Simulation of Laminar Pipe Flows

57:020 Mechanics of Fluids and Transport Processes CFD PRELAB 1

By Timur Dogan, Michael Conger, Maysam Mousaviraad, Tao Xing 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 PreLab 1 is to teach students how to use the CFD educational interface (ANSYS), be familiar with the options in each step of CFD Process, and relate simulation results to AFD concepts. Students will simulate **laminar** 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 wall shear stress. Students will compare simulation results with AFD data, analyze the differences and possible numerical errors, and present results in CFD Lab 1 report.



Flow chart for "CFD Process" for pipe flow

2. Simulation Design

In EFD Lab 2, you conducted experimental study for **turbulent** pipe flow. The data you have measured will be used for CFD Lab 1. In CFD PreLab 1, simulation will be conducted only for **laminar** circular pipe flows, i.e. the Reynolds number is less than 2300. Reynolds number based on pipe diameter and mean inlet velocity is **654.75** in the current simulation. CFD predictions of friction factor and fully developed axial velocity profile will be compared with AFD data.

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

Table 1 –	Geometry	dimensions
-----------	----------	------------





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 in details later. Uniform flow is 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 the outlet. Symmetric boundary condition will be applied on the pipe axis. Since the flow is laminar, turbulence models are not necessary.

Navigation Tips

- To zoom in and out use the magnifying glass with a plus sign in it and drag, from top left to bottom right over the are you wish to zoom.
- To look at a view plane, simply click on the arrow in the coordinate system identifier in the bottom right of the screen. i.e. if you wish to look at the XYplane, click on the Z Arrow.

3. Open ANSYS Workbench

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



3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop the **Geometry** component for the **Component Systems** on the left side of the screen into the **Project Schematic**. Rename the geometry by right clicking on the down arrow of the **Geometry** component and selecting **Rename**.

A Unsaved Project - Workbench		-	Along Along		00-	(1	A Unsaved Project - Workbench			1		00	-
File View Tools Units Extensions	wp []]]ing	ort Paseco	nnect 🚑 Refresh Project 💉 Update Project	Grow	et 🙆 Compact Mode		File View Tools Units Extensions	netp []imp	wrt] -Po Reco	rrect 😸 Refresh Project 🧚 Update Project	@Project	Compact Mode	
Toble • 0 X Redok Works • 0 X Redok Works • 0 X Redok Works • 0 X Rodok Works • 0 X Rodok Works • 0 X Stady-State Themail • 0 X Thraistic Workshill • 0 X Thraistic Workshill • 0 X Composet System • 0 X Addoph • 0 X Composet System • 0 X Robertal Data • 0 State System Robertal Data • 0 State 0 Data • 0 Detail Osta • 0 State 0 Data • 0 Robertal Data • 0 The Concertsion • 0 Robertal Model • 0 The Concert Model		Schemage Course soundables a			· 9	x	Color Color	Project	A A			- 9	×
Nicrosoft OfficeExcel	Nessag	n			+ 0	×	Nicrosoft OfficeExcel	Messag	ri i			* \$	×
n ^B Polyflow		A	0	c	D		n ^{III} Polyflow	1000	A		c	D	1
# Polyflow-Blow Molding	1	Type	Text	modatic	Date/Time	12	2 Polyflow - Blow Molding	1	Type	Text	sectato	Date/Time	1
rt ^{II} Polyflow-Extrusion	2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system.		10/12/2013 11:56:07 PM		Vev Al / Customian	2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system.		0/12/2013 11:56:07 PM	
Ready				Prov Prog	yess Hide S Messages	14	Ready			(C.)	how Progres	# Hide 5 Messages	18

3.3. Drag and drop two Mesh components and two Fluent components into the schematic as shown below. Rename the components as you did the geometry previously as per the as shown below. Make the connections as per below by dragging component to component.

🔥 Unsaved Project - Workbench	
File View Tools Units Extensions Help	
👔 New 🚰 Open 层 Save 🔊 Save As 📓 Import 🕹 Reconnect 🥔 Refresh Project 🍼 Update Project 🔇 Project 🚱 Compact Mode	
Toolbox • 7 X Project Schematic	- 4 X
🔁 Analysis Systems	
Component Systems	
Autodyn A C	
1 Geometry 1 Mesh 1 Line Fluent	
🔮 Engineering Data 🗧 2 🥪 Geometry ? 🖌 💶 2 💕 Mesh ?? 🖌 💶 2 🦃 Setup ?? 🖌	
External Connection pipe uniform 3 🗌 Solution 😨	
External Data laminar	
Truent Tuent (with TGrid meshing)	
A ICEM CFD	
Mechanical APDL	
Mechanical Model	
🚳 Mesh	
K Microsoft OfficeExcel	
+2 Polyflow	
🐉 Polyflow - Blow Molding 💌	
View All / Customize	
Ready 🔤 Show Progress) 💭 Sl	now 4 Messages

- 3.4. Create a Folder on the H: Drive called *CFD Pre-Lab and Lab 1*.
- 3.5. Save the project file by clicking **File** > **Save As...**
- 3.6. Save the project onto the H: Drive in the folder you just created and name it *CFD Pre-Lab and Lab 1 Pipe Flow*. (This will be used for both Pre-Lab 1 and Lab 1.)

