# **Simulation and Validation of Turbulent Pipe Flows**

#### ENGR:2510 Mechanics of Fluids and Transport Processes CFD LAB 1 (ANSYS 17.1; Last Updated: Oct. 10, 2016)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Andrew Opyd, 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 Lab 1 is to teach students how to use ANSYS, practice more options in each step of CFD Process, and relate simulation results to EFD and AFD concepts. Students will simulate **turbulent** pipe flow following the "CFD process" by an interactive step-by-step approach. The flow conditions will be the same as they used in EFD Lab2. 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 their own EFD data, analyze the differences and possible numerical/experimental errors, and present results in a CFD Lab report.



Flow chart for "CFD Process" for pipe flow

# 2. Simulation Design

In EFD Lab 2, you have conducted experimental study for **turbulent** pipe flow. The data you have measured include centerline pressure distribution and fully developed axial velocity profile. These data will be used in this Lab for comparisons with CFD predictions.

The problem to be solved is that of **turbulent** flows through a circular pipe. Reynolds number based on pipe diameter and inlet velocity should be **computed from your own EFD data** and is much higher than the Reynolds number used in CFD Prelab1.

Table 1 - Main particulars						
Parameter	Unit	Value				
Radius of Pipe	m	0.02619				
Diameter of Pipe	m	0.05238				
Length of the Pipe	m	6.096				



Figure 1 - Geometry

The problem formulation is similar to that in CFD PreLab1 and will not be repeated here. The Reynolds number is much higher and students need specify turbulence model, and thus more variables need to be specified in boundary conditions, as will be discussed in details later.

## **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 17.1 > Workbench 17.1

(Note: You may ignore the firewall warnings by closing the pop-up window)



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

A Unsaved Project - Workbench	- 🗆 🗙 🐧 Unsaved Project - Work	bench .	- 🗆 X
File View Tools Units Extensions Jobs Help	File View Tools Unit	a Extensions Jobs Help	
🚹 🥁 🛃 🔣	🗋 🐸 🛄 🔍 / 🖪 A	roject	
Dimport 29 Reconnect 💿 Refresh Project n/ Update Project	Dimport   +ip Reconnec	1 🕃 Refresh Project 🍠 Update Project 📲 ACT Start Page	
Toobox v 🛛 🗙 Project Schematic	V B X Toobex		- 0 X
(8) Analysis Systems	(B) Analysis Systems	A	
E Component Systems	Component Systems		
3 ACP (Post)	ACP (Post)	• A	
ACP (Pre)	(IS ACP (Pre)	1 🥪 Geometry	
Autodyn Create standalone system	🥶 Autodyn	2 🥔 Geometry ?	
👹 BladeGen	🚱 BladeGen	Rea	
en crx	(II) CFX	798	
Engineering Data	🥔 Engineering Data		
🗱 External Data	External Data		
External Model	🍘 External Model		
() Finite Element Modeler	(j) FiniteElementModeler		
Fluent	Fluent		
Fluent (with Fluent Meshing)	Fluent (with Fluent Mesh	sing)	
Geometry	🤪 Geometry		
ICEM CFD	SCEM CFD		
😰 Icepak	🐨 Icepak		
A Mechanical APOL	A Mechanical APOL		
Mechanical Model	Mechanical Model		
📦 Mesh	🍘 Mesh		
Microsoft Office Excel	Microsoft OfficeExcel		
1 <sup>9</sup> Polyflow	n <sup>2</sup> Polyflow		
2 <sup>10</sup> Polyflow - Blow Molding	22 Polyflow - Blow Molding		
Polyflow-Extrusion	Polyflow-Extrusion		
Results	Results		
System Coupling	System Coupling		
🛞 Turbo Setup	GS Turbo Setup		
G TurboGrid	turbeGnd	v	
Uses ADD		H (Casharina	
Vew Al / Custonize	T vev	ne / Substance	
Ready RE 30	b Monitor E Show Progress ( Show 1 Messages Ready	🔝 Job Monitor 😑 Show Progress 🔮	Show 1 Messages

3.3. Drag and drop a **Mesh** component and a **Fluent** component into the **Project Schematic** as shown below. Rename the components as "non-uniform" and "turbulent" for **Mesh** and **Fluent** components respectively.

- que	•••	, <b>e</b> rj	•															
Project	Sch	nemat	ic														<b>џ</b>	X
	•		Α			•		В			•		С					
	1	$\bigcirc$	Geometry			1	۲	Mesh			1		Fluent					
	2	$\Theta$	Geometry	?	4	2	$\bigcirc$	Geometry	?	4	2		Setup	?	4			
			Pipe			3	۲	Mesh	7	4	3	(iii)	Solution	7	4			
							n	on-uniform				1	turbulent					

3.4. Create connection as per below by dragging and dropping each component to the corresponding component.



- 3.5. Create a Folder on the network drive called "CFD Lab 1".
- 3.6. Save the project file by clicking **File** > **Save As...**
- 3.7. Save the project onto the network drive in the folder you just created and name it "*CFD Lab 1 Turbulent Flow*".

