Turbulent Flow and Heat Transfer in a Mixing Elbow

Abstract—*In this exercise, two-dimensional turbulent flow and heat transfer in a mixing elbow is modeled. The length of the outlet pipe can be specified. Coarse, medium, and fine mesh types are available. The inlet velocity and temperature of the fluid can be specified. A side stream with specified velocity and temperature joins the main flow stream in the elbow. The effect of the side stream on mixing within the elbow can be observed. Temperature dependent fluid density, viscosity, thermal conductivity, and specific heat can be specified. Pressure drop and temperature change from* Inlet1 *to the* Outlet *and mass imbalance are reported. Plots of velocity distribution, pressure distribution, temperature distribution, and wall Yplus are available. Velocity vectors, temperature contours, and streamlines can be displayed in the flow domain.*

1 Introduction

The mixing elbow configuration is commonly used in piping systems for power plants and process industries. It is important to predict the flow field and temperature field in the mixing region to ensure that the design performs as intended with respect to mixing.

Using this exercise, it is possible to visualize the mixing phenomenon in the elbow region as a function of inlet velocities and temperatures. Comparisons can be made for different inlet temperature and velocity combinations.

2 Modeling Details

The fluid region is represented in two dimensions (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 is achieved.

2.1 Geometry

The geometry consists of walls, two velocity inlets, and a pressure outlet. The flow domain is shown in Figure 2.1. The length of the outlet pipe (L2) can be specified whereas the other geometric dimensions are fixed.

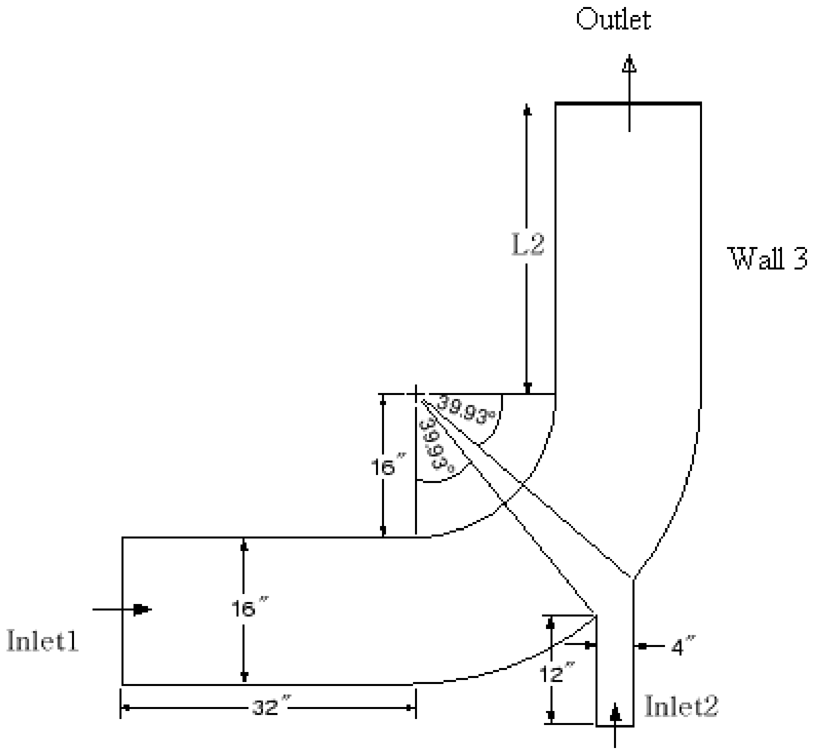

Figure 2.1: Geometry

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 > 20000	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 as follows:

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

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.