4. Geometry Creation

4.1. Right click on Geometry and from the drop down menu select New Geometry...



4.2. Select Meter for unit and click OK.

elect desired length	unit
🖲 Meter	C Foot
Centimeter	C Inch
C Milimeter	
C Micrometer	
Always use proj	ect unit
Always use sele	cted unit
Enable large mod	el support

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



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

Tree Outline 4	Graphics
⊡… √@ A: pipe	
⊡✓ <mark>沐</mark> XYPlane	
🔤 🖓 Look at	
	ndencies
VZPlai allo Rename	
🦾 🖓 0 Parts, υ воαles	

4.5. Select **Sketching** > **Rectangle**. Create a rectangle geometry as per below, make sure to start from the origin, the mouse arrow should change to a "P" when on the origin.



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



4.7. Click on H1under Details View, in the bottom left of the screen, and change H1 to 7.62*m*. Click on V2 and change it to 0.02619*m*.

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

4.8. Concept > Surface From Sketches, select the sketch by clicking on Sketch 1 in the Tree Outline and hit Apply in the Detatils View.



4.9. Click Generate. This will create a surface.

-	
🕅 A: pipe - DesignModeler	
File Create Concept Tools View	Help
] 🗟 📑 📑 🖚] ĐUndo 🔅	Redo 🛛 Select: 🆎 🍢 🕅 🗈 💽 💽 🥪 🗐
] S 🕂 Q 🕀 Q Q Q S	\$ ★ • •
	/x - ⊀ ≓
🖌 XYPlane 🛛 🔻 👗 Sketch1	👻 ಶ 🦪 🗸 Generate 🖓 Share Topology 😰 Para
📙 💽 Extrude 🚓 Revolve 🐁 Sweep	🚯 Skin/Loft 📋 🛅 Thin/Surface 🛛 💊 Blend 🔻 🔦 Cha
Tree Outline 4	Graphics
⊟, 🙆 A: pipe	
XYPlane	
Sketch1	
ZXPlane	
····· ↓ YZPlane	
SurfaceSk1	
Sketch1	
⊢, the second secon	
Surface Body	

4.10. File >Save Project. Save project and close the Design Modeler window.

5. Mesh Generation

5.1. From the **Project Schematic** right click on **Mesh** on the **Fluid Flow** (**Fluent**) component and select **Edit...**



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

A : CED Prelab 1 Laminar Flow - Meshing (ANSYS Academic	Teaching Advance	1	1 1 T 4 1	5 6 C B.	- 51 4		• ×
	M 1 #22 mt		1				
File Edit View Units Tools Help	eiviesn 👔 🏨	A 🚺 🕈 🛃 Work	sneet 1				
* * E + & E E E E E # + S +	ા હ સ∣ જ્	ଷ୍ପ୍ର୍ଙ୍	12 🗃 🖷 🚫 🗆 🗸				
🖵 🖵 Show Vertices 🖓 Wireframe 🛛 📕 Edge Coloring 👻 🦯	6• /1• /2• .	/s= /s= 🗶 🖻	How Thicken Annotations □	📲 Show Mesh 🛛 🍂	🕌 Random Colors	🐼 Annotation P	references
Mesh 💈 Update 🏾 🌚 Mesh 🔻 🔍 Mesh Control 👻 🔐	vletric Graph						
Outline	4					A N1	NC.
Filter: Name 🔻 🚯 🖉 🕀						AN:	SIZ
Project							R14.5
📄 🗑 Model (A3)						Acad	emic
Coordinate Systems							
Mesh							
Insert COD Me	ethod						
🥩 Update 🔍 🔍 Siz	ing						
🕺 Generate Mech	ontact Sizing						
	finement						
Preview Preview	anned Face Meshing						
Show Market	atch Control						
Create Pinch Controls	nch						†
🖉 Clear Generated Data 🛕 Inf	flation						-
Details of "Mask" @ Rename					_		
Defaults Start Recording	+						
Physics Preference CFD		Geometry Print	Preview & Report Preview /				
Solver Preference Fluent		Massagar					л v
Relevance 0		Test				Accociation	+ ^
± Sizing		TEX				Association	
Inflation	E						
Assembly Mesning							
P Patch Conforming Options							
Triangle Surface Mesher Program Controlled							
Advanced							
Defeaturing							
Statistics	-						
Press F1 for Help		🟓 No Messages	No Selection		Metric (m, kg, N, s, V	, A) Degrees r	ad/s Ce

5.3. Select your geometry by clicking the yellow box which says **No Selection**, the click on the geometry surface, and click **Apply**.



