Simulation of Flow Development in a Pipe

Chapter · December 2006

1 author:

V. T. T Nguyen
Ho Chi Minh University of Industry

Tutorial 4. Simulation of Flow Development in a Pipe

Introduction

The purpose of this tutorial is to illustrate the setup and solution of a 3D turbulent fluid flow in a pipe. The pipe networks are common in any engineering industry. It is important to know the development of a flow at the pipe entrance and pressure drop taking place along the pipe length. The flow of fluids in a pipe is widely studied fluid mechanics problem. The correlations for entry length and pressure drop are available in terms of flow Reynolds number.

This tutorial demonstrates how to do the following:

- Read an existing mesh file in FLUENT.
- Verify the grid for dimensions and quality.
- Add a new material from materials database.
- Define solver settings and perform iterations.
- Examine the results and compare them with experimental data.
- Visualize the flow field using animation tool.
- Control the view by reading a view file.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

Consider a pipe of diameter 1 m and a length of 20 m (Figure 4.1). The geometry is symmetric therefore you will model only half portion of the pipe. Water enters from the inlet boundary with a velocity of 0.015 m/sec. The flow Reynolds number is 15000.

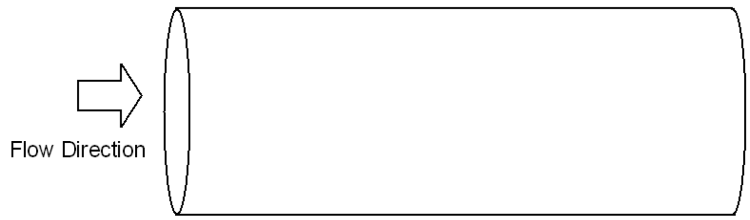


Figure 4.1: Problem Schematic

Preparation

1. Copy the files `pipe.msh` and `view-file` to your working folder.
2. Start the 3D double precision (3ddp) solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file, `pipe.msh`.

   File ⟶ Read ⟶ Case...

   FLUENT *will read the mesh file and report the progress in the console.*

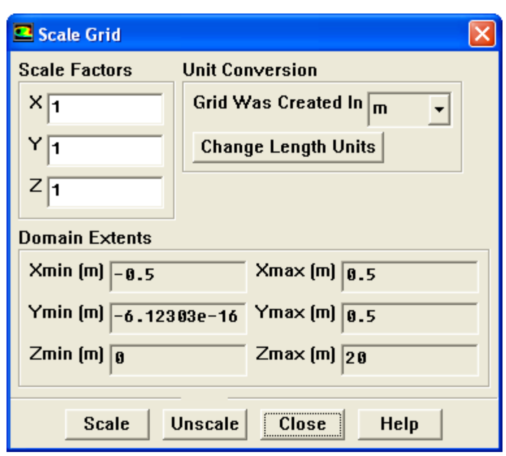
2. Check the grid.

   Grid ⟶ Check

   This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

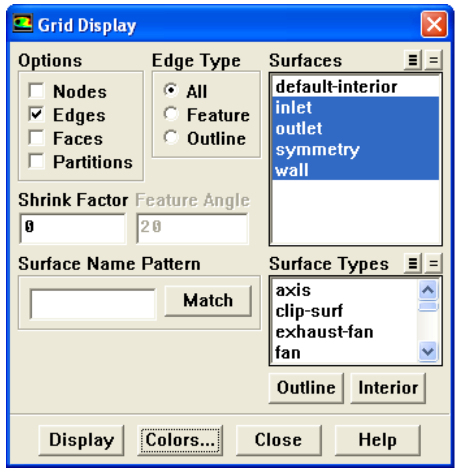
   Grid ⟶ Scale...



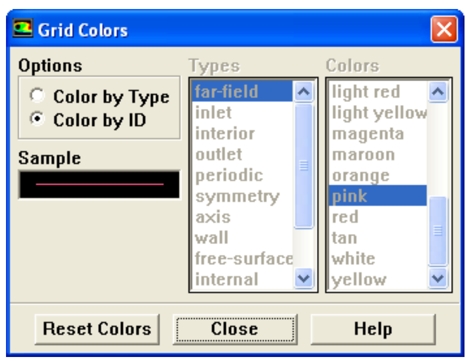
Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units.

4. Display the grid (Figure 4.2).

   Display ⟶Grid...



   (a) Click the Colors... button to open the Grid Colors panel.



      i. Select Color by ID in the Options group box.

      ii. Close the Grid Colors panel.

   (b) Click Display in the Grid Display panel.

   The hidden lines are visible in the Figure 4.2.

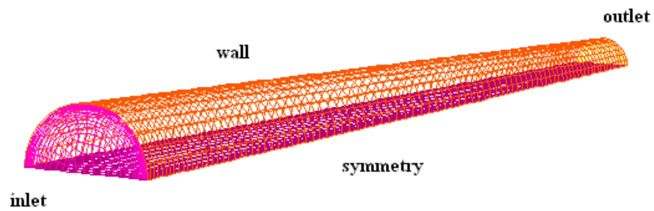
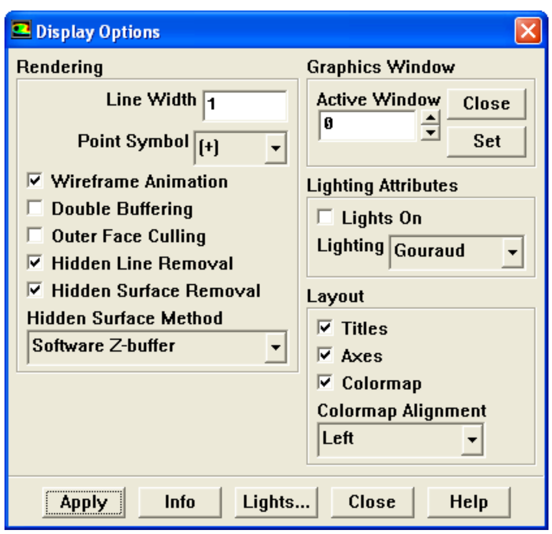


Figure 4.2: Grid Display

(c) Close the Grid Display panel.

5. Remove the hidden lines.

   Display →Options...



(a) Enable Hidden Line Removal.

(b) Select Software Z-buffer from the Hidden Surface Method drop-down list.

(c) Click Apply and close the Display Options panel.

The graphics window will get updated.

(d) Zoom-in the graphics window to get the display as shown in Figure 4.3.

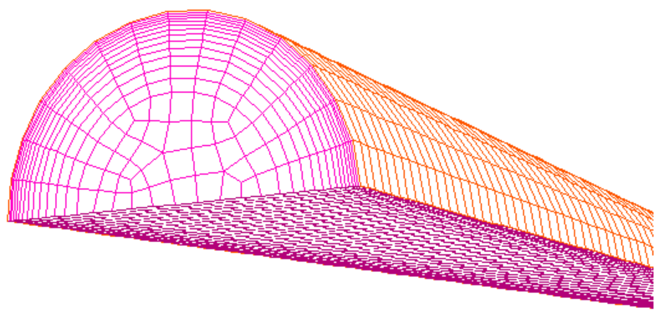


Figure 4.3: Grid Display—Zoomed-in View

Step 2: Models

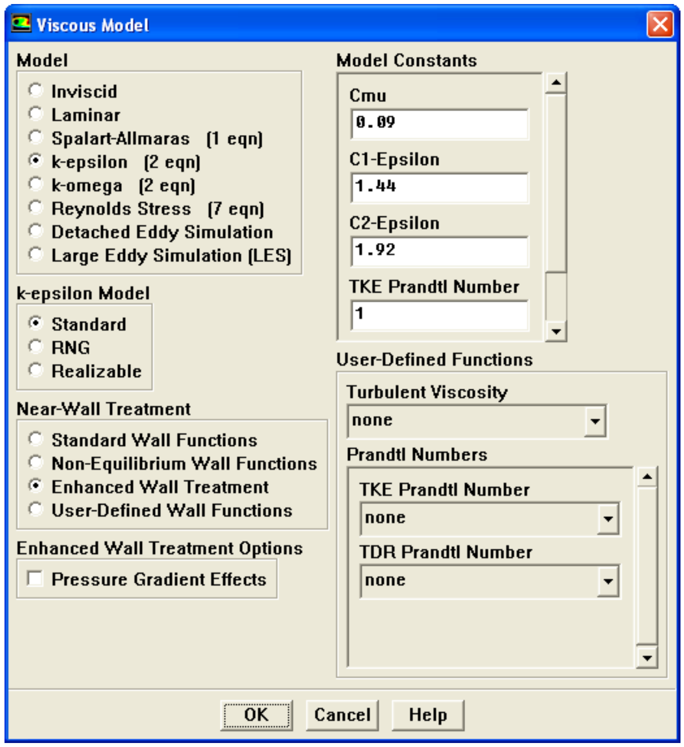
1. Retain the default solver settings.