# 4. Geometry

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



4.2. Make sure that Unit is set to Meter (default value).

🕦 A: Pipe - DesignModeler	
File Create Concept Tools	Units View Help
🖉 📑 📑 📫 🗍 🏵 Unda	✓ Meter
<u>5</u> ⊕€€€€€	Millimeter
• • • /1• /2•	Micrometer
XYPlane 🔻 🛧 None	Foot
🖪 Extrude 🏟 Revolve 🌜 S	Inch urfa
Point Donversion	Large Model Support
BladeEditor: 🔏 Import BGD 🧯	✓ Degree
<b>医玉</b> 昌 ( 京 <b>臣</b> )	Radian
Tree Outline P	Model Tolerance
A: Pipe	
XIPlane	
YZPlane	
📖 🖓 0 Parts, 0 Bodies	

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

🗊 A: Pipe - DesignModeler						
File Create Concept Tools Units View Help						
🖉 🔚 🛃 ඟ 🗍 💬 Undo 🔅 Redo 🛛 Select: 🌇 🍢 🔃 🔃 🔃 💽 💽 🗮 💭 🔪 🗮						
S ↔ Q ⊕ Q Q 🛱 🖆 🔸 📦 🔸 🕸						
■ • ■ • h • h • h • h • h • # ¤						
XYPlane 🔻 🗚 None 👻 💆 🛛 🧚 Generate 🖤 Share Topology 🔀 Parameters						
<b>R</b> Extrude 🚓 Revolve 💩 Sweep 🚯 Skin, <mark>New Sketch</mark> Thin/Surface 💊 Blend 🔻 💊 Chamfer 🔍 Slice						
Point Conversion						
BladeEditor: 🍰 Import BGD 🗿 Load BGD 🕢 Load NDF 🛛 式 FlowPath 🥒 Blade 💋 Splitter 🚽 VistaTF						
必还3目(京使						
Tree Outline						
Em S A: Pipe						
XYPlane						
ZXPlane						
🗤 🖓 0 Parts, 0 Bodies						

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

Tree Outline 4	Graphics
⊡… <b>√🚱</b> A: Pipe	
白 🧈 🖈 XYPI: 🙀 Look at	
ZXPI Show De	pendencies
YZPIa allo Rename	(F2)
🗤 🖓 0 Parts, 0 Bodies	

4.5. Select **Sketching** > **Constraints** > **Auto Constraints**. Enable the auto constraints option to pick the exact point as below.

	Draw	
	Modify	
	Dimensions	
	Constraints	▲
Auto Constraints		Global: 🔲 Cursor: 🔽
	Settings	
Sketching Modeling		

4.6. Select **Sketching** > **Draw** > **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.



4.7. 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.



4.8. Click on H1 under Details View and change it to 6.096m. Click on V2 and change it to 0.02619m.

**NOTE:** The actual length of the pipe is 30 feet (9.144m), However, in CFD simulation, we need to specify "outlet pressure", and we don't have a pressure transducer at the pipe outlet. So we choose the outlet of the pipe we will simulate to be the location of the last pressure transducer, which is 6.096 meters from the pipe inlet.

-	Details of Sketch1					
	Sketch	Sketch1				
	Sketch Visibility	Show Sketch				
	Show Constraints?	No				
Ξ	Dimensions: 2					
	H1	6.096 m				
	V2	0.02619 m				
-	Edges: 4					
	Line	Ln7				
	Line	Ln8				
	Line	Ln9				
	Line	Ln10				

4.9. Concept > Surface From Sketches, select the sketch by left clicking on Sketch1 in the Tree Outline and hit Apply in the Detatils View under Base Objects.



## 4.10. Click Generate. This will create a surface.

🔞 A: Pipe - DesignModeler	
File Create Concept Tools Units View Hel	lp
🔄 🛃 🛃 🖾 🗍 🗊 Undo 📿 Redo 🗍 Sel	lect: 🌇 🏷 🖪 🖪 🖪 🖉 🗐 🏹 🗐 🖉
	т́
XYPlane 💌 🛧 Sketch1 💌 ಶ	🚽 🗦 Generate 🛛 🗑 Share Topology 🛛 🚉 Parameters
🖪 Extrude 🏘 Revolve 🐁 Sweep 🚯 Skin/Lo	oft 🔄 🛅 Thin/Surface 💊 Blend 🔻 🥎 Chamfer 🏘 Slic
BladeEditor: 🆓 Import BGD 🕼 Load BGD 🕢	Load NDF 🛛 🚔 FlowPath 🥜 Blade 💋 Splitter 🚽 Vista
▶ ▲ S = ( 函 比	- · · · · ·
Tree Outline 🛛 🗣	Graphics
Image: Skippe A: Pipe A:	

4.11. File >Save Project. Save project and close window.

# 5. Mesh Generation

5.1. From the Project Schematic right click on Mesh component and select Edit...



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



5.3. Select your geometry by clicking on the yellow box which says **No Selection**, then select the geometry surface and click **Apply**. (Note: You can change orientation of your view by clicking the axis figure on the lower right corner. Press "F7" on your keyboard to restore to the "whole view". Zoom in by holding the right mouse button and selecting a region.)



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

😧 B : non-uniform - Meshing [ANSYS Academic Research]	-		$\times$			
File Edit View Un <u>its T</u> ools Help 🛛 🖂 🕶 🛛 🧚 Generate Mesh 🏥 📷 \Lambda 🞯 🔻 🎲 Worksheet i 🖡						
🐨 🎬 🕼 🖪 🖪 🔚   🚱 +   S ↔ Q ⊕   Q Q Q 🐺 /2 📾 📾 🏷   🗆 +						
🔎 Show Vertices 🛱 Close Vertices 6.1e-003 (Auto Scale) 👻 🆓 Wireframe 🌐 Show Mesh 🦗 🕌 Random Colors 🐼 Annotation Preferences	t→ t	$\rightarrow$ $\stackrel{\uparrow}{\bullet}\rightarrow$	<b>↓</b> ↓			
Assembly Center						
III Edge Coloring ▼ /₀ ▼ /₁ ▼ /₂ ▼ /₃ ▼ /₂ ▼ /₃ ▼ /₂ ▼ /□ Thicken Annotations						
Mesh 🧚 Update 🛛 🎕 Mesh 🔻 🎕 Mesh Control 🔻 🆚 Mesh Edit 👻 🚛 Metric Graph 🛛 🖾 Probe 🛛 📼 💷 🖗 💌						

