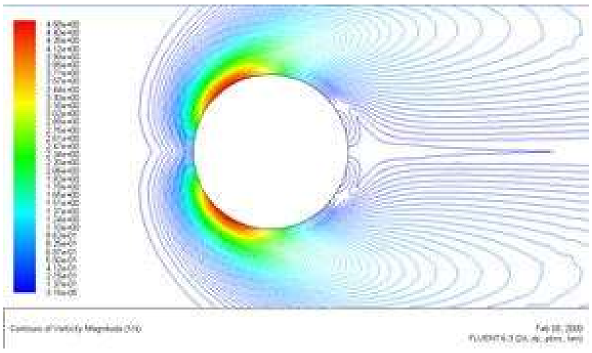
Enclosed streamlines at the back of cylinder clearly shows the recirculation region.

Plot Vorticity Magnitude

Let's take a look at the Pressure Coefficient variation around the cylinder. Vorticity is a measure of the rate of rotation in a fluid.

Display > Contours

Under ***Contours of***, choose ***Velocity..*** and ***Vorticity Magnitude***. Deselect the ***Filled*** option. Click ***Display***.



Higher Resolution Image Visit page in new window

Go to Step 7: Refine Mesh

See and rate the complete Learning Module

FLUENT - Steady Flow Past a Cylinder - Step 7

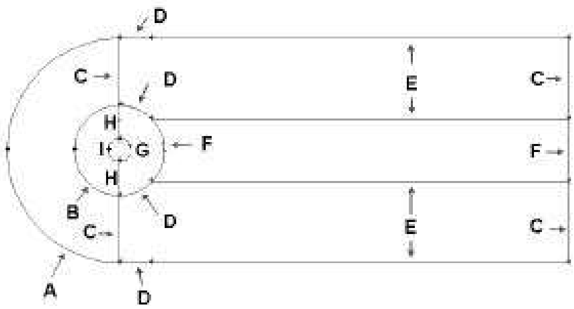
Step 7: Refine Mesh

It our mesh refine enough? Will the current mesh give us an accurate representation of the physical flow? Mesh density determine the truncation error of our simulation. With too little mesh density, the result we obtain will be inaccurate. With too many mesh elements, we will be wasting unnecessary computational power (long computational time!). To get the optimum mesh density, we do mesh sensitivity analysis.

For this purpose, we will create three model with different mesh density. We will create a coarser mesh and a finer mesh and compare all the three meshes together.

For coarse mesh, reduce the mesh density by half of the original mesh. For fine mesh, double the mesh density compared to the original density.

Follow Step 1 through Step 5. At step 2, use the following edge mesh properties instead.



Higher Resolution Image Visit page in new window

Label	Medium Mesh (Original)	Coarse Mesh	Fine Mesh

A	Interval Count: 36, Double First Length: 0.5	Interval Count: 24, Double First Length: 0.7	Interval Count: 54, Double First Length: 0.3
B	[illegible]	[illegible]	[illegible]
C	Interval Count: 30, First Length: 0.1	Interval Count: 20, First Length: 0.2	Interval Count: 45, First Length: 0.09
D	Interval Count: 18	Interval Count: 12	Interval Count: 27
E	Interval Count: 90, First Length: 0.1	Interval Count: 60, First Length: 0.15	Interval Count: 135, First Length: 0.07
F	Interval Count: 36	Interval Count: 24	Interval Count: 54
G	Interval Count: 72	Interval Count: 48	Interval Count: 108
H	Interval Count: 30	Interval Count: 20	Interval Count: 45
I	Interval Count: 36, Double First Length: 0.05	Interval Count: 24, Double First Length: 0.07	Interval Count: 54, Double First Length: 0.03

ⓘ You can also change the appropriate parameters (interval count and first length) in your previous journal file to obtain different meshes.

ⓘ To double the mesh density, the interval count is increased by factor of 1.5. The first length is decreased by a factor of about 1.5. To half the mesh density, the interval count is decreased by factor of 1.5. The first length is increased by a factor of about 1.5. Think about why it is so.

After that, you should have three meshes with following mesh elements.

	Coarse	Medium	Fine
No. of cells	6240	14040	31590

Drag Coefficient Comparison

For each mesh, take note of the drag coefficient and compare with each other.

	Coarse	Medium	Fine
Cd	2.4981	2.4941	2.4927
% dif	0.160378	0	0.056132

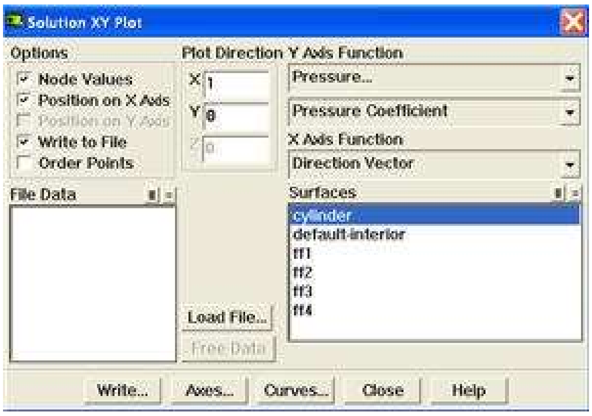
Taking medium as benchmark, we compare coarse and fine mesh. Note that the difference in drag coefficient between three meshes is negligibly small. For analysis of drag coefficient, it seems that coarse mesh provides enough resolution.

Pressure Coefficient Comparison

Let's analyze pressure coefficient plot around cylinder for three different mesh before we conclude which mesh to use. Let's start with a coarse mesh.

y Flow Past a Cylinder - Problem Specification

Change the ***Y Axis Function*** to ***Pressure...***, followed by ***Pressure Coefficient***. Then, select ***cylinder*** under ***Surfaces***. Check the option ***Write to File***.

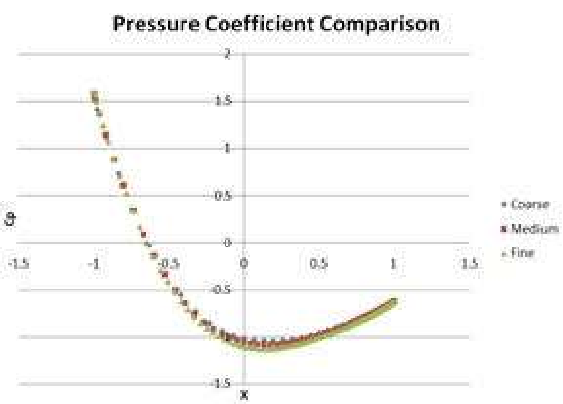


Click ***Write...***

Enter the file name and directory to save.

Do the same for medium and fine meshes by opening different case and data files.

After that, open the files that you have written with excel spreadsheet. Compile and plot the data accordingly.



Higher Resolution Image Visit page in new window

> ⓘ A quick and dirty way plotting three different mesh is to use FLUENT directly. **Plot > XY Plot...** You can ***Load File...*** that you have saved previously. However, using this way, you have less control of how you can present the data and is not recommended for report presentation.