   Define ⟶ Models ⟶ Solver...

   The problem is solved in steady state using pressure-based solver, therefore, retain the default solver settings.

2. Enable the standard $k - \epsilon$ turbulence model.

   Define ⟶ Models ⟶ Viscous...



(a) Select k-epsilon (2 eqn) from the Model list.

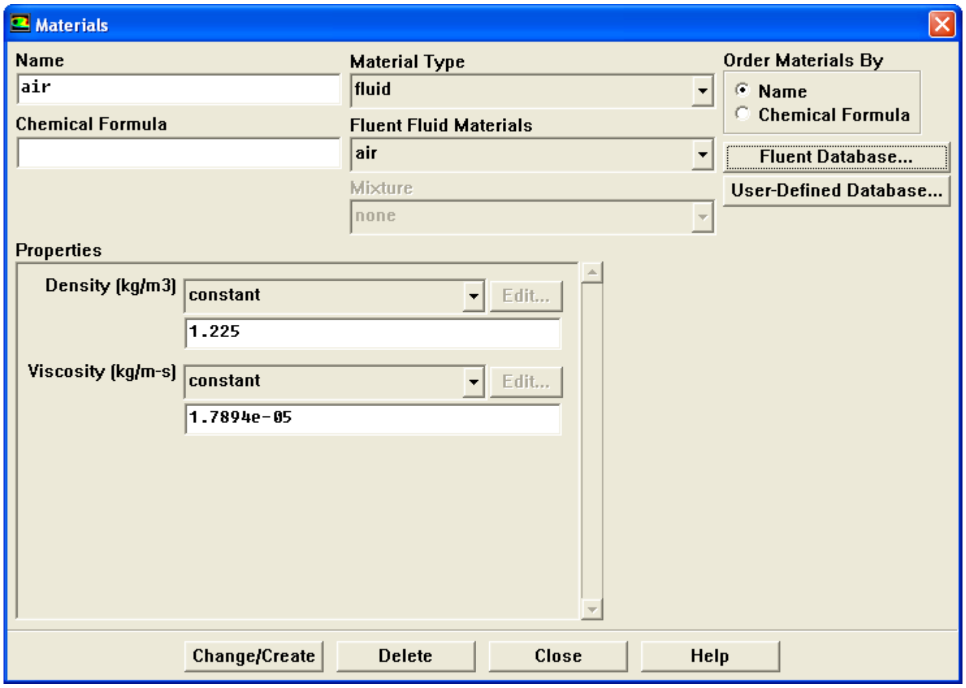
(b) Enable Enhanced Wall Treatment from the Near-Wall Treatment list.

(c) Click OK to close the Viscous Model panel.

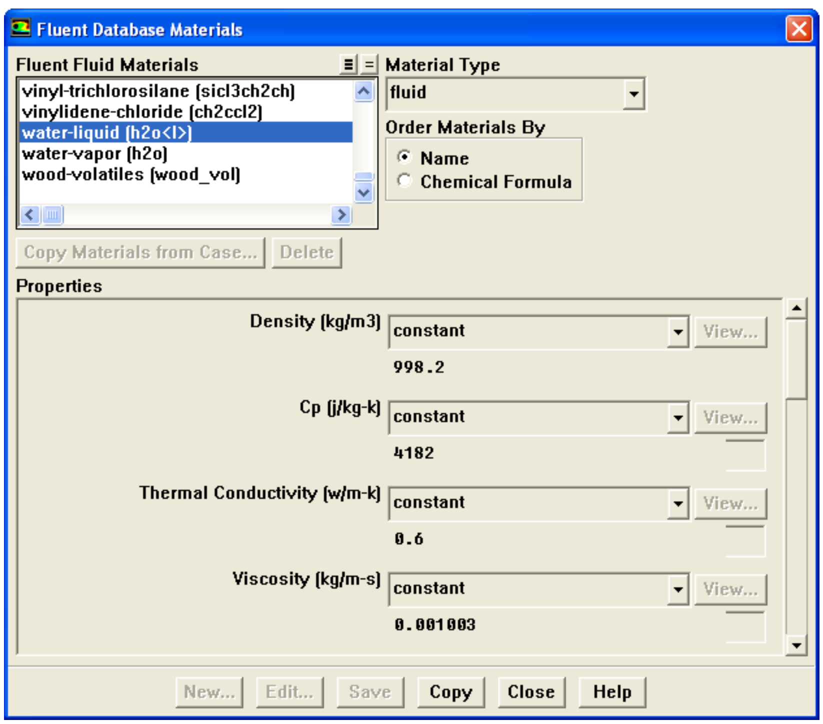
Step 3: Materials

1. Copy `water-liquid (h2o <l>)` from the FLUENT database.

   $\boxed{\text{Define}}$ ⟶ Materials...



(a) Click the Fluent Database... button to open the Fluent Database Materials panel.



i. Select `water-liquid (h2o <l>)` from the Fluent Fluid Materials selection list.

This will display the default settings for water-liquid.

ii. Click Copy and close the fluent Database Materials panel.

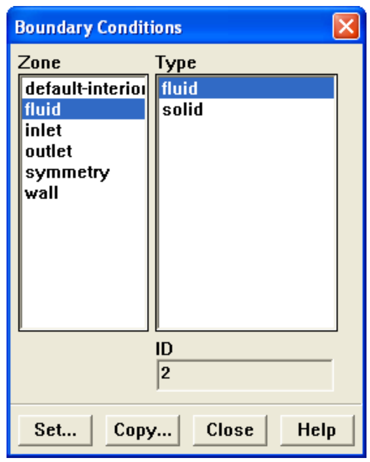
The Materials *panel will now display the copied information of water.*

(b) Click Change/Create and close the Materials panel.

Step 4: Boundary Conditions

1. Set the boundary conditions for fluid.

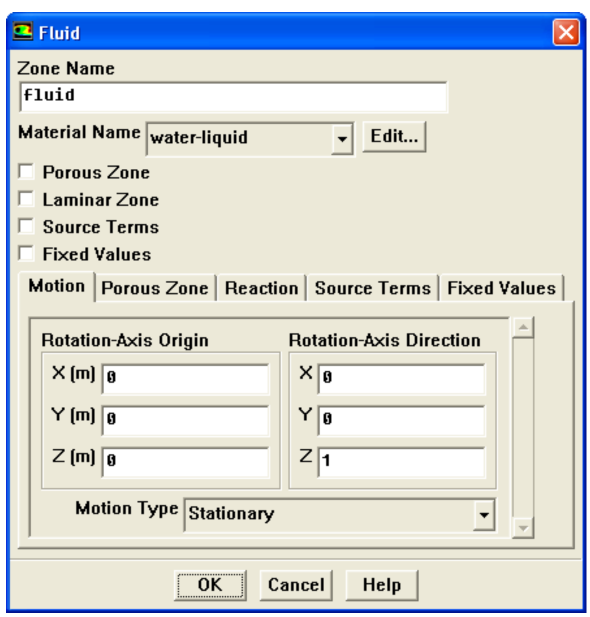
   Define ⟶Boundary Conditions...



(a) Select fluid from the Zone selection list.

The Type *will be reported as* fluid*.*

(b) Click the Set... button to open the Fluid panel.



i. Select water-liquid from the Material Name drop-down list.

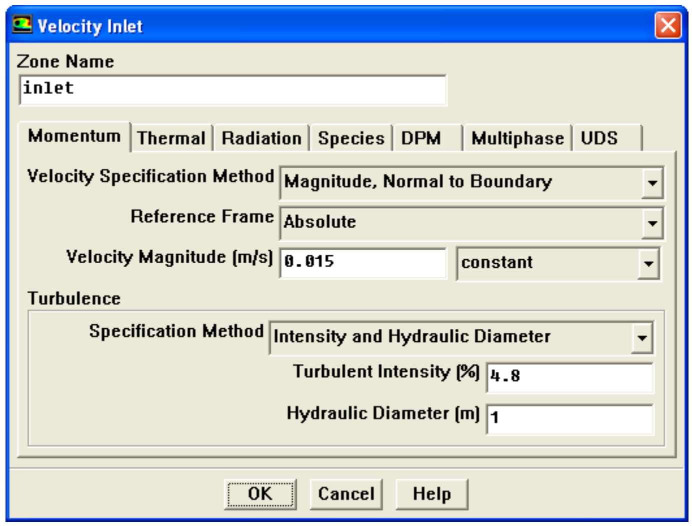
ii. Click OK to close the Fluid panel.

2. Set the boundary conditions for inlet.

(a) Select inlet from the Zone selection list.

The Type *will be reported as* velocity-inlet.