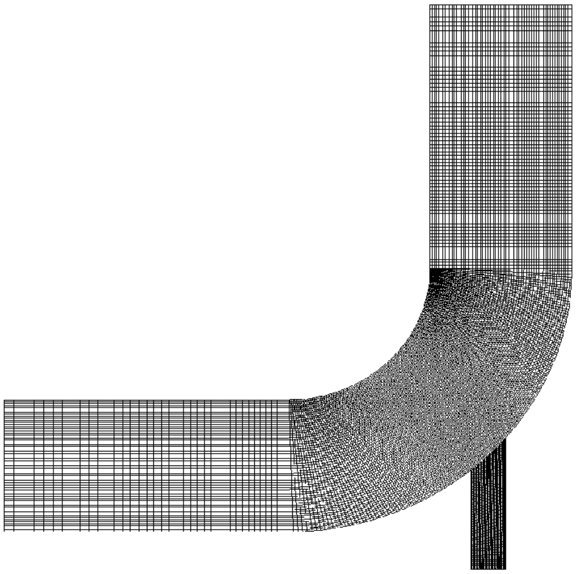

Figure 2.2: Mesh Generated by FlowLab

2.3 Physical Models for FLUENT

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

Re	**Model Used**
Re > 20000	$k-\epsilon$ model

Table 2.3: Turbulence Models Based on Pipe Reynolds Number

Note: *The turbulent flow conditions are specified even if the exercise is run using a Reynolds number less than 20000.*

2.4 Material Properties

The default properties provided in this exercise represent water. Temperature dependence for material properties, including density, viscosity, specific heat, and thermal conductivity can be specified through the following second order polynomial equation:

Property = A0 + (A1 × T) + (A2 × T^2)

where, A0, A1, and A2 are fluid dependent constants.

Two materials (air and water) are readily available in the exercise. A user-specified option is available for defining any additional material. The default values for the constants provided in the exercise are listed in Table 2.4. All the properties in the table are in SI units.

	A0	**A1**	**A2**
Air			
Density	1.6762173930448108	-2.1923479098913349e-03	8.1439756283488314e-07
Viscosity	7.0587110995982522e-06	4.1957623160204437e-08	-6.3559854911267468e-12
Specific Heat (Cp)	9.4235900447551762e+02	1.9317284714300231e-01	-9.0424058617075095e-07
Thermal Conductivity (K)	1.2342068184925949e-02	5.1944011288572976e-05	3.5801428546445151e-09
Water			
Density	8.9696916169663916e+02	1.0033551606212396	-2.2382670604180386e-03
Viscosity	1.2788719593450531e-02	-6.5107718802936428e-05	8.4504799548279901e-08
Specific Heat (Cp)	4.9535071710897391e+03	-5.0376115189154564	8.1953698475433333e-03
Thermal Conductivity (K)	-4.8376708074539942e-01	5.8597967250142054e-03	-7.3404856013554481e-06

Table 2.4: Polynomial Constants for Air and Water

2.5 Boundary Conditions

The following boundary conditions can be specified at both the inlets:

- Inlet fluid velocity
- Inlet fluid temperature
- Wall Roughness

The following boundary conditions are assigned in FLUENT:

Boundary	**Assigned as**
Inlet 1	Velocity inlet
Inlet 2	Velocity inlet
Outlet	Pressure outlet
All the walls including Wall 3	Wall

Table 2.5: Boundary Conditions Assigned in FLUENT

2.6 Solution

The Reynolds number at both inlets is calculated based on the specified boundary conditions and material properties.

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

This exercise is designed to compute turbulent flow and heat transfer in a mixing junction for Reynolds numbers greater than 20000. The $k - \epsilon$ turbulence model will be applied even if boundary conditions are specified such that Reynolds number is less than 20000. It is recommended to use this exercise only to simulate flows where Reynolds number is greater than 20000 at both inlets.

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 drop from Inlet1 to Outlet.
- Temperature change from Inlet1 to Outlet.
- Mass imbalance

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Velocity distribution at Outlet.
- Static pressure distribution at Wall3
- Temperature distribution at Outlet.
- Wall Yplus distribution.

Figures 4.1 and 4.2 show velocity distribution and temperature distribution respectively at the outlet section (Outlet).

