See discussions, stats, and author profiles for this publication at: https://www.researchgate.net/publication/273139027

## Simulation of Flow Development in a Pipe

Chapter · December 2006

CITATIONS 0		READS 4,425		
1 author	1 author:			
BK	V. T. T Nguyen Ho Chi Minh University of Industry 32 PUBLICATIONS 46 CITATIONS			
	SEE PROFILE			

Some of the authors of this publication are also working on these related projects:



An investigation on Francis Turbine by experiments View project

# 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.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case...}$ 

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

2. Check the grid.

 $Grid \longrightarrow Check$ 

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.

 $Grid \longrightarrow Scale...$ 

🗳 Scale Grid 🛛 🔀			
Scale Factors Unit Conversion			
X 1	Grid Was Created In 🖬 👻		
Y 1	Change Length Units		
Z 1			
Domain Extents			
Xmin (m) -0.5 Xmax (m) 0.5			
Ymin (m) -6.1230	93e-16 Ymax (m) 0.5		
Zmin (m) 👔	Zmax (m) 20		
Scale	Inscale Close Help		

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).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

💶 Grid Display		X
Options Nodes Edges Faces Partitions	Edge Type C All C Feature C Outline Eeature Angle 20	Surfaces E = default-interior inlet outlet symmetry wall
Surface Name Pattern Match		Surface Types = = axis clip-surf exhaust-fan fan Outline Interior
Display	Colors	Close Help

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

Grid Colors		X
Options	Types	Colors
C Color by Type Color by ID Sample	far-field inlet interior outlet periodic symmetry axis wall free-surface internal	light red light yellow maroon orange pink red tan white yellow
Reset Colors	Close	Help

- i. Select Color by ID in the Options group box.
- ii. Close the  $\mathsf{Grid}\xspace$  panel.
- (b) Click  $\mathsf{Display}$  in the  $\mathsf{Grid}$   $\mathsf{Display}$  panel.

The hidden lines are visible in the Figure 4.2.



Figure 4.2: Grid Display

- (c) Close the  $\mathsf{Grid}$  Display panel.
- 5. Remove the hidden lines.

 $\mathsf{Display} \longrightarrow \mathsf{Options}...$ 

Display Options	X	
Display Options      Rendering     Line Width 1     Point Symbol (+)     Wireframe Animation     Double Buffering     Outer Face Culling	Graphics Window Active Window Uighting Attributes Lighting Attributes	
Outer Face Culling         ✓ Hidden Line Removal         ✓ Hidden Surface Removal         Hidden Surface Method         Software Z-buffer	Lighting Gouraud  Layout          Image: Constraint of the second secon	
Apply Info Lights.	Colormap Alignment	

- (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  $\longrightarrow$  Models  $\longrightarrow$  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.

 $\boxed{\text{Define}} \longrightarrow \boxed{\text{Models}} \longrightarrow \boxed{\text{Viscous}} \dots$ 

- (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

Copy water-liquid (h2o <1>) from the FLUENT database.
 Define → Materials...

💶 Materials			
Name		Material Type	Order Materials By
air		fluid	O Name     O Chemical Formula
Chemical Formula		Fluent Fluid Materials	
1		air	Fluent Database
		Mixture	User-Defined Database
Bronarties		Inone	
Density (kg/m3) Viscosity (kg/m-s)	constant 1.225 constant 1.7894e-05	▼ Edit	
	Change/Create	Delete Close	Help

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

Fluent Database Materials	
Fluent Fluid Materials = = vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2) water-liquid (h2o<1>) water-vapor (h2o) wood-volatiles (wood_vol) Copy Materials from Case Delete Properties	Material Type fluid Order Materials By Name C Chemical Formula
Density (kg/m3) Cp (j/kg-k)	constant     View       998.2       constant
Thermal Conductivity (w/m-k)	4182 constant  ▼ View 0.6
Viscosity (kg/m-s)	constant         View           0.001003

i. Select water-liquid (h2o <1>) 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...

Boundary Condit	ions 🛛 🔀
Zone default-interiou fluid inlet outlet symmetry wall	Type fluid solid
	ID 2
Set Copy	/ Close Help

- (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.

🗳 Fluid 🛛 🔀
Zone Name
fluid
Material Name water-liquid 🗾 Edit
🗆 Porous Zone
Laminar Zone
Fixed Values
Motion   Porous Zone   Reaction   Source Terms   Fixed Values
Rotation-Axis Origin Rotation-Axis Direction
Y (m) 0 Y
Motion Type Stationary
OK Cancel Help

- 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.

