Verification of Laminar and Validation of Turbulent Pipe Flows

58:160 Intermediate Mechanics of Fluids CFD LAB 1

By Timur Dogan, Michael Conger, Maysam Mousaviraad and Fred Stern IIHR-Hydroscience & 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 ANSYS 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 ANSYS

2. Simulation Design

In CFD Lab 1, simulation will be conducted for **laminar and turbulent** pipe flows. Reynolds number is 655 for laminar flow and 111,569 for turbulent pipe flow, based on pipe diameter. The schematic of the problem and the parameters for the simulation are shown below.

Parameter	Unit	Value
Radius of Pipe	m	0.02619
Diameter of Pipe	m	0.05238
Length of the Pipe	m	7.62





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. Uniform grids will be used for the laminar flow whereas non-uniform grid will be used for the turbulent flow.

Grid	Grid Type	# of Divisions				
Ollu	Ond Type	Х	R			
8		453	45			
7		320	32			
6		227	23			
4	Uniform	113	11			
3		80	8			
2		57	6			
0		28	3			
Т	Non-uniform	564	15			

Tab	le 2 ·	- Grids
-----	--------	---------

Experimental, analytical and simulations will be compared. Additionally, detailed verification and validation study will be conducted. All the studies are detailed in the Table 3. In this manual, detailed instructions are given for the turbulent flow simulation and laminar flow simulations using non-uniform grid and uniform grid 8 respectively. Figures and data that needs to be saved are shown in Table 4.

Table 3 - Simulation matrix

Study	Grid	Model
V&V of friction factor and axial velocity profile	2,3,4	
V&V of friction factor	6,7,8	
V&V of friction factor	0,2,4	Laminar
V&V of friction factor	4,6,8	
Axial velocity, centerline velocity	8	
Axial velocity, centerline pressure, centerline velocity	Т	Turbulent

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).

		Convergence		
Grid	Flow	Limit	Figure	Data
Т	Turbulent	1.00E-06	*	
8	Laminar	1.00E-06	Residuals	**
8	Laminar	1.00E-05	Residuals	Wall Shear Stress
7	Laminar	1.00E-06		Wall Shear Stress
6	Laminar	1.00E-06		Wall Shear Stress
4	Laminar	1.00E-06		Wall Shear Stress
4	Laminar	1.00E-05		Wall Shear Stress
3	Laminar	1.00E-06		Wall Shear Stress
2	Laminar	1.00E-06		Wall Shear Stress
0	Laminar	1.00E-06		Wall Shear Stress
* Axia	al velocity p	orofile with EFD data	, normalize	d axial velocity profile at x=100D,
center	line pressur	e distribution with El	FD data, "c	enterline velocity distribution", contour
of axia	al velocity,	velocity vectors show	wing the de	veloping region and developed regions.
**Wal	l Shear Stres	s, velocity profile (10p	ts), centerlin	e velocity distribution
		•• •		•

Table 4 - Figures and data sets needed to be saved

3. Open ANSYS Workbench Template

3.1. Start > All Programs > ANSYS 14.5 > Workbench 14.5



3.2. Toolbox > Component Systems. Drag and drop Geometry, Mesh and Fluent components to Project Schematic as per below.



3.3. Right click on the upper corner of the components on the drop down arrow then select rename. Change the names as per below.



3.4. Create connections between component as per below. You can select components part and drop it onto the target component part to create connections.

Project S	chemati	ic																			
-		Α			Ŧ		в				•		С				•	•	D		
1	🧭 G	eometry			1	🧼 M	lesh				1	•	Fluent				1		Fluent		
2	🧭 G	eometry	? .		2	🧼 м	lesh	?		-	2		Setup	7	4	/	~ 2		Setup	7	4
		pipe		\setminus		unifor	m grik	8 6			3		Solution	7	4		з		Solution	7	4
				\								lan	ninar (1e-	6)				lar	minar (1e-	5)	
									\												
					•		Е				Ŧ		F								
					1	🧼 M	lesh				1	•	Fluent								
					2	🧼 м	lesh	?		-	2		Setup	7	4						
					ľ	non-uni	form	grid			3		Solution	7	4						
												turb	oulent (1e	-6)							

3.5. File > Save As. Save the workbench file to H drive. The H drive is shared between the computers in engineering labs.

Λ Un	saved Project - Workbench
File	View Tools Units Extensions Help
	New Ctrl+N
6	Open Ctrl+O
	Save Ctrl+S
R	Save As
	Save to Repository
	Open from Repository
0	Send Changes to Repository
0	Get Changes from Repository
	Manage Repository Project
4	Launch EKM Web Client
	Import
	Archive
•	Restore Archive
	Scripting •
	Export Report
	1 H:\CFD\CFD Lab 1\CFD Lab 1 Student.wbpj
	2 H:\CFD\CFD Lab 1\CFD Lab 1 Student V2.wbpj
	3 H:\CFD\CFD Lab 1\CFD Lab 1 Solutions V3.wbpj
	4 H:\CFD\CFD Lab 3\CFD Lab 3 V2.wbpj
L.	Exit Ctrl+Q

4. Geometry Creation

4.1. Right click **Geometry** and select **New Geometry**. (Since all the geometries are linked together, only one geometry creation is required)



4.2. Select Meter for unit and click OK.

ANSYS Workbench	_	x
Select desired length unit:		
	~	
• Meter	O Foot	
C Centimeter	C Inch	
C Millimeter		
C Micrometer		
Always use project unit	t	
Always use selected un	nit	
Enable large model supp	port	
ОК		

4.3. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.