5.4. Click on the edge button. This will allow you to select edges of your geometry.

(🗃 A :	CFD Pr	elab 1	Laminar	Flow - I	Meshing	ANS)	'S Acade	mic Teach	ning Ac	vanced]		-	۰.	-	π.,	4.14	6.6		-	ы. т .		x
	File	Edit	View	Units T	ools H	lelp 📗	••	📁 Gene	rate Mesh	n 🟥	R6¢ .	A) 🧭	- 🕼	Works	heet	ir								
		^{8,8,2}	k] ▼	la - 🖻		R	🏵	- S	+‡+ ⊕	Ð,	۹	Q (<u></u>	ISO	19 6	J 🖻	۲	-						
	۶	Show Ve	ertices	a ∰ Wir	efra Edo	e 📕 E	dge Co	loring 👻	6- 1	/i + .	2- 1	k • /	<u> </u>	< →	H T	hicken	n Annot	ations	¦≌ Show M	esh 🎄	🚼 Ran	dom Colors	Annotation Preferen	ices
	Mesh	n 🔣 Up	pdate	👘 Me	esh 🔻 🖡	🖲 Mes	n Contr	ol 🖣 🗌	Metric	Graph														

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

Dutline		.
Filter: Name	- 😰 🖉	
Project ☐ Model (☐ 6 ☐ 6 ☐ 6 ☐ 7	B2) ometry Surface Body ordinate Systems	
Ė <mark>∕</mark> ® <u>Me</u>	Insert	🕨 🌚 Method
~∎	誟 Update	🔍 Sizing
	誟 Generate Mesh	♀ Contact Sizing A Refinement
	Preview Show ジ Create Pinch Controls	Mapped Face Meshing Match Control
	⊘ Clear Generated Data alp Rename	A Inflation
	Start Recording	

5.6. Hold **Ctrl** button and select the top and bottom edge of the rectangle then click **Apply**. Specify details of sizing as per below in the **Details of "Edge Sizing" – Sizing** window.

D	Details of "Edge Sizing" - Sizing								
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							

5.7. Repeat step 5.5. Select the left and right edge of the rectangle and click **Apply** then change sizing parameters as per below.

De	Details of "Edge Sizing 2" - Sizing 🕈 🕈		
	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	

5.8. Click on **Generate Mesh** button. Click **Mesh** under **Outline**. The mesh should look like the mesh pictured below.



5.9. Change the edge names by selecting the edge, then right clicking on the edge 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.



5.10. File > Save Project. Save the project and close the window. Update mesh on Project Schematic by right clicking on Mesh and selecting Update.

6. Setup (Physics)

6.1. Right click Setup and select Edit...

🔥 Unsaved Project - Workbench	
File View Tools Units Extensions H	elp
🎦 New 对 Open 🛃 Save 📓 Save As	🕼 Import 🖗 Reconnect 🛿 Refresh Project 🍼 Update Project 🕜 Project 🕜 Compact Mode
Toolbox 🝷 🕂 🗙	Project Schematic 🗾 👻 🕂 🗙
Analysis Systems Design Assessment Electric Explicit Dynamics	
Ibuid Flow - Blow Molding (Polyflow) Fluid Flow - Extrusion (Polyflow) Fluid Flow (CFX) Fluid Flow (Fluent)	Geometry Uniform 3 @ du Edit Pre-Lab 1 La Register Startup Scheme File Import Fluent Case

6.2. Check Double Precision and click OK.



6.3. **Solution Setup** > **General** > **Check**. (Note: If you get an error message you may have made a mistake while creating you mesh. Review steps in mesh generation and make changes.)

Meshing	General	
Mesh Generation	Mesh	
Solution Setup Seneral Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Scale Display Solver Type © Pressure-Based © Density-Based Time	Check Report Quality
Solution Solution Methods Solution Controls	 Steady Transient 	 ● Planar ○ Axisymmetric ○ Axisymmetric Swirl
Solution Initialization Calculation Activities Run Calculation	Gravity	Units
Results	Help	
Graphics and Animations Plots Reports		

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

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

6.5. Solution Setup > Models > Edit... Make sure Laminar is selected and click OK.

Meshing	Models	Viscous Model
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	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 Spatart-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 Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Edit 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 Solution Setup	Materials	Name air	Material Type	Order Materials by
General Models Materials Phases Cell Zone Conditions	aluminum	Chemical Formula	= Fluent Fluid Materials air ▼ Mixture	Chemical Formula Fluent Database User-Defined Database
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Create/Edit Delete	Properties Density (kg/m3) constant 1.17 viscosity (kg/m-s) constant 1.872e-05	none •	
		Change/Create	Delete Close Help	

6.7. Solution Setup > Cell Zone Conditions > Zone > surface_body. Change type to fluid and click OK. Select Material Name as air and click OK. This should be defaulted to fluid.

Meshing	Cell Zone Conditions
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Zone Surface_body
Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Phase Type ID mixture solid 2 fluid fluid solid Parameters Operating Conditions Display Mesh Porous Formulation @ Superficial Velocity Physical Velocity
	Help

Iluid	X
Zone Name	
surface_body	
Material Name air Edit	
Frame Motion Source Terms	
Mesh Motion Fixed Values	
Porous Zone	
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Mu	ltiphase
This page is not applicable under current settings.	
OK Cancel Help	

6.8. **Solution Setup** > **Boundary Conditions** > **inlet** > **Edit...** Change parameters as per below and click **OK**. Table below shows the summary of

Inlet Boundary Condition			
Variable	u (m/s)	v (m/s)	P (Pa)
Magnitude	0.2	0	-
Zero Gradient	N	N	Y