Selocity Inlet	X
Zone Name	
inlet	
Momentum Thermal Radiation Species DPM Multipha	ise UDS
Velocity Specification Method Magnitude, Normal to Boundar	y •
Reference Frame Absolute	•
Velocity Magnitude (m/s) 0.015 constant	•
Turbulence	
Specification Method Intensity and Hydraulic Diameter	:r 🔻
Turbulent Intensity (%) 4.8	
Hydraulic Diameter (m) 1	
OK Cancel Help	

i. Enter 0.015  $\rm m/s$  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.

Pressure Outlet
Zone Name
outlet
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 👔 🔹 🔹
Backflow Direction Specification Method Normal to Boundary
Radial Equilibrium Pressure Distribution
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%) 4.8
Backflow Hydraulic Diameter (m) 1
OK Cancel Help

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  $\mathsf{OK}$  to close the  $\mathsf{Pressure}$   $\mathsf{Outlet}$  panel.
- 4. Close the Boundary Conditions panel.

#### **Step 5: Solution**

1. Set the solution controls.

Solve  $\longrightarrow$  Controls  $\longrightarrow$  Solution...

Solution Controls		×
Equations 📃	Under-Relaxation Factors	
Flow Turbulence	Pressure	0.3
	Density	1
	Body Forces	1
	Momentum	0.7
Pressure-Velocity Coupling	Discretization	
SIMPLE	Pressure	Standard 🗸
	Momentum	Second Order Upwind 🗸
	Turbulent Kinetic Energy	Second Order Upwind 🗸
	Turbulent Dissipation Rate	Second Order Upwind
	OK Default Cancel I	lelp

(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.

 $[\mathsf{Solve}] \longrightarrow [\mathsf{Initialize}] \longrightarrow \mathsf{Initialize}...$ 

Solution Initialization	X
Compute From Reference Frame inlet    Refative to Cel	l Zone
Initial Values	1
Gauge Pressure (pascal) g	-
X Velocity (m/s) 🔋	
Y Velocity (m/s) 🛛	-
Z Velocity (m/s) 0.015	•
Init Reset Apply Close Help	

- (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  $\longrightarrow$  Monitors  $\longrightarrow$  Residual...

Options	Storage			Plotting	
✓ Print ✓ Plot	lt	erations	1000 🛓	Wind	ow 🕴 🔺
	Normaliza	ation		Iterations	1000 🛨
	Γ	Normaliz	e 🗹 Scale	Axes	Curves
	Converge	nce Criter	ion		
	absolute		•		
continuity		V	1e-6		
x-velocity			1e-6		
y-velocity		$\checkmark$	1e-6		
z-velocity		$\overline{\mathbf{v}}$	1e-6		
k		$\overline{\mathbf{v}}$	1e-6		
epsilon			1e-6	— -	

- (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).



						E 🖬
Look in:	🗀 pipe		•	+ 🗈 č	* 📰 -	
My Recent Documents Desktop My Documents My Computer						
My Network Places	Case File Files of type:	pipe1.cas.gz Case Files			•	Cancel

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.

Solve  $\longrightarrow$  Iterate...

🖴 Iterate 🛛 🗙
Iteration
Number of Iterations 400
Reporting Interval 1
UDF Profile Update Interval 1
Iterate Apply Close Help

- (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  $\longrightarrow$  Iso-Surface...

Iso-Surface	X
Surface of Constant Grid X-Coordinate	From Surface
Min Max 0	wall
Iso-Values 0	From Zones I =
New Surface Name center-line Create Compute Mana	ge Close Help

(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.

Iso-Surface		X
Surface of Constant Grid X-Coordinate	From Surface	
Min (m) -0.5 Max (m) 0.5	outlet symmetry wall	
Iso-Values (m) Ø	fluid	
New Surface Name		
Create Compute	Manage Close Help	

- 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 postprocessing. 7. Create XY plots (Figure 4.5).

$Plot \longrightarrow XY Plot$
--------------------------------

Solution XY Plot		
Options	Plot Direction	Y Axis Function
✓ Node Values	Xg	Velocity 👻
Position on X Axis	Y	Z Velocity 🗸
Write to File	Z 1	X Axis Function
C Order Points		Direction Vector 🗸
File Data 📃 =		Surfaces
		center-line
		default-interior
		outlet
		symmetry
	Load File	wall
	Free Data	x=0
L		
Plot /	xes Cur	ves Close Help

- (a) Select center-line from the Surfaces selection list.
- (b) Set Plot Direction as  $X=0,Y=0,\,{\rm and}\,\,Z=1.$
- (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.