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

Outline			4		
Filter: Name	•				
🕼 🔄 🐎 🖽 🧕	1				
Project					
🗄 🖷 🚱 Model (B3)					
🖻 🗸 🖓 Geometr	ry Sera Ded				
u ⊡ v v Coordin	rface Boo ate Syste	ly ms			+
E	100 0 9000	113			
	Insert		•	¢	Method
-	Undate			8	Citize
2	Update	2		20 K	Sizing
3	Genera	ate Mesh		<sup>₩</sup> *	Contact Sizing 0.
Details of "Mesh"	Dravies		•		Refinement
- Display	Chave				Face Meshing
Display Style	Snow	Snow			Match Control
Defaults	Create Pinch Controls				Pinch
Physics Preferenc 🖉	Clear G	Generated Data		٨	Inflation
Relevance all	ຸ Renam	ie (F2)		-	
Shape Checking		and a children	_		Mesh Connection Group
Element Midside	Group All Similar Children			1	Manual Mesh Connection
Sizing	Start Recording				Contact Match Group
Inflation		-		۲	Contact Match
+ Advanced					Node Merge Group
				۰	Node Merge Sel
Scoping Method Geo					Node Move
Geometry 1 Ec					

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

D	Petails of "Edge Sizing" - Sizing						
-	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 5.5. Select the left edge then click **Apply** and change the parameters as per below.

De	Details of "Edge Sizing 2" - Sizing 7						
-	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					
	Reverse Bias	No Selection					

5.8. Repeat Step 5.5. Select the right edge then click **Apply** and change the parameters as per below.

De	Details of "Edge Sizing 3" - Sizing 4					
-	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				
	Reverse Bias	No Selection				

5.9. Click on **Generate Mesh** button and then select **Mesh** under **Outline**. The mesh should look like the mesh below.



5.10. Change the edge names by selecting the edge, then right clicking on the edge and selecting **Create Named Selection** from the drop down menu. Name left, right, bottom and top edges as *inlet*, *outlet*, *axis* and *wall* respectively then click **OK**. Your outline should look same as the figure below.

	-			Selection Name		$\times$
	Go To	•				
	Clear Generated	Data On Selected Bodies				
	Parts	•		Selection		×
	Filter Tree Pased	On Visible Redies				
		Un visible bodies		Apply selected geometry		
	las restrict View			<ul> <li>Apply geometry items of s</li> </ul>	ame:	
	Set			Size		
	Restore Default			Туре		
	( Zoom To Fit (F7)			Location X		
	🕼 Image To Clipbo	ard (Ctrl+ C)		Location Y		
	Cursor Mode View	*		Location 7		
0.020	👰 Look At					
	🙏 Create Coordina	te System		Analy To Commenda	- March Nadar	
	Create Named Select All (Ctrl+	election	Tim		g Mesh Nodes	
	Undate Geometr	ny	1			
		y non source	J	ОК	Cancel	
		Outline				
		Filter News	_			
		Filter: Name	•			
		j 🖉 < 🏷 🖽 🧧				
		Project				
		Geometry	1			
		🗸 🛄 Suri	face Bod	у		
		E Coordinat	te Syste	ms		
		⊡…, 🦓 Mesh				
			e Meshin o Sizioa	Ig		
		, √ ⊷, Lug , Ø, Eda	e Sizing	2		
		Edg	e Sizing	3		
		🖃 🖤 🎲 Named Se	elections			
		wall				
		axis	; +			
		Jin out	et			
		v				

5.11. 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...** 

Project	Sch	nemat	tic													
	•		Α				•		в		 -		с			
	1	$\Theta$	Geometry				1		Mesh		1	💶 Flu	uent			
	2	00	Geometry	~		-	2	00	Geometry	<ul> <li>Image: A second s</li></ul>	2	🍓 Se	tup 🕯		- h.	
			Dine		_		3		Mesh	1	3	So So	lution 1		Edit	
			ripe												Register Startup Scheme File	
								n	10n-uniform			turt	oulent		Import Eluent Case	•
															Import Fluent Case And Data	•
															Duplicate	
															Transfer Data From New	•
															Transfer Data To New	
														7	Update	

6.2. Check Double Precision and click OK.

Fluent Launcher (Setting Edit Only)		—		×
<b>ANSYS</b>	Flu	ient	Launc	her
Dimension 2D 3D Display Options Display Mesh After Reading Workbench Color Scheme Do not show this panel again	Options Double Precision Processing Options Serial Parallel			
軠 Show More Options				
OK Ca	ancel Help	•		

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



6.4. **Tree > Setup > General**. Choose **Axisymmetric** option shown below.