Meshing	Boundary Conditions	Velocity Inlet
Mesh Generation	Zone	Zone Name
Solution Setup		inlet
General Models Materials	inlet Interior-surrace_body outlet wall	Momentum Thermal Radiation Species DPM Multiphase UDS
Phases		Velocity Specification Method Magnitude, Normal to Boundary
Boundary Conditions		Reference Frame Absolute
Dynamic Mesh		Velocity Magnitude (m/s) 0.2 constant
Solution		Supersonic/Initial Gauge Pressure (pascal) 0 constant
Solution Methods Solution Controls Monitors Solution Initialization		OK Cancel Help
Calculation Activities Run Calculation	Phase Type ID	1e-10
Results	mixture velocity-inlet 7	1e-12
Plots	Edit Copy Profiles	0 100 2
Reports	Parameters Operating Conditions Display Mesh Periodic Conditions	

6.9. Solution Setup > Boundary Conditions > outlet > Edit... Change parameters as per below and click OK.

Outlet Boundary Condition			
Variable	u (m/s)	v (m/s)	P (Pa)
Magnitude	-	-	0
Zero Gradient	Y	Y	Ν

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

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

Wall Boundary Condition					
Variable u (m/s) v (m/s) P (Pa)					
Magnitude	0	0	-		
Zero Gradient	N	Ν	Y		

Wall
Zone Name
wall
Adjacent Cell Zone
surface_body
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film
Wall Motion Motion
Shear Condition
 No Slip Specified Shear Specularity Coefficient Marangoni Stress
Wall Roughness
Roughness Height (m) 0 constant -
Roughness Constant 0.5 constant v
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	axis	Operating Pressure (pascal)
General	inlet interior-surface body	97225.9
Models	outlot	Reference Pressure Location
Phases	Wall	
Cell Zone Conditions		e .
Mesh Interfaces		Y (m) 0
Dynamic Mesh		<u>e</u>
Reference Values		Z (m) 0
Solution		
Solution Methods		
Monitors		OK Cancel Help
Solution Initialization		
Calculation Activities	Phase Type ID	
Run Calculation	mixture wall - 5	
Results		
Graphics and Animations Plots	Edit Copy Profiles	
Reports	Parameters	
	Dirplay Mesh	
	Periodic Conditions	

6.12. Solution Setup > Boundary Conditions > axis. Make sure that axis is selected as per below.

Axis Boundary Condition					
Variable u (m/s) v (m/s) P (Pa)					
Magnitude	-	0	-		
Zero Gradient	Y	N	Y		



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

Meshing	Reference Values
Mesh Generation	Compute from
Solution Setup	▼
General Models Materials Phases	Area (m2) 0.002154869
Cell Zone Conditions Boundary Conditions Mesh Interfaces	Enthalpy (j/kg) 0
Reference Values Solution	Length (m) 0.05238
Solution Methods Solution Controls	Pressure (pascal)
Monitors Solution Initialization Calculation Activities	Temperature (k) 298.16 Velocity (m/s) 0.2
Results	Viscosity (kg/m-s) 1.872e-05
Graphics and Animations Plots Reports	Ratio of Specific Heats 1.4
	Reference Zone
	
	Help

7. Solution

7.1. Solution > Solution Methods. Change parameters as per below.

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

7.2. Solution > Monitors > Residuals – Print, Plot > Edit... Change convergence criterion to 1e-06 for all three equations as per below and click OK.

Meshing	Monitors	Residual Monitors		X
Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Create Edit Delete Surface Monitors	Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Residual Monitor Check Convergenc Continuity V V X-velocity V V Residual Values Iterations F	e Absolute Criteria 1e-06 1e-06 1e-06 Convergence Criterion absolute
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Create Edit Delete Volume Monitors	Iterations to Store	Scale Compute Local Scale	ql

7.3. Solution > Solution Initialization. Change parameters as per below and click Initialize.

Meshing	Solution Initialization	
Mesh Generation Solution Setup General Models	Initialization Methods O 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 Initial Values	•
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Gauge Pressure (pascal)	*
	Initialize Reset Patch	Ŧ
	Reset DPM Sources Reset Statistics	

7.4. Solution > Run Calculation. Change Number of Iterations to 1000 and click Calculate.

Meshing Mesh Generation Solution Setup	Run Calculation Check Case Preview Mesh Motion
General Models Materials	Number of Iterations 1000 Reporting Interval 1 •
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Profile Update Interval
Dynamic Mesh Reference Values	
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Help
Results Graphics and Animations Plots Reports	

7.5. Once the solution converges, click **OK**. (The residuals should be comparable to the ones below.)



NOTE: ANSYS determines when to stop a calculation based on the iteration number and convergent limit you specified. If: 1. The maximum iteration number is reached, but convergent limit is not reached, or 2. Convergent limit is satisfied, but maximum iteration number is not reached, ANSYS will terminate the computation.

File > Save Project. Save the project

8. Results

Please read exercises before continuing.

8.1. Displaying Mesh

Display> **Mesh**. Select all **Surfaces** you wish to be visible and select **Display** then click **Close**.

