

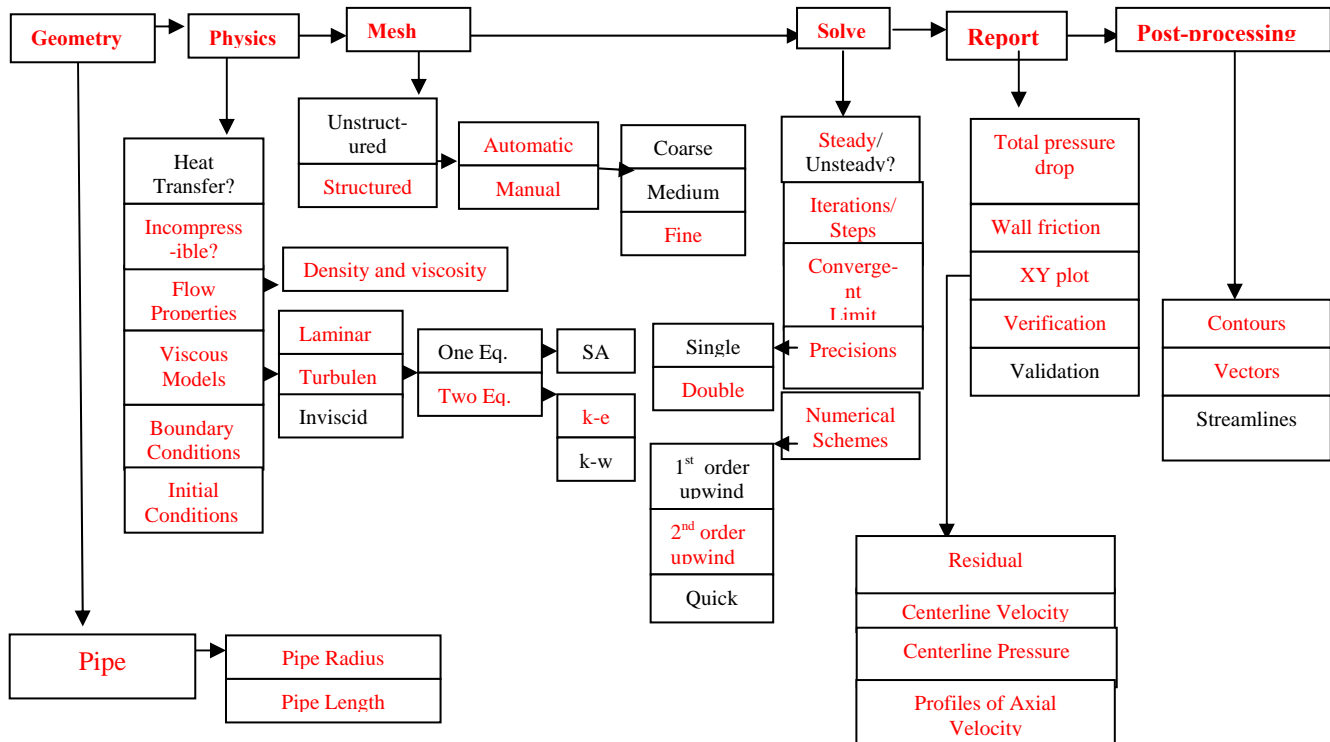
Verification of Laminar and Validation of Turbulent Pipe Flows

58:160 Intermediate Mechanics of Fluids CFD LAB 1

By Tao Xing and Fred Stern
 IIHR-Hydropscience & Engineering
 The University of Iowa
 C. Maxwell Stanley Hydraulics Laboratory
 Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD Lab 1 is to simulate steady **laminar** and **turbulent** pipe flow following the “CFD Process” by an interactive step-by-step approach. Students will have “hands-on” experiences using FlowLab to compute axial velocity profile, centerline velocity, centerline pressure, and friction factor. Students will conduct **verification studies for friction factor and axial velocity profile** of laminar pipe flows, including iterative error and grid uncertainties, and effect of refinement ratio on verification. Students will validate **turbulent pipe flow** simulation using EFD data, analyze the differences between laminar and turbulent flows, and present results in CFD Lab report.

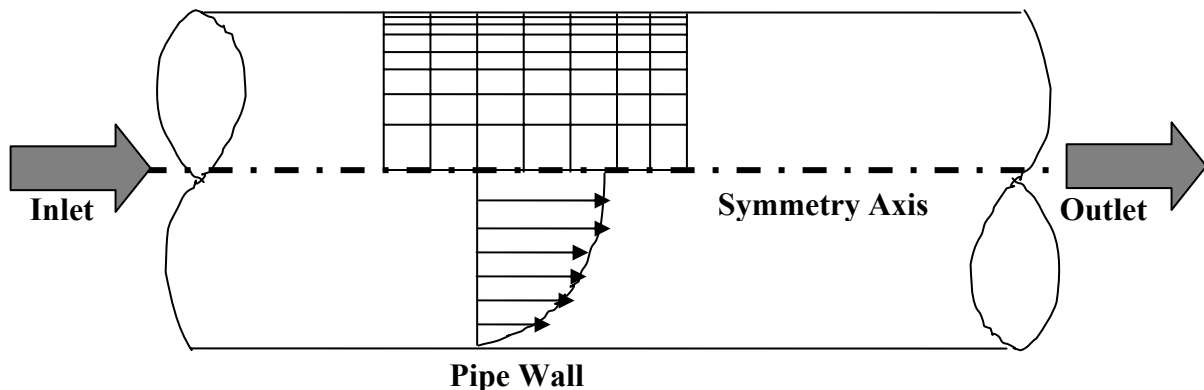


Flow Chart for ISTUE Teaching Module for Pipe Flow (red color illustrates the options you will use in CFD Lab 1)

2. Simulation Design

In CFD Lab 1, simulation will be conducted for **laminar and turbulent** pipe flows. Iterative error and grid uncertainties will be studied. Comparison between CFD and AFD for laminar flow, and CFD and EFD for turbulent flow will be performed.

The problem to be solved is that of laminar/turbulent flows through a circular pipe. Reynolds number is 655 for laminar flow and 111,569 for turbulent pipe flow, based on pipe diameter.

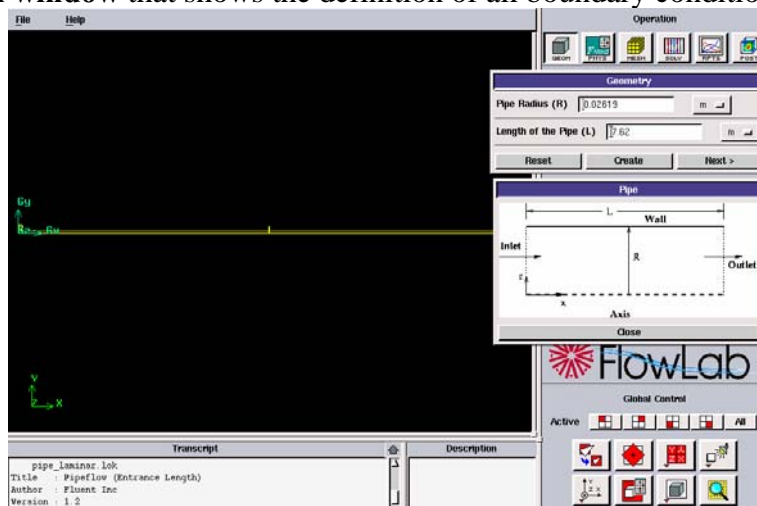


Since the flow is axisymmetric we only need to solve the flow in a single plane from the centerline to the pipe wall. **Boundary conditions** need to be specified include **inlet**, **outlet**, **wall**, and **axis**, as will be described details later. Uniform flow was specified at inlet, the flow will reach the fully developed regions after a certain distance downstream. No-slip boundary condition will be used on the wall and constant pressure for outlet. Symmetric boundary condition will be applied on the pipe axis.