~	🍓 Setup	General
	🔲 General	Mach
	> 🗄 Models	Mesh
	> 💩 Materials	Scale Check Report Quality
	> 🗇 Cell Zone Conditions	Display
	) J Boundary Conditions	Calua
	🛃 Dynamic Mesh	Solver
	🥏 Reference Values	Type Velocity Formulation
$\sim$	ᆒ Solution	Pressure-Based     Absolute
	🗞 Solution Methods	O Density-Based O Relative
	💰 Solution Controls	
	> 🖪 Monitors	Time 2D Space
	Report Definitions	Steady O Planar
	🔊 Report Files	Transient     Axisymmetric
	🔊 Report Plots	O Axisymmetric Swirl
	$P_{t=0}$ Solution Initialization	
	> 🗐 Calculation Activities	
	誟 Run Calculation	Gravity Units
$\sim$	🥪 Results	
	> 🧐 Graphics	
	> 🎞 Animations	Help
	N to Plate	



6.5. Tree > Setup > Models > Viscous (right click) > Model. Select Standard k-epsilon.

6.6. Tree > Setup > Materials > air > Create/Edit. Change the Density and Viscosity as per your experimental data. Click Change/Create then close the window. Use the flow property values in your EFD Lab2. If you only know the room temperature when you conducted EFD Lab 2, you can use the following website to easily get the corresponding density and dynamic viscosity for air: <a href="http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html">http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html</a>. Close the dialog box when finished.

Tree		Task Page	3	</th <th>🛀 Mesh</th> <th>×</th> <th></th>	🛀 Mesh	×	
× 🎕	Setup General Models Materials Cell Zone Conditions Soundary Conditions Dynamic Mesh	Materials Materials Fluid air Solid aluminum					ANSYS
~ 🕥	Create/Edit Materials						×
	Name		Material Type			Order Materials by	
	air		fluid		•	Name	
	Chemical Formula		Fluent Fluid Materials			O Chemical Formula	
			air		•	Fluent Database	
			Mixture			User-Defined Database	
>	Properties		none		~		
	Density (kg/m3)	constant	▼ Edit				
Ť 🌹		1.1885					
>	Viscosity (kg/m-s)	constant	▼ Edit				
>	- -	1.8396e-05					
>			]				
> 50-							
							_
		Chang	pe/Create Delete Close	Help			

If the room temperature is 24°, the density and viscosity are the values shown in the above panel. **NOTE:** viscosity used in ANSYS is the **dynamic viscosity** ( $kg/m \cdot s$ ), **NOT kinematic viscosity** ( $m^2/s$ )

6.7. Tree > Setup > Cell Zone Conditions > Zone > surface\_body. Change type to fluid and click OK. Select Material Name as air and click OK.

C:turbulent Fluent@L-COE208-0	1.engin.uiowa.edu [axi, dp, pbns, ske] [ANSYS Aca	demic Res				
Ø & @ ≥ ≠ ∧	₽→ MO=					
File Setting Up Domain	Setting Up Physics User-Defined Solving	Postpro				
🚌 🐼 🙀 🔽 Poly	Combin: Delete Appen: Mesh	🛃 Dynam				
h/S	eparat _ lace Mes Overset	Mixing P				
() Ualit in orc	Jjacency ctivate lace Zon	Turbo To				
Mesh	Zones Interfaces	Mesh I				
Tree	Task Page	×				
∽ 🍓 Setup	Cell Zone Conditions					
> B Models	Filter All 🔻					
> 🔠 Materials						
Cell Zone Conditions	Zone	_				
> J∓ Boundary Conditions	surface_body	_				
Beference Values			Eluid			
✓ incrementer values						
Solution Methods			Zone Name			
💥 Solution Controls			aufaca bady			1
> Monitors			sunace_body			
Report Files			Material Name air	▼ Edit		
Report Plots						
> 🗟 Calculation Activities						
Run Calculation Results			Mesh Motion	Fixed Values		
> 😗 Graphics	Phase Type ID		Porous Zone			
> Plots	mixture v fluid v 2		Poforonco Framo Mach	Mation Deraus Zone	2D Fan Zono	Embaddad LEC
> 🙀 Reports	Edit solid les		Melerence Frame Mesh	n Motion Porous Zone	SD Fall Zolle	Embedded LES
> B Parameters & Customization	Parameters Operating Conditions					
	Display Mesh				OK	Cancel Help
	Porous Formulation					

6.8. Tree > Setup > Boundary Conditions > inlet > Edit... Change velocity magnitude as per your experimental data. Change other parameters as below and click **OK**. You must calculate the inlet velocity u, which is uniform, based on the flow rate Q (m<sup>3</sup>/s) you computed in EFD Lab2, and cross

section area of the pipe $\pi D$	$^{2}/4$ , i.e. $u = 4Q/$	$\pi D^2$ .
----------------------------------	---------------------------	-------------

✓     Setup       □     General       >     Models       >     Office       >     Office       >     Office       >     Boundary Conditions       Ø     Dynamic Mesh        Reference Values       ✓     Solution	Boundary Conditions Filter All  Zone axis inlet interior-surface_body outlet wall	<ul> <li></li></ul>
Solution Methods Solution Controls Solution Controls Report Definitions Report Files Report Plots ↓ <sub>10</sub> Solution Initialization A Calculation Activities fun Calculation	Wan	Velocity Inlet  X Zone Name Inlet  Momentum Thermal Radiation Species DPM Multiphase Potential UDS  Velocity Specification Method Magnitude, Normal to Boundary  Reference Frame Absolute
<ul> <li>Results</li> <li>Graphics</li> <li>Animations</li> <li>Plots</li> <li>Reports</li> <li>Parameters &amp; Customization</li> </ul>	Phase Type ID mixture Velocity-inlet 7 Edit Copy Profiles Parameters Display Mesh Periodic Conditions	Velocity Magnitude (m/s) 33.81 constant  Supersonic/Initial Gauge Pressure (pascal) 0 constant  Turbulence Specification Method Intensity and Length Scale Turbulent Intensity (%) 0.01 P Turbulent Length Scale (m) 0.000294 P OK Cancel Help

6.9. Tree > Setup > Boundary Conditions > outlet > Edit... Change gauge pressure as per experimental data. Change other parameters as below and click OK. You need to transform the four pressure tap pressure values from "feet water" to "Pascal" and input pressure tap #4 value as the "outlet pressure". For example, if the pressure tap #4 has value of 0.2502 feet water, you need input 747 Pascal.