(b) Click the Set... button to open the Velocity Inlet panel.



i. Enter 0.015 m/s for Velocity Magnitude.

The Reynolds number is defined as:

$$R_e = \frac{U \times D \times \rho}{\mu} \tag{4.1}$$

for fluid properties of water-liquid and to have Re = 15000, the velocity should be set to 0.015 m/sec.

ii. Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop-down list.

iii. Enter 4.8% for Turbulent Intensity.

Turbulent Intensity can be calculated as:

$$T.I. = 0.16 \times Re^{-1/8} \tag{4.2}$$

iv. Enter 1 m for Hydraulic Diameter.

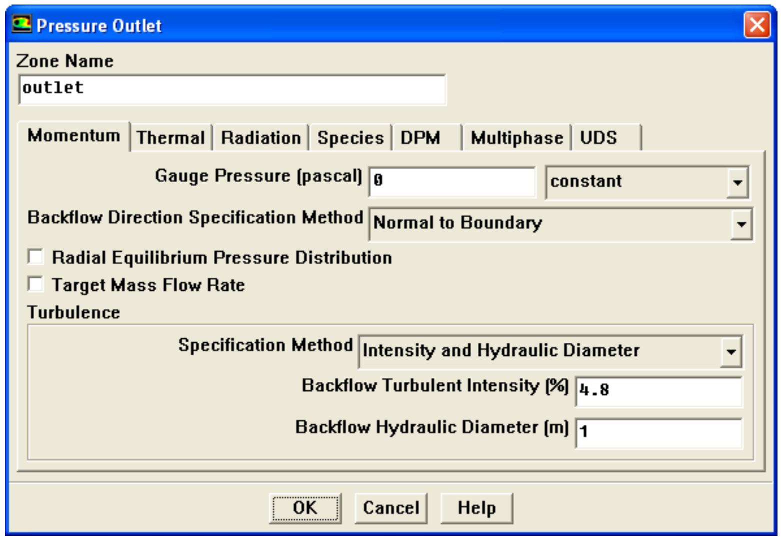
v. Click OK to close the Velocity inlet panel.

3. Set the boundary conditions for outlet.

   (a) Select outlet from the Zone selection list.

   The Type *will be reported as* pressure-outlet.

   (b) Click the Set... button to open the Pressure Outlet panel.



Use the default value for gauge pressure since the outlet is maintained at atmospheric pressure.

   i. Select Intensity and Hydraulic Diameter from the Turbulence Specification Method drop-down list.

   ii. Enter 4.8% for Turbulent Intensity.

   iii. Enter 1 m for Hydraulic Diameter.

   These values of turbulence parameters will be used only if reverse flow occurs at the outlet.

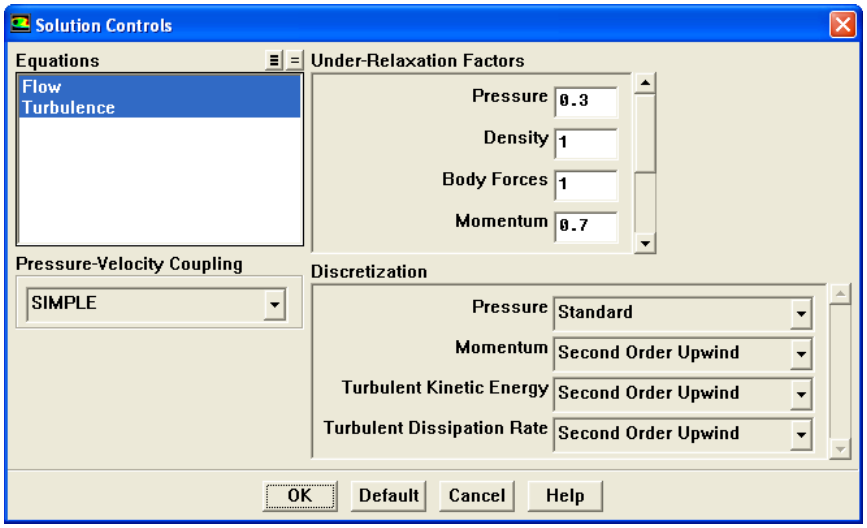
   iv. Click OK to close the Pressure Outlet panel.

4. Close the Boundary Conditions panel.

Step 5: Solution

1. Set the solution controls.

   Solve ⟶ Controls ⟶ Solution...



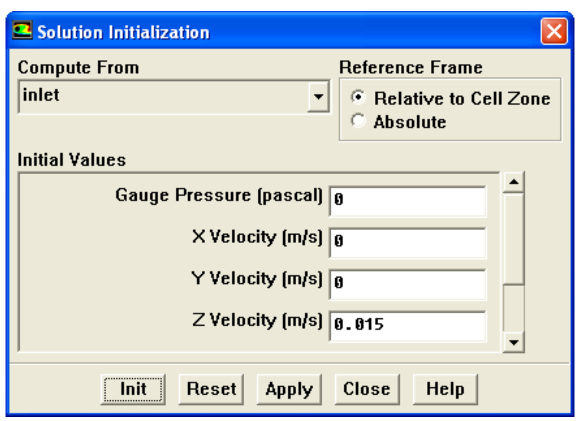
(a) Select Second Order Upwind from the Momentum, Turbulence Kinetic Energy, and Turbulence Dissipation Rate drop-down lists.

Since the flow is not very complex, you can use higher order discretization schemes directly. In case of complex flows, it is recommended to obtain a converged solution using first order schemes before switching to higher order schemes.

(b) Click OK to close the Solution Controls panel.

2. Initialize the flow.

Solve ⟶ Initialize ⟶ Initialize...

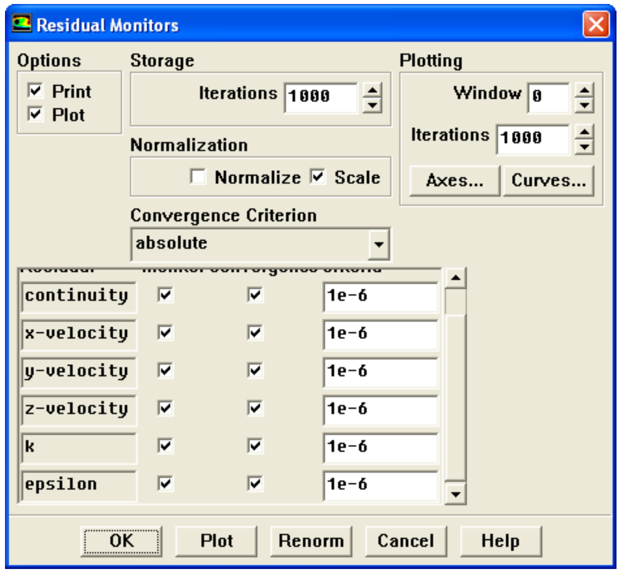


(a) Select inlet from the Compute From drop-down list.

(b) Click Init and close the Solution Initialization panel.

3. Enable plotting of residuals during the calculation and change the convergence criteria.

Solve ⟶ Monitors ⟶ Residual...



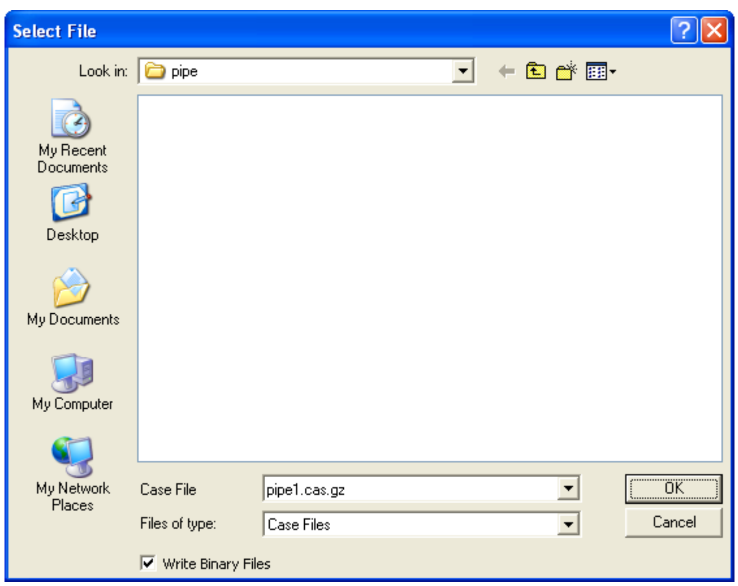
(a) Enable Plot in the Options group box.

(b) Enter `1e-6` for Absolute Criteria for all the equations.

(c) Click OK to close the Residual Monitors panel.

4. Save the case file (`pipe1.cas.gz`).

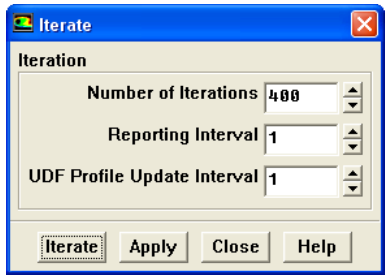
File ⟶ Write ⟶ Case...