Display Report Parallel View Help	Mesh Display	×
Graphics and Animations Plots Residuals Options	Options Edge Type Surfaces Nodes Image: All model All model Image: All model Faces Outline Outline Image: All model Partitions Outline wall wall	
Scene Views Lights Colormap	Shrink Factor Feature Angle 0 20 Surface Name Pattern Name Surface Test	
Annotate Zone Motion DTRM Graphics	Match Surface Types Outline Interior Axis Cip-surf	
Import Particle Data PDF Tables/Curves Reacting Channel/Curves	exhaust-fan Ifan	-
Mouse Buttons	Display Colors Close Help	

Zoom in to the inlet by using the magnifying glass with a plus sign in the middle of it. The mesh should look like the one below.



8.2. Saving Pictures

To save a picture of the screen, select **File > Save Picture...** Make sure all the parameters are set similar to the ones below and click **Save...** (To preview the picture, before you save click **Preview** in the **Save Picture** window)

Save Picture			×
Format	Coloring	File Type	Resolution
 EPS JPEG 	 Color Gray Scale 	Raster Vector	Width 960
 PPM PostScript 	Monochrome		Height 720
TIFF PNG	Options		
© VRML © Window Dump	 Landscape Orie White Backgrou 	intation Wind	dow Dump Command
Save	Apply Pr	eview Clos	se Help

Name the File as *CFD Pre-Lab1 Laminar Pipe Flow Mesh*, navigate to the CFD Pre-Lab 1 file you created and save it in that file. Then close the **Save Picture** window.



8.3. Displaying and Saving Residuals

To display the residuals click **Solution** > **Monitors** > **Residuals** – **Print**, **Plot** > **Edit**... > **Plot** then click **Cancel**.

A:CFD Prelab 1 Lamina	ar Flow Fluent [axi, dp, pbns, lam] [ANSYS Academic Tea	aching Advanced]	
File Mesh Define Sc	olve Adapt Surface Display Report Parallel Vie	ew Help	
🚺 🗐 💌 🖬 🕶 💽	@ 5 � € € / € 次		
Meshing	Monitors	1: Scaled Residuals	
Mesh Generation	Residualsy Statistic and Force Honitors		ANSTS 115 14
General	Residuals - Print, Plot	1e+00	
Models Materials		16-01	
Phases Cell Zone Conditions		1e-02 -	
Boundary Conditions Mesh Interfaces	Create Edit Delete	1e-03 -	
Dynamic Mesh Reference Values	Surface Mönitors	1e-04 -	
Solution		1e-05	
Solution Methods		1e-06	
Monitors		16-07	
Calculation Activities Run Calculation	Volume Monitors	1e-08	
Results		1e-09 0 100 200 300 400	500 600
Graphics and Animations Plots		Iterations	
Reports	Residual Monitors	×	J
	Crea Options Ec	quations	Jul 29, 2013
9	Conve	esidual Monitor Check Convergence Absolute Criteria	(i, dp, pbns, lam)
	Vindow	continuity V 1e-06	:00:32 46 ^
	1 Curves Axes	x-velocity V 1e-06	:01:34 45
	Iterations to Plot	y-velocity V 1e-06	:01:00 45
		esidual Values Convergence Criterion	:00:38 45
	Help Iteration to Stars		:01:55 45
		▼ Scale	:01:32 45
		Compute Local Scale	time/ite :00:58 44
	OK Plot	Renormalize Cancel Help	:00:47 44 :00:37 44
		555 8.2278e-07 1.0404e-06 1.6062e-09	J:00:30 44 0:01:53 44
,		556 8.3473e-07 1.0269e-06 1.5879e-09 557 8.2048e-07 1.0139e-06 1.5691e-09	0:01:30 44 0:01:12 44 -
1		< III	<u>بر</u>

You can save this picture the same way you saved the mesh. Name it *CFD Pre-Lab 1 Laminar Pipe Flow Residuals History* and save it to the folder you created on the H: Drive.

8.4. Plotting and Saving Results

•

To plot results, click **Results** > **Plots** > **XY Plot** > **Set Up...**

To plot the Centerline Pressure Distribution, copy the parameters as per below and click **Plot**.

A:CFD Prelab 1 Lamin	ar Flow Fluent [axi, dp, pbns, lam] [ANSYS Academic	Teaching Advanced]
File Mesh Define S	olve Adapt Surface Display Report Parallel	View Help
i 💼 i 💕 🕶 🖬 🔻 🞯) @∥S ↔ Q 🕑 🖊 🖗 Հ 🖷 ▾ 🗖 י	-
Meshing	Plots	1: Static Pressure
Mesh Generation	Dista	axis ANSYS
Solution Setup	XY Plot	4.00e-01
General	Histogram	
Models	File Profiles:	3.50e-01 -
Materials	Profile Data - Unavailable	
Cell Zone Conditions	Interpolated Data	3.00e-01 -
Boundary Conditions		2.50e-01 -
Mesh Interfaces		
Reference Values		Static 2.00e-01
Solution		Pressure (nascal) 4 co- of
Solution Methods		(pascal) 1.508-01
Solution Controls		1.00e-01 -
Solution Initialization		
Calculation Activities		5.00e-02
Run Calculation	Set Up	0.000+000
Results		0 1 2 3 4 5 6 7 8
Plots	Help	Position (m)
Reports		
	Solution	XY Plot
	Options	Plot Direction Y Axis Function
	V Node Va	alues X 1 Pressure V pbns, lam)
	Position Position	on X Axis
	Write to	File X Axis Function 58 45
	Order P	oints Z 0 Direction Vector - 34 45
	File Data	
		axis 48 45
		interior-surface_body 38 45
		outlet 155 45
		:32 45
		1000 File 58 44
		Free Data New Surface V 44
		37 44 38 44
		Mot Axes Curves Close Help 53 44
		557 8-28488-87 1-81398-86 1-56918-89 8-81-12 44
		I I I I I I I I I I I I I I I I I I I