✓     Setup       □     General       >     B       >     Models       >     □       Cell Zone Conditions       >     □       Dynamic Mesh        Reference Values       ✓     ■	Boundary Conditions Filter All Zone axis inlet interior-surface_body outlet	
Solution Methods Solution Controls Monitors Report Definitions Report Files Report Plots	wall	Zone Name Outlet  Momentum Thermal Radiation Species DPM Multiphase Potential UDS Backflow Reference Frame Absolute
<ul> <li>Calculation Activities</li> <li>Run Calculation</li> <li>Results</li> <li>Graphics</li> <li>Graphics</li> <li>Animations</li> <li>Plots</li> <li>Reports</li> </ul>	Phase Type ID mixture pressure-outlet Edit Copy Profiles	Gauge Pressure (pascal 119 constant   Backflow Direction Specification Method Normal to Boundary  Average Pressure Specification  Target Mass Flow Rate  Turbulence  Specification Method K and Ension
Parameters & Customization	Parameters Display Mesh Periodic Conditions Help	Backflow Turbulent Kinetic Energy (m2/s2) 1 constant  Backflow Turbulent Dissipation Rate (m2/s3) 1 constant  OK Cancel Help

6.10. Tree > Setup > Boundary Conditions > wall > Edit... Change roughness height as per experimental data. Change other parameters as below and click OK. Input the pipe roughness of the pipe you used in EFD Lab2. For example, user inputs 0.000025 m for smooth pipe.

<ul> <li>✓ Setup</li> <li>General</li> <li>&gt; Models</li> <li>&gt; Materials</li> <li>&gt; Cell Zone Conditions</li> </ul>	Boundary Conditions Filter All V Zone		×
III Boundary Conditions     Dynamic Mesh     Reference Values     Solution     Solution Methods	axis inlet interior-surface_body outlet wall	Zone Name wall Adjacent Cell Zone	
<ul> <li>Solution Controls</li> <li>Solution Controls</li> <li>Report Definitions</li> <li>Report Files</li> <li>Report Files</li> <li>Calculation Activities</li> <li>Run Calculation</li> <li>Solution</li> <li>Graphics</li> <li>Graphics</li> <li>Reports</li> <li>Parameters &amp; Customization</li> </ul>	Phase     Type     ID       mkture     wall     5       Edit     Copy     Profiles       Parameters     Operating Conditions     Operating Conditions	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Wall Motion Stationary Wall Moving Wall Shear Condition No Silp Specified Shear Specified Shear Specularity Coefficient Marangoni Stress Wall Roughness Roughness Height (m) 0.000025 Roughness Constant Roughness Constant 0.5 Constant Wall Rough	Potential
	Help	OK Cancel Help	

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

<ul> <li>✓ Setup         <ul> <li>General</li> <li>B Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> </ul> </li> <li>Dynamic Mesh         <ul> <li>Reference Values</li> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> </ul> </li> </ul>	Boundary Conditions Filter Al  Zone axis inlet interior-surface_body outlet wall	5 ↔ @ ⊕ ∧
<ul> <li>Solution Controls</li> <li>Solution Controls</li> <li>Monitors</li> <li>Report Definitions</li> <li>Report Plots</li> <li>Solution Initialization</li> <li>Calculation Activities</li> <li>Run Calculation</li> <li>Results</li> <li>Solution S</li> <li>Plots</li> <li>Plots</li> <li>Reports</li> <li>Parameters &amp; Customization</li> </ul>	Phase       Type       ID         mixture       wall       5         Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions	Operating Conditions     Pressure     Operating Pressure (pascal)     97225.9     P      Reference Pressure Location     X (m) 0     P     Y (m) 0     P     OK Cancel Help

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

✓ Setup	Boundary Conditions
> 🗄 Models	Filter All 👻
A Materials     Gell Zone Conditions	Zone
> 🗍 💷 Boundary Conditions	axis
💋 Dynamic Mesh	inlet
Reference Values	interior-surface_body
✓ iii Solution	outlet
🗞 Solution Methods	wall
💰 Solution Controls	
> 🔊 Monitors	
Report Definitions	
Report Files	
Report Plots	
↓ Solution Initialization	
> 🗊 Calculation Activities	
💈 Run Calculation	
✓	
> (9) Graphics	Dhase Tree ID
> III Animations	
> C Plots	
>   Reports	Edit Copy Profiles
> B Parameters & Customization	Parameters
	Operating Conditions
	Display Mesn Periodic Conditions

6.13. **Tree > Setup > Reference Values**. Change parameters as per experimental data. Parameters in blue are constant and should be entered as seen below. Parameters in red are from experimental data.





6.14. Tree > Solution > Solution Methods. Change parameters as per below.

6.15. Tree > Solution > Monitors > Residuals. Right click Residual and select Edit... Change convergence criterion to 1e-06 for all five equations as per below and click OK.

