Flow Through an Expansion

Abstract—*In this exercise, a two-dimensional expansion is modeled. The flow is assumed to be axisymmetric. The radius and the length of the inlet and exit pipes is specified. Coarse, medium, and fine mesh types are available. The inlet velocity, the density, and the viscosity of the fluid can be changed within limits. The number of iterations, the convergence limit, and the position of XY plots can be specified. Total pressure at the inlet, total pressure at the outlet, velocity at the outlet, and frictional force on the wall are reported. The following plots are available: residuals; centerline velocity, static pressure, and total pressure; exit-pipe wall static pressure, total pressure, and shear; and radial profiles of axial velocity. Contours of pressure, velocity, and stream function can be displayed. A velocity vector plot is also available.*

1 Introduction

A sudden expansion creates a separation of the boundary layer from the wall, which results in significant pressure loss. The magnitude of the pressure loss depends upon the Reynolds number and the geometric expansion ratio. In most cases, it is desired to minimize pressure loss in a piping system, since the pressure loss correlates to lost energy. In this exercise, it is possible to assess the effects of boundary layer separation on pressure loss.

2 Modeling Details

The fluid region is represented in two dimensions. The procedure for solving the problem is:

- Create the geometry.
- Set the material properties and boundary conditions.
- 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. The length and the radius of the inlet and exit pipe can be specified. The flow field is assumed to be axisymmetric and solved in two dimensions.

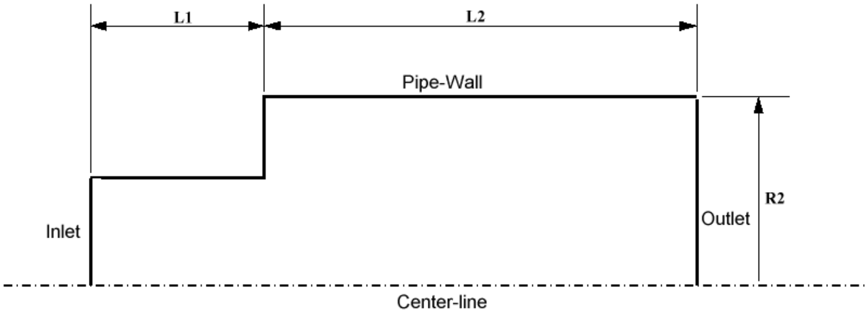

Figure 2.1: Schematic of Flow Domain

The length of the inlet pipe, $L1$, should be at least 10 times the inlet radius, $R1$. The length, $L2$, should be greater than 50 times the difference between $R2$ and $R1$. These geometric criteria are evaluated and a warning message is displayed if violated. Three faces are created to facilitate meshing.

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 < 1000	Laminar	First Cell Height = Inlet Radius/30
2000 ≤ Re ≤ 40000	Turbulent, Enhanced Wall Treatment	Yplus < 10
Re > 40000	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 (Figure 2.2)

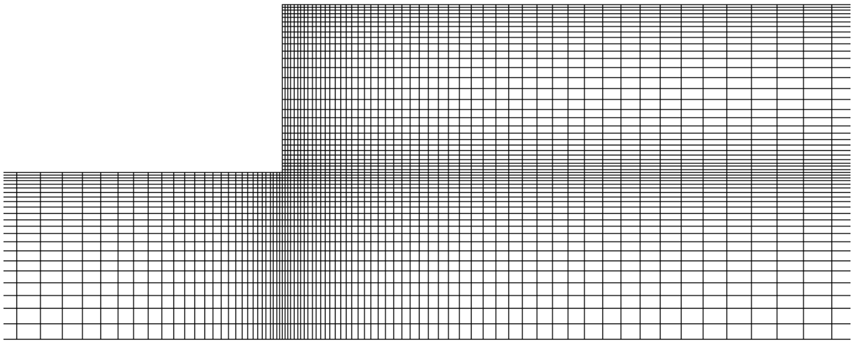

Figure 2.2: Mesh Generated by FlowLab

2.3 Physical Models for FLUENT

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

Re	**Model Used**
Re $<$ 1000	Laminar flow
$1000 \leq$ Re ≤ 1800	Low Reynolds number $k{-}\epsilon$ model
$1800 <$ Re ≤ 40000	$k - \epsilon$ model
Re > 40000	$k - \epsilon$ 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 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 the data to a `neutral file` and `.xy` plot files. GAMBIT reads the `neutral` file for postprocessing.

3 Scope and Limitations

It is recommended not to exceed Reynolds numbers of 1.0e+6.

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:

- Total pressure at inlet
- Total pressure at outlet
- Average velocity at the outlet
- Total frictional force on the wall
- Discharge coefficient
- Mass imbalance

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Centerline velocity distribution
- Centerline static pressure distribution

- Centerline total pressure distribution
- Exit pipe static pressure distribution
- Exit pipe total pressure distribution
- Exit pipe X-wall shear distribution
- Radial profiles of axial velocity
- Wall Yplus distribution *

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

Figure 4.1 shows total pressure versus axial position, as an example.