💿 A: Fluid Flow (FLUENT) - DesignModeler	and the second second	
File Create Concept Tools View Help		
🛛 🔄 🔚 🖾 🗍 💬 Undo 📿 Redo	Select: 🆎 💱 💽 💽 💽 😂 🗍 S	💠 Q. Q. Q. Q. 🐺 🗼 📦 🗖
XYPlane 🔻 🗚 None 😽	📁 📋 🧚 Generate 🛛 🖤 Share Topology 🕴 💽 Extruc	de 🚓 Revolve 🐁 Sweep 🚯 Skin/Loft 📗 Thin/Surface 🔷
Tree Outline	New Sketch ^{hics}	
A: Fluid Flow (FLUENT) XVPlane XVPlane XVPlane V2Plane V2Plane		

4.4. Right click **XYPlane** and select **Look at**.



4.5. Select **Sketching** > **Rectangle**. Create a rectangle geometry as per below.



4.6. Select **Dimensions** > **General**. Click on top edge then click anywhere. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.



4.7. Click on H1 under Details View and change it to 7.62 *m*. Click on V2 and change it to 0.02619 *m*.

Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 2	
H1	7.62 m
🗌 V2	0.02619 m
Edges: 4	
Line	Ln15
Line	Ln16
Line	Ln17
Line	Ln18
	Details of Sketch1 Sketch Sketch Visibility Show Constraints? Dimensions: 2 Dimensions: 2 V2 Edges: 4 Line Line Line

4.8. Concept > Surface From Sketches and select the sketch and hit Apply.



4.9. Click Generate. This will create a surface.



4.10. **File** > **Save Project**. Save project and close window.



4.11. If you see the lightning sign next to **Geometry** in the workbench then right click on **Geometry** and click **Update** as shown below. If you don't see the check mark after you update then you may have made a mistake when you created the geometry.

Project Schematic		
T Geor	A	✓ B 1 ₩ Mesh
2 😡 Ge	DM	Edit Geometry
1 1		Replace Geometry
6	b	Duplicate
		Transfer Data From New
		Transfer Data To New
	7	Update
1	9	Refresh
		Reset 9
8	jþ	Rename
		Properties
		Quick Help
		Add Note

5. Mesh Generation

5.1. Right click **Mesh** and select **Edit**.

▼ A	-	в	С	▼ D
1 🥩 Geometry	1 🍻 M	lesh 1	💶 Fluent	1 🛄 Fluent
2 🕼 Geometry 🗸	• 2	Edit	tup ?	2 🎉 Setup 😨
	↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	Duplicate Transfer Data To New Update Clear Generated Data Refresh Reset Rename Properties Quick Help Add Note	r (1e-6)	laminar (1e-5)

5.2. Right click on **Mesh** then select **Insert** > **Mapped Face Meshing**.

] File Edit View Units Tools Help ∳Ge	nerate Mesh 🏙 🔝 🎯 🕶 🗊 W
刘 🛱 Show Vertices 📲 Wireframe 🛛 🖽 Edge Colo	oring \checkmark $/_1 \checkmark$ $/_2 \checkmark$ $/_3 \checkmark$ $/_3 \checkmark$
] Mesh 🛭 🗦 Update 🕴 🍘 Mesh 🔻 🔍 Mesh Contro	I ▼ 🔤 📶 Metric Graph 🛛 🖓 Options
Outline Project Outline Note: Surface Body Outline Outline	# Mesh 3/11/2013 11:16 AN
Insert ✓ <td>Image: Sizing Image: Sizing <t< td=""></t<></td>	Image: Sizing Image: Sizing <t< td=""></t<>
alp Kename	

5.3. Select your geometry and click **Apply**.

De	Details of "Mapped Face Meshing" - Mapped Face Meshing							
Ξ	Scope							
	Scoping Method	Geometry Selection	n					
	Geometry	Apply	Cancel					
	Definition							
	Suppressed No							
	Method Quadrilaterals							
	Radial Number of Divisions Default							
	Constrain Boundary	No						



5.4. Click on the **Edge Button**. This will allow you to select edges of your geometry.

1	🗭 B :	B : uniform grid 8 - Meshing [ANSYS Academic Teaching Advanced]																				
	File	Edit	View	Units	Тоо	ls He	lp 📗	••	🥠	Gener	rate N	/lesh	t	RPS	A	🧭 🗸	D	Work	sheet	i,		
I		8, Y, Z N	k •	₽\$.		6		I 🥰	j •	S	+‡+	€	Ð	0	Q	Q	Q,	ISO	12	j 🖻	🗞 🗖	•
l	ן ד	Show \	Vertices	; ; ; ; ; ; ;	Nirefr	Edge	[]] E	dge C	olorin	ng 👻	6	- /	íт.,	/₂ ◄	13-	k	- 7	/ -	⊷ 1	Thicker	Annotations	: □à Show
l	Mes	י 🥠	Update	6	Mesh	-	, Mesi	n Cont	trol 🔻		H Me	tric G	iraph									

5.5. Right click on **Mesh** then select **Insert** > **Sizing**.

Outline		4
Filter: Name	- 🔯 🔬	Ŧ
Project	B2) ometry Surface Body ordinate Systems	
	Insert 🔸	🎲 Method
	誟 Update	🔍 Sizing
	🧚 Generate Mesh	🦌 Contact Sizing
	Preview Show Show Show Streate Pinch Controls	Mapped Face Meshing Match Control
	Clear Generated Data alp Rename	A Inflation
	Start Recording	
1		

5.6. Hold Ctrl and select the top and bottom edge then click **Apply**. Specify details of sizing as per below.

Laminar

De	Details of "Edge Sizing" - Sizing 4				
Ξ	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	453			
	Behavior	Hard			
	Bias Type	No Bias			

Turbulent

De	etails of "Edge Sizing" - Sizing 4		
Ξ	Scope		
	Scoping Method	Geometry Selection	
	Geometry	2 Edges	
	Definition		
	Suppressed	No	
	Туре	Number of Divisions	
	Number of Divisions	564	
	Behavior	Hard	
	Bias Type	No Bias	

5.7. Repeat step 5. Select the left and right edge and click **Apply** for uniform grid flow and change sizing parameters as per below. Change the sizing parameters separately for non-uniform grid as per below. Make sure to select edges individually when changing sizing parameters for non-uniform grid.