🗸 🍓 Setup	Moni	itors			2		
General	Pioli				- <b>↓</b>		
> 🗄 Models	Residi	uais, Statistic and Fo					
> 💩 Materials	Resid	duals - Print. Plot	<u> </u>				
> 🗇 Cell Zone Con	ditions State	Statistic - Off					
> J∓ Boundary Con	ditions						
Dynamic Mesh	ו						
Reference Value	Ies	ato 💌 Edit Do	lata				
<ul> <li>Solution</li> <li>Solution Math.</li> </ul>	ode U	Luit De	lete		( Carlor and Carlor an		
Solution Meth	ous Surfac	ce Monitors			0		
<ul> <li>Monitors</li> </ul>							
Residual					L Z		
🔊 Drag					101		
🕓 Lift							
🔊 Moment					10-		
Surface	Crea	te Edit Del	ete		P 35.		_
Residual Monito	rs						×
Options		Equations					
Print to Console		continuity	$\checkmark$	$\checkmark$	1e-06		^
Plot		x-velocity		$\checkmark$	1e-06		
Window		y-velocity			1e-06		
1 🗘 🔿	urves Axes	k			10-06		
V I Iterations to Dist					10-00		
		epsilon			1e-06		~
		Residual Values			Convergenc	e Criterion	
		Normalize		Iterations	absolute	-	
Iterations to Store				5 🜲			
1000 ≑		Scale					
		Compute Local	Scale				
	OK	Plot Renormalize	Cano	el Help			

6.16. **Tree > Solution > Solution Initialization**. Change gauge pressure and axial velocity as per experimental data and click **Initialize**. Parameters in blue are constant and should be entered as seen below. Parameters in red are from experimental data.





> 🍓 Setup	Run Calculation
Solution Methods	Check Case Update Dynamic Mesh
<ul> <li>Solution Controls</li> <li>Monitors</li> <li>Report Definitions</li> <li>Report Files</li> <li>Report Plots</li> <li>Solution Initialization</li> <li>Calculation Activities</li> </ul>	Number of Iterations 1000 Profile Update Interval 1 Data File Quantities Acoustic Signals
<ul> <li>Kun Calculation</li> <li>Results</li> <li>Graphics</li> <li>Animations</li> <li>Plots</li> <li>Reports</li> <li>Parameters &amp; Customization</li> </ul>	Calculate

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



6.19. File > Save Project. Save the project.

# 7. Results

Please read exercises before continuing.

#### 7.1. 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					Х
Format EPS JPEG PPM	Coloring File Typ Color Rast Gray Scale Vector Monochrome		rpe Resolution ster Width 960 🜩 ctor Height 720 🜩		
<ul> <li>PostScript</li> <li>TIFF</li> <li>PNG</li> <li>HSF</li> </ul>	Options Landscape Orie White Backgrou	ntation und	Win imp	dow Dump Command ort -window %w	
O VRML Window Dump					
	Save Apply	Preview	Clo	se Help	

#### 7.2. Displaying Mesh

Setting Up Domain > Display. Select all Surfaces you wish to be visible and select Display then click Close.



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.



Name the File as "*CFD Lab 1 Mesh*", navigate to the CFD Lab 1 file you created and save it in that file. Close the **Save Picture** window.

## 7.3. Displaying and Saving Residual History

To display the residuals click <b>Tree &gt; Solution</b> > <b>Monitors</b> .	Right click <b>Residuals</b> select <b>Edit</b> ,	click
Plot then click Cancel.		

<ul> <li>Setup</li> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> <li>Monitors</li> <li>Residual</li> <li>Drag</li> <li>Lift</li> <li>Moment</li> </ul>	Monitors Residuals, Statistic and F Residuals - Print, Plot Statistic - Off Create  Edit D	iorce Monitors Helete		5 ↔ @ ⊕ ≯ @	SIGUE conti -x-vel -y-vel -k epsil	e-021	
Image: Surface       Image: Surface         Image: Surface       Image: Su	Console Console Curves Axes s to Plot Store Curves	Equations continuity x-velocity y-velocity k epsilon Residual Values Normalize Scale Compute Local Plot Renormalize	Scale	Iterati 5 cel H	∑ ∑ ∑ ons €lp	1e-06         1e-06         1e-06         1e-06         1e-06         2000         Convergence Criterion         absolute	× *

You can save this picture the same way you saved the mesh. Name it "*CFD Lab 1 Residuals*" and save it to the folder you created on the network drive.

#### 7.4. Plotting Centerline Pressure Distribution

To plot results, click **Results** > **Plots**. Right click **XY Plot**, then click **edit...** To plot the Centerline Pressure Distribution, copy the parameters as per below. Next click **Load File...**, select the file named *"pressure-EFD-turbulent-pipe.xy"* and click **OK**. Then click **Plot**. This .xy file can be downloaded from the class website below. The plot should look similar to the one below.



You can save this picture the same way you saved the mesh. Name it "*CFD Lab 1 Center Line Pressure with EFD*" and save it to the folder you created on the network drive.

## 7.5. Plotting Centerline Velocity Distribution

To plot results, click **Results** > **Plots**. Right click **XY Plot**, then click **edit**... To plot Centerline Velocity Distribution, copy the parameters as per below and click **Plot**.

Solution XY Plot				×
Options	Plot Direction	Y Axis Functio	on	
Node Values	X 1	Velocity		-
Position on X Axis	Y 0	Axial Velocity	1	<b>~</b>
Position on Y Axis     Write to File	Z 0	X Axis Functio	on	
Order Points		Direction Vec	ctor	-
	_	Surfaces	[1/5]	<b>) =</b> =
File Data [0/0] 🐚 🔳 💻	Load File Free Data	axis inlet interior-surfa outlet wall	ice_body	
		New Sullac	C .	
Plo	Axes Cu	Irves Close	e Help	

Save the picture as you did for the mesh and call it "*CFD Lab 1 Centerline Velocity Distribution*" and save it in the folder you created.