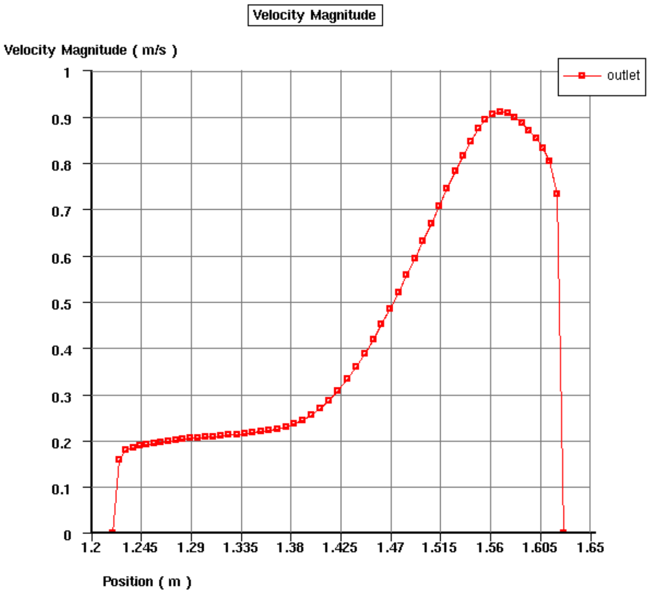

Figure 4.1: Velocity Distribution at Outlet

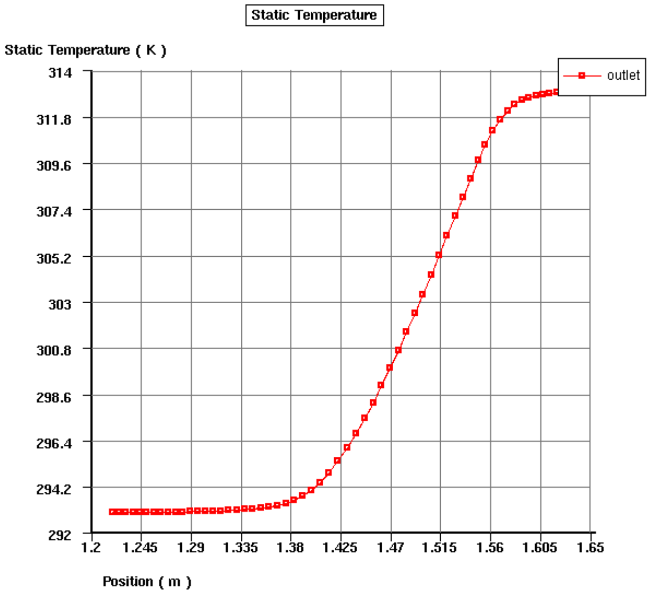

Figure 4.2: Temperature Distribution at Outlet

4.3 Contour Plots

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

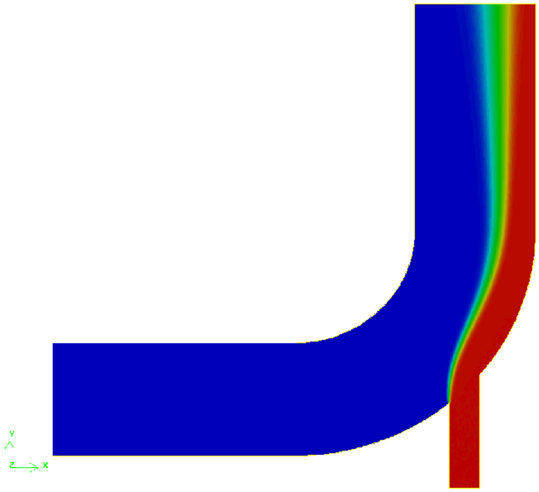

Figure 4.3: Contours of Static Temperature

5 Comparative Study

Figure 5.1 shows temperature distribution at the outlet section for various Inlet2 velocities. Temperature gradient at the outlet is shown to be dependent upon Inlet2 velocity.

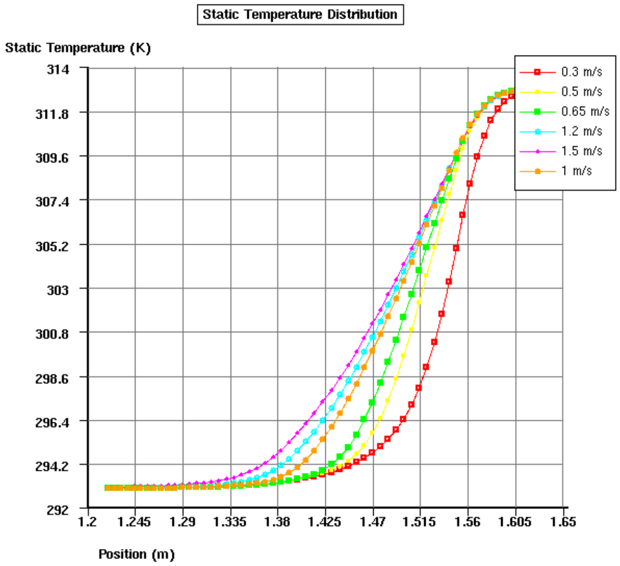

Figure 5.1: Temperature Distribution at Outlet for Different Inlet2 Velocities

6 Reference

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