# Developing Flow in a Pipe

**Abstract**—*In this exercise, developing flow in a pipe is modeled. The pipe is represented in two dimensions, assuming axisymmetric flow. The radius and the length of the pipe can be changed. Coarse, medium, and fine mesh types are available. The flow can be solved with or without heat transfer. The material properties for flow without heat transfer (viscosity and density) and for flow with heat transfer (viscosity, density, specific heat, and thermal conductivity) can be specified. Inlet velocity, inlet temperature, heat flux, and wall temperature can be specified as boundary conditions. The exercise reports total pressure drop, wall friction force, total heat flux, and the temperature rise. The plots for centerline velocity, pressure, and temperature distributions, wall friction factor, temperature, and Nusselt number distributions are available. Velocity and temperature profiles at specified axial positions can be displayed. Contours of velocity, pressure, temperature, and stream function can be displayed. A velocity vector plot is also available.*

## 1  Introduction

In this exercise, boundary layer development can be studied with respect to developing flow in a pipe. The effects of heat transfer can be included in the solution. The boundary layer can be visualized using velocity vector and temperature contour plots when heat transfer is considered. The Prandtl number and the Reynolds number can be varied through material property selection and boundary condition settings, which will allow you to inspect the corresponding effects upon boundary layer development.

## 2  Modeling Details

The pipe is represented in two dimensions by a rectangle (Figure 2.1). When heat transfer is enabled, a wall temperature or heat flux boundary condition is specified along the pipe wall. 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 is achieved.

### 2.1  Geometry

The geometry consists of a pipe wall, a centerline, an inlet, and an outlet (Figure 2.1). The pipe is created from a set of four edges forming a rectangle. The faces are built from these edges. The radius and the length of the pipe are specified.
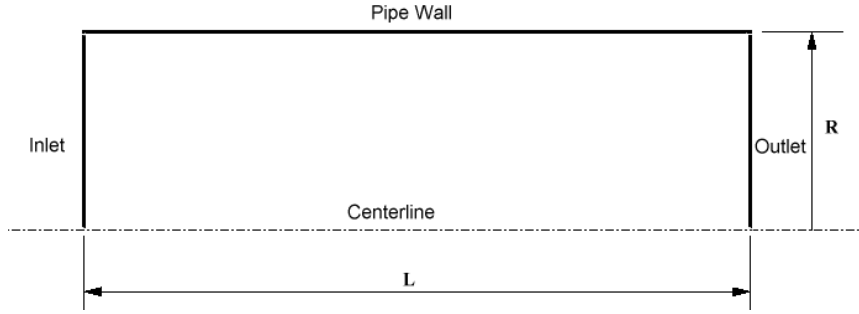
Figure 2.1: Geometry

## 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:

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

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

| Reynolds Number | Flow Regime | Yplus/First Cell Height |
|---|---|---|
| Re $\leq$ 2000 | Laminar | First Cell Height = Pipe Radius/38 |
| 2000 < Re < 15000 | Turbulent, Enhanced Wall Treatment | Yplus < 5.0 |
| Re $\geq$ 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:

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

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.
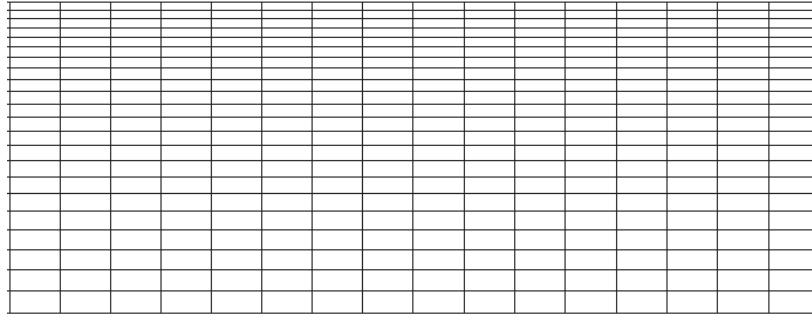


Figure 2.2: Mesh Generated by FlowLab

## 2.3 Physical Models for FLUENT

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

| Re | Model Used |
|---|---|
| Re < 2000 | Laminar flow |
| $2000 \leq$ Re $< 10000$ | $k - \omega$ model |
| $10000 \leq$ Re $< 15000$ | $k - \epsilon$ model |
| Re $\geq 15000$ | $k - \epsilon$ 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 fluid material is air. The following material properties can be specified:

- Density

- Viscosity

- Thermal conductivity *

- Specific heat *

Other materials such as Glycerin, Water, and a User Defined fluid can also be selected.
* *To be specified when the heat transfer is on.*

## 2.5   Boundary Conditions

The following boundary conditions can be specified:

- Inlet velocity

- Inlet temperature *

- Wall heat flux or wall temperature *

- Wall Roughness **

   * *To be specified only when the heat transfer is on.*

   ** *To be specified only when the flow is modeled as turbulent.*

The following boundary conditions are assigned in FLUENT:

| Boundary | Assigned as |
|----------|-------------|
| Inlet | Velocity Inlet |
| Outlet | Pressure Outlet |
| 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

The flow is in transition in the Reynolds number range of 2000 to 10000. In order to predict the correct flow and temperature field, the $k - \omega$ turbulence model was used. However, the accuracy of predicted results may vary by more than five percent from experimental correlation in this 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.

**Note:**  *Turbulent flows develop very fast; hence the value of entry length is significantly smaller as compared to laminar cases. It is advisable to reduce the pipe length while running turbulent cases for smaller run time and accurate entry length predictions.*

## 4   Exercise Results

### 4.1   Reports

The following reports are available:

- Total pressure drop along the pipe length

- Frictional force at the wall

- Total heat flux on the wall *

- Temperature rise *

- Mass imbalance

    *Available only when the heat transfer is on.

### 4.2   XY Plots

The plots reported by FlowLab include:

- Residuals

- Velocity distribution along the axis

- Pressure distribution along the axis

- Temperature distribution along the axis *

- Wall friction factor distribution

- Wall temperature distribution *

- Wall Nusselt number distribution *

- Wall Yplus distribution **

- Profiles of axial velocity

- Profiles of temperature *

    *  Available only when the heat transfer is on.
    ** Available only when the flow is modeled as turbulent.

Figure 4.1 presents centerline velocity distribution, as an example.
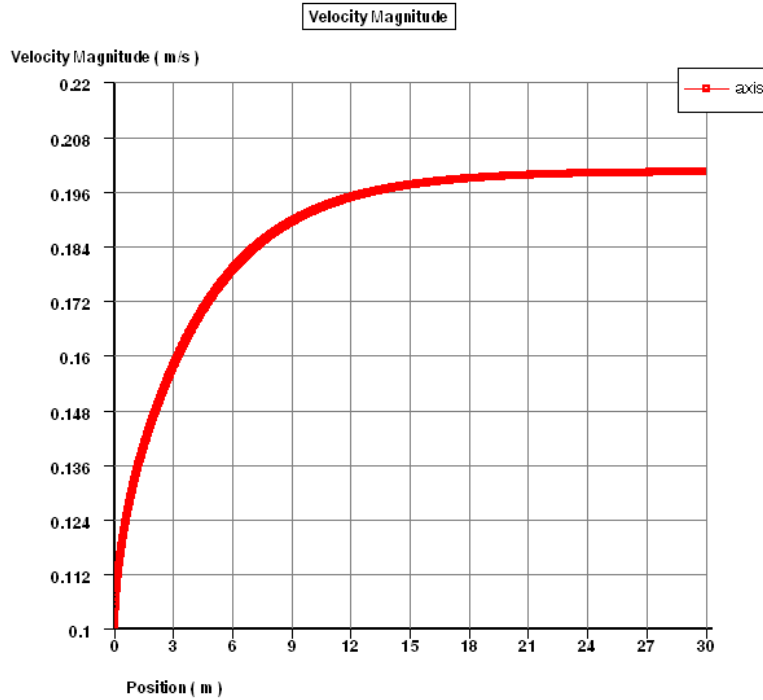
Figure 4.1: Centerline Velocity Distribution for Re = 1500