7.6. Plotting Wall Shear Stress Distribution

To plot results, click **Results** > **Plots**. Right click **XY Plot**, then click **edit**... To plot the Wall Shear Stress Distribution, copy the parameters as per below and click **Plot**.

Solution XY Plot				×
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0	Y Axis Function Wall Fluxes Wall Shear Stres X Axis Function Direction Vecto Surfaces	ss r [1/5]	• • •
File Data [0/0] 😉 🔳 🖃	Load File Free Data	axis inlet interior-surface_ outlet wall	_body	
Plot	Axes Cu	rves Close	Help	

Save the picture as you did for the mesh and call it "CFD Lab 1 Wall Shear Stress Distribution" and save it in the folder you created.

## 7.7. Plotting Profiles of Axial Velocity at All Axial Locations (3 pages long)

To plot Profiles of Axial Velocity at All Axial Locations with EFD Data, click **Setting Up Domain** > **Surface** > **Create** > **Line/Rake...** 

💶 C:turbulent Fluent@L-COE208-01.engin.uiowa.edu [axi, dp, pbns, ske] [ANSYS Academic Research] — 🛛 🛛 🛛						$\times$
۵ 🗞 🧌 🕫 🖇 💧						
File Setting Up Domain S	Setting Up Physics User-Defined	Solving	Postprocessi	ing Viewing 4	▶ ⊘ ⊘ ॥ .	ANSYS
Display 💽 📩 🎦	Transform Make Polyhedra Smooth/Swap n Units Reorder	Zones Ir	nterfaces M	Mesh Adapt Iodels	Surface	
Me	sh	•	•		<b>.</b>	
ree	Task Page		×	🦳 🔁 Static Press	🕂 Create 🖕	
🖻 🍓 Setup	Blata		S	axis	Zone	ANSYS
Solution	PIOLS		÷‡+	•Expe	Partition	Asidenia
<ul> <li>Results</li> </ul>	Plots			1.80e+031	Imprint	
> 😗 Graphics	XY Plot		Q		· · · · · · · · · · · · · · · · · · ·	
> Animations	Histogram File		Ð,	1.60e+03	Point	
	Profiles:		*	1.000 000	Line/Rake	
Profile Data	Profile Data - Unavailable			1.400+031	Plane	
Interpolated Data	Interpolated Data		(1)	1.406103	Quadric	
FFT				1 20e+03	lso-Surface	
XY Plot			્	1.200700	lso-Clin	
Histogram			达。	1.000+02		
Reports Parameters & Customization			600	tatic	Transform	

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

Line/Rake Surface	×
Options Type Line Tool Reset	<ul> <li>Number of Points</li> <li>10 +</li> </ul>
End Points	
x0 (m) 0.5238	x1 (m) 0.5238
y0 (m) 0	y1 (m) 0.02619
z0 (m) 0	z1 (m) 0
Select Point	s 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**.

Next you should open the file **"axialvelocityEFD-turbulent-pipe.xy"** with a text editor and replace the velocity values with the values you recorded experimentally. Save the file.

Go back to XY plot. Click **Load File...** and select "**axialvelocityEFD-turbulent-pipe.xy**", which is the file you just modified, and click **OK**. Change Parameters as per below. Make sure to select **inlet** as well.

Solution XY Plot				×	
Options ✓ Node Values ✓ Position on X Axis ○ Position on Y Axis ○ Write to File ○ Order Points File Data [1/1] ► ■ = Velocity Magnitude	Plot Direction X 0 Y 1 Z 0	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vect Surfaces outlet wall x=100d x=10d x=20d x=40d x=60d	n n for [7/11]		
Plot Axes Curves Close Help					

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**.

Curves - S	olution XY Plot	×			
Curve # 0 💽 Sample	Line Style Pattern Color foreground  Weight 1	Marker Style Symbol (*) Color foreground Size 0.3			
Apply Close Help					





Save the picture as you did for the mesh and call it "*CFD Lab 1 Axial Velocity at All Axial Locations with EFD Data*" and save it in the folder you created.

#### 7.8. Plotting Velocity Vectors

Click **Results** > **Graphics** > **Vectors** > **Set Up...** To plot the velocity vectors at the region flow begin to become fully developed, copy the parameters as per below and click **Display**. Zoom into the region where the flow is almost fully developed.

1 A			5								
> 💘 Setup	Graphics and Anin	nations									
> Mi Solution	Graphics		÷		4.18e+01						
✓ ♥ Results	Mash		۲	_	4 08e+01						
Graphics	Contours		<u> </u>								
Miesn	Vectors		⊕ <b>.</b>		3.98e+01						
Contours	Pathlines				3 806+01						
Pay Vectors	Particle Tracks				5.036.01						
Particle Tracks					3.79e+01						
> Animations	Set Up		<b>W</b>	_	3.600+01			→			-
> I Plots			Q .								
> 🔂 Reports				_	3.59e+01						
> 🍰 Parameters & Customizatio	Vectors		×	-	3 50e+01						
	Ontions	Vactors of			0.000 0						
		Velocity			3.40e+01						
	Auto Range	Color by			3.30e+01						
	Clip to Range	Velocity			0.01.01						
	Auto Scale	Velocity			3.210+01						-
	Draw Mesh	Axial Velocity	•		3.110+01		>	——->	>	—	
C	tida	Min (m/s) Max (m/	s)		0.040	F		N N	r N	P N	
		22.37938 41.758	19		-3.01e+01	t	>				
	cale Skin	Surfaces [0/11]	) = =		2.920+01>		>	<del>}</del>		— — <del>— —</del>	
	0.75 0	axis	^		2 820+01						
	Vector Options	inlet			2.026.0	t	>				
	vector Options	interior-surface_body			2.72e+01	0	<u>_</u>	6	▶	6	
	Custom Vectors	line-9			2.63e+01	l		V			
		outlet	~		2.000 0.						
S	Surface Name Pattern	New Surface -			2.53e+01						
	Match	New Surface +			2.43e+01						
		Surface Types [0/31]	> = =		0.00-1.04						
		axis	^		2.330+01						
		clip-surf			2.24e+01			0		0.02 (m)	
		exhaust-fan									
_		fan	¥								
	<b>D</b> 1	Computer Close 11.1									
	Display	Compute Close Help		ocitv							
			y-ver	locity							
			k								
											_

