Verification of Laminar and Validation of Turbulent Pipe Flows

ME:5160 Intermediate Mechanics of Fluids CFD LAB 1 (ANSYS 19.4; Last Updated: Aug. 9, 2019)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Sung-Tek Park, 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 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 "CFD Process" for pipe flow

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, respectively. The schematic of the problem and the parameters for the simulation are shown below.

Table 1 - Main Particulars				
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/Mesh	Grid/Mesh	# of Divisions			
	Туре	Х	R		
8	Uniform	453	45		
7		319	32		
6		226	23		
4		113	11		
3		80	8		
2		56	6		
0		28	3		
Т	Non-uniform	564	15		

Table 2 - Grids

Experimental, analytical results, and simulation results 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 laminar flow simulation and turbulent flow simulation using uniform grid 8 and non-uniform grid respectively. For the rest of the simulations, the grid and simulation setups have been provided with workbench uploaded on the class website:

- (1) go to "http://user.engineering.uiowa.edu/~me_160/"
- (2) go to "CFD Labs" tab
- (3) go to "CFD Lab1: Pipe Flow" tab
- (4) download "CFD Lab1 Workbench" by clicking "Download"

Please refer to the exercise at the end of the manual to *determine* the data and figures that need to be saved before you analyze (postprocess) any result. Even though the manual shows every possible step for analyzing the data at Section 7 & 8, only certain subsections (e.g. 7.3, 7.4, 7.7) will be required for each exercise.

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

Table 3 - Simulation Matrix

All analytical data (AFD) and experimental data (EFD) needed for the comparison with laminar and turbulent flow CFD results, respectively, can be downloaded from the class website again:

(1) go to "http://user.engineering.uiowa.edu/~me_160/"

- (2) Click RMB on "axialvelocityAFD-laminar-pipe.xy" and select "Save link as..."
- (3) Click RMB on "axialvelocityEFD-turbulent-pipe.xy" and select "Save link as..."
- (4) Click RMB on "pressure-EFD-turbulent-pipe.xy" and select "Save link as..."

3. Open ANSYS Workbench Template



3.1. Start > All Programs > ANSYS 19.1 > Workbench 19.1

- **3.2.** You can ignore all the pop-ups by clicking "Cancel" if you see any.
- **3.3. Toolbox** > Component Systems. Click and Drag & Drop [Geometry], [Mesh] and [Fluent] components to Project Schematic as per below.



3.4. Click on the drop down arrow and select **Rename**. Change the names as per below to avoid any confusion during the work.



3.5. Create connections between component as per below. To make connections, click and drag the [Geometry ?] box to the [Mesh ?] box, and the [Mesh ?] box to the [Setup ?] box as per below.



3.6. File > **Save As**. Save the workbench file to H drive (i.e. <u>home.iowa.uiowa.edu</u> drive). The H drive is shared between the computers in engineering labs.



4. Geometry Creation

- D 1 🖪 Fluent 🚺 Fluent 1 🧼 Mesh 1 2 🍓 Setup 🛛 🖓 🧧 24 2 🎡 Setup se New SpaceClaim Geometry... 3 🎧 Solution 👕 🖌 3 巓 Solution 😤 🖌 New DesignModeler Geometry... laminar (1e-5) laminar (1e-6) Import Geometry Duplicate Transfer Data From New ۲ Transfer Data To New ۲ 1 Update 2 🎡 Setup ? 🛓 Update Upstream Components 3 👔 Solution ? 🖌 Refresh 2 turbulent (1e-6) Reset ab Rename Properties Quick Help Add Note
- **4.1.** Right click **Geometry** and select **New DesignModelerGeometry...** (Since all the geometries are linked together, only one geometry creation is required)

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

🏧 A: pipe - DesignModeler				
File Create Concept Tools	Units View Help			
] 🔄 🔒 📕 📫 🗍 🏵 Undo	✓ Meter	R R 🖪 🖬 🕯	8+ ≣∭ S+Q+	t 🔍 🔍 🔍
	Centimeter			
XYPlane 🔻 🛧 None	Millimeter	🖤 Share Topology	Parameters Reference	🚓 Revolve 🚺
BladeEditor: 🔏 Import BGD 🔮	Micrometer	FlowPath 🛛 🥩 Blade	💋 Splitter 🚽 VistaTFExport	🛰 ExportPoint
皮下る = (涼皮	Inch		💽 🖗 🖋 📾 🧭	
Tree Outline	Large Model Support	P	Graphics	
→ → A: pipe → XYPlane → XZPlane	✓ Degree Radian			
0 Parts, 0 Bodies	Model Tolerance			