💶 Axes - Solution XY	Plot	×
Axis • X • Y Label	Number Format Type general Precision 3	Major Rules Color foreground - Weight 1
Options └ Log V Auto Range V Major Rules V Minor Rules	Range Minimum Ø Maximum Ø	Minor Rules Color dark gray Weight 1
	Apply Close Help	]

- i. Enable Major Rules and Minor Rules in the Options group box.
- ii. Click Apply.
- iii. Select  ${\sf Y}$  in the  ${\sf Axis}$  group box.
- iv. Enable Major Rules and Minor Rules in the  $\mathsf{Options}\xspace$  group box.
- v. Click  $\mathsf{Apply}\xspace$  and close the  $\mathsf{Axes}\text{-}\mathsf{Solution}\xspace$   $\mathsf{YPlot}\xspace$  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,

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,  $\tau$  = Shear stress on the wall

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

Surface  $\longrightarrow$  Iso-Surface...

💶 lso-Surface	×
Surface of Constant Grid Z-Coordinate Min Max	From Surface
lso-Values	x=0       From Zones       Image: state s
New Surface Name wall-outlet Create Compute Manag	je Close Help

- (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 lso-Surface panel.

10. Report surface integral.

$\begin{array}{c} Report \longrightarrow Surface Integrals \end{array}$
---

Surface Integrals	X	
Report Type Area-Weighted Average	Field Variable Wall Fluxes	
Surface Types = = axis clip-surf	Wall Shear Stress	
exhaust-fan fan 🕑	center-line default-interior inlet	
Match	outlet symmetry wall	
	xall-outlet x=0	
Area-Weighted Average (pascal) 0.0008491806		
Compute Write Close Help		

- (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 dropdown 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  $\longrightarrow$  Contours...

Contours		X
Options	Contours of	
🗹 Filled	Turbulence	-
✓ Node Values ☐ Global Range	Wall Yplus	•
🗹 Auto Range	Min	Max
Clip to Range	2.271785	5.910382
Draw Profiles	Surfaces	==
	outlet	<u>~</u>
	symmetry	
	wall-outlet	
Surface Name Pattern	×=0	<b>~</b>
	Surface Types	<u>=</u> =
Match	axis	<u>~</u>
	exhaust-fan	
	fan	<b>~</b>
Display Compute Close Help		

- (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  $\longrightarrow$  Views...

Views		
Views	Actions	Mirror Planes = =
back hottom	Default	symmetry
front	Auto Scale	
isometric left	Previous	
right	Save	Define Plane
ωμ	Delete	Periodic Repeats
Sa∨e Name	Read	Define
view-0	Write	
Apply Camera Close Help		



Figure 4.6: Contours of Wall Yplus on wall

- (a) Select symmetry from the Mirror Planes selection list.
- (b) Click  $\mathsf{Apply}\xspace$  and close the  $\mathsf{Views}\xspace$  panel.
- 3. Display the velocity contours on surface x=0 (Figure 4.7).



- (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  $\longrightarrow$  Iso-Surface...

Iso-Surface	X
Surface of Constant	From Surface 📃 🗐
Grid	▼ inlet
Z-Coordinate	<ul> <li>outlet</li> <li>symmetry</li> </ul>
Min Max	wall wall-outlet
0	x=0
lso-Values	From Zones 📃 📃
0.1	fluid
New Surface Name	
z=0.1	
Create Compute Ma	nage Close Help

(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 lso-Surface panel.
- 5. Display the velocity contours on surface z=0.1 (Figure 4.8). Display  $\longrightarrow$  Contours...

Contours	×
Options	Contours of
✓ Filled	Velocity 👻
✓ Node Values ✓ Global Range	Velocity Magnitude 👻
🗹 Auto Range	Min Max
Clip to Range	0
Draw Profiles     Draw Grid	Surfaces <u>=</u> =
Levels Setup	symmetry
20 🔶 1 🔶	wall-outlet
Surface Name Pattern	x=0
	Surface Types = =
Match	
	exhaust-fan
	fan 🕑
Display Co	mpute Close Help

- (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  $\mathsf{Draw}\xspace$  Grid to open the  $\mathsf{Grid}\xspace$  Display panel.

💶 Grid Display		×
Options	Edge Type	Surfaces 📃 =
<ul> <li>□ Nodes</li> <li>□ Edges</li> <li>□ Faces</li> <li>□ Partitions</li> <li>Shrink Factor F</li> </ul>	All     Feature     Outline	center-line default-interior inlet outlet symmetry wall well scalat
0	20	×=0
Surface Name	Pattern	Surface Types 🔳 =
	Match	axis clip-surf exhaust-fan fan
		Outline Interior
Display	Colors	Close Help

- 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.

💶 Profile Options	X	
Reference Value	Projection Dir.	
0	X	
Scale Factor 25	YØ	
,	Z 1	
Apply Close Help		

- 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  $\longrightarrow$  Views...

