

FLUENT - Turbulent Pipe Flow - Step 4

Problem Specification

1. Pre-Analysis & Start-Up
 2. Geometry
 3. Mesh
 - 4. Setup (Physics)**
 5. Solution
 6. Results
 7. Verification & Validation
- Problem 1



Useful Information

[Click here](#) for the FLUENT 12 version.

Step 4: Setup (Physics)

If you have skipped the previous mesh generation steps 1-3, you can download the mesh by right-clicking on [this link](#). Save the file as *pipe100x30.msh*. You can then proceed with the flow solution steps below.

Launch FLUENT

Lab Apps > FLUENT 6.3.26

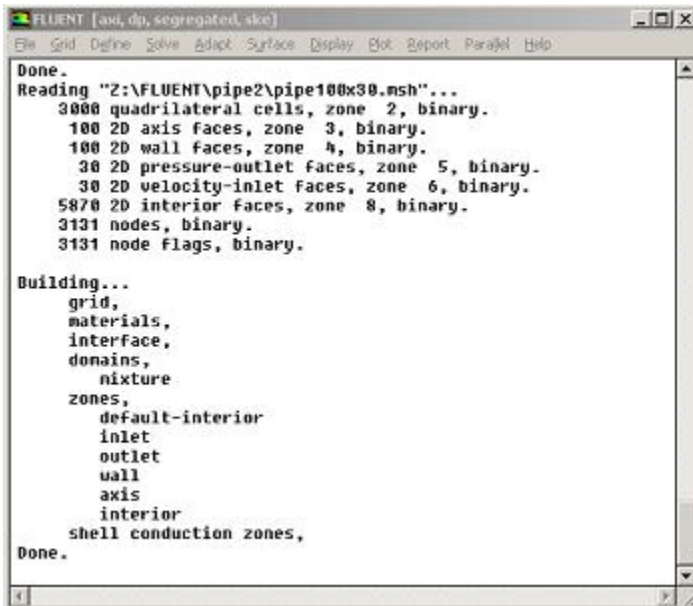
Select **2ddp** (2D, double-precision version) from the list of options and click **Run**.

Import File

Main Menu > File > Read > Case...

Navigate to your working directory and select the pipe100x30.msh file. Click **OK**.

The following should appear in the FLUENT window:



Check the number of nodes, faces (of different types) and cells. There are 3000 quadrilateral cells in this case. This is what we'd expect since we used 30 divisions in the radial direction and 100 divisions in the axial direction while generating the grid. So the total number of cells is $30 \times 100 = 3000$.

Also, take a look under zones. We can see the four zones **inlet**, **outlet**, **wall**, and that we defined in **centerline** GAMBIT.

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. Then select:

Main Menu > Grid > Info > Size

The following summary about the grid should appear:

Grid Size

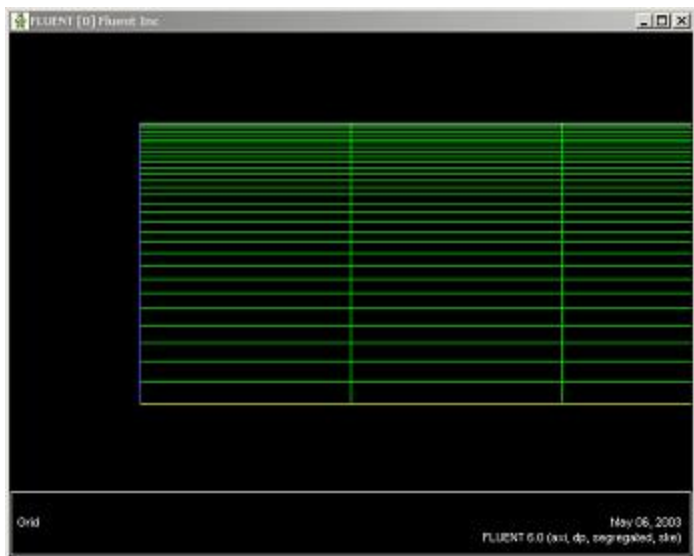
Level	Cells	Faces	Nodes	Partitions
0	3000	6130	3131	1

1 cell zone, 5 face zones.

Let's look at the grid:

Main Menu > Display > Grid...

Make sure all 5 items under **Surfaces** are selected. Then click **Display**. Remember that we can zoom in using the middle mouse button. Zoom in and admire the grid. How many divisions are there in the radial direction?



[Higher Resolution Image](#)

Recall that you can look at specific components of the grid by choosing the entities 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. Use this feature and make sure that the boundary labels correspond to the correct geometric entities.

Close the *Grid Display Window* when you are done.