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Centerline Pressure Distribution* and save it in the folder you created.



To plot Centerline Velocity Distribution, copy the parameters as per below and click **Plot**.

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Centerline Velocity Distribution* and save it in the folder you created.



To plot the Wall Shear Stress Distribution, copy the parameters as per below and click Plot.

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Wall Shear Stress Distribution* and save it in the folder you created.

To plot Profiles of Axial Velocity at All Axial Locations with AFD Data, click **Surface** > **Line/Rake...**

i	A:CFD Prelab 1 Lamina	r Flow Fluent	[axi, dp, pbns, lam] [ANSYS	Academic Tea	aching Advanced]	-	-		-	L		a x
	File Mesh Define So	lve Adapt	Surface Display Report	Parallel Vi	ew Help							
	i 📖 i 📂 🕶 🖬 👻 🚳	0 S -	Zone									
t	Meshing	Plots	Partition		1: Skin Friction Coefficient	-						
L	Mesh Generation	Plata	Point		• wall							ANSYS
I	Solution Setup	XY Plot	Line/Rake		1.40e-01							
L	General	Histogram	Plane									
I	Models Materials	Profiles: Profile Dat	Quadric		1.20e-01							
L	Phases Cell Zone Conditions Boundary Conditions	Interpolat FFT	Iso-Surface Iso-Clin		1.00e-01							
Ш	Mesh Interfaces		and endow									
L	Dynamic Mesh		Transform		Skin s 00e.02							
I	Reference Values Solution		Manage		Friction							
I	Solution Methods	- I		-	6.00e-02							
I	Solution Controls Monitors											
L	Solution Initialization				4.00e-02							
L	Calculation Activities	Cattle										
	Results	ser up			2.00e-02							
	Graphics and Animations					0 1	2	3 4	5	6	7	8
	Plots Reports	Help						Position	(m)			

Change x and y values as per below, name the surface, and click **Create**. Repeat this for all lines shown in the table.

Line/Rake Surface	×
Options Type Line Tool Reset	Number of Points
	x1 (m) [0, 5220
0.5238	0.5238
y0 (m) 0	y1 (m) 0.02619
z0 (m) 0	z1 (m) 0
Select Poin	ts 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

When all lines are created, click **Close**.

Click **Results** > **Plots** > **XY Plot** > **Set Up...** Click **Load File...** and select axialvelocityAFD-laminar-pipe.xy, which can be found from the class website. Click **OK**.

Solution XY Plot			×
Options Options Option on X Axis Position on Y Axis Write to File	Plot Direction X 1 Y 0	Y Axis Function Wall Fluxes Skin Friction Coefficient X Axis Function	• •
File Data	Ζ_0	Direction Vector Surfaces inlet interior-surface_body outlet wall	
۰ III کې ا	Load File Free Data	x=100d x=10d x=20d x=40d New Surface ▼	*
Plot	Axes	Curves Close Help	



Change Parameters as per below. Make sure to select inlet as well.

Click **Curves...** > Change the **Pattern** to the pattern seen below and click **Apply**. Incriment the **Curve #** by one and repeat. Do this for curves 0 through 7 then click **Close**.

Solution XY Plot	X
Solution XY Plot Options Plot Direction Curve # Curves - Solution XY Plot Curve # Curve # Pattern File Di Sample Color foreground Weight 1	Y Axis Function Velocity Marker Style Symbol (*) Color foreground Size 0.3
Plot Axes	Close Help

Click Plot.