Views		
Views	Actions	Mirror Planes = =
back bottom	Default	symmetry
front	Auto Scale	
isometric left	Previous	
right ton	Save	Define Plane
No P	Delete	Periodic Repeats
Save Name	Read	Define
left	Write	
Apply Camera Close Help		

- (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.

 $\mathsf{Display} \longrightarrow \mathsf{Scene} \ \mathsf{Animation}...$ 

💶 Animate	
Playback Playback Mode Play Once Start Frame Increment End Frame 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Key Frame Keys       Image: Add Delete All
Write/Record Format Key Frames	Hardcopy Options
Write Read Clo	ose Help

(a) Click the Add button.

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

(b) Set the animation frame.

 $\mathsf{Display} \longrightarrow \mathsf{Scene...}$ 

Names <u>=</u> =	Geometry Attribute	s Scene Compositio
center-line profile-8-velocity-ma	Type profile	☐ Overlays ☐ Draw Frame
	Display	Frame Options
	Transform	
	Iso-Value	
Delete Geometry	Time Step	

- i. Select profile-8-velocity-magnitude from the Names selection list.
- ii. Click the Iso-Value... button to open the Iso-Value panel.

💶 Iso-Value		
Geometry Name profile-8-vel	locity-m	
Min (m) Ø	Value (m) 10	Ma× (m)
۲		
Apply Close Help		

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

Create line surfaces to visualize the velocity vectors at different axial locations.
 Surface → Line/Rake...

Line/Rake Surface		X				
Options Type Line Tool Reset	Number of Points ▼ 10					
End Points		_				
×0 (m) 0	×1 (m) g					
у0 (m) в	y1 (m) 0.5					
z0 (m) g	z1 (m) 🔋					
Select Points with Mouse						
New Surface Name						
line=0						
Create Manage Close Help						

- (a) Enter 0 for x0, y0, and z0.
- (b) Enter  $0,\,0.5,\,\mathrm{and}~0$  for  $x1,\,y1,\,\mathrm{and}~z1$  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  $0.5,\,1,\,1.5,\,2,\,2.5,\,{\rm and}$  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).

Display  $\longrightarrow$  Vectors...

Vectors			X			
Options	Vectors of					
Node Values	Velocity 👻					
Global Range	Color by					
✓ Auto Range	Velocity	•				
✓ Auto Scale ✓ Draw Grid	Velocity Magnitude					
	Min (m/s)	Max (m/s)				
Style arrow	0.002046185	0.01791342				
Scale 1	Surfaces					
Skip 👩 🔺	line=1 line=1 E		<u>^</u>			
• • •	line=2					
Vector Options	line=2.5					
Custom Vectors	outlet		~			
Surface Name Pattern	Surface Types		II.			
	axis		^			
Match	exhaust-fan					
	fan		~			
Display Compute Close Help						

- (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.

💶 Grid Display		×			
Options Nodes Edges Faces Partitions Shrink Factor Ø	Edge Type C All C Feature C Outline C eature Angle	Surfaces = = line=2.5 outlet symmetry wall wall-outlet x=0 z=0.1			
Surface Name	Pattern Match	Surface Types = = axis clip-surf exhaust-fan fan Outline Interior			
Display	Colors	Close Help			

- 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.

	1.79e-02									
	1.71e-02									
	1.63e-02									
	1.55e-02									
	1.47 e-02									
	1.39e-02									
	1.32e-02									
	1.24e-02									
	1.16e-02									
	1.08e-02	8	2							
	9.98e-03	E		E	E	E		Ē		
	9.19e-03	E			Ē	8	6	Ē.,		
	8.39e-03		-			-		-		
	7.60e-03									
	6.81e-03									
	6.01e-03									
	5.22e-03									
	4.43e-03									
	3.63e-03	Y								
	2.84e-03	<u> </u>	z							
	2.05e-03	~	-							
Veloc	ity Vectors (	Colored	By Vel	ocity N	lagniti	ude (r	n/s)			ELLIENT 6 2 /2d dn nhno oko)
										FLOEINI 0.5 (30, 0p, pbris, ske)

Figure 4.10: Velocity Vectors

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

Display  $\longrightarrow$  Views...

Views		×			
Views	Actions	Mirror Planes = =			
back bottom	Default	symmetry			
front	Auto Scale				
isometric left	Previous				
right top	Save	Define Plane			
view	Delete	Periodic Repeats			
Sa∨e Name	Read	Define			
view	Write				
Apply Camera Close Help					

- (a) Click Read... in the Actions group box to open the Select File dialog box.
  - i. Select the view-file and click  $\mathsf{OK}.$
- (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

View publication