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

🙀 A: pipe - DesignModeler	
File Create Concept Tools Units View Help	
🛛 🛃 📕 🖾 🗍 💬 Undo 📿 Redo 🛛 Select: 🆎 🍢 💽 💽 💽 🔍	}- ∭∭]\$ ় ় @ @ @ @
■ • ■ • <i>ト</i> • <i>ト</i> • <i>ト</i> • <i>ト</i> • <i>K</i> • <i>K</i>	
🛛 XYPlane 🛛 🔻 🖌 None 🚽 💹 🗍 🧚 Generate 🖤 Share Topology	😰 Parameters 🛛 🕞 Extrude 💏 Revolve 🔞
BladeEditor: 🆓Import BGD 📳 Load BGD New Sketch 😂 FlowPath 🥜 Blade	🖋 Splitter 🚽 VistaTFExport 📉 ExportPoints
遼丞 ड≣ (函慶 ▼	• 🖗 🖋 📾 🧭
Tree Outline 4	Graphics
□-√@ A: pipe ↓ ★ XYPlane	

4.4. Right click Sketch1 and select Look at.

Tree Outline		џ
A: pipe	ne Q Always Show Sketch THIDE Sketch	
	Show Dependencies Delete Generate (F5) allo Rename (F2)	

4.5. Enable the auto constraints option to pick the exact point as below. Select **Sketching** > **Constraints** > **Auto Constraints** > make sure Cursor is selected.

File Create Concept Tools Units View Help	File Create Concept Tools Units View Help
27 🔜 🔜 📫 🖸 Undo 📿 Redo 🛛 Select: 🌇 🎝 🕇 🕞 🕞 🕞	🖉 🔲 🗐 👘 💭 Unda 🖉 Reda 🛛 Selecti 🌇 🦭 🖓 🗔 🗔 🖿 🖉
xmane v sketchi v s j venerate vanare ropology	XYPlane V Sketch1 V Concernent Share lopology
BladeEditor: 🎬 Import BGD 👔 Load BGD 🖾 Load NDF 📑 FlowPath 🌈 Blade	BladeEditor: 👹 Import BGD 🛛 Load BGD 🛛 Load NDF 🛛 🚍 FlowPath 🌈 Blade
<u>医</u> E S ≡ (京 E -	▶ 巫 S = (寂 ⊭
Tree Outline 🛛 🕈	Sketching Toolboxes 📮
⊡…,√@ A: pipe	Draw
□····································	Modify
ZY Diane	Dimensions
X YZPlane	Constraints
👘 0 Parts, 0 Bodies	TTT Fived
	777 Horizontal
	Vertical
	✓ Perpendicular
	→ Tangent
	Coincident 🖓
	Midpoint
	ா Symmetry
	(Concentric
	A Equal Radius
	ve≌Equal Length
	Rute Constraints
	CON AUTO CONStraints
	Cutting
	Settings
Sketching Modeling	Sketching Modeling
Details View 9	Details View 4
Details of Sketch1	Details of Sketch1
Sketch Sketch1	Sketch Sketch1
Sketch Visibility Show Sketch	Sketch Visibility Show Sketch
Show Constraints? No	Show Constraints? No

4.6. Select **Sketching** > **Draw** > **Rectangle**. Create a rectangle geometry as per below. The cursor will show "P" when it is on the origin point.



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



4.8. Click on **H2** under **Details View** and change it to 7.62*m*. Click on **V1** and change it to 0.02619*m*. Always omit units ("m" for this time) when you put in values.

D	etails View	ф.
Ξ	Details of Sketch1	
	Sketch	Sketch1
	Sketch Visibility	Show Sketch
	Show Constraints?	No
Ξ	Dimensions: 2	
	H2	7.62 m
	V1	0.02619 m
Ξ	Edges: 4	
	Line	Ln7
	Line	Ln8
	Line	Ln9
	Line	Ln10

4.9. Concept > Surfaces From Sketches and select Sketch1 from the Tree Outline and hit Apply on Base Objects under Details view.



4.10. Click Generate. This will create a surface.



- 4.11. File > Save Project. Save project and close window.
- **4.12.** If you see the lightning sign next to **Geometry** in the workbench then right click on the **Geometry** and click **Update** as shown below. If you don't see the check mark after the update, then you may have made a mistake when you were creating the geometry.



5. Mesh Generation

5.1. Right click on Mesh and select Edit.



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



5.3. Select the pipe geometry by clicking anywhere on the pipe surface, then click the yellow box that says "No Selection" and click **Apply**. (From now on, rotate the view to xy-plane by clicking z-axis of 3D axis located at right bottom of the screen. You can drag and drop with right mouse button to zoom in. You can press F7 to restore the view.)



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



5.5	. Right	click	on Mesh	then s	select l	[nsert >	Sizing.

