FLUENT - Laminar Pipe Flow Step 4

(i) **Useful Information**

Click here for the FLUENT 12

Step 4: Setup (Physics)

If you have skipped the previous mesh generation steps 1-3, you can download the mesh by right-clicking here. Save the file as *pipe.msh*. You can then proceed with the flow solution steps below.

Launch Fluent 6.0

Start > Programs > Fluent Inc. Products > 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*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.
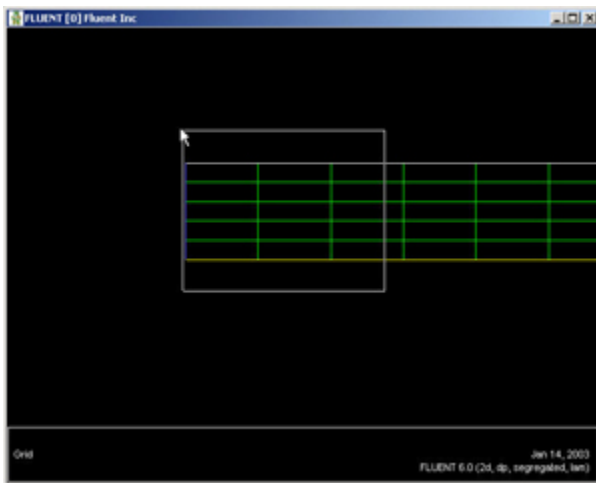
Higher Resolution Image

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:
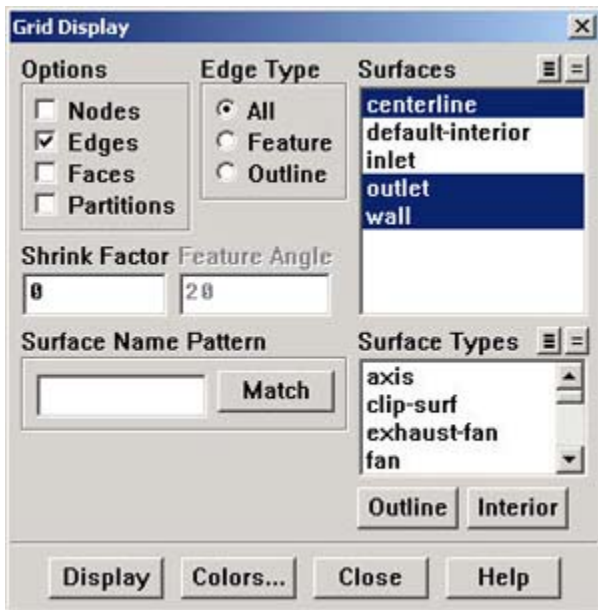
These options will display the graph:
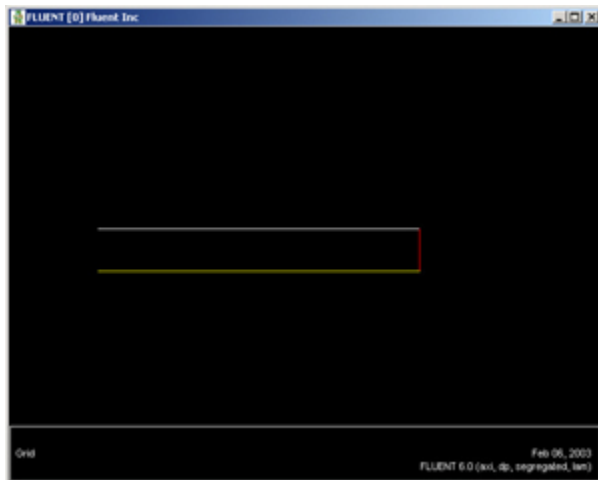
Higher Resolution Image

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**.
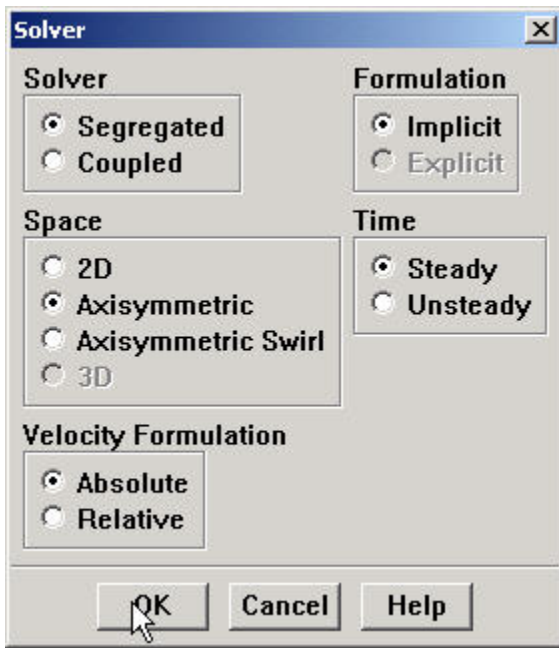
Main Menu > Define > Models > Viscous

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**.
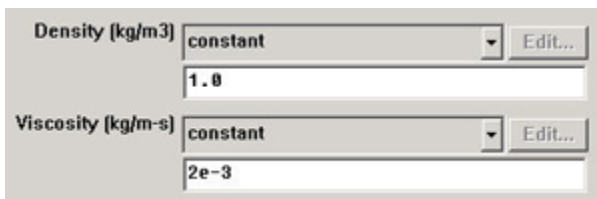

Click **Change/Create**. Close the window.

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.
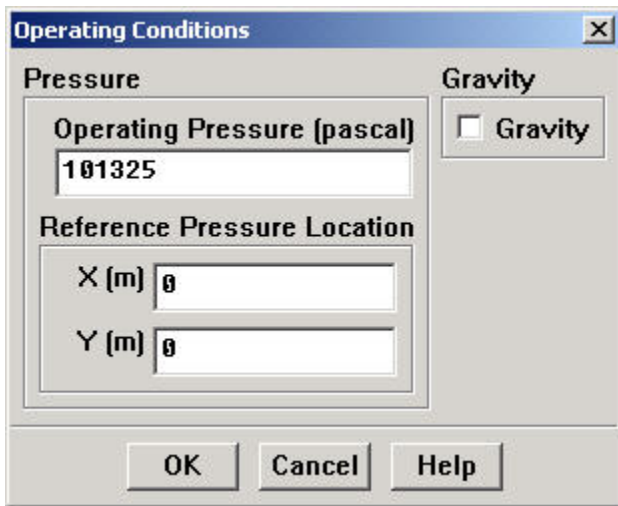
Define Boundary Conditions

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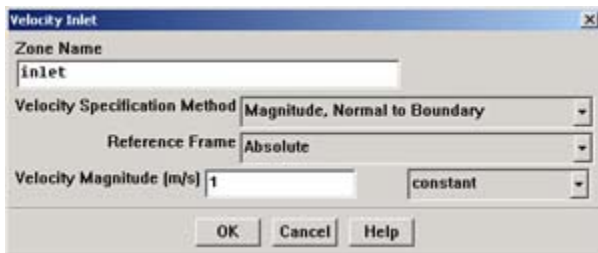
Click on *Set...*. Enter 1 for *Velocity Magnitude*. Click *OK*. This sets the velocity of the fluid entering at the left boundary.

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.

Go to Step 5: Solve!

See and rate the complete Learning Module

Go to all FLUENT Learning Modules