All analytical data (AFD) for Laminar Pipe Flow and EFD data for turbulent pipe flow can be downloaded from the class website (http://css.engineering.uiowa.edu/~me_160).

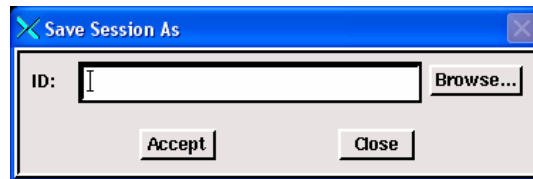
3. CFD Educational Interface

Right after you launch FlowLab 1.2.10, the following interface will be shown. The top right corner illustrates the CFD processes: **Geometry**→**Physics**→**Mesh**→**Solve**→**Reports**→**Post-processing**. There is also a **sketch window** that shows the definition of all boundary conditions and coordinates.



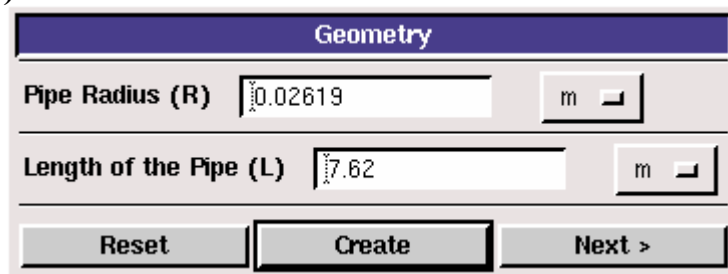
If you close the sketch window and want to see it again, you can click <<File>>→<<Problem Overview>>.

You MUST save your work regularly to avoid any possible lost of your data and jobs. <<File>>→<<Save As>>. Then use the <<Browse>> button to locate the directory where you want to save. It is recommended that you created your own folder in the FlowLab working directory: C:\Documents and Settings\Fluidslab\myflowlab\YOURNAME\.



4. CFD Process

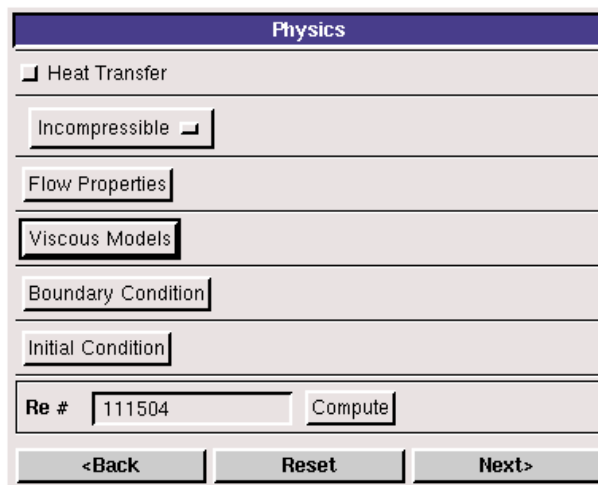
Step 1: (Geometry)



1. Radius of pipe (0.02619 m)
2. Length of pipe (7.62 m)

Click <<Create>>, after you see the pipe geometry created, click <<Next>>.

Step 2: (Physics)



The Reynolds number shown in the above figure is for “laminar” pipe flow case, for “turbulent” pipe flow, the Reynolds number will be different based on the inlet velocity you specified.

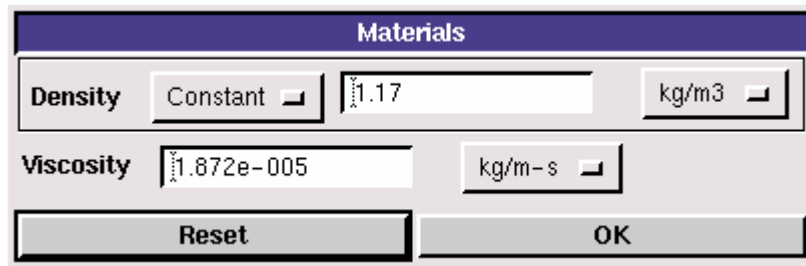
1. With or without Heat Transfer?

Thermal effects are not considered in CFD Lab 1, turn **OFF** <<Heat Transfer >> button.

2. Incompressible or compressible

Choose “Incompressible”, which is the default setup.

3. Flow Properties



use the values shown in the above figure. Input the values and click <<OK>>.

4. Viscous Model



In CFD Lab 1, both laminar model and turbulent model (k-e) will be used, follow exercise notes for specifications, and click <<OK>>. **Note: For each simulation, you can only choose one model, either laminar model OR turbulent k-e model.**

5. Boundary Conditions

At “Inlet”, FlowLab use zero gradient for pressure and fix the velocity to be **0.2 m/s** and **34.08 m/s**, for laminar and turbulent pipe flows, respectively. Use default values for “intensity” and “length scale”.

Inlet			
Variables	u (m/s)	v (m/s)	P (Pa)
Magnitude	0.2	0	-
Zero Gradient	N	N	Y
<input type="checkbox"/> Import inlet profile			
Profile	Browse...		
Reset		OK	

Laminar

Inlet					
Variables	u (m/s)	v (m/s)	P (Pa)	Intensity	Length scale
Magnitude	34.08	0	-	0.01	0.000294
Zero Gradient	N	N	Y	N	N
<input type="checkbox"/> Import inlet profile					
Profile	Browse...				
Turbulent Specification			Intensity and length scale		
Reset			OK		

Turbulent

At “**Axis**”, FlowLab use zero gradient for axial velocity and Pressure and specify the magnitude for radial velocity to be zero. Read all the values and click <<**OK**>>.

Axis			
Variables	u (m/s)	v (m/s)	P (Pa)
Magnitude	-	0	-
Zero Gradient	Y	N	Y
Reset		OK	

Laminar

Axis					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m ² /s ²)	e (m ² /s ³)
Magnitude	-	0	-	-	-
Zero Gradient	Y	N	Y	Y	Y
Reset			OK		

Turbulent

At “**Outlet**”, FlowLab uses magnitude for pressure and zero gradients for axial and radial velocities and turbulent quantities. For pressure magnitude, use “0” for laminar flow and “400 Pa” for turbulent flow, click <<**OK**>>.

Outlet					
Variables	u (m/s)	v (m/s)	P (Pa)		
Magnitude	-	-	0		
Zero Gradient	Y	Y	N		
Reset			OK		

Laminar

Outlet					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m ² /s ²)	e (m ² /s ³)
Magnitude	-	-	400	-	-
Zero Gradient	Y	Y	N	Y	Y
Reset			OK		

Turbulent

At “**Wall**”, no-slip boundary conditions are fixed for both axial and radial velocity, gradients for other variables are zero. For turbulent pipe flow, pipe roughness also needs to be specified, which is 2.5×10^{-5} m in this lab. Read the panel, input pipe roughness and click <<OK>>.

Wall			
Variables	u (m/s)	v (m/s)	P (Pa)
Magnitude	0	0	-
Zero Gradient	N	N	Y
Reset		OK	

Laminar

Wall					
Variables	u (m/s)	v (m/s)	P (Pa)	k (m ² /s ²)	e (m ² /s ³)
Magnitude	0	0	-	0	0
Zero Gradient	N	N	Y	N	N
Wall Roughness <input style="width: 100px;" type="text" value="2.5e-005"/> m					
Reset			OK		

Turbulent

6. Initial Conditions

Use the default setup for initial conditions.