Retain the default enabled Write Binary Files *option so that you can write a binary file. The* `.gz` *option will save zipped files, this will work on both,* Windows *as well as* LINUX/UNIX *platforms.*

5. Iterate the solution.

   $\boxed{\text{Solve}}$ ⟶Iterate...



   (a) Set Number of Iterations to 400.

   (b) Click Iterate to start the calculation.

   The solution converges in approximately 215 iterations. The residuals plot is shown in Figure 4.4.

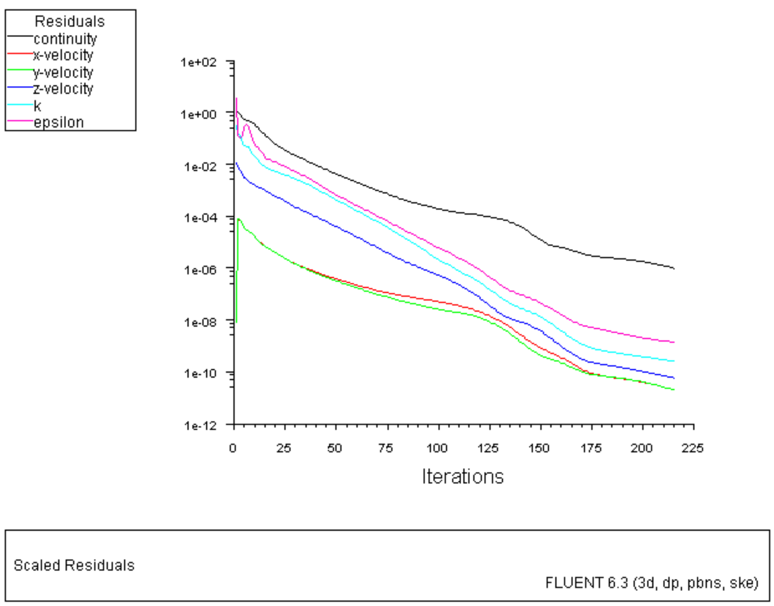
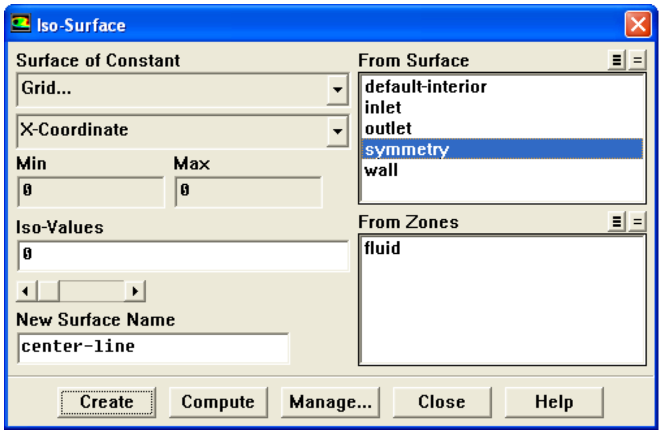


Figure 4.4: Scaled Residuals

   (c) Close the Iterate panel.

6. Create an isosurface to plot the variation of velocity along the axis of the pipe.

   Surface ⟶Iso-Surface...



   (a) Select symmetry from the From Surface selection list.

   An axis will lie at intersection of symmetry and x= 0 plane. To create such line surface you need to select symmetry *from the* From Surface *selection list.*

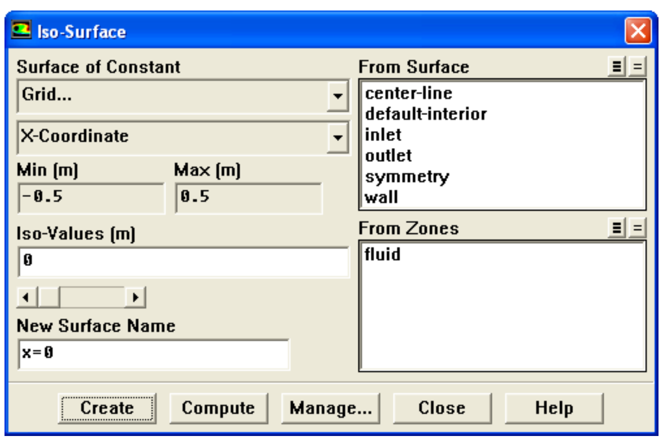
   (b) Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.

   (c) Enter 0 for Iso-Values.

   (d) Enter `center-line` for New Surface Name.

   (e) Click Create.

   (f) Create another surface at x=0.



   i. Deselect symmetry from the From Surface selection list.

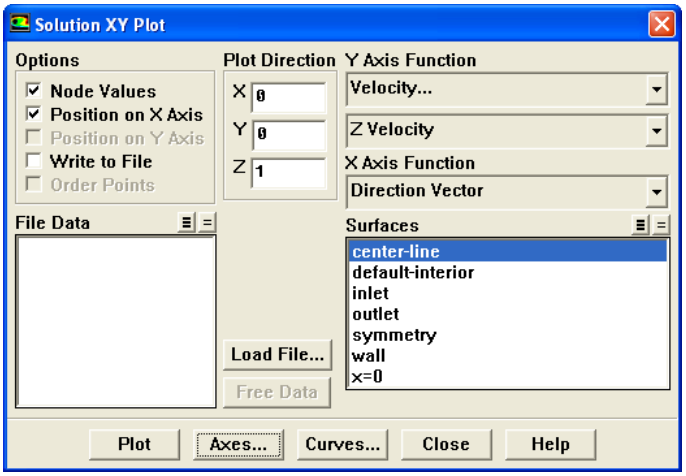
   ii. Enter x=0 for New Surface Name.

   iii. Click Create and close the Iso-Surface panel.

   This surface will be used for displaying contours of velocity during post-processing.

7. Create XY plots (Figure 4.5).

   Plot ⟶XY Plot...

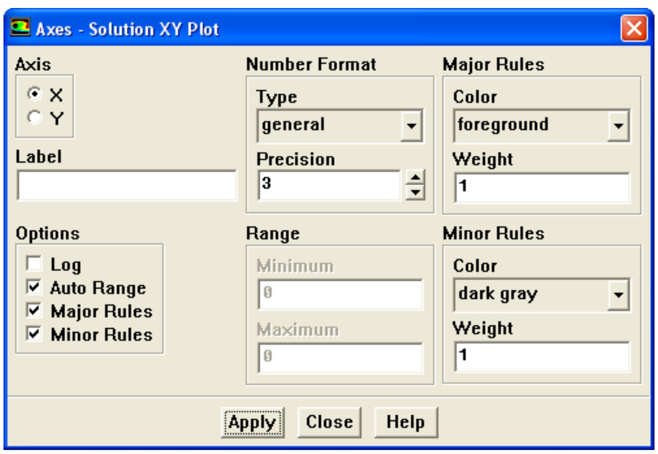


   (a) Select center-line from the Surfaces selection list.

   (b) Set Plot Direction as X = 0,Y = 0, and Z = 1.

   (c) Select Velocity... and Z Velocity from the Y Axis Function drop-down list.

   (d) Click the Axes... button to open the Axes-Solution XY Plot panel.



i. Enable Major Rules and Minor Rules in the Options group box.

ii. Click Apply.

iii. Select Y in the Axis group box.

iv. Enable Major Rules and Minor Rules in the Options group box.

v. Click Apply and close the Axes-Solution XY Plot panel.

(e) Click Plot and close the Solution XY Plot panel.

Figure 4.5 can be used to calculate the entry length of the pipe. The distance along the pipe length where velocity reaches 99.9% of its final value is called as the entry length. From the above plot the value of entry length comes to be 15.19 m, which matches closely with the values reported in the literature [3].