Uniform Grid 8

De	etails of "Edge Sizing 2" - Sizing 📍				
Ξ	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	45			
	Behavior	Hard 🔹			
	Bias Type	No Bias			

Non-uniform Grid Left Edge

De	Details of "Edge Sizing 2" - Sizing 4					
Ξ	Scope					
	Scoping Method	Geometry Selection				
	Geometry	1 Edge				
	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	15				
	Behavior	Hard				
	Bias Type					
	Bias Option	Bias Factor				
	Bias Factor	3.1117				

Non-uniform Grid Right Edge

D	Details of "Edge Sizing 3" - Sizing 4						
	Scope	Scope					
	Scoping Method	Geometry Selection					
	Geometry	1 Edge					
Definition							
	Suppressed	No					
	Туре	Number of Divisions					
	Number of Divisions	15					
	Behavior	Hard					
	Bias Type						
	Bias Option	Bias Factor					
	Bias Factor	3.1117					

5.8. Click on Generate Mesh button and select Mesh under Outline.



Uniform Grid 8	Non-uniform Grid

5.9. Change the edge names by right clicking and selecting **Create Named Selection**. Name left, right, bottom and top edges as inlet, outlet, axis and wall respectively. Your outline should look same as the figure below.

	Insert	
	Go To	
P	Hide Body	
0	Suppress Body	
	Isometric View	
ISO	Set	
ISO	Restore Default	
۹	Zoom To Fit	
	Cursor Mode	
	View	
ØQ.	Look At	
*	Create Coordinate System	
÷Ġ	Create Named Selection	
Ŷ	Select All	
	Undata Consister from Source	





5.10. **File** > **Save Project**. Save the project and close the window. Update **Mesh** on Workbench if necessary.



6. Solve

6.1. Right click **Setup** and select **Edit**.

Project Schematic			
Project Schematic	I B I		Edit Edit Register Startup Scheme File Import Fluent Case Duplicate Transfer Data From New Transfer Data To New Update Refresh Reset Rename Properties
		<u>j</u> d	Rename Properties Quick Help Add Note

6.2. Under options check **Double Precision** and click **OK**.

E Fluent Launcher (Setting Edit Only)	
ANSYS	Fluent Launcher
Dimension	Options Double Precision Processing Options Serial Parallel
Show More Options	ancel <u>H</u> elp 🔻

6.3. **Solution Setup** > **General** > **Check**. (Note: If you get and error message you may have made a mistake while creating you mesh)



6.4. Solution Setup > General > Solver. Choose options shown below.

Meshing	General	
Mesh Generation	Mesh	
Solution Setup General Models Materials Phases	Scale Display Solver	Check Report Quality
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Solution	Time Steady	2D Space
Solution Methods Solution Controls Monitors	Transient	Axisymmetric Axisymmetric Swirl
Solution Initialization Calculation Activities Run Calculation	Gravity	Units
Results	Help	
Graphics and Animations Plots Reports		

6.5. Solution Setup > Models > Edit. Select parameters as per below.

Laminar flow

Meshing	Models	Viscous Model
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	Models Multiphase - Off Energy - Off Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off	Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) K-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) OK Cancel Help
Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Edit	

Turbulent flow

Meshing	Models	Viscous Model
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Models Multiphase - Off Energy - Off Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off Edit	Model Model Constants Inviscid Laminar Spalart-Allmaras (1 eqn) (1-Epsilon) k-epsilon (2 eqn) I.44 Transition K+4-omega (3 eqn) 1.44 Transition SST (4 eqn) Scale-Adaptive Simulation (SAS) k-epsilon Model Standard Standard RNG Realizable User-Defined Functions Non-Equilibrium Wall Functions Turbulent Viscosity Scalable Wall Functions Treatment User-Defined Wall Treatment Inone User-Defined Wall Functions TRE Prandtl Number Inone Inone User-Defined Wall Functions Inone Model Inone User-Defined Wall Functions Inone Image: Cancel Help

6.6. Solution Setup > Materials > air > Create/Edit... Change the Density and Viscosity as per below and click Change/Create. Close the dialog box when finished.

Meshing	Materials	Create/Edit Materials	· · · · · · · · · · · · · · · · · · ·	X
Mesh Generation	Materials	Name	Material Type	Order Materials by
General	air Solid	Chemical Formula	fluid	 Name Chemical Formula
Materials Phases	aluminum		Fluent Fluid Materials	Fluent Database
Cell Zone Conditions			Mixture	User-Defined Database
Mesh Interfaces		Properties	THE	
Reference Values		Density (kg/m3) constant	Edit	
Solution Solution Methods Solution Controls Monitors		Viscosity (kg/m-s) constant	Edt	
Calculation Activities Run Calculation		1.872e-05		
Results				
Plots				
Reports	Create/Edit Delete			
	Help	1 Change/Crea	ate Delete Close Help	

6.7. Cell Zone Conditions > Zone > surface_body. Change type to fluid and click OK. Select Material Name as air and click OK.