Initial Condition			
Variables	P (Pa)	u (m/s)	v (m/s)
Magnitude	0	0.2	0
Reset		OK	

Laminar

Initial Condition					
Variables	P (Pa)	u (m/s)	v (m/s)	k (m ² /s ²)	e (m ² /s ³)
Magnitude	400	34.08	0	0.09	16
Reset			OK		

Turbulent

After specifying all the above parameters, click <<Compute>> button and FlowLab will automatically calculate the Reynolds number based on the inlet velocity and pipe diameter you input. **Note: For turbulent pipe flow, the outlet pressure is 400 Pa, you may specify pressure magnitude in “initial condition” to be 400 Pa to speed up convergence.** Click the <<Next>>, this takes you to the next step, “Mesh”.

Step 3: (Mesh)

Mesh	
Mesh option	
<input checked="" type="checkbox"/>	Structured
<input type="checkbox"/>	Unstructured
Mesh option	
<input type="checkbox"/>	Automatic
<input checked="" type="checkbox"/>	Manual
Select Edge	NR <input type="text" value="8"/>
	NX <input type="text" value="234"/>
<input type="button" value=" <Back"/>	<input type="button" value=" Reset"/>
<input type="button" value=" Create"/>	<input type="button" value=" Next>"/>

For CFD Lab 1, “**Structured**” meshes will be generated using either “**Automatic**” or “**Manual**” generations (see exercises at the end of this document for details). NR and NX is the number of intervals in axial and radial directions. For “**Automatic**” generation, just choose the mesh density: “coarse”, “medium” or “fine”, and click <<Create>>, FlowLab will automatically create the mesh you required and display the grid information NR, NX. For manually generating mesh, you should first choose the <<Manual>> button, and then the following panel will be shown:

Mesh

Mesh option

Structured
 Unstructured

Mesh option

Automatic
 Manual

Select Edge NX

NR 44
NX 451

<Back Reset Create Next>

NR

Distribution function Uniform

Number of Intervals 44

Reset Create Close

NX

Number of Intervals 451

Reset Create Close

Choose “**Uniform**” distribution for both axial and radial directions and use the appropriate numbers required in the exercise notes, click <<**Create**>> buttons in “NR”, “NX” and “Mesh” windows, then the mesh will be generated and displayed in the graphic window.

NOTE.: NR and NX are the number of **intervals** in each direction, as required by Flowlab. Therefore, remember to subtract 1 from the number of grid points when typing NX or NR (‘n’ points define ‘n-1’ intervals)

Step 4: (Solve)

Solve

Solver

Steady
 Unsteady

Iterations 10000

Convergence Limit 1e-006

Radial Profile x/D Position 1 10
Radial Profile x/D Position 2 20
Radial Profile x/D Position 3 40
Radial Profile x/D Position 4 60
Radial Profile x/D Position 5 100

Precision

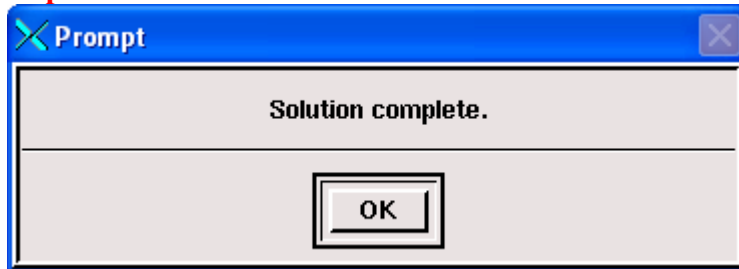
Single
 Double

Numerical schemes 2nd order

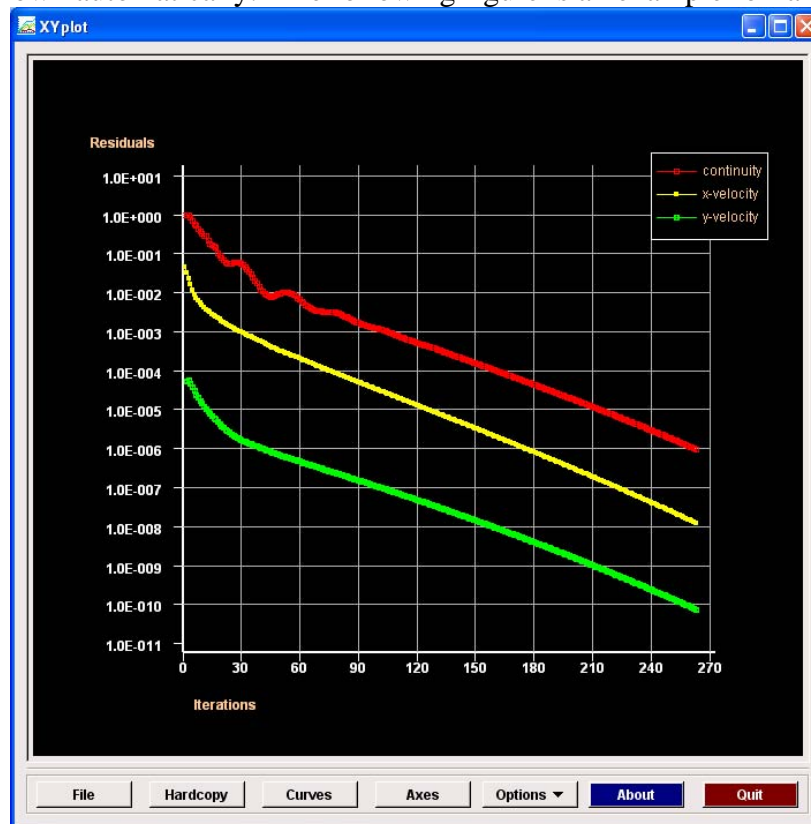
New
 Restart

< Back Reset Iterate Next >

The flow is steady, so turn ON the “**Steady**” option and turn OFF the “**Unsteady**” option. Specify the iteration number and convergence limit to be **10000** and **10⁻⁶**, respectively, EXCEPT in exercise note 1, where the effect of convergent limit (**10⁻⁵** and **10⁻⁶**) is studied. Use **10, 20, 40, 60, 100** for **radial profile x/D positions** and choose “**Double precision**” with “**2nd order scheme**”. Use “**New**” calculation for this Lab. Click <<**Iterate**>>, FlowLab will start the calculation, whenever you see the window, “**Solution Complete**”. Click <<**OK**>>.



The iterative history of residuals for continuity equation, x and y momentum equations, and turbulent quantities will be shown automatically. The following figure is an example for laminar pipe flow.