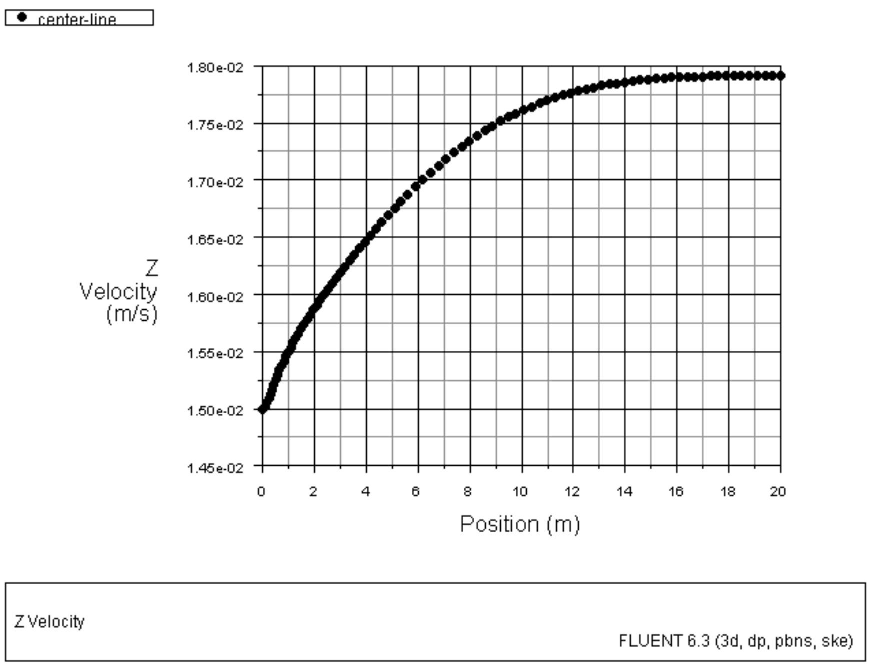


Figure 4.5: Plot of Z-velocity on center-line

8. Calculation of friction factor at outlet.

   Friction factor (f) is used to indicate the pressure drop in a pipe [2]. It is defined as:

$$f = \frac{\Delta P \times 2 \times D}{L \times \rho \times v^2} \tag{4.3}$$

   where,

ΔP	=	*Pressure drop along pipe length (L)*
D	=	*Pipe diameter*
v	=	*Average velocity in the pipe section*
ρ	=	*Fluid density*

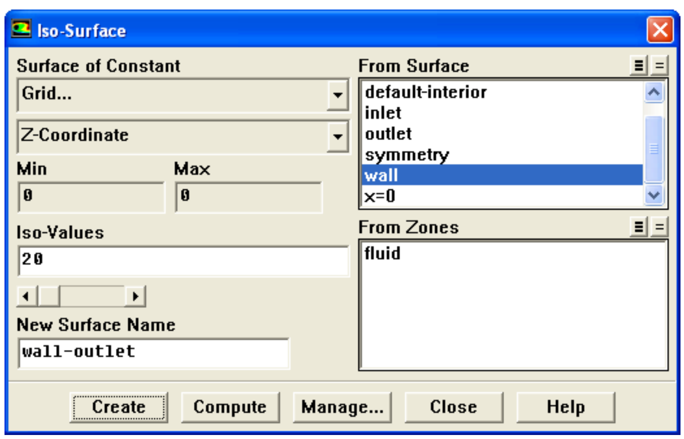
   After balancing the pressure and shear forces the same expression takes the form:

$$f = \frac{8 \times \tau}{\rho \times v^2} \tag{4.4}$$

   where, τ = *Shear stress on the wall*

9. Create a line surface at intersection of wall and outlet.

   Surface ⟶Iso-Surface...



   (a) Select wall from the From Surface selection list.

   (b) Select Grid... and Z-Coordinate from the Surface of Constant drop-down lists.

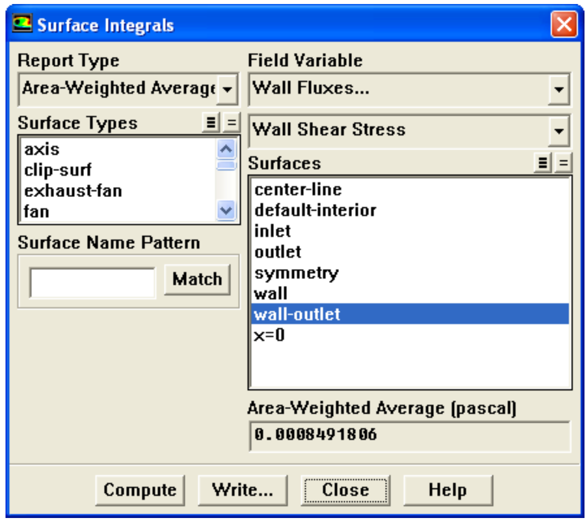
   (c) Enter 20 for Iso-Values.

   (d) Enter `wall-outlet` for New Surface Name.

   (e) Click Create and close the Iso-Surface panel.

10. Report surface integral.

    Report ⟶Surface Integrals...



(a) Calculate the average shear stress at the outlet.

i. Select Area-Weighted Average from the Report Type drop-down list.

ii. Select Wall Fluxes... and Wall Shear Stress from the Field Variable drop-down lists.

iii. Select wall-outlet from the Surfaces selection list.

iv. Click Compute.

The value of shear stress will be updated in the Area Weighted Average *field as* 0.0008491806.

(b) Calculate average velocity at outlet.

i. Select Mass-Weighted Average from the Report Type drop-down list.

ii. Select Velocity... and Z Velocity from the Field Variable drop-down list.

iii. Select outlet from the Surfaces selection list.

iv. Click Compute and close the Surface Integrals panel.

The value of bulk velocity will be updated in the Mass-Weighted Average *field as* 0.01559819.

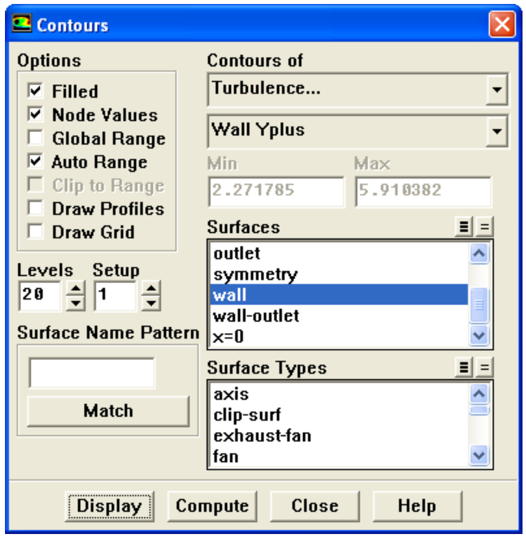
Using Equation 8, friction factor can be calculated as:

$$f = \frac{8 \times \tau}{\rho \times v^2} = \frac{8 \times 0.000849}{998.2 \times (0.015599)^2} = 0.02796 \tag{4.5}$$

Step 8: Postprocessing

1. Display filled contours of wall Yplus (Figure 4.6).

   Display ⟶Contours...



   (a) Select Turbulence... and Wall Yplus from the Contours of drop-down lists.

   (b) Select wall from the Surfaces selection list.

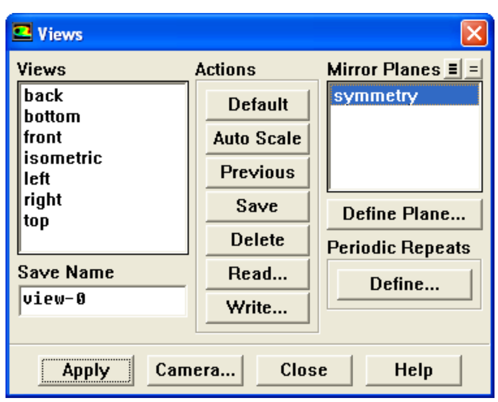
   (c) Enable Filled and disable Global Range in the Options group box.

   (d) Click Display and close the Contours panel.

   The Yplus value for most of the domain is less than 5, except for the cells near inlet, where it is slightly higher. This shows that enhanced wall treatment is acceptable as a wall function.

2. Enable the mirror plane to view the complete geometry.

   Display ⟶Views...