	Meshing	Cell Zone Conditions	
	Mesh Generation	Zone	-
	Solution Setup	surface_body	
	General		
	Materials		
	Phases		
	Boundary Conditions		
	Mesh Interfaces		
	Reference Values		
	Solution		
	Solution Methods Solution Controls		
	Monitors		
	Calculation Activities		
	Run Calculation	Phase Type ID	
	Results Graphics and Animations	mixture V Solid V	
	Plots	Edit	
	Reports	Parameters Operating Conditions	
		Display Mesh	
		Porous Formulation Superficial Velocity 	
		O Physical Velocity	
		Help	
-			~
Fluid			~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
Zone Name			
surface_body			
Material Name			
ar	-	Lait	
Frame Motion So	ource Terms		
Porque Zope	xed values		
Defenses Freeze Luc		let u vele a le se le	In the large of the
Reference Frame Me	ash Motion Porous Zone	Embedded LES Reaction Source Terms Fixe	d Values Multiphase
This page is not applic	able under current setting	s.	
		OK Cancel Help	

6.8. Solution Setup > Boundary Conditions > inlet > Edit... Change parameters as per below and click OK. (Note: Change inlet velocity to 0.2 m/s for laminar flow)

Laminar flow

Meshing	Boundary Conditions	Velocity Inlet
Mesh Generation	Zone	Zone Name
Solution Setup	axis	inlet
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	rilet interior-surface_body outlet wall	Momentum Thermal Radiation Species DPM Multiphase UDS Velocity Specification Method Magnitude, Normal to Boundary Reference Frame Absolute Velocity Magnitude (m/s) 0.2 constant v Supersonic/Initial Gauge Pressure (pascal) 0 constant v
Solution Controls Monitors Solution Initialization Calculation Activities		OK Cancel Help
Run Calculation Results Graphics and Animations Plots Reports	Phase Type ID mixture Velocity-inlet 7 Edit Copy Profiles Parameters Operating Conditions	1e-12 0 100 2

Turbulent flow

Meshing	Boundary Cor	nditions		elocity Inlet			X
Mesh Generation	Zone			Name			
Solution Setup	axis			t			
General Models Materials Phases	inlet interior-surface_t outlet wall	oody		mentum Thermal Radia	tion Species DPM	Multiphase UDS	
Cell Zone Conditions Boundary Conditions				Refe	rence Frame Absolute	,	·
Mesh Interfaces Dynamic Mesh				Velocity Mag	nitude (m/s) 34.08		onstant 🔻
Reference Values Solution				personic/Initial Gauge Press	sure (pascal) 0		onstant 👻
Solution Methods				bulence			
Solution Controls Monitors				Specificati	ion Method K and Epsil	on	
Solution Initialization Calculation Activities		_		Turbulent Kinetic Energy	gy (m2/s2) 0.01	con	stant 👻
Run Calculation Results	mixture	velocity-inlet •	ID 7	Turbulent Dissipation Ra	ite (m2/s3) 0.000294	con	stant 👻
Graphics and Animations Plots	Edit	Copy Profiles]				
Reports	Parameters	Operating Conditions]		OK Car	Help	
	Display Mesh	Periodic Conditions					

6.9. Solution Setup > Boundary Conditions > outlet > Edit. Change parameters as per below and click OK. (Note: Outlet pressure is 0 Pa for laminar flow)

Laminar flow

Meshing	Boundary Conditions
Mesh Generation	Zone Zone Name
Solution Setup	axis outlet
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Inlet Interior surface_body wall Womentum Thermal Radiation Species DPM Multiphase UDS Gauge Pressure (pascal) 0 constant • Backflow Direction Specification Method Normal to Boundary Average Pressure Specification Target Mass Flow Rate
Solution	
Solution Methods Solution Controls Monitors	OK Cancel Help
Solution Initialization Calculation Activities Run Calculation	Phase Type ID 1e-08
Results Graphics and Animations Plots	mixture v pressure-outlet v 8 Edit Copy Profiles
Reports	Parameters Operating Conditions 1e-12 Display Mesh Periodic Conditions 0 100 200 300 400 500 60

Turbulent flow

Meshing	Boundary Conditions	Pressure Outlet
Mesh Generation	Zone	Zone Name
Solution Setup	axis	outlet
General Models Materials Phases Cell Zone Conditions	inlet interior-surface_body outlet wall	Momentum Thermal Radiation Species DPM Multiphase UDS Gauge Pressure (pascal)
Boundary Conditions Mesh Interfaces		Backflow Direction Specification Method Normal to Boundary
Dynamic Mesh		Average Pressure Specification
Reference values		Target Mass Flow Rate
Solution Solution Methods Solution Controls		Turbulence Specification Method K and Epsilon
Monitors Solution Initialization		Backflow Turbulent Kinetic Energy (m2/s2) 1 constant
Calculation Activities Run Calculation	Phase Type ID	Backflow Turbulent Dissipation Rate (m2/s3)
Results	mixture v pressure-outlet v 8	
Graphics and Animations Plots Reports	Edit Copy Profiles	OK Cancel Help
	Display Mesh Periodic Conditions	