Save the picture as you did for the mesh and call it "*CFD Lab 1 Velocity Vectors at The Region Flow Begin to Become Fully Developed*" and save it in the folder you created. Close the **Vectors** window.

#### 7.9. Plotting Axial Velocity Contours

Click **Results** > **Graphics** > **Contours** > **Set Up...** To plot the Contours of Axial Velocity, 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 Lab 1 Contours of Axial Velocity*" and save it in the folder you created.

## 7.10. Exporting Axial Velocity Profile at x=100d

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				×	
Options	- Plot Direction	Y Axis Function			
✓ Node Values	X 0	Velocity		-	
Position on X Axis	Y 1	Axial Velocity		•	
	Z 0	X Axis Function			
Order Points		Direction Vecto	r	-	
		Surfaces	[3/11]	> = =	
File Data [0/0] 🗎 🔳 🖃		wall		^	
	Load File	x=100d			
	Free Data	x=10d			
		x=20d			
		x=40d			
		x=60d			
				*	
		New Surface	•		
· · · · · · · · · · · · · · · · · · ·					
Write Axes Curves Close Help					

Name the file "CFD Lab 1 Developed Axial Velocity Profile" and leave the Files of Type: as XY Files.

## 7.11. Exporting Wall Shear Stress Distribution

Options	- Plot Direction -	Y Axis Function	l		
Node Values	X 1	Wall Fluxes		•	
Position on X Axis	YO	Wall Shear Stress			
	Z 0	X Axis Function			
		Direction Vector			
		Surfaces	[1/11]	> ≡ =	
	Load File Free Data	line-9 outlet <b>wall</b> x=100d x=10d x=20d x=40d New Surface	Ŧ	~	
Write	Axes 0	Curves Close	Help		

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

Name the file "CFD Lab 1 Wall Shear Stress Distribution" and leave the Files of Type: as XY Files.

### 7.12. Normalizing Velocity Profile

- Open the Excel sheet you created in CFD Pre-Lab 1 from your folder on the network drive.
- Open the "CFD Lab 1 Developed Axial Velocity Profile.xy" file you just created in a text editor program.
- Copy the velocity and position data from the text program and paste it into the Excel sheet, make sure to label it accordingly.
- Normalize the velocity profile by dividing every velocity by the Centerline Velocity (Max Velocity).
- Edit the plot to include only AFD Velocity Profile (Laminar) and CFD Velocity Profile (Turbulent).
- File > Save As, name the file "*CFD Lab 1 Developed Axial Velocity Profile.xlsx*" and save it into your CFD Lab 1 folder on the network drive.

## 8. Exercises

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

# **Simulation and Validation of Turbulent Pipe Flow**

## 8.1. Validation of CFD using EFD data

Follow instructions and setup the simulation case using the same flow conditions as in your EFD Lab2, use the values in instruction part for other parameters, iterate the simulation until it converges. Find the relative error between EFD friction factor and friction factor computed by CFD, which is computed by:

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

The equation for the friction factor is C=8\* $\tau/(r*U^2)$ , where 'C' is the friction factor, ' $\tau$ ' is wall shear stress, 'r' is density and 'U' is the inlet velocity. Use the wall shear stress value at the end of the pipe, found in the exported wall shear stress distribution. Use ANSYS Fluent to show the comparison between CFD and EFD on: fully developed axial velocity profile and pressure distribution along the pipe.

**Figures need to be saved:** (1) residual history, (2) centerline pressure with EFD, (3) profiles of axial velocity at all axial locations (x/d=10, 20, 40, 60, 100) with EFD data, (4) centerline velocity distribution, (5) wall shear stress distribution, (6) contour of axial velocity and (7) velocity vectors (pick up the region where flow begins to become fully developed).

Data need to be saved: (1) wall friction factor in the developed region, (2) developing Length

#### 8.2. Normalized developed axial velocity profile

- 8.2.1. Export the axial velocity profile data at x=100d only.
- 8.2.2. Use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude, which is the maximum value of the profile. Plot the normalized turbulent and laminar flow

#### 8.3. Questions need to be answered in CFD Lab 1 report

- 8.3.1. Where is the mesh clustered? Near the wall or near the axis? Why?
- 8.3.2. In the developed region, is the pressure constant? Where are the minimum and maximum pressure locations? What does the axial velocity profile look like? Where does it reach maximum and minimum?
- 8.3.3. Where is the developing region? How can you determine the length of the developing region? (hint: centerline velocity). What is the difference for axial velocity profile between "developing" region and "developed" region?
- 8.3.4. Compare the normalized axial velocity profiles for laminar and turbulent pipe flows at x=100d, discuss the difference of axial velocity gradient (du/dy) on the wall, which is larger?
- 8.3.5. What are the correct boundary conditions (velocities u, v and pressure) on the pipe wall and on the pipe centerline?
- 8.3.6. Summarize your findings and use the knowledge you learned from classroom lectures, textbooks and ANSYS figures to answer questions.