					0					
File Ho	ome	Mesh	Display	Selec	tion	Auto	omation			
Duplicate	iut iopy aste i line	× Delete Q Find ₽ <mark>₽</mark> Tree ▼	Generate Mesh	凰 Nan 凃 Coo ☞ Con	ned Se ordinat iment	lection e Systen Inse	i ima n i i i i i i i i i i i i i i i i i i i	ages * ction PI notatio	ane on	<mark>m ft</mark> Units
Outline			्र म् 🗆 >	< §)	Q ()		📦 🌍		0 - 🤞	•
Project* → →	i (B2) ieome ieome iateria iateria) etry Gurface Body als nate System:	s							
	leshi	Insert		•	-	Metho	d			T.
	9 2	Update			1	Sizing				
	۶	Generate I	Mesh		Ų.	Conta	Sizing			
		Preview		×	A	Refine	I	Con	trol size	-relate
	_	Show		•	()	Face N	E	sph	ere or bo	dy of
	2	Create Pin	ch Controls			Match				
		Group All	Similar Childr	ren		Pinch	(1) Pre	ess F1 f	or help.	_
	2	Clear Gene	erated Data			Inflatio	n			
	ab	Rename		F2		Mesh (Connecti	on Gro	up	
		Start Reco	rding		1	Manua	Mech (onned	tion	

5.6. Hold Ctrl and select the top and bottom edge then click **Apply** in the **Details** box for **Geometry** on the right. Specify details of sizing as per below depending on the case.

т			٠			
L	_/ a	m		n	a	r

De	Details of "Edge Sizing" - Sizing 7				
-	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
-	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	453			
-	Advanced				
	Behavior	Hard			
	Capture Curvature	No			
	Capture Proximity	No			
	Bias Type	No Bias			

Turbulent

De	Details of "Edge Sizing" - Sizing 🛛 🖓						
Ξ	Scope						
	Scoping Method	Geometry Selection					
	Geometry	2 Edges					
Ξ	Definition						
	Suppressed	No					
	Туре	Number of Divisions					
	Number of Divisions	564					
Ξ	- Advanced						
	Behavior	Hard					
	Capture Curvature	No					
	Capture Proximity No						
	Bias Type	No Bias					

5.7. Repeat step 5.5. 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.

Details of "Edge Sizing 2" - Sizing 🛛 🖓					
Scope					
Scoping Method	Geometry Selection				
Geometry	2 Edges				
- Definition					
Suppressed	No				
Туре	Number of Divisions				
Number of Divisio	ns 45				
- Advanced					
Behavior	Hard				
Capture Curvature	No				
Capture Proximity	No				
Bias Type	No Bias				

Uniform Grid 8

De	etails of "Edge Sizing 2" - Sizing 🛛 🖓						
-	Scope						
	Scoping Method	Geometry Selection					
	Geometry	1 Edge					
-	Definition						
	Suppressed	No					
	Туре	Number of Divisions					
	Number of Divisions	15					
-	Advanced						
	Behavior	Hard					
	Capture Curvature	No					
	Capture Proximity	No					
	Bias Type						
	Bias Option	Bias Factor					
	Bias Factor	3.1117					
	Reverse Bias	No Selection					

Non-uniform Grid Left Edge

Non-uniform Grid Right Edge

De	Details of "Edge Sizing 3" - Sizing 🛛 🕴 🕂					
-	Scope					
	Scoping Method	Geometry Selection				
	Geometry	1 Edge				
	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	15				
Ξ	- Advanced					
	Behavior	Hard				
	Capture Curvature	No				
	Capture Proximity	No				
	Bias Type					
	Bias Option	Bias Factor				
	Bias Factor	3.1117				
	Reverse Bias	No Selection				



5.8. Click on Generate Mesh button and click Mesh under Outline to show mesh.

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

_				
	Insert	•		
	Go To	· · ·	Selection Na	me X
Ŷ	Hide Body	F9		
	Filter Iree Based On Vis	ible Bodies		
G	Suppress Body		Selection	×
	Isometric View			
*	Set			lasted as a state
Â	Restore Default	H 67	Apply se	lected geometry
Q	Zoom To Selection	7	Apply ge	cometry items of same:
í	Image To Clipboard	Ctrl+C	Size	
	Cursor Mode	cuir c	Type	
	View			
49.	Look At		Loca	tion X
*	Create Coordinate Syste	em	Local	tion Y
	Create Named Selection	N		tion 7
6	Select All	Create Named Selection	(N)	
1	Select Mesh by ID	Create a Name	Selection for the selected geometry entities in the	
ş	Generate Mesh On Sele	I graphical inter selection and	ce (bodies, faces, etc.). You can specify a name for the Apply of an apply of the selected geometry.	Y To Corresponding Mesh Nodes
	Clear Generated Data C			
	Parts	 Press F1 for help. 		OK Cancel
_				

Uniform Grid 8

Non-uniform Grid



5.10. File > Save Project. Save the project and close the window. Update mesh by clicking RMB on Mesh and clicking Update on Workbench.



6. Solve

- Α В D ▼ ▼ C 1 1 1 1 😥 Mesh Eluent 🎾 Geometr 2 🔤 Geometry 🗸 🖌 2 🎯 Mesh 🗸 🎡 Setup 4 Edit.. 3 🕼 Solutio uniform grid 8 pipe Register Startup Scheme File... laminar (1 Import Fluent Case And Data ۲ Import Fluent Case ۲ Duplicate Е • Transfer Data From New 1 1 ۲ E Eluer Mach Transfer Data To New 2 🍓 Setup ۲ 2 🎯 Mesh 2 3 😭 Solutio non-uniform grid Update 9 Update Upstream Components turbulent (Clear Generated Data 🔹 Refresh Reset ab Rename Properties Quick Help Add Note
- 6.1. Right click Setup and select Edit.