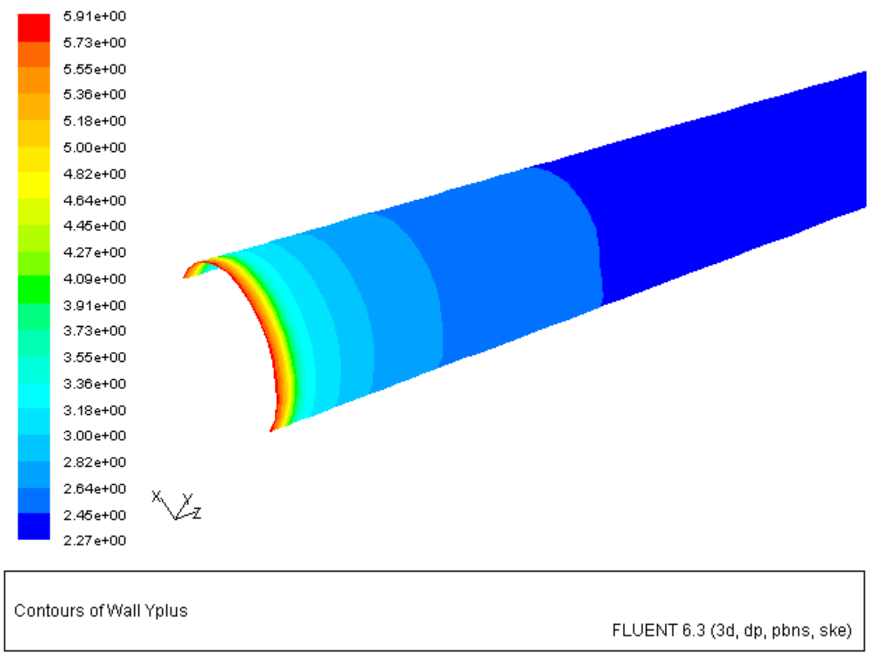


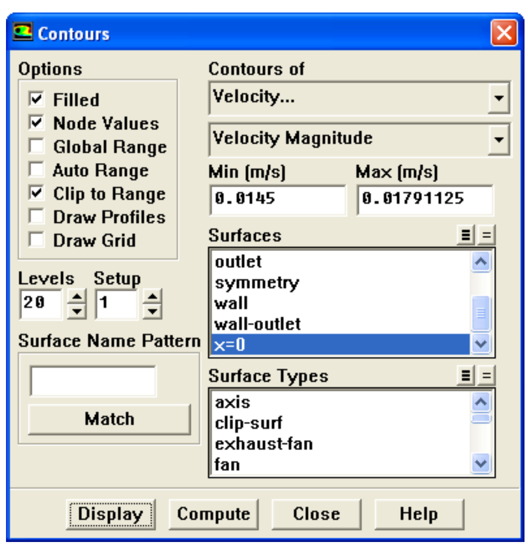
Figure 4.6: Contours of Wall Yplus on wall

(a) Select symmetry from the Mirror Planes selection list.

(b) Click Apply and close the Views panel.

3. Display the velocity contours on surface x=0 (Figure 4.7).

$\boxed{\text{Display}}\longrightarrow$Contours...



(a) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.

(b) Select only x=0 from the Surfaces selection list.

(c) Click Compute.

(d) Disable Auto Range in the Options group box.

(e) Enter 0.0145 for Min and retain the default value for Max.

(f) Click Display and close the Contours panel.

Figure 4.7 shows the development of velocity along the length of pipe.

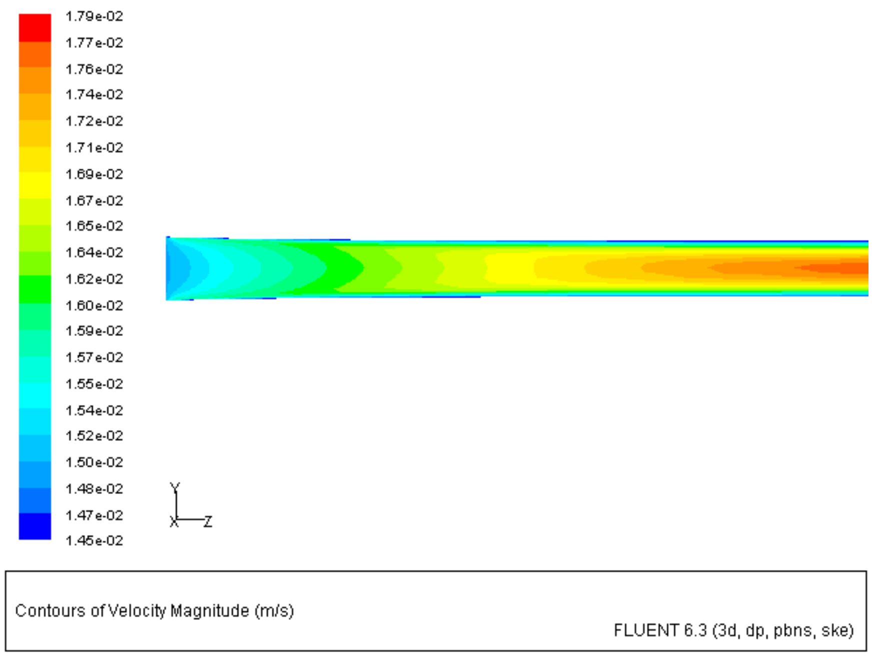
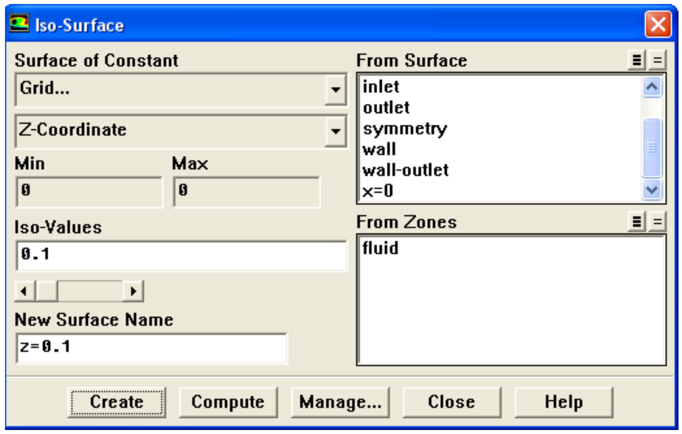


Figure 4.7: Contours of Velocity Magnitude on x=0

4. Create another isosurface.

Surface ⟶Iso-Surface...



(a) Select Grid... and Z-Coordinate from the Surface of Constant drop-down lists.

Note: *Please make sure that no surface is selected from the* Surfaces *selection list.*

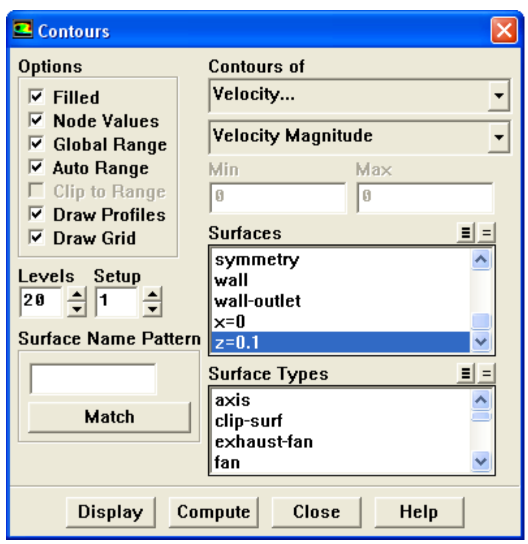
(b) Enter `0.1` for Iso-Values.

(c) Enter `z=0.1` for New Surface Name.

(d) Click Create and close the Iso-Surface panel.

5. Display the velocity contours on surface z=0.1 (Figure 4.8).

Display →Contours...

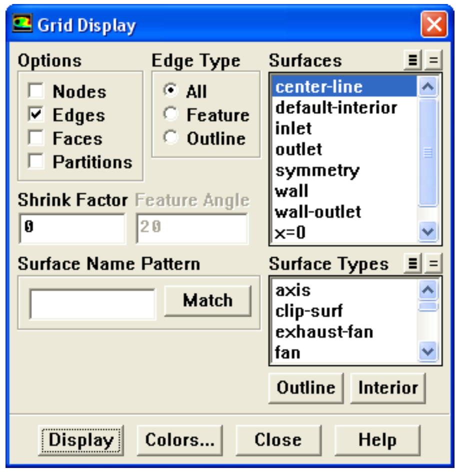


(a) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.

(b) Select only `z=0.1` from the Surfaces selection list.

(c) Enable Global Range and Auto Range in the Options group box.

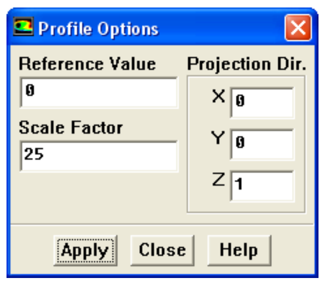
(d) Enable Draw Grid to open the Grid Display panel.



i. Deselect all the surfaces and select center-line from the Surfaces selection list.

ii. Click Display and close the Grid Display panel.

(e) Enable Draw Profiles in the Options group box to open the Profile Options panel.



i. Set Projection Dir. to 0, 0, 1 respectively.

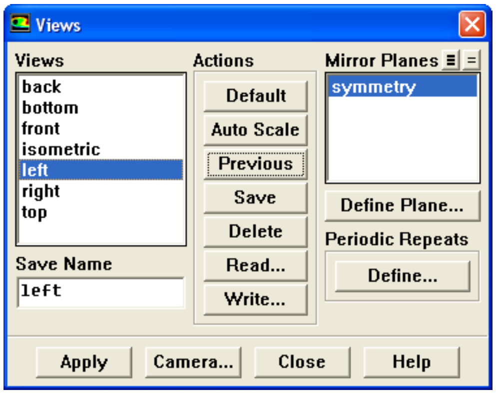
ii. Set Scale Factor to 25.

iii. Click Apply and close the Profile Options panel.

(f) Click Display and close the Contours panel.