6.10. Solution Setup > Boundary Conditions > wall > Edit... Change parameters as per below and click OK.

Turbulent flow

Meshing	Boundary Conditions	Wall	23
Mesh Generation	Zone	Zone Name	
Solution Setup	axis	wall	
General Models Materials Phases	interior-surface_body outlet wall	Adjacent Cell Zone surface_body	
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values		Momentum Thermal Radiation Species DPM Multiphase UDS Wall Motion Motion Image: Stationary Wall Image: Relative to Adjacent Cell Zone	Wall Film
Solution		Chara Carativar	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Phase Type ID	Specified Shear Specularity Coefficient Marangoni Stress	
Results	mixture 👻 wall 🔻 5	Wall Roughness	
Graphics and Animations Plots Reports	Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions	Roughness Height (m) 2.5e-5 constant Roughness Constant 0.5 constant	• •
	Help	OK Cancel Help	

6.11. Solution Setup > Boundary Conditions > Operating Condition. Change parameters as per below and click OK.

Meshing	Boundary Conditions	Operating Conditions
Mesh Generation	Zone	Pressure Gravity
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Peference Values	axis inlet interior-surface_body outlet wall	Operating Pressure (pascal) 97225.9 Reference Pressure Location X (m) 0 Y (m) 0 P Z (m) 0
Solution		P
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities		OK Cancel Help
Run Calculation	Phase Type ID	
Results	mixture val 5	
Graphics and Animations Plots Reports	Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions	

6.12. Solution Setup > Reference Values. Change parameters as per below.

Meshing	Reference Values	
Mesh Generation	Compute from	
Solution Setup		•
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Reference Values Area (m2) Density (kg/m3) Enthalpy (j/kg)	0.002154869
Dynamic Mesh Reference Values Solution	Length (m)	0.05238
Solution Methods Solution Controls	Pressure (pascal)	0
Monitors Solution Initialization	Temperature (k) Velocity (m/s)	298.16
Run Calculation Results	Viscosity (kg/m-s)	1.872e-05
Graphics and Animations Plots Reports	Ratio of Specific Heats	1.4
	Reference Zone	
		•
	Help	

Laminar flow

Turbulent flow

Meshing	Reference Values							
Mesh Generation	Compute from							
Solution Setup	· · · · · · · · · · · · · · · · · · ·							
General	Reference Values							
Models Materials	Area (m2)	2.154869e-3						
Phases Cell Zone Conditions	Density (kg/m3)	1.17						
Mesh Interfaces	Enthalpy (j/kg)	0						
Reference Values	Length (m)	0.05238						
Solution	Pressure (pascal)							
Solution Methods		0						
Monitors Solution Initialization	Temperature (k)	298.16						
Calculation Activities Run Calculation	Velocity (m/s)	34.08						
Results	Viscosity (kg/m-s)	1.872e-5						
Graphics and Animations Plots Reports	Ratio of Specific Heats	1.4						
Reports	Reference Zone							
		•						

6.13. **Solution** > **Solution** Methods. Change parameters as per below.

Turbulent flow

	4
Meshing	Solution Methods
Mesh Generation	Pressure-Velocity Coupling
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions	Scheme SIMPLE Spatial Discretization Gradient Green-Gauss Cell Based
Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities	Pressure Second Order Momentum Second Order Upwind Turbulent Kinetic Energy Second Order Upwind Turbulent Dissipation Rate Second Order Upwind
Run Calculation Results Graphics and Animations Plots Reports	Transient Formulation Non-Iterative Time Advancement Frozen Flux Formulation Pseudo Transient High Order Term Relaxation Options Default

Laminar flow

Meshing	Solution Methods					
Mesh Generation	Pressure-Velocity Coupling					
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Scheme SIMPLE Spatial Discretization Gradient Green-Gauss Cell Based Pressure Second Order Momentum	*				
Solution <u>Solution Methods</u> Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Second Order Upwind	Ŧ				
Results Graphics and Animations Plots Reports	Transient Formulation Non-Iterative Time Advancement Frozen Flux Formulation Pseudo Transient High Order Term Relaxation Options Default					

6.14. **Solution** > **Monitors** > **Residuals** > **Edit**. Change convergence criterion to 1e-6 for all five equations as per below and click **OK**. (Note: for iterative error study you will need to use 1e-5)

Meshing	Monitors	Residual Monitors				23
Mesh Generation Solution Setup General Models Materials Phases	Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off	Options Øptions Øptint to Console Øplot Window 1 Total Curves Axes	Equations Residual continuity x-velocity	Monitor Check Conve	rgence Absolute Criteria 1e-06 1e-06	^
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Create Edit Delete Surface Monitors	Iterations to Plot	Residual Values	Iterations	1e-06 Convergence	Triterion
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Bun Calulation	Create Edit Delete Volume Monitors	Iterations to Store	Scale Compute Loca Renormalize	I Scale	Help	

Laminar flow

Tubulent flow

Meshing	Monitors	Residual Monitors					X
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Create Edit Delete Surface Monitors	Options Vrint to Console Vindow 1 Iterations to Plot 1000 Curves Axes	Equations Residual continuity x-velocity y-velocity k	Monitor Ch	eck Convergence V V V	Absolute Criteria 1e-6 1e-6 1e-6 1e-6	
Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Create) Edit) Delete Volume Monitors	Iterations to Store	Residual Values Normalize Scale Compute Loco t Renormalize	al Scale	terations 5 (V)	Convergence Cr absolute	•

6.15. Solution > Solution Initialization. Change parameters as per below and click Initialize. (Note: use 0 Pa and 0.2 m/s for laminar flow for pressure and velocity respectively)

