

FLUENT - Laminar Pipe Flow 4 Set Up Problem in FLUENT

Content

Step 4: Set Up Problem in FLUENT

Launch Fluent 6.0

Lab Apps > FLUENT 6.3.26

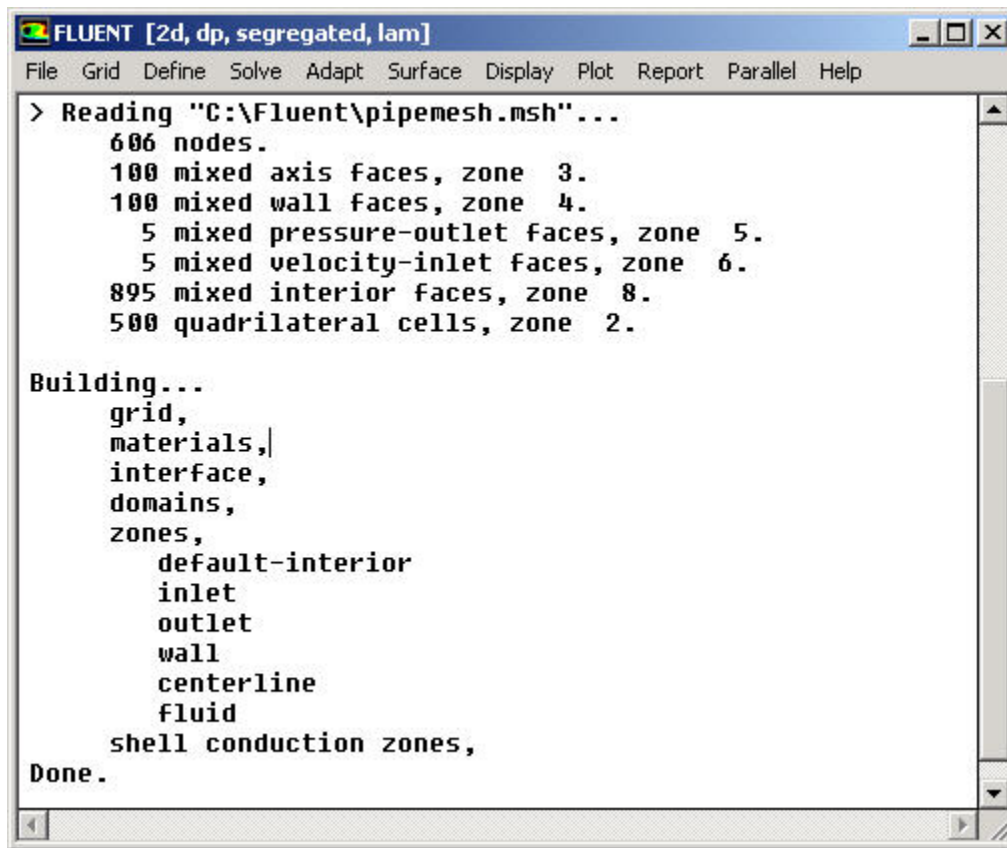
Select 2ddp from the list of options and click Run.

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 pipe.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:



Check the number of nodes, faces (of different types) and cells. There are 500 quadrilateral cells in this case. This is what we expect since we used 5 divisions in the radial direction and 100 divisions in the axial direction while generating the grid. So the total number of cells is $5 \times 100 = 500$.

Also, take a look under zones. We can see the four zones inlet, outlet, wall, and centerline 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 statistics should appear:

Grid Size

Level	Cells	Faces	Nodes	Partitions
0	500	1105	606	1

1 cell zone, 5 face zones.

Display the grid:

Main Menu > Display > Grid...

Make sure all 5 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.

Some of the operations available in the graphics window are:

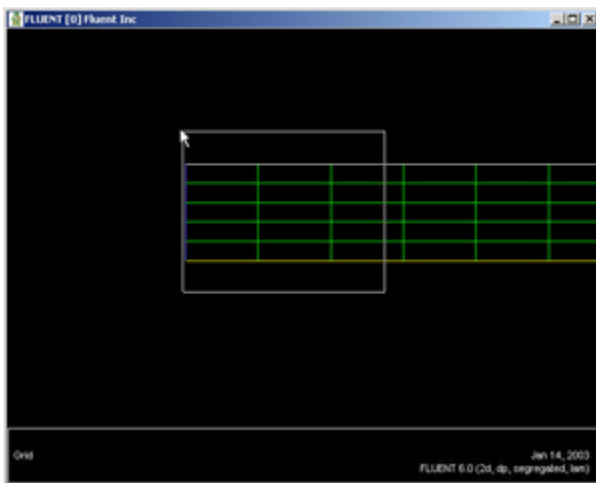
Translation: The grid can be translated in any direction by holding down the Left Mouse Button and then moving the mouse in the desired direction.

