Developed Flow in a Pipe

Abstract—*In this exercise, fully developed flow in a pipe is modeled. The pipe is represented in 2D and an axisymmetric boundary condition is applied. The radius and the length of the pipe can be specified. Coarse, medium, and fine mesh types are available. Material properties (viscosity and density) can be specified within limits. The mass flow rate of the fluid is specified as a boundary condition. The exercise reports friction factor, mean inlet velocity, and total friction force on the wall. A plot of radial distribution of velocity is available. Contours of velocity and stream function can be displayed. A velocity vector plot is also available.*

1 Introduction

Hydrodynamically developed flow is achieved in a pipe after a certain length when the effect of viscosity reaches the center of the pipe. After this point, the flow is essentially one-dimensional. FlowLab solves the problem using periodic boundaries. Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature.

2 Modeling Details

The pipe is represented in 2D by a rectangle. The pipe geometry is displayed in Figure 2.1. The procedure for solving the problem is:

1. Create the geometry.
2. Set the material properties and boundary conditions.
3. Mesh the domain.

FlowLab creates the geometry and mesh, and exports the mesh to FLUENT. The boundary conditions and flow properties are set through parameterized case files. FLUENT converges the problem until the convergence limit is met or the number of iterations specified by the user is achieved.

2.1 Geometry

The geometry consists of a pipe wall, a centerline, and periodic inlet and outlet boundaries. The radius and length of the pipe can be specified.

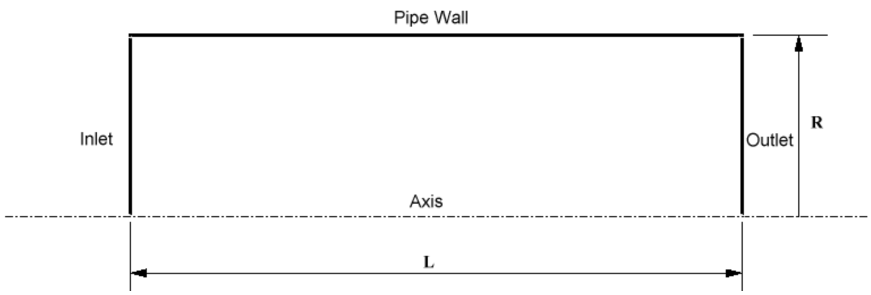

Figure 2.1: Schematic of the Flow Domain

2.2 Mesh

Coarse, medium, and fine mesh types are available. Mesh density varies based upon the assigned Refinement Factor. The Refinement Factor values for the mesh densities are given in Table 2.1.

Mesh Density	Refinement Factor
Fine	1
Medium	1.414
Coarse	2

Table 2.1: Refinement Factor

Using the Refinement Factor, First Cell Height is calculated with the following formula:

$$First\ Cell\ Height = Refinement\ Factor \left[\frac{Yplus \times (Characteristic\ Length^{0.125} \times Viscosity^{0.875})}{(0.199 \times Velocity^{0.875} \times Density^{0.875})} \right] \quad (2\text{-}1)$$

Reynolds number is used to determine Yplus. Yplus values for turbulent flow conditions are summarized in Table 2.2.

Reynolds Number	Flow Regime	Yplus/First Cell Height
Re ≤ 2000	Laminar	First Cell Height = Pipe Radius/38
2000 < Re < 15000	Turbulent, Enhanced Wall Treatment	Yplus < 5.0
Re ≥ 15000	Turbulent, Standard Wall Functions	Yplus > 30

Table 2.2: Flow Regime Vs. Reynolds Number

The number of intervals along each edge is determined using geometric progression and the following equation:

$$Intervals = INT \left[\frac{Log\left\{ \frac{Edge\_length \times (Growth\_ratio - 1)}{First\ Cell\ Height} + 1.0 \right\}}{Log(Growth\_ratio)} \right] \quad (2\text{-}2)$$

The edges are meshed using the First Cell Height and the calculated number of intervals. The entire domain is meshed using a map scheme. The resulting mesh is shown in Figure 2.2.

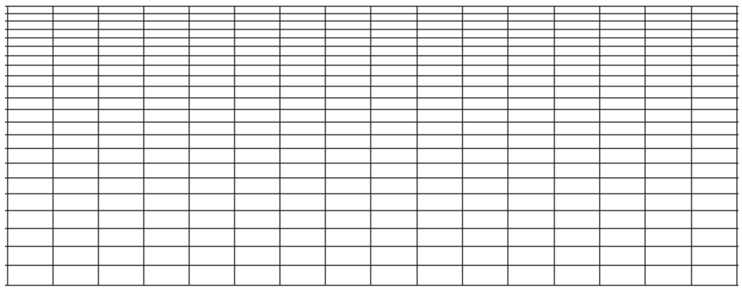

Figure 2.2: Mesh Generated by FlowLab

2.3 Physical Models for FLUENT

Based on the Reynolds number, the following physical models are recommended:

Re < 2000	Laminar flow
2000 ≤ Re < 10000	$k - \omega$ model
10000 ≤ Re < 15000	$k - \epsilon$ model
Re ≥ 15000	$k - \epsilon$ model

Table 2.3: Turbulence Models Based on Pipe Reynolds Number

If turbulence is selected in the Physics form of the Operation menu, the appropriate turbulence model and wall treatment is applied based upon the Reynolds number.