6.2. Under options check Double Precision and click OK.

Fluent Launcher 2019 R2 (Setting Edit	Only) — 🗆 🗙
ANSYS	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Do not show this panel again ACT Option Load ACT	Options Double Precision Processing Options Serial Parallel
吾 Show More Options	
ОК С	ancel Help 🔻

6.3. Fold the upper tool box by clicking the button inside the red box to avoid any confusion. For this section 6, the "tree outline" on the left side bar will be used only.



6.4. Tree > Setup > General > Check. You may ignore the *warning* messages if pop up. (Note: If you get an *error* message you may have made a mistake while creating your mesh)

🖉 🕷 🖬 🕾 🦻 🚺 🛤						
<u>F</u> ile Domain Physi	cs User-Defined	Solution	Results	View	Parallel 🔶 🔺	Q 🛛 🖉
Mesh		Zones		Interfaces	Mesh Models	Adapt
Display] Scale ♥ Transform ↓ □ Separa	ne 🚽 📑 Delete ate 🚽 📅 Deactivate	← Append ■ Replace Mesh	✓ Mesh ✓ Overset	Dynamic Mesh	Refine / Coarsen
🖋 Units Check- Quality - 4	Make Polyhedri 👉 Adjace	ency 🕂 Activate	Replace Zone.		Turbo Topology	👓 More 🚽
Outline View	Task Page		× 🗈		Mesh	
Filter Text	General		(?)			
Setup Image: Constraint of the set of the	Mesh Scale Check Display Units Solver Type V Pressure-Based Density-Based Time 21 Steady Transient Gravity	Report Quality elocity Formulation Absolute Relative D Space Planar Axisymmetric Axisymmetric Swirl	Console x-coo Volume : minim tot. Face ar: minim checkin Done.	Ininate: min (m rdinate: min (m rdinate: min (m) statistics: um volume (m3): um volume (m3): a statistics: um face area (m um face area (m g mesh	<pre> Q Q 0 </pre>	 (m) = 7.620000e+00 x (m) = 2.619000e-02



6.5. Setup > General > Solver. Choose an option shown below.

Axis Boundary Condition								
Model	Model Laminar Turbulent							
Variable	u	v	Р	u	v	Р	k	e
variable	[m/s]	[m/s]	[Pa]	[m/s]	[m/s]	[Pa]	$[m^2/s^2]$	$[m^2/s^3]$
Magnitude	-	0	-	-	0	-	-	-
Zero Gradient	Y	Ν	Y	Y	Ν	Y	Y	Y

(above table explains the adaption of axisymmetric condition for the "axis" boundary condition)

6.6. Tree > Setup > Models > Viscous (Laminar) (double click). Select parameters as per below and click OK.



Outline View	Viscous Model	×
Filter Test	Model	Model Constants
Filter Text Setup General Models Multiphase (Off) Faciation (Off) Radiation (Off) Heat Exchanger (Off) Species (Off) Solidification & Melting (Off) Structure (Off) Electric Potential (Off) Electric Potential (Off) Electric Potentials Electric Potentials Electric Potential (Off) Reference Values Named Expressions Named Expressions Solution % Methods	Model Inviscid Laminar Spalart-Allmaras (1 eqn) K-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Enhanced Wall Functions Enhanced Wall Treatment Meated Lachae	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 TDR Prandtl Number 1.3 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number TDR Prandtl Number TDR Prandtl Number
Methods Controls Report Definitions Monitors Cell Registers Ritialization Calculation Activities	Menter-Lechner User-Defined Wall Functions Options Production Kato-Launder Production Limiter	none
Run Calculation Results Surfaces Capabian	ОК	ancel Help

6.7. Tree > Setup > Materials > Fluid > air (double click). Change the Density and Viscosity as per below and click Change/Create. Close the dialog box when finished.

Outline View	Task Page		× 💽		
Filter Text	Materials		(?)		
 Setup ⊗ General ⊗ Models Multiphase (Off) ④ Energy (Off) ④ Energy (Off) Sediation (Off) 	Materials Fluid air Solid aluminum				
Heat Exchanger (Off)	Create/Edit Materials				×
 ↔ ≯ Discrete Phase (Off) 	Name		Material Type		Order Materials by
Solidification & Melting (Off)	air		fluid	•	Name Chaminal Formula
Structure (Off)	Chemical Formula		Fluent Fluid Materials		
Electric Potential (Off)			air	•	Fluent Database
Fluid			none	•	User-Defined Database
() Ar Solid		Properties			_
Cell Zone Conditions		Density (kg/m3)	constant	▼ Edit	
 Boundary Conditions Dynamic Mesh 			1.17		
Reference Values		Viscosity (kg/m-s)	constant	▼ Edit	
Keference Frames Mamed Expressions			1.872e-05		
 Solution Solution Methods Controls Report Definitions ♥ Monitors ♥ Outperiod 			<u> </u>		
Cerr Registers Cerr Reg		Chang	e/Create Delete Close	e Help	

6.8. Tree > Setup > Cell Zone Conditions(Double click) > Zone > surface_body. Change type to fluid, make sure air is selected and click OK.