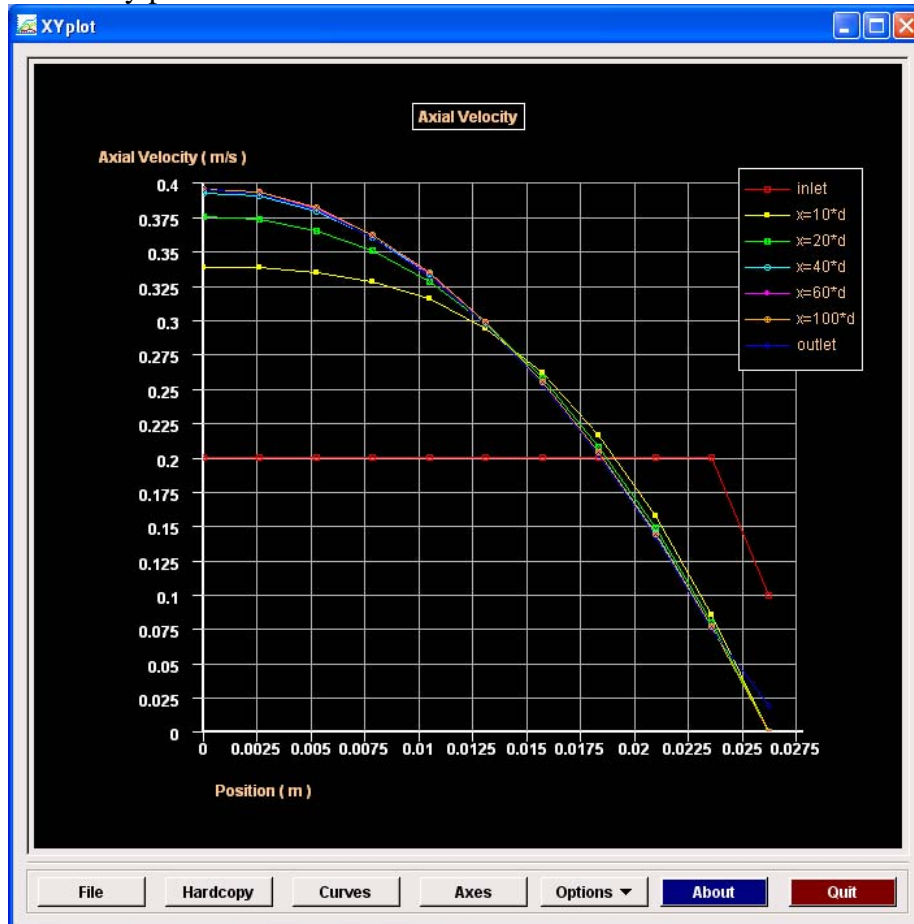


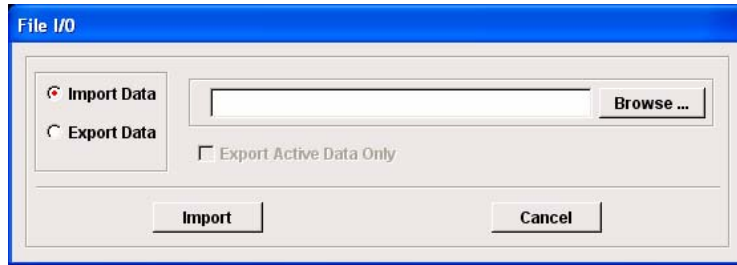
Step 5: (Reports)

For laminar pipe flow, record the “**Wall Friction Factor**” and compare its value with the AFD data. Import AFD data for axial velocity profile and compare with CFD predictions in the XY plot.

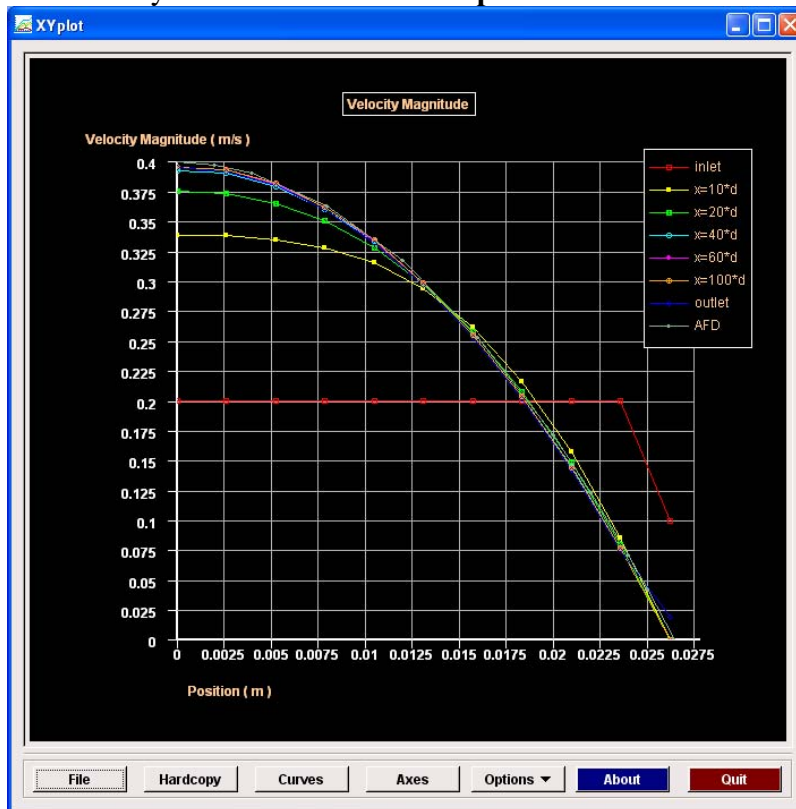
Reports	
Total Pressure Drop	0.358095 Pa
Wall Friction Force	0.000739372 N
Total Heat Flux	150.796 W
Temperature Rise	19.5519 °C
Verification and Validation	Open
XY Plots	
Profiles of Axial Velocity	Plot
<input type="button" value=" < Back"/> <input type="button" value=" Close"/>	

“XY Plots” provide options to plot residuals, axial velocity profile (axial velocity vs. radial locations), and centerline pressure/velocity distribution (pressure/velocity vs. x). The following figure shows the example for axial velocity profile at different streamwise locations:

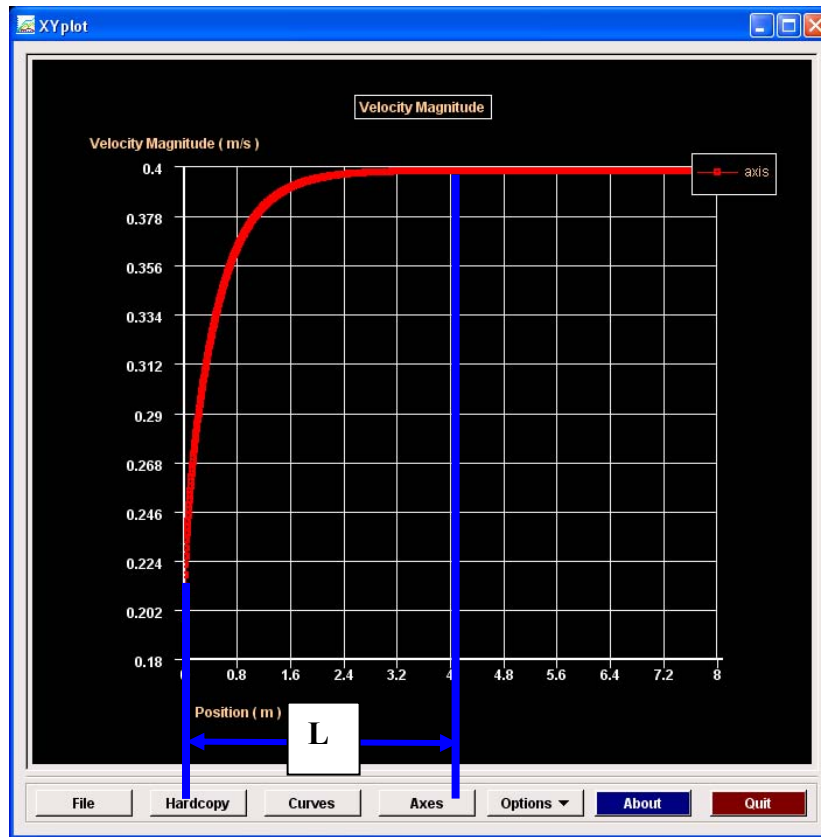




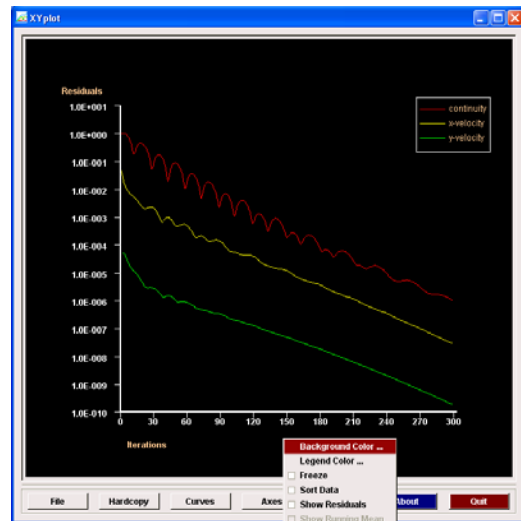
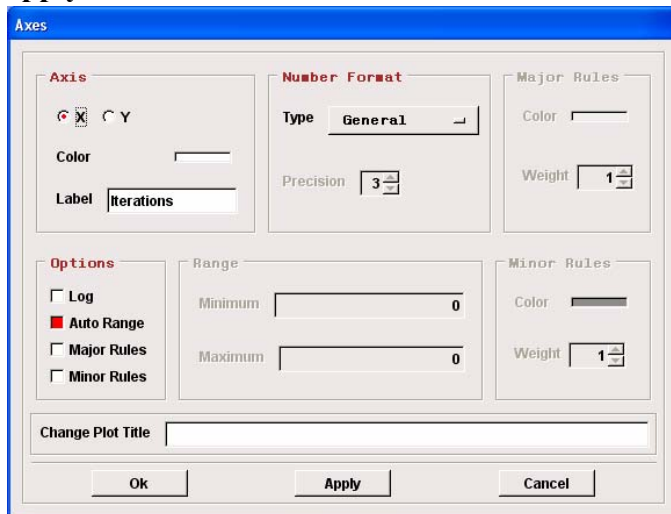
To import the AFD data and plot together with CFD results, just click <<**File**>> button and use the browse button to locate the file you want and click <<**Import**>>.



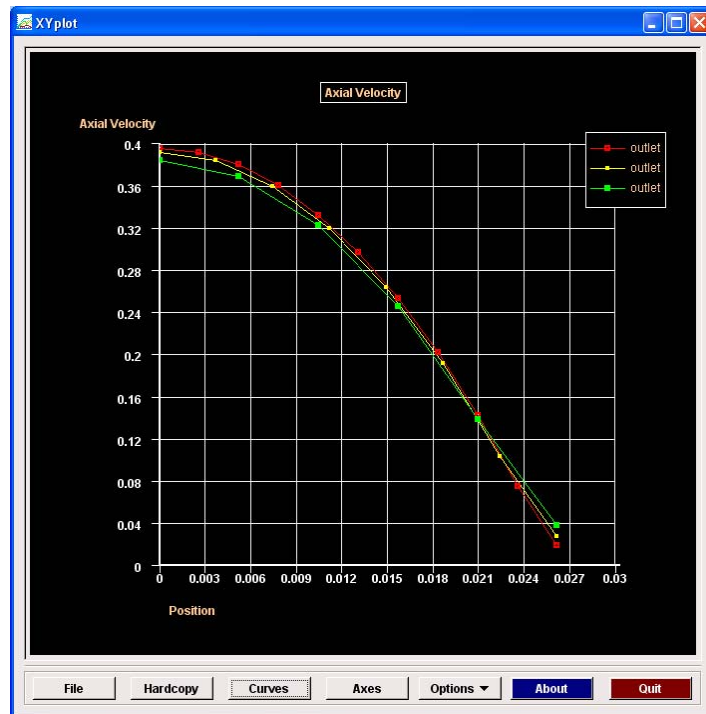
“Developing length” L of laminar/turbulent pipe flow can be determined through the use of the XY plot for centerline velocity, as shown below. The distance from the pipe inlet ($x=0.0$ to the streamwise location where the centerline velocity does not change any more) is the “developing length”.



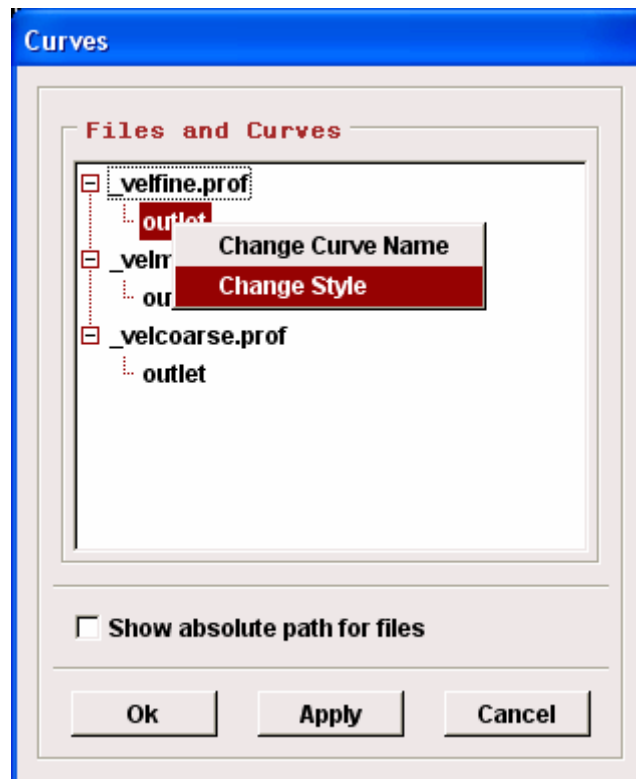
To turn OFF the X and Y grids in the figure, you can click <<Axes>> and the following window will appear. Turn OFF the <<Major Rules>> and <<Minor Rules>> for both X and Y axis, click <<Apply>> and <<OK>>.



In verification of axial velocity profile, three different color profiles are plot on one single XY plot, as shown below.



It is difficult to tell which curve is for which mesh since the legend is the same. You can click <<curves>> and get the following interface. Right click the “outlet” under the mesh you are interested in (e.g. fine mesh) and select <<change style>>, it will show the symbol used for the solution on that mesh.



To save the XY plot and pasted into WORD, you can click the XY plot, press and hold the <<Alt>> key and hit <<Print Screen Sys>>, then in WORD, click <<Paste>>.

If you want to save the XY plot as a *.JPG, *.TIFF, or *.BMP file, you can click <<**Hardcopy**>>, choose the figure format you need and use the <<**Browse**>> button to specify the folder you need (NOT the path shown in the figure). It is **strongly recommended** that you choose the “**Printer Friendly Colors**” before you <<**Save**>>.



You can also click <<**Curves**>> to choose which curve will be displayed, press and hold <<**Ctrl**>> and left button of the mouse for multiple choices.

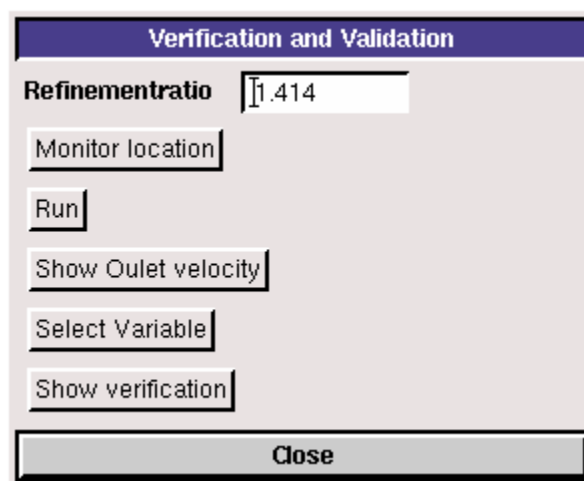
In this Lab, the AFD data for axial velocity profile of fully developed laminar pipe flow, EFD data for axial velocity profile of developed turbulent pipe flow, and the EFD data for pressure distribution along pipe length for turbulent pipe flow can be downloaded from the class website:

http://css.engineering.uiowa.edu/~me_160/

with the following file names: “axialvelocityAFD-laminar-pipe.xy;”
“axialvelocityEFD-turbulent-pipe.xy”, and “pressureEFD-turbulent-pipe.xy”.