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.

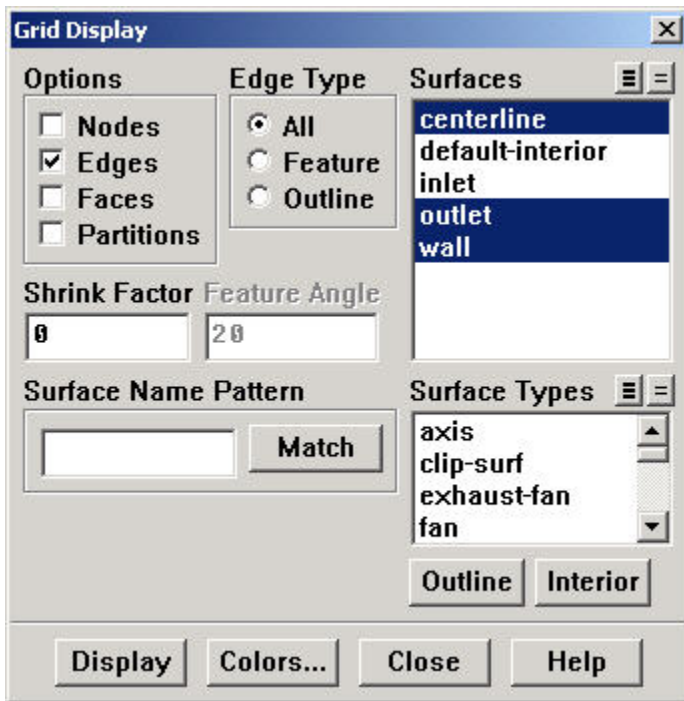
Use these operations to zoom into the grid to obtain the view shown below.

Note: The zooming operations cannot be performed without a middle mouse button.



[fluent:Click here for larger image](#)



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. For example, the wall, outlet, and centerline boundaries have been selected in the following view:



These options will display the graph:



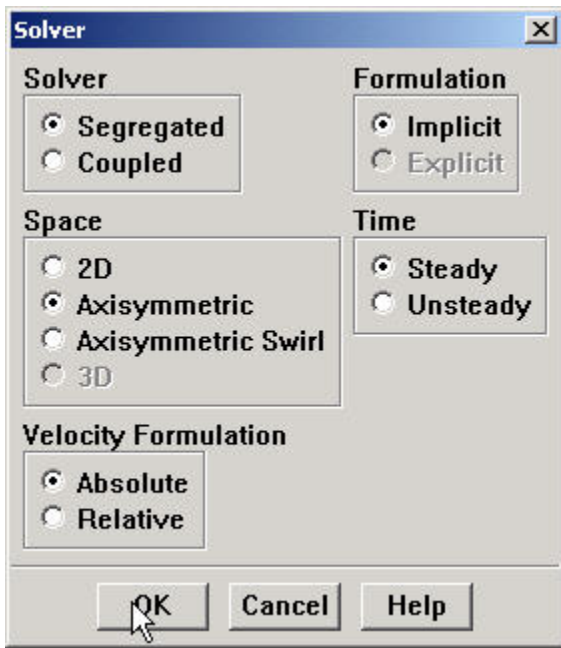
fluent:[Click here for larger image](#)

For convenience, the  button next to Surfaces selects all of the boundaries while the  deselects all of the boundaries at once. Close the *Grid Display Window* when you are done.

Define Solver Properties

Main Menu > Define > Models > Solver

Choose Axisymmetric under Space. We'll use the defaults of pressure based ("segregated", in older versions) solver, implicit formulation, steady flow and absolute velocity formulation. 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.

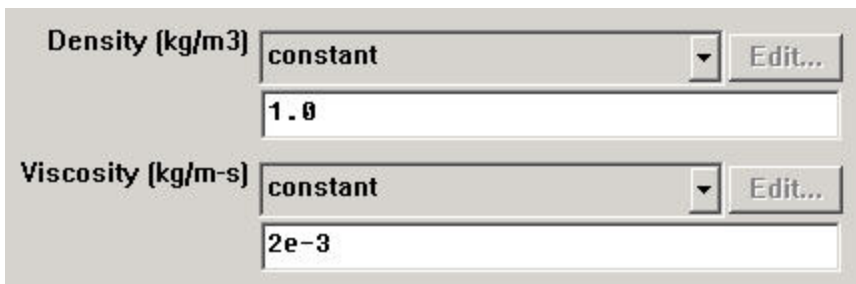
Main Menu > Define > Models > Energy

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 1.0 and Viscosity to 2e-3. These are the values that we specified under Problem Specification. We'll take both as constant.