2.4 Material Properties

The default material is air. The following material properties can be specified:

- Density
- Viscosity

Other materials such as Glycerin, Water, and a User Defined fluid can also be selected.

2.5 Boundary Conditions

The mass flow rate of the fluid can be specified. The following boundary conditions are assigned in FLUENT:

Boundary	Assigned as
Inlet	Periodic
Outlet	Periodic
Centerline	Axis
Pipe Wall	Wall

Table 2.4: Boundary Conditions Assigned in FLUENT

2.6 Solution

The mesh is exported to FLUENT along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file. Instructions for the solver are provided through a `journal file`. When the solution is converged or the specified number of iterations is met, FLUENT exports data to a `neutral` file and to `.xy` plot files. GAMBIT reads the neutral file for postprocessing.

3 Scope and Limitations

Transitional flow occurs between Reynolds numbers of 2000 and 10000. To improve accuracy of predictions, the $k - \omega$ turbulence model was applied. However, a prediction error of five percent or more exists in the transitional region.

Difficulty in obtaining convergence or poor accuracy may result if input values are used outside the upper and lower limits suggested in the problem overview.

4 Exercise Results

4.1 Reports

The following reports are available:

- Darcy's friction factor
- Mean inlet velocity
- Total frictional force on the wall
- Mass imbalance

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Axial velocity distribution along pipe radius
- Wall Yplus distribution *

* *Available only when the flow is modeled as turbulent.*

An Axial distribution of velocity along the pipe radius is shown in Figure 4.1.

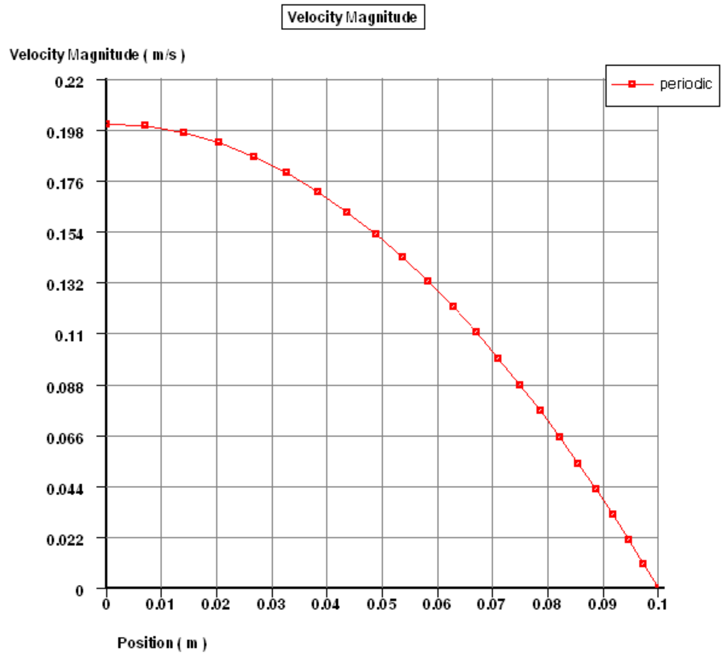

Figure 4.1: Radial Profile of Velocity for Re = 1500

4.3 Contour Plots

Contour plots of velocity magnitude, streamlines, radial velocity, and axial velocity are available. In addition, velocity vector plots can be displayed (Figure 4.2).

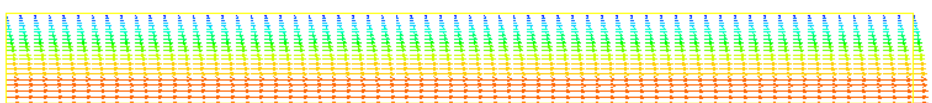

Figure 4.2: Velocity Vectors for Re = 1500

5 Verification of Results

Table 5.1 presents results generated using a pipe radius of 0.1 m, a pipe length of 1 m, and the fine mesh option. Reported values of Darcy's friction factor are compared to correlation in the table.

Re	Computed Value	Correlation Value [1]
200	0.3193	0.3200
500	0.1277	0.1280
800	0.0799	0.0800
1000	0.0639	0.0640
1500	0.0426	0.0426
2000	0.03195	0.0320
4000	0.0430	0.0389
5000	0.0391	0.0376
6000	0.0361	0.0356
10000	0.02971	0.0316
20000	0.02532	0.026
40000	0.02143	0.022
60000	0.0195678	0.0204
120000	0.0168036	0.0176

Table 5.1: Predicted Values of Darcy's Friction Factor Vs. Experimental Correlation

6 Sample Problems

1. The default Reynolds number for this exercise is 1500, and the default material properties represent air. Run this case and compare the reported results with the literature.

2. Change the material properties to those for *water*. Vary the inlet boundary condition to achieve Re = 1500. Run the case. Is there a difference in the value of friction factor reported for this case versus the value reported in the first case with air?

3. Investigate the velocity profiles for both laminar and turbulent flow cases. Are they different? Why?

4. Is there any correlation between friction factor and Reynold's number?

7 Reference

[1] *White, Frank M.*, "Viscous Fluid Flow", International Edition, McGraw-Hill, 1991.

[2] *Adrian Bejan*,"Convective Heat Transfer", John Wiley and Sons, 1994.

[3] *Schlichting, H.*, "Boundary-Layer Theory", 7th Edition, McGraw-Hill, 1979.