Verification and Validation

In CFD Lab 1, you will conduct verification studies using the manually mesh generation options. (read exercises at the end of this document for details). The V&V panel is shown below. Whenever you manually create a mesh, that mesh will be automatically used as the **default “fine”** mesh for verification.



First input the refinement ratio you will use for the “coarse”, “medium”, and “fine” meshes.

“Monitor location” is used to specify the locations for line monitors (verification of axial velocity profile), click that will pop out the following window:

Line monitors		
Point no	X location (m)	Y location (m)
1	7	0
2	7	0.005
3	7	0.01
4	7	0.015
5	7	0.02
6	7	0.021
7	7	0.022
8	7	0.023
9	7	0.024
10	7	0.025

x locations are specified as 7 meters, which is inside the developed region and y location is the distance from pipe centerline to the pipe wall (type in the y locations shown above).

Click <<Check>>, FlowLab will check the locations you input and a warning message will be displayed if any point is out of the domain permitted.

Click <<Close>> and then Click <<Run>>, FlowLab will conduct simulations in the order of “fine mesh”→ “medium mesh”→ “coarse mesh”. The information on which mesh is solving now will be displayed in the left bottom window.

Whenever the V&V completed, you can select which variables you want to be shown for verification results. In this Lab, you can choose either “friction factor”, or “line monitors” for axial velocity profiles.

For friction factor, choose “friction factor” under button “select variables” click “show verification”, you will get two tables: verification results table and the errors table. You need save both using <<Alt+PrintScreen>> and paste into WORD document.

Verification					
Variable	Rg	Pg	Cg	Ug(%S)	Ugc(%S)
Fric factor	0.547303	1.73994	0.827642	1.65873	1.44613

Errors			
Mesh	Coarse	Medium	Fine
Fric factor	0.0939669	0.0957911	0.0967785
e(%)	N/A	1.86344	1.0203
ei(%)	0	0	0

Calculate ei

For line monitor for axial velocity profile, choose “line monitors” and click “show verification”.

Verification					
Variable	Rg	Pg	Cg	Ug(%S)	Ugc(%S)
Velocity	0.510295	1.94204	0.960229	0.0261921	0.00243784

Data for verification	
IEI(%)	3.79557
Import AFD data	

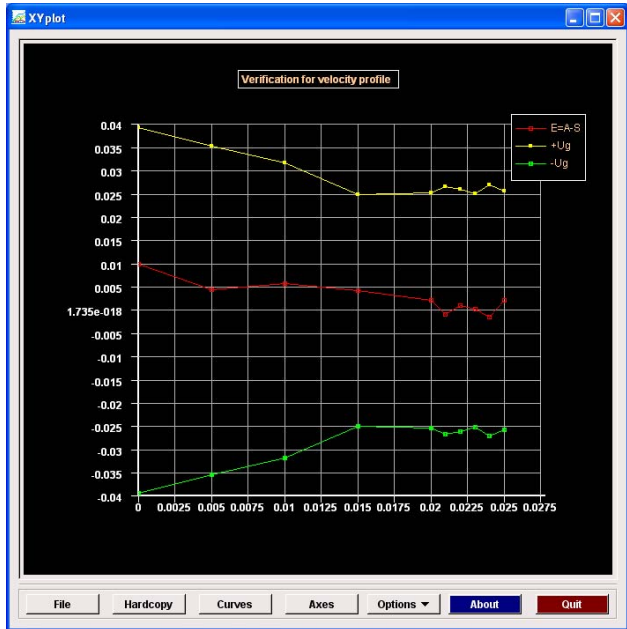
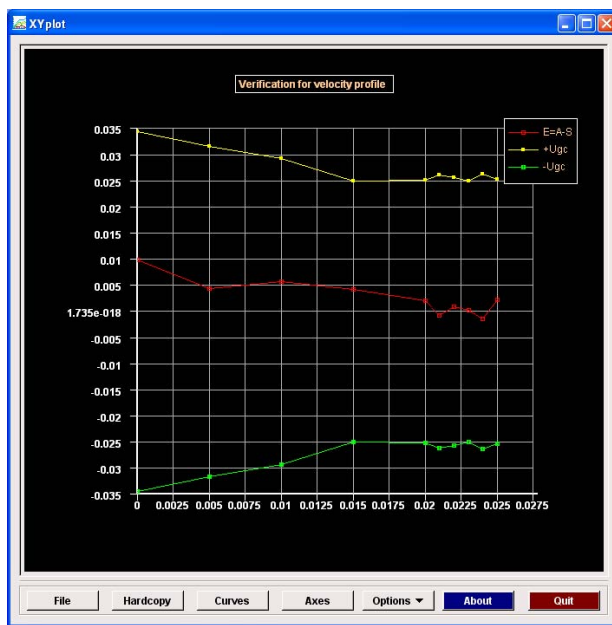
The first table is for profile-averaged UA results for axial velocity profile, the second is the global error $|E|$, ie. The difference between AFD and CFD results. (NOTE: the results shown above may be different from the results you should have).

Click <<Import AFD data>>, then input the 10 axial velocity values, as shown below:

AFD data for line monitors	
Point no	AFD data
1	0.4
2	0.385
3	0.342
4	0.269
5	0.167
6	0.143
7	0.118
8	0.092
9	0.064
10	0.036

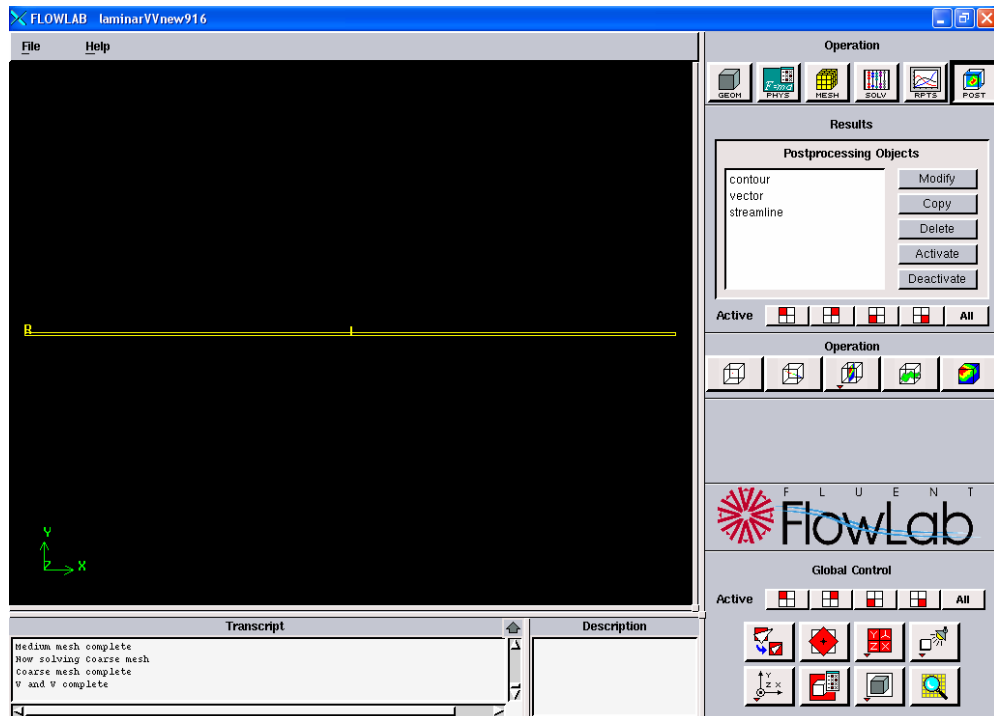
Close

Click <<Do Verification>>, the following figures will be shown. For the explanation of R_g , P_g , and U_g , etc., you can refer to the CFD lecture “Introduction to CFD”.