6. Set the view.

Display ⟶Views...



(a) Select left from the Views selection list.

(b) Click Previous in the Actions group box.

Graphics window will get updated. Zoom-in to get the view as shown in Figure 4.8.

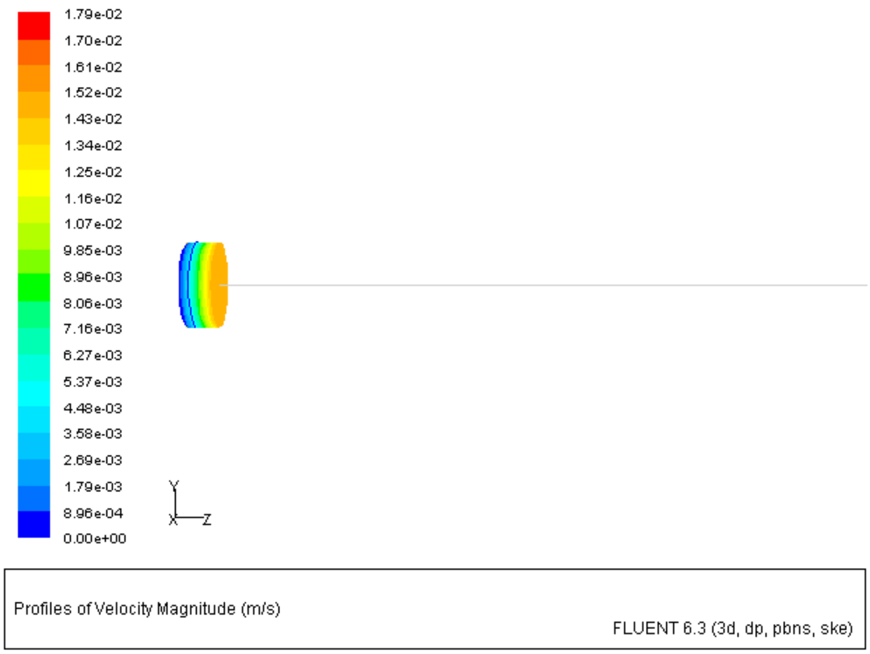
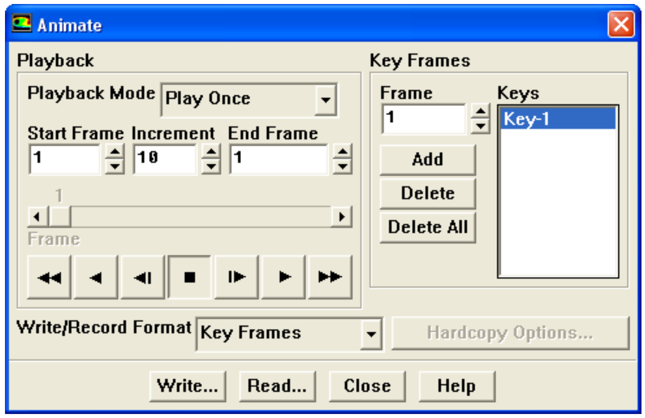


Figure 4.8: Profiles of Velocity Magnitude on z=0.1

7. Create an animation.

   Display ⟶Scene Animation...

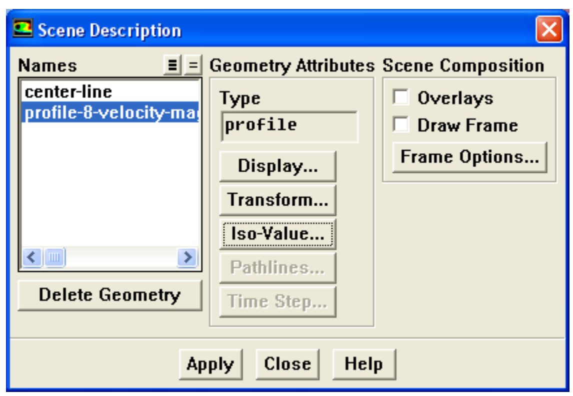


   (a) Click the Add button.

   The first frame named Key-1 *is added in the* Keys *selection list.*

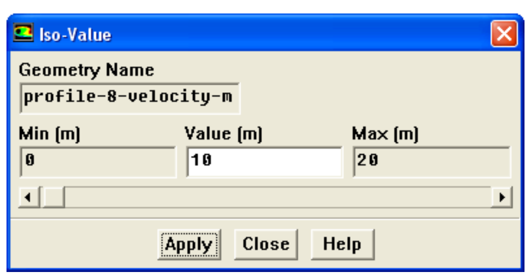
   (b) Set the animation frame.

   Display ⟶Scene...



   i. Select profile-8-velocity-magnitude from the Names selection list.

   ii. Click the Iso-Value... button to open the Iso-Value panel.



   A. Enter 10 for Value.

B. Click Apply and close the Iso-Value panel.

The display window will get updated as shown in Figure 4.9.

iii. Close the Scene Description panel.

(c) Enter 150 for Frame and click the Add button, in the Animate panel.

The last frame named Key-150 *is added in the* Keys *selection list.*

(d) Click  button to visualize the variation in velocity in the graphics window.

Select MPEG *from the* Write/Record Format *drop-down list and click the* Write... *button to create a movie file of this animation.*

(e) Close the Animate panel

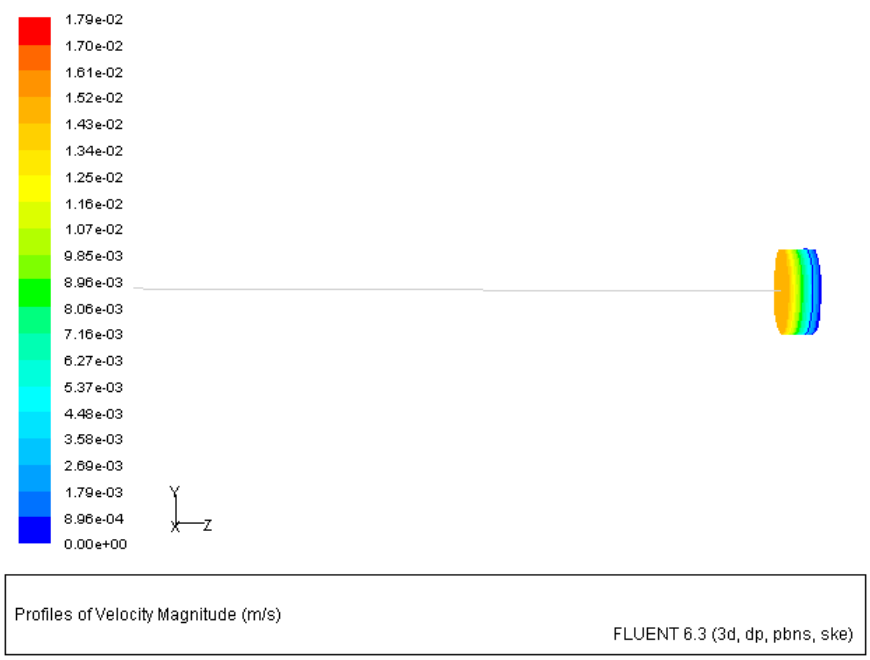
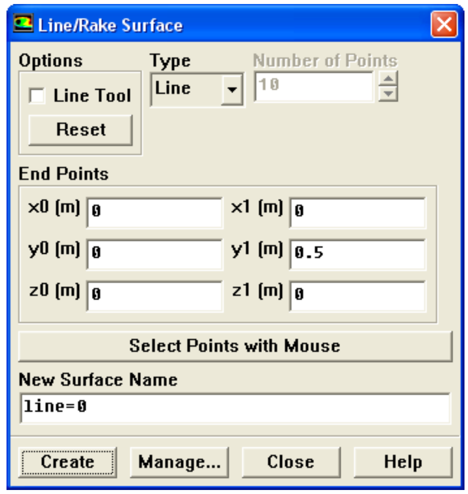


Figure 4.9: Profiles of Velocity Magnitude—Frame 150

8. Create line surfaces to visualize the velocity vectors at different axial locations.

   $\boxed{\text{Surface}}$ ⟶Line/Rake...



   (a) Enter 0 for x0, y0, and z0.

   (b) Enter 0, 0.5, and 0 for x1, y1, and z1 respectively.

   (c) Enter `line=0` for New Surface Name.

   (d) Click Create.