<u>F</u> ile Domain Phy	ysics User-Defined	Solution Re	sults Vie	ew Parallel	🛶 🔺 🔍 🛛 🔋 📕
Mesh Display i Info , P Units Check, Quality ,	Scale Scale	Zones ne → ➡ Delete ate → ➡ Deactivate ↓ ency ➡ Activate	+ Append ↓ → Replace Mesh ↓ → Replace Zone	Interfaces Mesh M Image: Mesh Image: Dynamic Image: Overset Image: Dynamic <th>odels Adapt c Mesh Refine / Coarsen Planes ************************************</th>	odels Adapt c Mesh Refine / Coarsen Planes ************************************
Outline View	Task Page		8		Mesh
Filter Text	Cell Zone Conditions	(0		
 Setup [®] General [®] Models [®] Models [®] Muthpase (Off) [®] Fairsgy (Off) [®] Viscous (Laminar) [®] Radiation (Off) [®] Assistance (Off) [®] Species (Off) [®] Solidification & Melting ([®] Acoustics (Off) [®] Structure (Off) [®] Structure (Off) [®] Electric Potential (Off) [®] Materials [®] Fluid [®] Fluid [®] Solid [®] Cell Zone Conditions [®] [®]	Zone Filter Text surface_body Fluid Zone Name Surface_body Material Name air Frame Motion Sou Mesh Motion Fixe Porous Zone Reference Frame Mesh M	Edit rce Terms d Values	a Zane Embedded LES	Reaction Source Terms Fi	X X X X X X X X X X X X X X X X X X X
 surface_body (fluid, id=2) Boundary Conditions Dynamic Mesh 		L	OK Cancel Help		
 ☑ Reference Values ☑ I. Reference Frames ☞ Named Expressions ☑ Solution ③ Methods ✓ Controls ☑ Report Definitions ④ Monitors ☑ Cell Registers ☑ Initialization 	Phase mixture Type fluid Edt Parameters Display Mesh	D 2 Profiles	Console Face area a minimum checking m	Q Q	0e-04 9e-02

6.9. Tree > Setup > Boundary Conditions > inlet (double click). Change parameters as per below and click OK.

File Domain Physics User: Defined Solution Results View Parallel Mesh Cones Interfaces Mesh Interfaces Mesh ① Info @@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@@		Lar	ninar flo ^s	W		
Mesh Zones Interfaces Mesh	<u>File</u> Domain Physic	s User-Define	d Solution	Results	View	Parallel
Outine View Task Page Filter Text Image: Sector of the sector	Mesh Display Info - Check- Quality - A	Scale	Zo Combine - Delete Separate - Deact	nes e [+] Append tivate Beplace	Mesh	faces Mesh M esh // Dynamic rerset // Mixing F
Filter Text Boundary Conditions © Setup © Models © Models © Filter Text @ Nocustors (off) © Filter Text I text Exchanger (Off) © Solidification & Melting (A constarce (off) © Solidification & Melting (A constarce (off) © Filter Text Waterials © Provide Magnitude, Normal to Boundary © Materials © Soliditions © Soliditions © Soliditions © Soliditinetion-suratice body (filidi, id=2)	Outline View	Task Page			20110111	
Setup General	Filter Text	Boundary Conditions		()		
fro Named Expressions	● Setup I General ● Models I Energy (Off) I Electric Potential (Off) I Interior-surface body (fluid, id=2) I Electric Potencounditions I Interior-surface body (fluid, id=2) I Interior-surface body (fluid, id=3) I Interior-surface body (fluid, id=5) I Interior-surface body (fluid, id=5) I Nomic Mesh Reference Values I Reference Values I Reference Values I Reference Frames I Reference Frames </td <td>Zone Filter Text axis interior-surface_body outlet wall Cone Name inlet Momentum Them Velocity Ve Supersonic/Initial Ga Phase Type velocity Edit</td> <td>al Radiation Spec y Specification Method M Reference Frame A looity Magnitude (m/s) ouge Pressure (pascal) OK Inter Porfiles</td> <td>es DPM Mu lagnitude, Hormal to B bsolute .2 Cancel (Help) Cons Done Done</td> <td>Potential oundary sole e area statist ininum face as akinum face as akinum face as</td> <td>UDS V V V V V V V V V V V V V</td>	Zone Filter Text axis interior-surface_body outlet wall Cone Name inlet Momentum Them Velocity Ve Supersonic/Initial Ga Phase Type velocity Edit	al Radiation Spec y Specification Method M Reference Frame A looity Magnitude (m/s) ouge Pressure (pascal) OK Inter Porfiles	es DPM Mu lagnitude, Hormal to B bsolute .2 Cancel (Help) Cons Done Done	Potential oundary sole e area statist ininum face as akinum face as akinum face as	UDS V V V V V V V V V V V V V