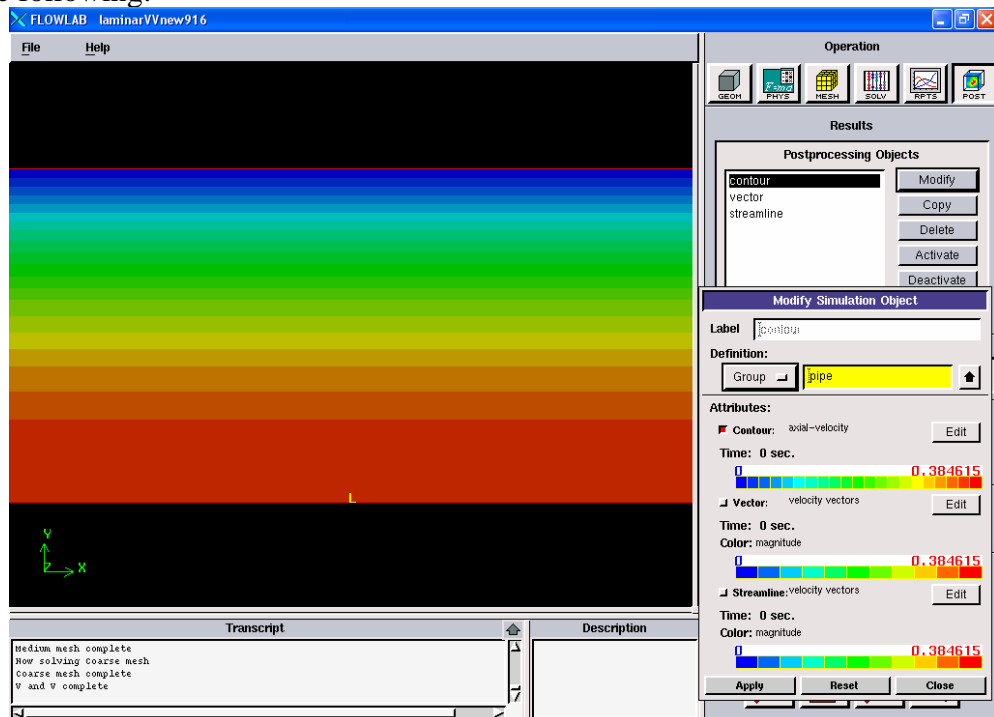


Step 6: (Post-processing)

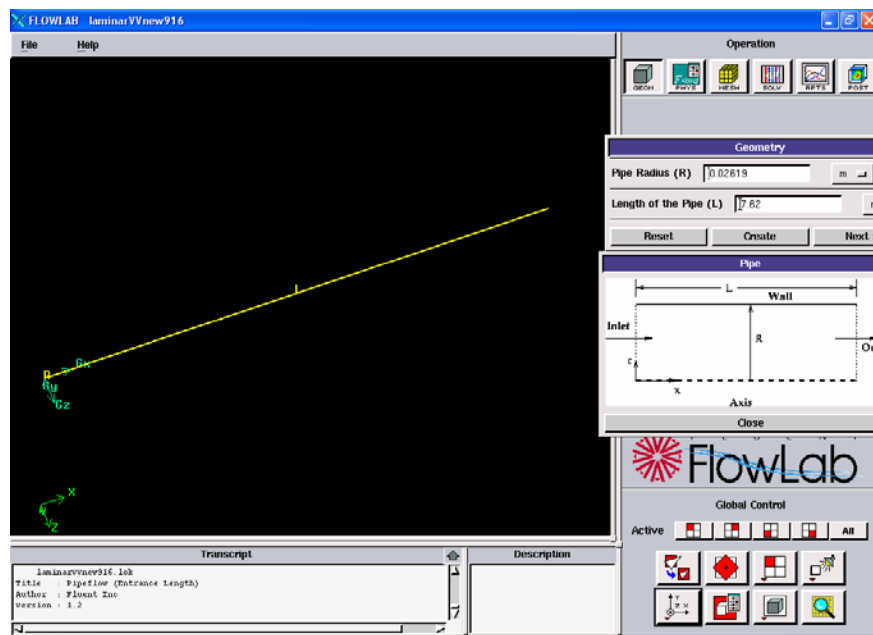
Use the “contour”, “vector” buttons to show pressure contours and velocity vectors.





To show pressure contour, choose “contour”, click <<Activate>>, contour for axial velocity will be shown. You can then click <<Modify>> button to select different variables if you need. For a better view, **press and hold** the right button of the mouse and drag towards you, you will have a figure similar to the following:

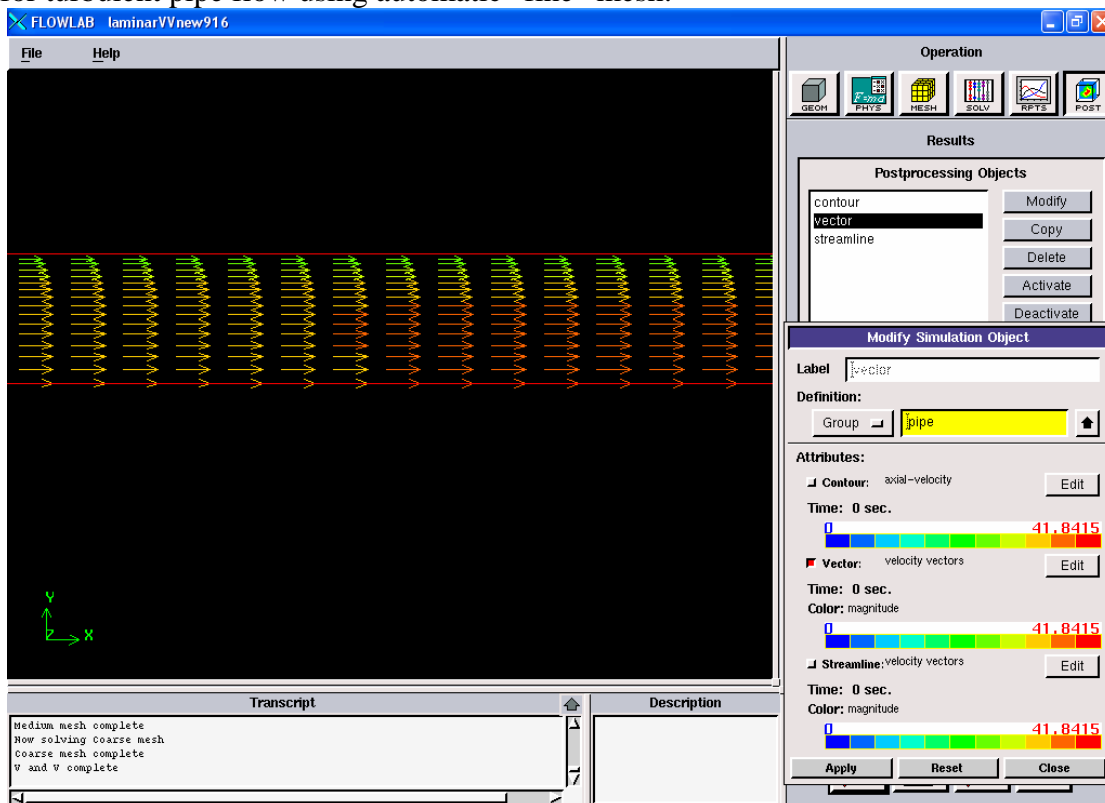


NOTE: If you happen to rotate the figure, such as the following:

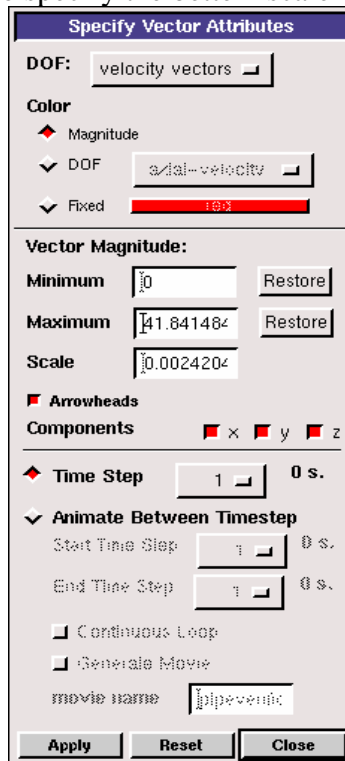


You can always use the button  to align the pipe with the coordinate and use  to view the full size of the pipe.