## 4.3 Contour Plots

Contour plots of pressure, temperature, velocity, and stream function are available. Velocity vector plots can also be displayed.
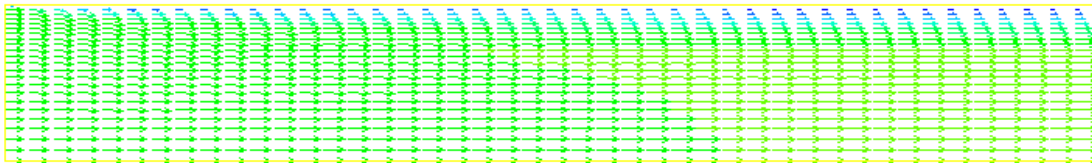


Figure 4.2: Velocity Vectors for Re = 1500

## 5 Verification of Results

The following results (Table 5.1 and Table 5.2) were obtained using the fine mesh option, default material properties, a radius of 0.1 m, a length of 30 m, and the default heat flux boundary condition of 30 W/m$^2$. Inlet velocity was varied to obtain the required Reynolds number. Darcy's friction factor values were calculated from FlowLab results and are reported in Table 5.1.

FlowLab predicted Nusselt numbers are compared with experimental correlation in Table 5.2.

| Re | Computed Value | Correlation Value [1] |
|---|---|---|
| 200 | 0.3194 | 0.32 |
| 500 | 0.1277 | 0.128 |
| 800 | 0.0799 | 0.08 |
| 1000 | 0.0638 | 0.064 |
| 1500 | 0.0426 | 0.04267 |
| 2000 | 0.032 | 0.032 |
| 4000 | 0.0430 | 0.0389 |
| 5000 | 0.0391 | 0.0376 |
| 6000 | 0.0363 | 0.0356 |
| 10000 | 0.0297 | 0.0316 |
| 20000 | 0.0253 | 0.026 |
| 40000 | 0.02144 | 0.022 |
| 60000 | 0.01957 | 0.0204 |
| 120000 | 0.0168 | 0.0176 |

Table 5.1: Predicted Values of Darcy's Friction Factor

| Re | Computed Value | Correlation Value [2] |
|---|---|---|
| 200 | 4.37 | 4.36 |
| 500 | 4.37 | 4.36 |
| 800 | 4.365 | 4.36 |
| 1000 | 4.3634 | 4.36 |
| 1500 | 4.365 | 4.36 |
| 2000 | 4.375 | 4.36 |
| 4000 | 16.67 | 12.566 |
| 5000 | 19.17 | 16.133 |
| 6000 | 21.55 | 18.977 |
| 10000 | 30.06 | 29.943 |
| 20000 | 54.7 | 51.046 |
| 40000 | 92.3 | 87.392 |
| 60000 | 126.2 | 121.853 |
| 120000 | 216.0 | 209.695 |

Table 5.2: Predicted Values of Nusselt Number

The Gnielinski correlation was used for comparison with FlowLab predicted Nusselt numbers:

$$Nu = \frac{(f/8)(Re - 1000)Pr}{1 + 1.27\sqrt{(f/8)}(Pr^{2/3} - 1)} \quad (5\text{-}1)$$

where,

$Re$ = Reynold's number
$Pr$ = Prandtl Number
$f$ = Darcy's friction factor

The value of $f$ is based on the following equations:

$f$ = $64/Re$ for laminar flow
$f$ = $0.316 \times Re^{-1/4}$ for turbulent flow ($Re \leq 20000$)
$f$ = $0.814 \times Re^{-1/5}$ for turbulent flow ($Re \geq 20000$)

## 6   Sample Problems

1. Using default exercise settings, vary the magnitude of wall heat flux and investigate trends in thermal boundary layer thickness.

2. Change the thermal boundary condition to constant wall temperature and run additional cases. Investigate how specified wall temperature affects thermal boundary layer characteristics.

3. Using incremental steps, increase inlet velocity to achieve a Reynolds number of 1.0e+6, running cases at the intermediate points as velocity increases. Observe that Nusselt number and skin friction coefficient on the pipe surface increases with Reynolds number. Enable heat transfer and assess how inlet velocity effects the reported Nusselt number.

## 7   Reference

[1] *White, Frank M.*, "Viscous Fluid Flow", International Edition, McGraw-Hill, 1991, p 423.

[2] *Incropera F. P. and DeWitt David P.*, "Fundamentals of Heat and Mass Transfer", John Wiley and Sons, Fourth Edition.

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