Outline View	Task Page 🛞 🚺
Filter Text	Boundary Conditions
Setup Image: Constant of the set of	Zone Filter Text To
	Vélocity Inlet X Zone Name inlet
	Momentum Thermal Radiation Species DPM Multiphase Potential LDS Velocity Specification Method Magnitude, Normal to Boundary * Velocity Magnitude (m/s) 34.08 * Supersonic/Initial Gauge Pressure (pascal) 0 * * Turbulence * * Turbulent Intensity 4%0 0.01 * Turbulent Lingth Scale (m) * *
wall (wall, id=8) wall (wall, id=8) Opnamic Mesh Reference Values A Reference Values A Reference Values Monte Expressions Montors Report Definitions Q Monitors Report S	OK Cancel Help Phase moture * Edt Copy Profiles Parameters Operation conditions

Inlet Boundary Condition									
Model Laminar Turbulent									
Variable	u [m/s]	v [m/s]	P [Pa]	u [m/s] v [m/s] P [Pa] Intensity Length Scale					
Magnitude	0.2	0	-	34.08	0	-	0.01	0.000294	
Zero Gradient	N	N	Y	N	N	Y	N	N	

6.10. Tree > Setup > Boundary Conditions > outlet (double click) or click Edit.... Change parameters as per below and click OK.





Outlet Boundary Condition											
Model	Model Laminar Turbulent										
Variable	u [m/s]	v [m/s]	P [Pa]	u [m/s]	u [m/s] v [m/s] P [Pa] k $[m^2/s^2]$ e $[m^2/s^3]$						
Magnitude	-	-	0	-	-	400	1	1			
Zero Gradient	Y	Y Y N Y Y N Y Y									

6.11. Tree > Setup > Boundary Conditions > wall (double Click) Change parameters as per below and click **OK**. No need to change for laminar cases.

Eile	Domain		Physic	s User-De	sfined	Solution	Re	sults	Vie	w P	arallel	Design	•	
		Mesh					Zones			Interfaces	Mesh	Models	,	Adapt
Display	AT .	ATR.		Scale	😂 Combine	• - m [*] •	elete	Append		Mesh	🛃 Dynan	nic Mesh	Refir	ne / Coars
(i) Infn	Ø	۲	<u>c0</u>	Transform -	Co Separat	- m D	eactivate	B Replace Mest		M Overset	C Mixing	Planes		
C and t	Cherk-	Quality			- Separat	* • ± •	incovoie			Fair Oversec	and and a second	T Home of the		
Units	Checky	Quanty	• 0	Make Polyhedra	• ∲• Adjacen	KY ⊞ A	stivate	Heplace Zone	·		Turbo	Topology	ooo More	9
Outline View				Task Page				8						
Filter Text				Boundary Condi	tions									
Setup				Zona Eilter Text				-						
Generation (Control of the control o	ài la			Cone (Filter Fext			- <u>co</u> c	9						
	5 ultiphare (Of	*		axis										
() En	erav (Off)	~		inlet	and a									
Vis	scous (Lamin	ar)		outlet	soay									
🔨 Ra	diation (Off)			wall										
👫 He	at Exchange	r (Off)												
tů, sp	ecies (Off)			🔼 Wall							×			
💿 🥕 Dis	screte Phase	(Off)		Zone Name										
So So	lidification 8	Melting	(011)	wall										
AD AD	oustics (Off)			Adjacent Cell Zo										
24 Str 1911 Cla	ructure (OII) estric Dotonti			surface hody										
- A Mater	rials	aii (Oli)		sandee_booy										
⊖ ₽ Flu	id			Momentum 1	hermal Radia	stion Specie	s DPM	Multiphase UDS	W	all Film Potential	Structure			
	air			Wall Motion	Motic	n								
📀 🔐 So	lid			Ctationary	wall wall	alation to Ad	incent Coll 2							
😑 🖽 Cell Zo	one Conditio	ins		Maving Wa		verduve to Au	Jocenii Ceni Z	one						
🗄 sur	rface body (fluid, id=3	!)	- moning the	211									
😑 🖽 Bound	dary Conditio	ins		Shear Conditi	n									
🖽 aoi	is (axis, id=7)		No Sin										
ink	et (velocity-in	nlet, id=5;		Coacified 1	bear									
int int	erior-surface	E_body (in	ter	Concularity	Confficient									
a ou	flet (pressur	e-outlet, i	d=6)	O Marganeri	Cheere									
iii wa	ill (wall, id=8	9	_	- Marangoni	Stress									
Dynam Dynam	ance Malues			Wall Roughne	55									
(+) K Refere	ence Frames			Roughporr Ho	abt (m) a					-				
f Name	ence rraines ed Expression	ns		Roughness Hei	gur (m) 0					-				
 Solution 				Roughness C	ionstant 0.5					Ψ.				
% Metho	ods													
🔀 Contro	ols						_							
🔊 Repor	rt Definitions						OK Canc	el Help				L. 🚹 🛛	-	5. 6
• Q Monit	lors					- E								
Cell Ri	egisters			_		_		Console						
🔜 Initialia	zation			Phase Typ	e	1D		V-000	dir	ate: min (m)	- 0.00000	0e+00, max	(=) = 2	619000
Calcula	ation Activitie	is .		mbture 👻 wa	1	* 8		Volume	stat	istics:				
Run Ci	alculation					_		minim	am v	volume (m3):	9.789934e-	06		
Surfac	nes			Edit	Сору	Profiles		maxim	1	rolume (m3):	9.789934e-	-06		
- Junac								tota		orene (ma):		· · · ·		