Turbulent flow

Initialization Methods Hybrid Initialization Standard Initialization Compute from Reference Frame Relative to Cell Zone Absolute	•
Initial Values Gauge Pressure (pascal) 400 Axial Velocity (m/s) 34.08 Radial Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 0.09 Turbulent Dissipation Rate (m2/s3) 16 Initialize Reset Patch Perset DPM Sources Perset Statistice	E
	Initialization Methods Hybrid Initialization Standard Initialization Compute from Reference Frame Relative to Cell Zone Absolute Initial Values Gauge Pressure (pascal) 400 Axial Velocity (m/s) 34.08 Radial Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 0.09 Turbulent Dissipation Rate (m2/s3) 16 Initialize Reset Patch Reset DPM Sources Reset Statistics

Laminar flow

Meshing	Solution Initialization
Mesh Generation	Initialization Methods
General Models	 Hybrid Initialization Standard Initialization
Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Compute from Reference Frame Relative to Cell Zone Absolute
Solution	Initial Values
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Gauge Pressure (pascal) 0 Axial Velocity (m/s) 0.2 Radial Velocity (m/s) 0
	Initialize Reset Patch Reset DPM Sources Reset Statistics

6.16. **Solution** > **Run calculation**. Change number of iterations to 1000 and click **Calculate**.

7. Post Processing

Displaying Mesh

Display > Mesh



Select all the surface you want to display, lines and points you create can be displayed here as well.

Plotting and Printing Residuals

Display > **Residuals**. Click on **Plot** button then click on **OK**.



Residual Monitors					X	
Options	Equations					
Print to Console	Residual	Monitor C	heck Convergence	Absolute Criteria	*	
V Plot	continuity	V		1e-06		
Window	x-velocity			1e-06		
Iterations to Plot	y-velocity			1e-06		
1000	k			1e-06	-	
	Residual Values			Convergence Cr	iterion	
Iterations to Store	Normalize		Iterations	absolute	•	
	Scale					
	Compute Loca	l Scale				
OK Plot Renormalize Cancel Help						



File > Save Picture. Using option as per below save the residuals.

	💶 J:	Fluent Fl	uent [axi	, dp, pb	ns, ske			
	File	Mesh	Define	Solve	Adap			
		Refresh Save Pro	Input Dat oject	ta				
		Read			۱.			
		Write			•			
		Import			•			
		Export			•			
		Solution	n Files					
		EM Map	oping		•			
		FSI Map	ping		•			
		Save Pie	cture					
		Data Fil	e Quantit	ies				
		Close Fl	uent					
Sava Dictura								X
					_	1		
Format	Coloring		File T	ype	Res	olution		_
JPEG	Gray S	Scale		Kaster Vector		Widt	ⁿ 960	
 PPM PostScript 	Monoc	hrome				Heigh	t 720	
TIFF PNG	Options							
VRML Window Dump	✓ Lands ✓ White	cape Or Backgr	rientatio ound	n ['indow I mport -	oump Con window 9	nmand ‰w	
Save	Appl	у	Preview		lose	Help		

-

Creating Points and Lines

Surface > **Point**. Change x and y values as per below click **Create**. Repeat this for other lines shown in the table below.

Point Surface	X
Options	Coordinates
Point Tool	x0 (m) 7
Reset	y0 (m) 0
	z0 (m) 0
s	elect Point with Mouse
New Surface Name	e
point-1	
Create	anage Close Help

Point Name	x0	y0
point-1	7	0
point-2	7	0.005
point-3	7	0.010
point-4	7	0.015
point-5	7	0.020
point-6	7	0.021
point-7	7	0.022
point-8	7	0.023
point-9	7	0.024
point-10	7	0.025

Surface > **Line**/**Rake**. Change x and y values as per below click **Create**. Repeat this for other lines shown in the table below.

I Line/Rake Surface				
Options Type Number of Points Line Tool Reset				
End Point	ts			
x0 (m)	0.5238 ×1 (m) 0.5238			
y0 (m)	0 y1 (m) 0.02619			
z0 (m)	0 z1 (m) 0			
	Select Points with Mouse			
New Surface Name				
x=10d				
Create Manage Close Help				

Surface Name	X0	Y0	X1	Y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.238	0	5.238	0.02619

Plotting Results

Results > **Plots** > **XY Plot** > **Setup**. Select **inlet**, **outlet**, and the lines you created and change setting as per below then click **Plot**.

Problem Setup	Plots	Solution XY Plot
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Plots YY Plot Histogram File Profiles: Profile Data - Unavailable Interpolated Data FFT	Options Plot Direction Y Axis Function ♥ Node Values X 0 ♥ Position on X Axis Y 1 Position on Y Axis Y 1 Order Points Z 0 File Data Image: Section of the section of
Graphics and Animations Plots Reports	Set Up	Plot Axes Curves Close Help

XY Plot > **Setup** > **Curves**. For Curve # 0 select the **Pattern** as per below and click **Apply**. Repeat this for all the curves 0 through 7.

Curves - S	olution XY Plot	X		
Curve #	Line Style Pattern Color Orange Weight 1	Marker Style Symbol (*) Color orange Size 0.3		
Apply Close Help				

Download the data for the Simulation from the class website (http://www.engineering.uiowa.edu/~me_160/).

XY Plot > **Setup** > **Load File**. Select axialvelocityEFD-turbulent-pipe.xy and click **OK**.



Results > **Plots** > **XY Plot** > **Setup**. Change Y function to **Pressure...** and select **axis** then click **Plot**.

Solution XY Plot		×
Solution XY Plot Options Options Option X Axis Position on X Axis Virite to File Order Points File Data	Plot Direction X 1 Y 0 Z 0	Y Axis Function Pressure Static Pressure X Axis Function Direction Vector Surfaces axis inlet interior-surface_body outlet wall x=100d
Plot	Load File Free Data	x=20d

Load experimental data for the centerline pressure.



Select axis and change Plot Direction as per below. Then plot the figure.

