

FLUENT - Unsteady Flow Past a Cylinder - Step 4

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. Validate the Results

Step 4: Set Up Problem in FLUENT

Launch Fluent

Lab Apps > FLUENT 6.3.26

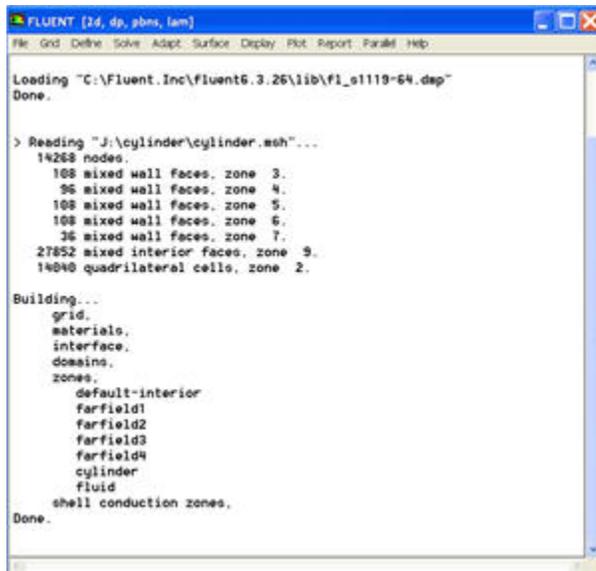
Select **2ddp** from the list of options and click **Run**.

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:



```
FLUENT [2d, dp, pbns, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

Loading "C:\Fluent.Inc\fluent6.3.26\lib\fl_s1119-64.dsp"
Done.

> Reading "J:\cylinder\cylinder.msh"...
14268 nodes.
108 mixed wall faces, zone 3.
96 mixed wall faces, zone 4.
108 mixed wall faces, zone 5.
108 mixed wall faces, zone 6.
36 mixed wall faces, zone 7.
27852 mixed interior faces, zone 9.
14040 quadrilateral cells, zone 2.

Building...
grid.
materials.
interface.
domains.
zones.
  default-interior
  farfield1
  farfield2
  farfield3
  farfield4
  cylinder
  fluid
  shell conduction zones.
Done.
```

[Higher Resolution Image](#)

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:

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.

i Graphics Window Operation

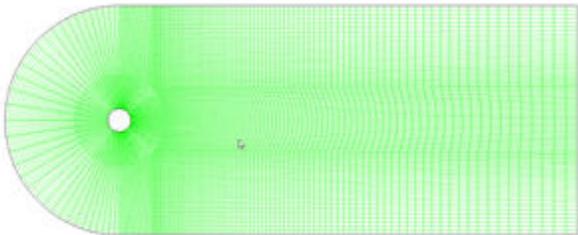
Translation: The grid can be translated in any direction by holding down the **Left Mouse Button** and then moving the mouse in the desired direction.

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 Image](#)

✓ White Background on Graphics Window

To get white background go to:

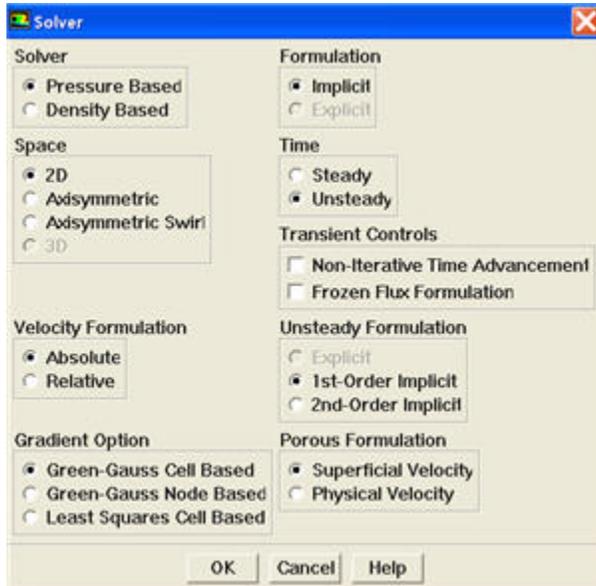
Main Menu > File > Hardcopy

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



Under **Time**, select **Unsteady**. We will use the default **1st-Order Implicit Unsteady Formulation** for for now. Click **OK**.

Main Menu > Define > Models > Viscous

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

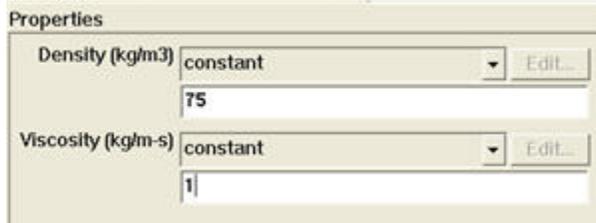
Main Menu > Define > Models > Energy

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 75 and **Viscosity** to 1. These are the values that we specified under **Problem Specification**.

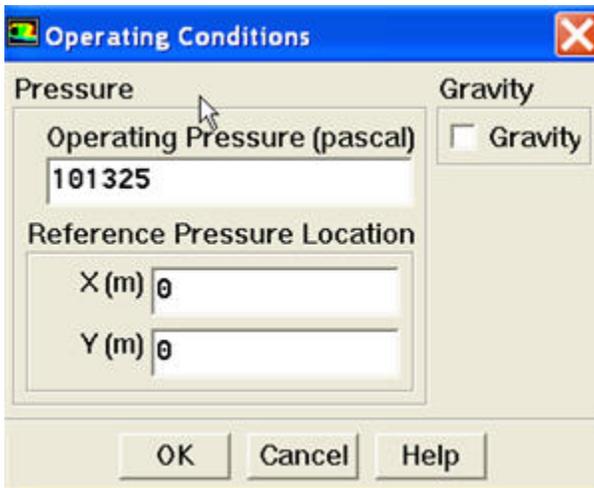


Click **Change/Create**. Close the window.

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

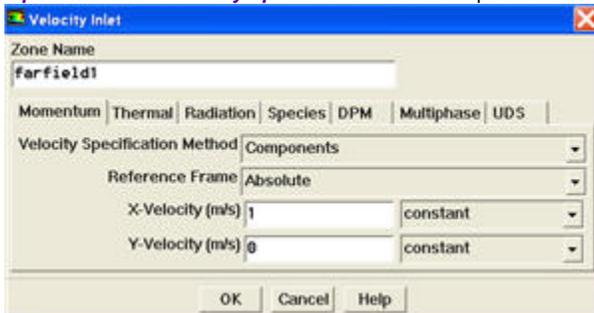
We'll now set the value of the velocity at the inlet and pressure at the outlet.

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](#)