Laminar flow



Wall Boundary Condition										
Model		Laminar Turbulent								
Variable	u [m/s]	v [m/s]	P [Pa]	u [m/s]	v [m/s]	P [Pa]	k [m²/s²]	e [m ² /s ³]	Roughness	
Magnitude	0	0	-	0	0	-	-	-	2.50E-05	
Zero Gradient	Ν	Ν	Y	Ν	Ν	Y	N	N	-	

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

<u>F</u> ile Domain Phy	vsics User-D	efined Solution	on Resul	ts Vie	w
Mesh Display i Info + Other Quality +	Scale	Separate → ⊟ Separate → ⊟	Zones	Append - Replace Mesh	Interfaces
Outline View	Task Page		1	×	
Filter Text	Boundary Cond	itions	(?)		
 Setup General Models Multiphase (Off) Energy (Off) Fadiation (Off) Radiation (Off) Heat Exchanger (Off) Species (Off) Foisrete Phase (Off) Solidification & Melting (Acoustics (Off) Solid Electric Potential (Off) Solid Electric Potential (Off) Solid Cell Zone Conditions surface body (fluid, id=2) Boundary Conditions axis (axis, id=7) inter(or-surface_body (int 	Zone Filter Text axis inlet interior-surface_ outlet wall Pressure Operating P 97725.9 Reference X (m) 0 Y (m) 0 Z (m) 0	body conditions ressure (pascal) Pressure Location	Gravity Gravity	Console	<u>2</u> Q 🛃
 in outlet (pressure-outlet, i) wall (wall, id=8) Dynamic Mesh Reference Values 	Phase Typ mixture V wa	all • 8 Copy Profiles.		y-coordin Volume stat minimum v maximum v	ate: min (n istics: volume (m3) volume (m3)
Areference Frames for Named Expressions Solution	Parameters Display Mesh	Operating Condition	ns	total v Face area s minimum f maximum f	volume (m3) statistics: ace area (m ace area (m ace area (m

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

		24				
<u>F</u> ile	Domain	Physic	s User-	Defined S	olution	Result
 Displa Info Units. 	IY • Office · Check•	lesh Quality - 4	Scale Transform Make Polyhedra	Image: Separate Image: Separate Image: Adjacency.	Zones Delete Deactivate Activate	"
Outline Vie	2W		Task Page			
Filter Tex	d		Reference Va	ues		?
○ Setup ② G ③ ♥ ♥ № ● ● ● ● ● ● ● ● ● ● ● ● <	eneral Addels Vaterials Cell Zone Condition Soundary Condition axis (axis, id=7) axis (axis, id=7) axis (axis, id=7) interior-surface_ interior-surface_ wall (velocity-inl interior-surface_ wall (velocity-inl wall	is is body (inter -outlet, id=6)	Compute from Ref	erence Values Area (m2) Density (kg/m3) Enthalpy (j/kg) Length (m) Pressure (pascal) Temperature (k) Velocity (m/s) Viscosity (kg/m-s) io of Specific Heats	0.002154869 1.17 0 0.05238 0 288.16 0.2 1.872e-05 1.4	
۱ 👸 ۲ 🎇	vlethods Controls Conort Definitions		Reference Zone			•
	report Deminuons					

Laminar flow

Turbulent flow

<u>F</u> ile	Domain	Physic	s User-	Defined S	olution	Results
Displa Displa Onits.	y • Ø Check•	Mesh	Scale Transform - Make Polyhedra	Separate € Separate € Adjacency.	Zones Delete Deactivate Activate	t
Outline Vie Filter Tex	ew d		Task Page Reference Va	lues		?
- Setup ⊮ G + ♥ № + ≝ №	eneral 10 dels vlaterials		Compute from	erence Values		•
	Cell Zone Conditio	ons ons		Area (m2) Density (kg/m3)	0.002154869	
	axis (axis, id=/ inlet (velocity-ii)) nlet, id=5) a bady (inter		Enthalpy (j/kg)	0	
2	outlet (pressur	e_body (inter e-outlet, id=6)		Pressure (pascal)	0	
	ynamic Mesh	,)		Temperature (k) Velocity (m/s)	288.16 34.08	=
+ 14 F	Reference Frames	DC.		Viscosity (kg/m-s)	1.872e-05	
 Solution No 	n Nethods	115	Rat Reference Zone	tio of Specific Heats	1.4	
0 % F € @ N	Controls Report Definitions Monitors					•