Solution XY Plot	1.00000	W L AND THE R. LEWIS CO. L.	X
Options Vode Values Position on X Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0 Load File Free Data Axes	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vector Surfaces axis inlet interior-surface_body outlet wall x = 100d x = 20d New Surface ▼ Curves Close Help	
axis			



Exporting Data

Select **Plots** > **XY Plot**. Then change parameter as per below and click **Write**. This will export the shear stress along the wall of the pipe. You will need this data to compute the shear stress coefficient at the developed region.

Problem Setup	Plots	Solution XY Plot		[23
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Plots XY Plot Histogram File Profiles: Profile Data - Unavailable Interpolated Data FFT	Options Vode Values Position on X Axis Position on Y Axis Vitte to File Order Points	Plot Direction X 1 Y 0 Z 0	Y Axis Function Wall Fluxes Wall Shear Stress X Axis Function Direction Vector	•
Dynamic Mesh Reference Values		File Data 🔳 🗏		Surfaces	
Solution				inlet interior-surface body	Â.
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities			Load File	outlet wall x=10d x=20d	T T
Run Calculation			Free Data	New Surface 🔻	
Graphics and Animations Plots Reports	Set Up	Write	Axes	Curves Close Help	

Plotting Vectors and Contours

Results > **Graphics and Animations** > **Vectors** > **Set Up...** Change the vector parameters as per below and click **Display**.

Problem Setup	Graphics and Animations	Vectors	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Graphics Mesh Contours Vectors Pathlines Partide Tracks	Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh	Vectors of Velocity Color by Velocity Axial Velocity
Dynamic Mesh Reference Values Solution	Set Up	Style	Min (m/s) Max (m/s) 22.47792 41.61796
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Animations Sweep Surface Scene Animation Solution Animation Playback	Image: State of the state o	Surfaces
Graphics and Animations Plots Reports	Set Up Options Scene Views	Surface Name Pattern Match	New Surface Surface Types axis clip-surf
	Lights Colormap Annotate	Display	Compute Close Help

4.16e+01		TANSYS
4.07e+01		Noncommercial use only
3.97e+01		
3.87e+01		
3.78e+01		
3.68e+01		
3.59e+01		
3.49e+01		
3.40e+01		
3.30e+01		
3.20e+01		
3.11e+01		
3.01e+01		
2.92e+01		
2.82e+01		
2.73e+01		
2.63e+01		
2.53e+01		
2.44e+01		
2.34e+01		
2.25e+01		
Velocity Vectors Colored By A	∞ial Velocity (m/s)	Mar 11, 2013
		ANSYS FLUENT 13.0 (axi, dp, pbns, ske)

Results > **Graphics and Animations** > **Contours** > **Set Up...** Change the vector parameters as per below and click **Display**.

	Orachian and the impatience	Contours	×
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Graphics and Animations Graphics Mesh Contours Vectors Partiles Particle Tracks Set Up	Contours of Contours of Contours of Velocity Velocity Velocity Axial Velocity Min (m/s) Max (m/s) Draw Profiles Draw Mesh Surfaces axis inlet	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Anima Plots Reports	Animations Sweep Surface Scene Animation Solution Animation Playback Set Up	Levels Setup interior-surface_body 20 1 outlet wal x=100d Surface Name Pattern New Surface v Match Surface Types axis dip-surf exhaust-fan fan	
	Options Scene Views Lights Colormap Annotate	Display Compute Close Help	
	4 16e-01 3 95e-01 3 76e-01 3 76e-01 3 76e-01 2 39e-01 2 39e-01 2 312e-01 2 312e-01 3 312e-	CANSYS	
	Contours of Avial Velocity (m/s)	Mar 11, 2013 ANSYS FLUENT 13.0 (axi, dp. pbns, ske)	

Close window and save workbench file.

V&V Instructions

V&V Instructions for Velocity Profile

Right click **Solution** > Select **Edit...**



Create reference points

Results > **Plots** > **XY Plot** > **Set Up...**

💶 B:Laminar (1e-6) Fluent	: [axi, dp, pbns, lam] [ANSYS Academic Teaching Advan
File Mesh Define Sol	ve Adapt Surface Display Report Parallel Vie
i 📖 i 📂 🕶 🖬 🕶 🔟	❷∥S ፼€ € ↗∥® 洙 III ▾ □ ▾
Meshing	Plots
Mesh Generation	Plots
Solution Setup	XY Plot
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Histogram File Profiles: Profile Data - Unavailable Interpolated Data FFT
Solution Solution Methods Solution Controls Monitors	
Solution Initialization Calculation Activities	
Run Calculation	Set Up
Results	
Graphics and Animations Plots Reports	Help

Change parameters as per below and click Write... Make sure to select points 1 through 10.

Solution XY Plot			X
Options	Plot Direction	Y Axis Function	
Node Values	X 0	Velocity	
Position on X Axis Position on Y Axis	Y	Axial Velocity	-
Write to File		X Axis Function	
Order Points	2 0	Direction Vector	•
File Data 🔳 🗏]	Surfaces	
		point-3 point-4	^
		point-5 point-6	
		point-7	
		point-9	
	Load File	Iwall	*
	Free Data	New Surface 🔻	
Write	Axes	Curves Close Help	

Name file according to which grid solution you are using.



Open file using Wordpad, copy points to input into V&V Excel file.



Paste value into V&V Excel file according to its y position and its grid number. Use the Keep Text Only paste function by right clicking in the cell and selecting it from the paste options.