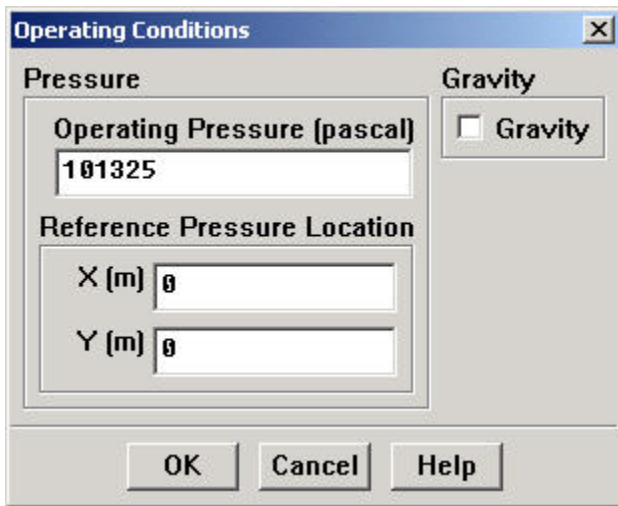
Click Change/Create. Close the window.

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.



The **Operating Conditions** dialog box has two main sections: **Pressure** and **Gravity**. In the **Pressure** section, the **Operating Pressure (pascal)** is set to 101325. Below it, the **Reference Pressure Location** is defined with X (m) and Y (m) both set to 0. In the **Gravity** section, the **Gravity** checkbox is unchecked. At the bottom are **OK**, **Cancel**, and **Help** buttons.

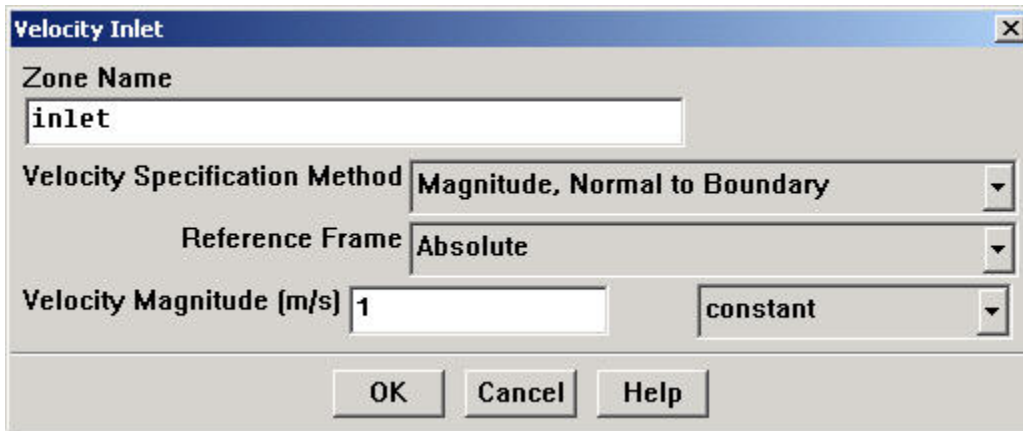
Define Boundary Conditions

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

Main Menu > Define > Boundary Conditions...

We note here that the four types of boundaries we defined are specified as zones on the left side of the *Boundary Conditions Window*. The *centerline* zone should be selected by default. Make sure it is, then make sure the Type of this boundary is selected as axis and click Set.... Notice that there is nothing to set for the axis. Click OK.

Move down the list and select inlet under Zone. Note that FLUENT indicates that the Type of this boundary is velocity-inlet. Recall that the boundary type for the "inlet" was set in GAMBIT. If necessary, we can change the boundary type set previously in *GAMBIT* in this menu by selecting a different type from the list on the right.



The **Velocity Inlet** dialog box contains the following settings: **Zone Name** is 'inlet'. **Velocity Specification Method** is set to 'Magnitude, Normal to Boundary'. **Reference Frame** is set to 'Absolute'. **Velocity Magnitude (m/s)** is set to 1, and the **constant** checkbox is checked. At the bottom are **OK**, **Cancel**, and **Help** buttons.

Click on Set.... Enter 1 for Velocity Magnitude. Click OK. This sets the velocity of the fluid entering at the left boundary.

The (absolute) pressure at the outlet is 1 atm. Since the operating pressure is set to 1 atm, the outlet gauge pressure = outlet absolute pressure - operating pressure = 0. Choose outlet 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 wall under Zones and make sure Type is set as wall. Click on each of the tabs and note that only momentum can be changed under the current conditions. This will not be so under later exercises so make a note of the location of these options. Click OK.

Click Close to close the *Boundary Conditions* menu.