A:CFD Pre-Lab 1 Lamin	ar Pipe Flow Fluent [axi, d	o, pbns, lam] [ANSYS Acade	mic Teaching Advanced]		23
File Mesh Define Sol	lve Adapt Surface Dis	play Report Parallel Vi	ew Help		
i 📖 i 📂 ד 🖬 ד 🚳	🞯 📴 🕀 🗨 🥕	🎚 🍭 🏷 📑 🕶 🚽			
Meshing	Plots		1: Axial Velocity	•	
Mesh Generation	Plots				SYS R14.5
Solution Setup	XY Plot			4 50o 01	demic
General Models	File			4,00001	
Materials	Profile Data - Unavailable		• AFDT	4.00e-01 +	
Cell Zone Conditions	FFT			2 50 - 01	
Boundary Conditions Mesh Interfaces				3.508-01	
Dynamic Mesh				3.00e-01 -	
Solution					
Solution Methods			Ax	z.50e-01	
Monitors			Veloci	ity 2.00e-01	
Solution Initialization Calculation Activities			(m/	/s)	
Run Calculation	Set Up			1.508-01	
Results Graphics and Animations				1.00e-01 -	
Plots	Help				
Reports				5.00e-02	
				0.00e+00	
Solution	XY Plot			0 0.005 0.01 0.015 0.02 0.025 0.03	
Options	Plot Direction	Y Axis Function		Position (m)	
V Node Va	alues X 0	Velocity			
Position	on Y Axis Y 1	Axial Velocity			
Order P	voints Z 0	Direction Vector		Oct 10, 20	013
File Data		Surfaces		ANSYS Fluent 14.5 (axi, dp, pbns, la	am)
Velocity Ma	gnitude	interior-surface_body outlet	^ ^		_
		wall x=100d			he
		x=10d x=20d	E		-
	[Land Els.]	x=40d x=60d		-activate-item "Line/Rake Surface*PanelButtons*PushButton1(OK)")	
	Load File	New Conference			
	Free Data	INEW SUFFACE		-activate-item "Line/Rake Surface*PanelButtons*PushButton2(Cancel)")	
	Plot Axes	Curves Close	Help		
			¥	k-activate-item "HenuBar*WriteSubMenu*Stop Journal")	
			•	III.	F

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Axial Velocity at All Axial Locations with AFD Data* and save it in the folder you created. Close the **Solution XY Plot** window.

8.5. Plotting and Saving Graphics

Click Results > Graphics and Animations > Vectors > Set Up...

To plot the velocity vectors at the region flow begin to becomes fully developed, copy the parameters as per below and click **Display**. Zoom into the region where the flow is almost fully developed.



Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Velocity Vectors at the Region Flow Begins to Become Fully Developed* and save it in the folder you created. Close the **Vectors** window.

Click Results > Graphics and Animations > Contours > Set Up...

To plot the Contours of Radial Velocity, copy the parameters as per below and click **Display**. Zoom in to the pipe inlet to see the contours of radial velocity.

G	raphics and Animations	1: Contours of Radial Velocit	•		
ston G	raphics	9 13=05			ANSY
6	tesh	4.876.04			
	Vectors	-1.06a-03			
	Pathines Particle Tracks	-1 64+03			
10		.2 226.03			
1.15		-2.80+//3			
1	Set Up	.3 38+03			
		3.050.03			
A	nimations	4.53+.03			
	Scene Animation	-4.530-03			
	Solution Animation Playback	-5.11e-03			
<		6.036-03			
*		0.27603			
	Gettin	Contours		X	
Sons	an april (Cotions	Contours of	1	
0	Ontions Scene Views	2 Filed	Velocity	-	
č	Linhts Colormon Annotate	Node Values	Dadial Valority		
-	(Second Second S	Global Range	Raudi Velouty		
1		Clip to Range	-0.01146814 9.128	967e-05	
1	tielo.	Draw Profiles	hand the second se		
		E Sourcest	axis		
		Levels Setur	iniet	1	
		20 0 1 0	outiet		
			x=100d	-	Oct 10, 201
		Surface Name Pattern	New Conference		ANSYS Fluent 14.5 (axi, dp, pbns, lan
		Match	[ver surace *]]		
		A	Surface Types		e o sta america a metro ana
			dp-suf	n l	rface*Frane4(End Points)*Frane1*Table1*RealEntry4
			fan	*	
			2		PanelButtons=PushButton1(OK)")

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Contours of Radial Velocity* and save it in the folder you created. Close the **Contours** window.

8.6. Exporting Results

```
To export Results, click Results > Plots > XY Plot > Set Up...
```

To export the Developed Axial Velocity Profile at x=100d, copy the parameters as per below and click **Write...**

Solution XY Plot		X
Ontions	Plot Direction	Y Axis Function
Vode Values	x o	Velocity
Position on X Axis	×	Axial Velocity
Write to File	1	X Axis Function
Order Points	ZO	Direction Vector 🗸
File Data 🔳 🗏		Surfaces 🔳 🗐
	Load File	axis inlet interior-surface_body outlet wall x=100d x=10d x=20d v
	Free Data	New Surface V
Write	Axes	Curves Close Help

Name the file *CFD Pre-Lab 1 Laminar Pipe Flow Developed Axial Velocity Profile* and leave the Files of Type: as XY Files. Click **OK**.

Select File			×
Look in:	🔒 CFD Pre-Lab	- G 🖻 🖻 🎞 -	
Recent Places	axialvelocityAFD		
Network	XY File	CFD Pre-Lab 1 Developed Axial Velocity Profile 🔻	ОК
	Files of type:	XY Files •	Cancel

To export the wall shear stress distribution, copy the parameters as per below and click **Write...**

Solution XY Plot	5	×
Options	Plot Direction	Y Axis Function
Node Values	X 1	Wall Fluxes 👻
Position on X Axis	YO	Wall Shear Stress 🗸
Write to File		X Axis Function
Order Points	ZO	Direction Vector 🗸
File Data		Surfaces 🔳 🗏
	Load File	axis inlet interior-surface_body outlet wall
	Free Data	New Surface 🔻
Write	Axes	Curves Close Help

Name the file *CFD Pre-Lab 1 Laminar Pipe Flow Wall Shear Stress Distribution* and leave the Files of Type: as XY Files. Click **OK.** Close the Solution XY Plot.