Pgest	2							
rg	1.41421	36						
		(Grids 2,3,4					
y (m)	Sg1		Sg2	Sg3	А	Eg2 [%]	Eg3 [%]	Eg4 [%]
0	0.39671	L3			0.400000	****	100.000000	100.000000
0.005					0.385000	*****	100.000000	100.000000
0.01					0.342000	*****	100.000000	100.000000
0.015			Paste Opt	tions:	0	*****	100.000000	100.000000
0.02			عر		0	*****	100.000000	100.000000
0.021			LÁ		0	*****	100.000000	100.000000
0.022			K T .		0.118000	*****	100.000000	100.000000
0.023			Keep Text	Only (1)	0.092000	*****	100.000000	100.000000
0.024					0.064000	*****	100.000000	100.000000
0.025					0.036000	*****	100.000000	100.000000

Repeat this process for the remaining y location points and then the two remaining grid solutions. All yellow cells should be filled.

V&V Instructions for Friction

Right click **Solution** > Select **Edit...**



Results > **Plots** > **XY Plot** > **Set Up...**

E B:Laminar (1e-6) Fluent [axi, dp, pbns, lam] [ANSYS Academic Teaching Advan								
File Mesh Define So	lve Adapt Surface Display Report Parallel Vie							
i 📖 i 📂 🕶 🖬 🕶 🎯	Ø ∰ € € €							
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Plots Plots XY Plot Histogram File Profile Data - Unavailable Interpolated Data FFT							
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Set Up Help							

Change parameters as per below and click Write...

Solution XY Plot	X
Options Plot Direction Y Axis Fill Image: Position on X Axis X 1 Wall Fill Image: Position on X Axis Y 0 X Axis Fill Image: Position on X Axis Y 0 X Axis Fill Image: Position on X Axis Position on X Axis Y 0 X Axis Fill Image: Position on X Axis Image: Position on X Axis Y 0 X Axis Fill Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Position on X Axis Image: Posi	unction xes ▼ ear Stress ▼ unction n Vector ▼ s ■ = -surface_body fface ▼ Close Help

|--|

Select File				×
Look in:	\mu Fluent	•	G 🤌 📂 🛄 -	
(Pa)	Name	*	Date modified	Туре
Recent Places		No items match your s	search.	
Desktop				
Libraries				
Computer				
Network	•	III		4
INCLWOIK	XY File	Grid7 Wall Shear Stress	- (ОК
	Files of type:	XY Files	•	Cancel

Open file with Notepad and copy wall shear stress at the x location of 7.62m.

Wall Sherar Stress Grid 7 (1e	-06)							
File Edit Format View Help (title "Wall Shear Stress") (labels "Position" "Wall Shear Stres:								
((xy/key/labe) Wall 7.62 0.000570371 7.59619 0.0005714(7.57237 0.0005717 7.54856 0.0005713 7.52475 0.0005713 7.52475 0.0005714	Undo Cut Copy							
7.45331 0.0005714 7.4295 0.0005714 7.40569 0.0005714 7.38188 0.0005714 7.35806 0.0005714 7.33425 0.0005714	Paste Delete Select All							
·	Right to left Reading order Show Unicode control characters Insert Unicode control character							
	Open IME Reconversion							

Paste the value into corresponding cell in the V&V template.



Make sure when pasting you select **Keep Text Only** and you select the proper cell corresponding to the grid number.

	Wall		
Grid	Shear	c	
	Stress		
0		0	
2		0	
3		0	
4		0	
6		0	
7		0	
8		0	

Repeat this process for the remaining six grids. Each yellow cell should be filled.

8. Exercises

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

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

8.1. Iterative error studies: Use grid #4 and #8 with laminar flow conditions. 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 export wall shear stress data. Then use the wall shear stress at the developed region to calculate the friction factor. The equation for the friction factor is C=8* $\tau/(r*U^2)$. Where C is the friction factor, t is wall shear stress, r is density and U is the inlet velocity. Discuss the effect of convergent limit on results for these two meshes

Mesh No.	f (10 ⁻⁵)	F(10 ⁻⁶)
4	(%)	(%)
8	(%)	(%)

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 (452×44) 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.
- ANSYS case need to be saved: mesh 8 with convergent limit 10^{-6}
- 8.2. Verification study for friction factor of laminar pipe flow: Run the simulations with the meshes shown in the table. Using mesh 4 as the "fine" mesh, and run verification with grid refinement ratio 1.414 and convergence limit 10⁻⁶. Compute the parameters in the table (Refer to class website for V&V instructions). 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?

- Figure need to be saved: Figures and tables from V&V spread sheet.
- Data need to be saved: the above table with values
- 8.3. 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 and convergence limit 10⁻⁶.

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: Figures and tables from V&V spread sheet.
- Data need to be saved: the above table with values
- 8.4. Verification study of axial velocity profile: Use mesh 4 as the "fine mesh", use grid refinement ratio 1.414 and convergence limit 10⁻⁶. Follow the V&V for velocity how to in the post processing section. Save the figures and discuss if the simulation has been verified.

• Figures need to be saved: Figures showing Ug, Ugc with |E|. Discuss which mesh solution is closest to the AFD data, why?

• Data need to be saved: None.

8.5. Simulation of turbulent pipe flow

Run simulation with convergence limit 10^{-6} 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.

8.6. 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.

8.7. Questions need to be answered in CFD Lab report

- 8.7.1. Answer all the questions in exercises 1 to 6
- 8.7.2. Analyze the difference between CFD/AFD and CFD/EFD and possible error sources.
- 8.7.3. Analyze the difference between ANSYS predictions and your own calculations (using formula in CFD lecture) for order of accuracy and grid uncertainties.