To view velocity vectors, click **<<Close>>** on the modify panel, and select **<<Vector>>** with **<<Activate>>**, if you don't want pressure contour you plot before to be shown, just choose **<<Contour>>** and then click **<<Deactivate>>**. The following figure shows an example of velocity vectors for turbulent pipe flow using automatic "fine" mesh:



Note that, you can press and hold the middle button of the mouse and move to left or right to choose the intersection while the flow begins to become fully developed. Just as shown above. You can also use the <<Edit>> button for vectors to specify the better “scale” value for velocity vector length.



5. Exercises

You need complete the following assignments and present results in your lab reports following the lab report instructions

Verification Study of Laminar Flow and Validation of Turbulent Flow

* 1-4 and 6 are for laminar flows, 5 is for turbulent flows

* You must save each case file for each exercise using “file”→ “save as”

1. Iterative error studies: using laminar flow conditions presented in the instructions, use 2nd order scheme with “laminar” model, “double precision” and create uniform meshes using “manual” function. Use two different convergent limits 10^{-5} and 10^{-6} and fill in the following table for the values on friction factors. Find the relative error between AFD friction factor (0.097747231) and friction factor computed by CFD, which is computed by:

$$\left| \frac{Factor_{CFD} - Factor_{AFD}}{Factor_{AFD}} \right| \times 100\%$$

To get the value of $Factor_{CFD}$, you need first generate XY plot for “Wall Friction Factor Distribution”, then use <<File>>→<<Export data>>→<<Browse>> to create the data file. Then use EXCEL to open the data file and pick the value close to the pipe exit or inside the fully developed region. Discuss the effect of convergent limit on results for these two meshes

Mesh No.	Grid points (X×R)	f (10 ⁻⁵)	F(10 ⁻⁶)
4	113×11	(%)	(%)
8	452×45	(%)	(%)

NOTE: (1). X and R should be NX+1 and NR+1. So, when you can create mesh manually, you need use NX, NR (112×10) for mesh 4 and (451×44) for mesh 8.

- **Figure need to be saved:** residuals history for mesh 8 for two convergent limits.
- **Data need to be saved:** the above table with values.
- **FlowLab case need to be saved:** mesh 8 with convergent limit 10⁻⁶

2. Verification study for friction factor of laminar pipe flow: Using mesh 4 as the “fine” mesh, and run verification with grid refinement ratio 1.414. Either take note or hardcopy the FlowLab verification results and fill in the following table. You should note that you ONLY need manually create the “fine” mesh (mesh 4), FlowLab will automatically create the corresponding “coarse” (2) and “medium” (3) meshes for you based on the refinement ratio you specify in the V&V panel. Using Mesh 8 as the “fine” mesh and repeat the above procedure using the same grid refinement ratio 1.414.

Meshes	Pg	Cg	Ug(%)	Ugc (%)
2,3,4				
6,7,8				

Which set of meshes is closer to the asymptotic range (i.e. Cg close to 1.0)? Which set has a lower grid uncertainty (Ug)? Which set is closer to the theoretical value of order of accuracy(2nd order). For the fine mesh 8, also compare its relative error of the friction factor (the one using convergent limit 10⁻⁶ in the table in exercise 1) with the grid uncertainty for 6,7,8, which is higher and what does that mean?

NOTE: (1). You must create mesh 4 and 8 manually.

(2). For exercise 2, you don’t need specify line monitor locations.

- **Figure need to be saved:** FlowLab “Errors” panel and “Verification” panel for both set of meshes.
- **Data need to be saved:** the above table with values

3. Effect of grid refinement ratio on verification results (friction factor): Still use mesh 4 and 8 as the “fine mesh”, but run verification with grid refinement ratio 2 for laminar pipe flow.

Meshes	Pg	Cg	Ug(%)	Ugc (%)
0,2,4				
4,6,8				

Compared to results in 2, which set of meshes is sensitive to grid refinement ratio? Why?

- **Figures need to be saved:** FlowLab “Errors” panel and “Verification” panel for both set of meshes.
- **Data need to be saved:** the above table with values

4. Verification study of axial velocity profile: Use mesh 4 as the “fine mesh”, use grid refinement ratio 1.414. First, input the line monitor locations, and then click “run”. After that, choose line monitors under “select variables”, and click “show verification results” button. Save the results. “Import AFD data” for the 10 points location you just

specified. And click “Do Verification” results. Save the figures and discuss if the simulation has been verified.

- **Figures need to be saved:** two figures showing U_g , U_{gc} with $|E|$, FlowLab “Verification” panel (with R_g , P_g , etc.), by clicking “show outlet velocity” in the V&V panel, you will see the axial velocity profiles for three meshes, import AFD data and save the figure. Discuss which mesh solution is closest to the AFD data, why?
- **Data need to be saved:** None.

5. Simulation of turbulent pipe flow

Change the following parameters:

- Turbulent model: $k-\varepsilon$
- Outlet pressure 400 Pa (*Change also the initial conditions to speed convergence up*)
- Inlet velocity: 34.08 m/s

Create a new mesh choosing “automatic” and “fine” options. Run simulation and compare with EFD data on axial velocity profile and pressure distribution along the pipe. Export the axial velocity profile data at $x=100D$, use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude at $x=100D$. Plot the normalized velocity profile in EXCEL and paste the figure into WORD.

- **Figures need to be saved:** Axial velocity profile with EFD data, normalized axial velocity profile at $x=100D$, centerline pressure distribution with EFD data, “centerline velocity distribution”, contour of axial velocity, velocity vectors showing the developing region and developed regions.
- **Data need to be saved:** Developing length and compared it with that using formula 6.6 in textbook.

6. Comparison between laminar and turbulent pipe flow

Compare the results of laminar pipe flow using mesh 8 in exercise 1 (convergent limit 10^{-6}) with results of turbulent pipe flow in exercise 5. Analyze the difference in normalized axial velocity profile and developing length for laminar and turbulent pipe flows.

NOTE: (1). Since you have finished laminar simulation using mesh 8 in exercise 1, you can just open the case file you saved and output the figures and data you need.

- **Figures need to be saved:** Axial velocity profile with AFD data, normalized axial velocity profile at $x=100D$, “centerline velocity distribution” for laminar flows.
- **Data need to be saved:** Developing length for laminar pipe flow and compared it with that using formula 6.5 in textbook.

7. Questions need to be answered in CFD Lab report

- 7.1. Answer all the questions in exercises 1 to 6
- 7.2. Analyze the difference between CFD/AFD and CFD/EFD and possible error sources.
- 7.3. Analyze the difference between FlowLab predictions and your own Calculations (using formula in CFD lecture) for order of accuracy and grid uncertainties.