File > Save Project. Save the project and close the Fluent window.

- 8.7. Normalizing Velocity Profile
 - Open excel from Start Menu.
 - Click **File** > **Open**, navigate to your folder you created on the H: Drive.
 - Change the file type to all files.
 - Select the file CFD Pre-Lab 1 Laminar Pipe Flow Developed Axial Velocity Profile and click **Open**.
 - Select **Yes** on the **Microsoft Excel** message.

Microsoft	Excel
	The file you are trying to open, 'CFD Pre-Lab 1 Developed Axial Velocity Profile', is in a different format than specified by the file extension. Verify that the file is not corrupted and is from a trusted source before opening the file. Do you want to open the file now?
	Yes No Help
	Was this information helpful?

- Make sure delimited is selected and click Next>.
- Make sure that **Tab** and **Space Delimiters** are checked and hit **<u>Finish</u>**.

below.		
Delimiters		
✓ <u>T</u> ab		
Semicolon	Treat consecutive delimiters as one	
Comma		
Space	Text gualifier:	
Other:		
Other:		
Other:	Axial Velocity)	
Other:	Axial Velocity) Position Axial Velocity)	
Other:	Axial Velocity) Position Axial Velocity)	
Qther:	Axial Velocity) Position Axial Velocity) x=100d) 0.393408	

• In Cell C5 enter the formula as seen in the Formula Bar below. Then take the fill handle and drag to the end of the data. This normalizes the velocity profile from the max velocity.

File Home In			Inse	nsert Page Layout Formulas Data								
Cut			Calibri			- 11	Ŧ	A A	- =			
	- C	— □ Copy ▼			-				_			
🚽 💞 Format F			inter B I U		Ţ	*						
	Clipboard				F	ont			G			
[C5		- (0		f_x	=B5/	\$B\$5					
	1	A		В		С		Formula Bar				
	1	(title A		Axial Velocity)								
	2	(labels	Position		Axi	Axial Velocity						
	3											
	4	((xy/key/label x		x=100d)								
	5	0	0		8	1						
	6	0.000582		0.399309								
	7	0.001164		0.3987	2							
	8 0.001746			0 397739								

- Insert a Scatter Plot With Smooth Lines and Markers.
- For the x-axis use the radial position, and for the y-axis use the normalized velocity.
- Name it CFD Velocity Profile (Laminar).
- You can move this plot to a new tab by clicking on the chart **Chart Tools** > **Design** > **Move Chart Location** > **New Sheet** > **OK**
- Next open the file Normalized-velocity-AFD-laminar-pipe.xy in TextPad, highlight the data and paste it into your Excel spread sheet next to the CFD velocity profile data.
- Plot this in the same way as the other set on the existing plot and call this AFD Velocity Profile (Laminar).
- Create axis titles and make sure the legend is shown. You should move the legend to the bottom of the chart. Call the axes *Normalized Velocity* [-] and *Radial Position* [m].

• Save this Sheet by selecting File > Save As, name it *CFD Pre-Lab 1 Developed Axial Velocity Profile*.

9. Exercises

You must complete all the following assignments and present results in your CFD Lab 1 reports following the CFD Lab Report Instructions.

Simulation of Laminar Pipe Flow

• You need use CFD Lab1 Report Template.doc to save all the figures and data

1. Compare CFD with AFD on friction factor

Use the instructions to generate the mesh and setup then iterate the simulation until it converges. Find the relative error between AFD friction factor (0.097747231) and friction factor computed by CFD, which is computed by:

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

To get the value of $Factor_{CFD}$, you need first write to file the wall Shear Stress Distribution. Then use EXCEL to open the data file and pick the value close to the pipe exit or inside the fully developed region. Next use the equation $C=8*\tau/(\rho*U^2)$ to solve for the Friction Factor. Where C is the friction factor, τ is wall shear stress, ρ is density and U is the inlet velocity.

- **Figures need to be saved:** 1. Residual history, 2. centerline pressure distribution, 3. centerline velocity distribution, 4. Wall shear stress distribution, 5. profiles of axial velocity at all streamwise locations (x/D=10,20,40, 60,100) with AFD data, 6. contour of radial velocity, and 7. velocity vectors (pick up the region where flow begins to become fully developed).
- **Data need to be saved:** shear stress in the developed region, developing length. Here, developing length is defined as the length from pipe inlet to the axial location where the centerline velocity does not change any more.

2. Normalized developed axial velocity profile

- 2.1. Export the axial velocity profile data at x=100d following the instructions in Step 8.6.
- 2.2. Use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude, which is the maximum value on that profile. Plot the normalized velocity profile in EXCEL and paste the figure into WORD, together with other figures you made in Exercise 1.

3. Questions need to be answered when writing CFD Lab 1 report

- 3.1. Can you use centerline pressure distribution to determine the "developing length"? Why?
- 3.2. What is the value for radial velocity at developed region?
- 3.3. Summarize your findings in CFD Lab report and try to relate them to your classroom lectures or textbooks.