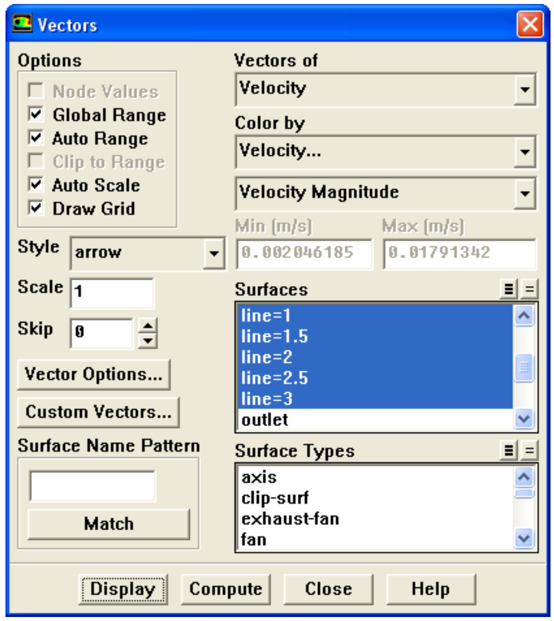
   (e) Similarly, create 6 more surfaces with values of both, z0 and z1, set to 0.5, 1, 1.5, 2, 2.5, and 3.

   Name the surfaces accordingly, e.g., line=0.5, line=1 *etc.*

   (f) Close the Line/Rake Surface panel.

9. Display velocity vectors (Figure 4.10).

   $\boxed{\text{Display}}$ ⟶Vectors...

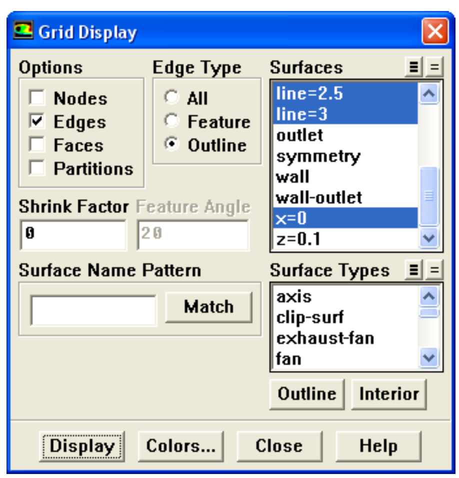


(a) Select Velocity from the Vectors of drop-down list.

(b) Select Velocity... and Velocity Magnitude from the Color by drop-down lists.

(c) Select all the line/rake surfaces created in the previous step from the Surfaces selection list.

(d) Enable Draw Grid in the Options group box to open the Grid Display panel.



i. Select all the line surfaces and x=0 surface from the Surfaces selection list.

ii. Enable Outline in the Edge Type group box.

iii. Click Display and close the Grid Display panel.

(e) Click Display and close the Vectors panel.

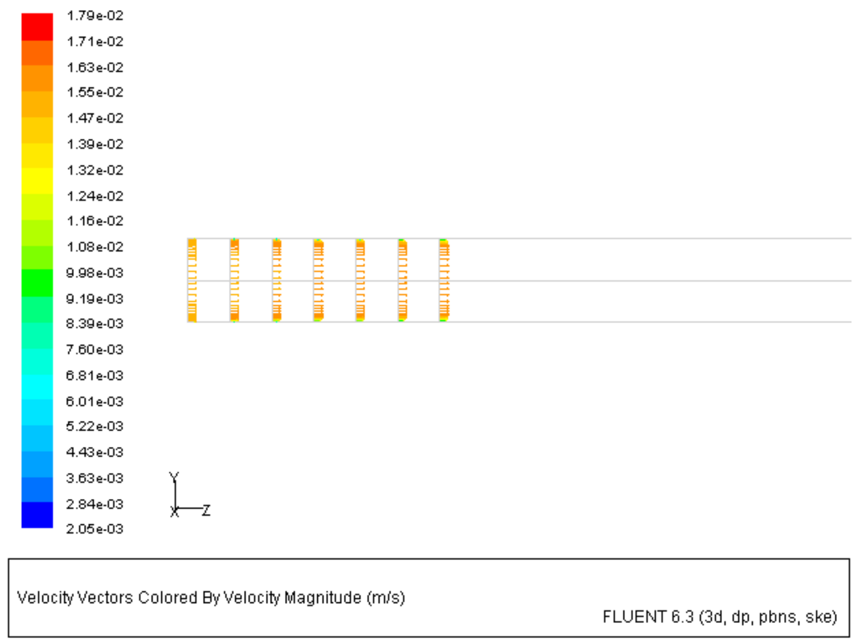
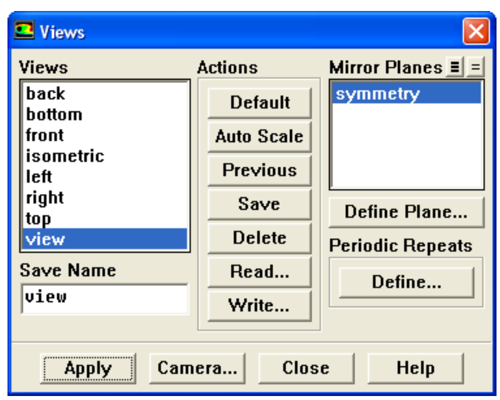


Figure 4.10: Velocity Vectors

10. Change the view to get a closer view of velocity vectors.

    Display →Views...



(a) Click Read... in the Actions group box to open the Select File dialog box.

    i. Select the view-file and click OK.

(b) Select view from the Views selection list.

(c) Click Apply and close the Views panel.

The display window will get updated as shown in Figure 4.11.

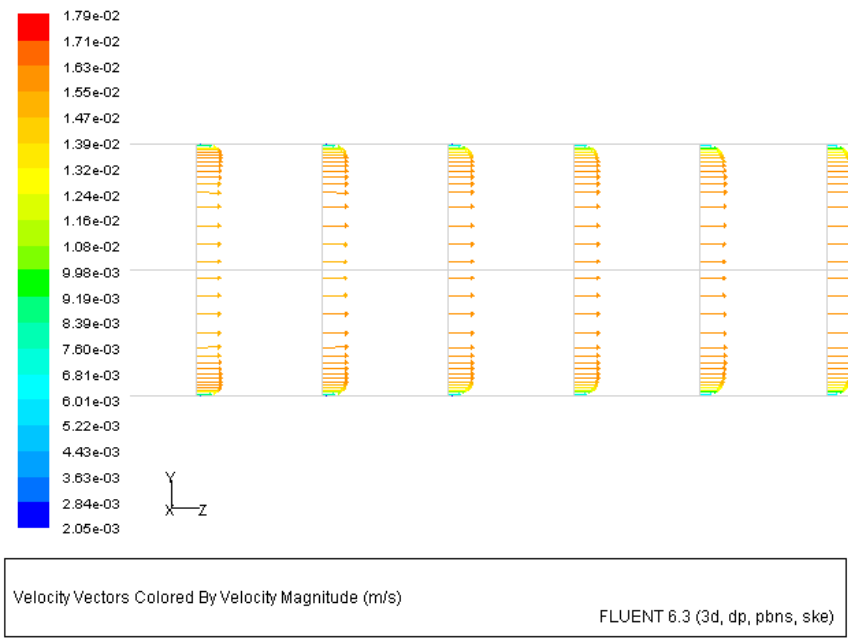


Figure 4.11: Velocity Vectors Close-up

11. Save the case and data files (`pipe2.cas.gz` and `pipe2.dat.gz`).

    File ⟶ Write ⟶Case & Data...

Summary

This tutorial demonstrated the use of symmetry boundary condition to reduce the computational time without compromising the flow physics. Entry length and friction factor values obtained from FLUENT matches to those obtained from correlations. You also learned to use scene animation to visualize the development of flow field in the domain.

References

1. White Frank M., *Viscous Fluid Flow*, International Edition, McGraw-Hill, 1991, p 423.
2. Schlichting H., *Boundary-Layer Theory*, 7th Edition, McGraw-Hill, 1979.
3. John M. Cimbala and Yunus A. Cengel, *Fluid Mechanics Fundamentals and Applications*, 1st Edition, McGraw-Hill, 2005.

Exercises/ Discussions

1. Find the maximum Reynolds number for which current pipe length is sufficient to obtain a fully developed flow at outlet.
2. Compare the results using different wall functions.
3. Study the effect of roughness on the friction factor.
4. Change the velocity values and model laminar flow conditions.
5. Compare the results with pressure outlet boundary and outflow. How do they differ?
6. Conduct multiple runs by increasing the static pressure at the outlet. How does it influence the flow?
7. Simulate a periodic flow using the same mesh by creating a periodic zone using inlet and outlet boundaries.
8. Apply a fully developed flow profile using profile files or user defined functions.

Links for Further Reading

- http://www.owlnet.rice.edu/~mech372/handouts/viscous_flows_pipes.pdf
- http://www.engr.psu.edu/ce/hydro/hill/teaching/java/pipeflow.html
- http://www.cartage.org.lb/en/themes/Sciences/Physics/Mechanics/FluidMechanics/RealFluids/Laminar/Laminar.htm
- http://instruct1.cit.cornell.edu/courses/fluent/pipe2/index.htm