Define Governing Equations

Main Menu > Define > Models > Solver

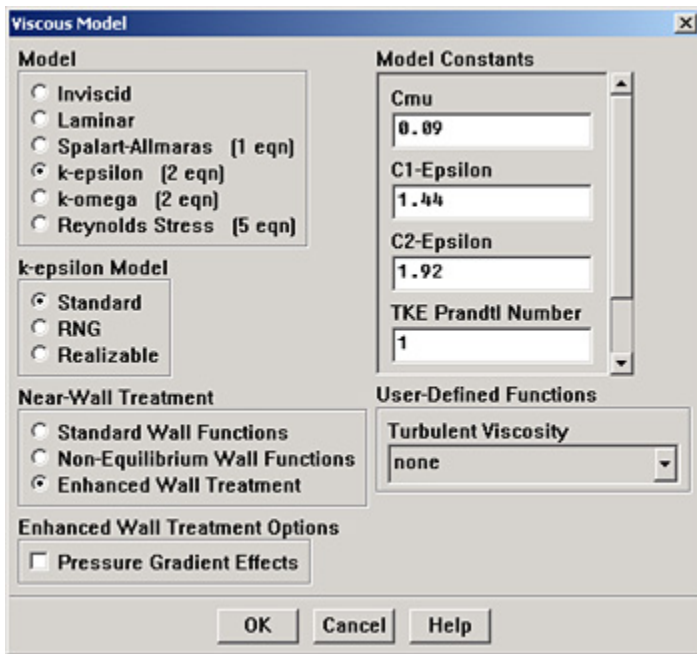
Choose **Axisymmetric** under **Space**. As in the laminar pipe flow tutorial, we'll use the defaults of segregated solver, implicit formulation, steady flow and absolute velocity formulation. Click **OK**.

Main Menu > Define > Models > Energy...

The energy equation can be turned off since this is an incompressible flow and we are not interested in the temperature. Make sure no tick mark appears next to **Energy Equation**.

Main Menu > Define > Models > Viscous...

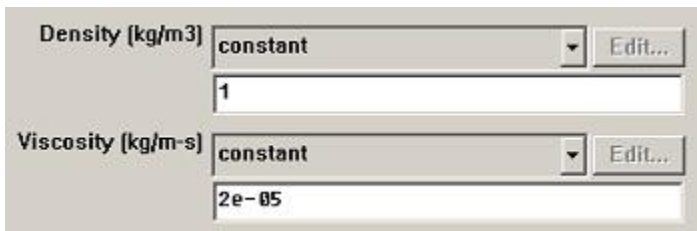
Choose **k-epsilon (2eqn)**. Notice that the window expands and additional options are displayed on choosing the **k-epsilon** turbulence model. Under **Near-Wall Treatment**, pick **Enhanced Wall Treatment** so that we may get a more accurate result.



Click **OK**.

Main Menu > Define > Materials...

Change **Density** to 1.0 and Viscosity to 2×10^{-5} . These are the values in the [Problem Specification](#). We'll take both as constant.



Click **Change/Create**.

Define Boundary Conditions

Main Menu > Define > Operating Conditions...

Recall that for all flows, FLUENT uses the 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**.

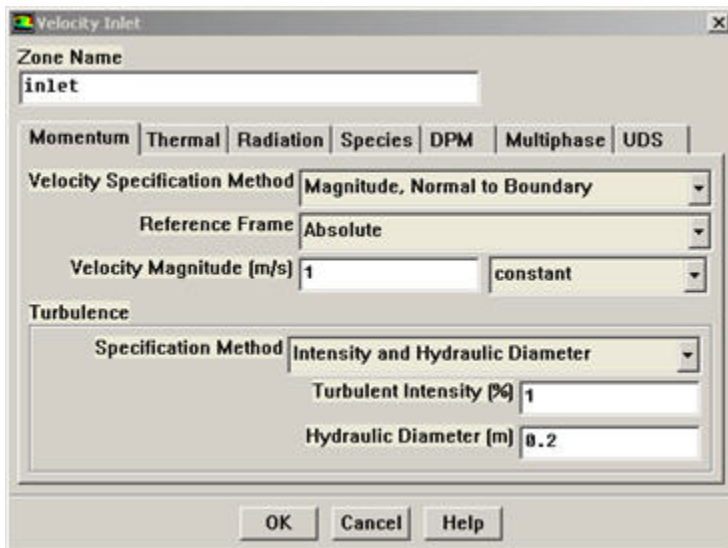
Click **Cancel** to leave the default in place.

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

Main Menu > Define > Boundary Conditions...

The four types of boundaries we defined are specified as zones on the left side of the *Boundary Conditions Window*. Recall that we don't need to set any parameters for the **centerline** and **wall** zones. Verify this by selecting each of these two zones and clicking on **Set...**

Choose inlet and click on **Set...** Enter 1 for **Velocity Magnitude**. This indicates that the fluid is coming in normal to the inlet at the rate of 1 meter per second. Select **Intensity and Hydraulic Diameter** next to the **Turbulence Specification Method**. Then enter 1 for **Turbulence Intensity** and 0.2 for **Hydraulic Diameter**. Click **OK** to set the velocity.



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 defaults in place.

Note: Backflow in the *Pressure Outlet* menu refers to flow *entering* through an outlet boundary. This is not likely to happen in this case. So we don't have to set the backflow parameters.

This completes the boundary condition specification. **Close** the *Boundary Conditions* menu.

Go to [Step 5: Solution](#)

[See and rate the complete Learning Module](#)

Go to [all FLUENT Learning Modules](#)