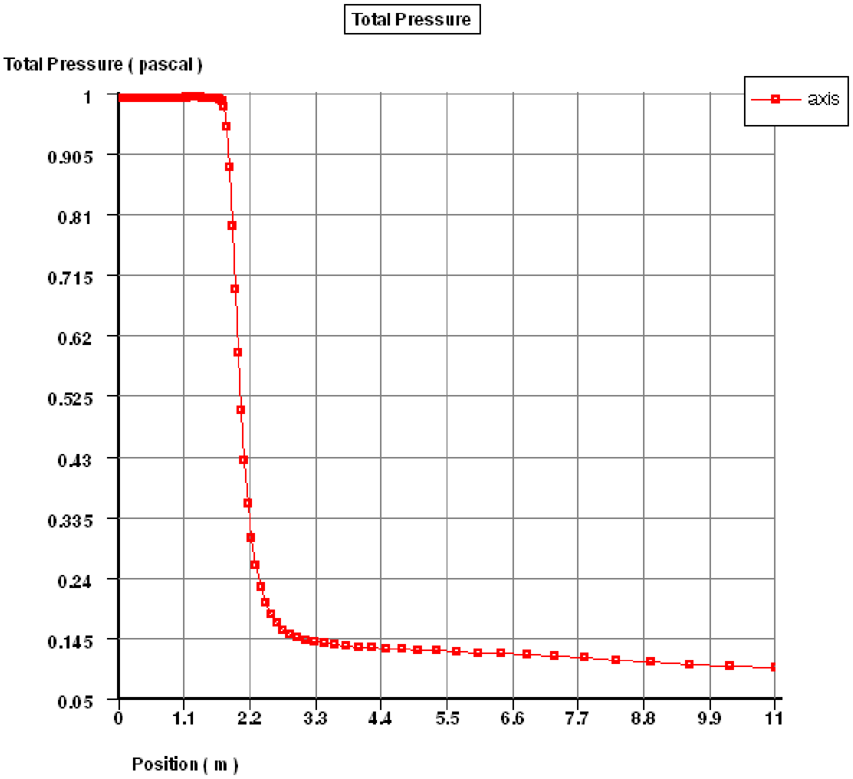

Figure 4.1: Centerline Total Pressure as a Function of Distance at Re = 20000

4.3 Contour Plots

Contour plots of pressure, total pressure, axial velocity, radial velocity, velocity magnitude, and stream function are available. Figure 4.2 presents contours of stream function, as an example.

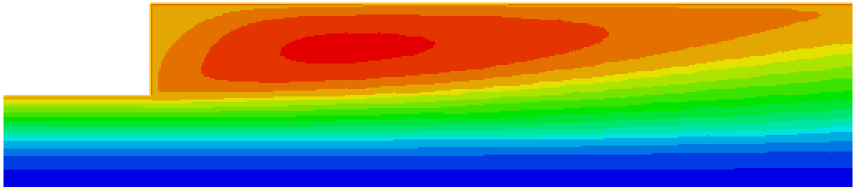

Figure 4.2: Contours of Stream Function at Re = 20000

5 Verification of Results

Table 5.1 presents FlowLab results generated using the fine mesh option, an inlet radius of 0.1 m, and a Reynolds number of 20000. The discharge coefficient (k) is calculated using the following equation:

$$k = \frac{\left\{2\frac{(p_1 - p_2)}{\rho} + (V_1^2 - V_2^2)\right\}}{V_1^2} \tag{5-1}$$

where,

p_1 = Static pressure at the expansion
p_2 = Static pressure at the reattachment point
V_1 = Mass averaged velocity at the expansion
V_2 = Mass averaged velocity at the reattachment point
ρ = Fluid density

Expansion Ratio	**Predicted Value**	**Experimental Correlation[1]**
1.5	0.2777	0.3080
2	0.5549	0.5625
2.5	0.7023	0.7050
3	0.7855	0.7900
4	0.8771	0.8780

Table 5.1: Predicted Discharge Coefficient Values Vs. Experimental Correlation

6 Sample Problem

1. Run a case and evaluate the solution using the default Reynolds number of 20000 and the default material properties (for air).
2. Run several cases while increasing the inlet velocity to assess the effect of Reynolds number upon predicted results.
3. Modify the geometry by changing the inlet and outlet radii. Observe how pressure loss increases with the expansion ratio (the separation bubble is an indicator of head loss).

7 Reference

[1] *Masey, B. S.*, "Mechanics of Fluid", 6^{th} Ed., Ch. 7.

[2] *Shames, I. H*, "Mechanics of Fluid", 3^{rd} Ed., Ch. 9.