# **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 & Vallidation Problem 1



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:
```

```
_ [] X
ELUENT [axi, dp, segregated, s
        Grid Define
                                    Adapt
                                                                     Elot
                                                                             Report Parallel Help
 Done.
                                                                                                                                   *
  Reading "Z:\FLUENT\pipe2\pipe100x30.msh"..
        ding "2:\FLUENT\pipe2\pipe100x30.msh"...
3000 quadrilateral cells, zone 2, binary.
100 2D axis faces, zone 3, binary.
100 2D wall faces, zone 4, binary.
30 2D pressure-outlet faces, zone 5, binary.
30 2D velocity-inlet faces, zone 6, binary.
5870 2D interior faces, zone 8, binary.
1131 podes, binary.
         3131 nodes, binary.
3131 node flags, binary.
  Building...
            grīd,
materials,
            interface,
            donains,
                 nixture
            zones,
                 default-interior
                 inlet
                 outlet
                 uall
                  axis
                 interior
            shell conduction zones,
  Done.
30
```

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\*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.

Viscous Model	×
Model	Model Constants
C Inviscid C Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Reynolds Stress (5 eqn) k-epsilon Model Standard RNG Realizable	Cmu         •           0.09         •           C1-Epsilon         •           1.44         •           C2-Epsilon         •           1.92         •           TKE Prandtl Number         •
Near-Wall Treatment	User-Defined Functions
<ul> <li>Standard Wall Functions</li> <li>Non-Equilibrium Wall Functions</li> <li>Enhanced Wall Treatment</li> </ul>	Turbulent Viscosity
Enhanced Wall Treatment Options	
OK Cano	Help

Click OK.

# Main Menu > Define > Materials...

Change Density to 1.0 and Viscosity to \*2e-5. These are the values in the Problem Specification. We'll take both as constant.

Density (kg/m3) Viscosity (kg/m-s)	constant	*	Edit	
	1			
	constant	•	Edit	
	2e-05			

# 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 Hyd raulic Diameter. Click OK to set the velocity.

Velocity Inlet			ļ.
Zone Name			
inlet			
Momentum   Thermal   Radiat	ion Species DPM	Multiphase   UDS	1
Velocity Specification Method	Magnitude, Normal	to Boundary	•
Reference Frame	Absolute		•
Velocity Magnitude (m/s)	1	constant	•
Turbulence			_
Specification Method	Intensity and Hydrau	lic Diameter	•
	Turbulent Intensity	129 1	-
	Hydraulic Diameter	(m) 0.2	-
		· · /laure	
OK	Cancel Help	•	

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