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



Laminar flow

<u>F</u> ile Domain F	Physics	User-Defined	Solution	Results	Vie
Mesh Display Dinfo - Check- Quality -	Scale C Transfor	m 🖕 🖓 Comi m 🗣 🖵 Sepa lyhedra	Zones Dine Delete rate Activate	[+ A e ■ R . ■ R	append teplace Mesh teplace Zone
Outline View Filter Text	Task Pa Solution	ge n Method <i>s</i>		×	
 Setup ③ General ④ Models ④ Models ④ Models ④ Materials ④ Cell Zone Conditions ④ Optimic Mesh ⑦ Reference Values ④ Methods ✓ Controls ⑤ Named Expressions ⑤ Solution ⑤ Methods ♥ Controls ⑧ Report Definitions ④ Calculation Activities ④ Initialization ♥ Calculation Activities ● Run Calculation ● Results ● Graphics ♥ Plots ♥ Animations ● Plots ● Planameters & Customization 	Pressu Schem SIMPL Spatial D Gradien Green- Pressu Secon Turbule Secon Turbule Secon Turbule Secon Turbule Secon Turbule Secon Turbule Secon	re-Velocity Couplin e E Discretization it -Gauss Cell Based e d Order tum d Order Upwind ent Kinetic Energy d Order Upwind ent Dissipation Rate d Order Upwind nt Formulation it Formulation en Flux Formulation ped-Face Gradient CC Order Term Relaxat	rg rg icement ion Options	*	

6.15. Tree > Solution > Monitors > Residual (double click). Change convergence criterion to 1e-6 for all three and five equations as per below for laminar and turbulent cases respectively and click **OK**. (Note: for iterative error study you will need to use 1e-5)

Lammar now										
Outline View	Task Page	×								
Filter Text ⇒ Setup	Monitors Report Definition quantities can be monitored during solution when they are included in Rej Files or Report Plots. Specifying Convergence Conditions allows yo to define stop conditions for the solver based on Report Definition convergence.	port u								
 Image: Boundary Conditions Image: Dynamic Mesh Image: Reference Values Image: Reference Values Image: Reference Frames Image: Reference Frames	Residual Monitors Options Print to Console Plot Window Curves Rerations to Plot	Equations Residual continuity x-velocity y-velocity	Monitor V V V V	Check Com V V	× vergence Absolute Criteria 1e-06 1e-06 1e-06					
 ► Report Files ✓ Report Plots ✗ Convergence Conditions ✔ Cell Registers ♣ Initialization ◆ € Calculation Activities E Run Calculation 	1000 Iterations to Store 1000	Residual Values Normalize Scale Compute Lo	s Itera 5 cal Scale	ations	Convergence Criterion absolute Convergence Conditions					
 Results	ОК	Plot Renorma	alize Cancel	Help						

Turbulent flow

Outline View	Task Page	×			
Filter Text Setup General Models Materials Ellow Conditions Ellow Conditions	Monitors Report Definition quantities can be monitored during solution when they are included in Repo Files or Report Plots. Specifying Convergence Conditions allows you to define stop conditions for the solver based on Report Definition convergence.	rt			
 Dynamic Mesh Defense Volume 	Residual Monitors				×
 Kererence Values L. Reference Frames Solution Methods Controls Report Definitions Monitors Report Files Report Plots Convergence Conditions 	Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Residual continuity x-velocity y-velocity k epsilon	Monitor	Check Convergen	nce Absolute Criteria 1e-06 1e-06 1e-06 1e-06 1e-06
 ✓ Cell Registers initialization ◆ Calculation Activities ○ Results ✓ Surfaces ✓ Graphics 	Iterations to Store	Residual Value Normalize ✓ Scale Compute Lo	Ite	Conservations	nvergence Criterion
Plots Animations Reports	ОК	Plot Renorm	alize	I Help	

Laminar flow





Laminar flow



6.17. Tree > Solution > Run calculation. Change number of iterations to 1000 and click Calculate.



6.18. File > save project. Make sure to save the project for later use.

7. Results

This section shows how to analyze your results in Fluent. You do not need to do all of the analysis for every case. Please refer to exercises at the end of this manual to determine what analysis you need to do for each simulation.

7.1. Saving Picture

File > Save Picture. Your current display can be saved as a picture file by adjusting formats or resolutions like below and by clicking **Save**. Use this function whenever you need to save pictures for the report.

<u>F</u> ile	Domain	P	hysic	s User-I	Defined	So	ution	
Refresh	Input Data							Zones
Recorde	ed Mesh Operations			Scale		Combine 🖕	H	Delete
Save Pro	oject)	<u>_</u>	Transform 💂		Separate 🖕	Ē	Deactivate
Reload		ity 🖵	4	Make Polyhedra	4.	Adiacency		Activate
Sync Wo	orkbench		Ť	Task Dasa	1 .			
Read				тазк Раде				
Write				Run Calculation	1			
		-						
Import				Check Case		Update Dy	namic	Mesh
Export								
Solution	Files			Options				
Interpol	ate			Data Sampl	ing for S	Steady Statis	tics	
EM Map	oping I	•		Sampling Interv	/al	anling Option	-	
FSI Map	ping I	•		1		iping Option	s	
Save Pic	ture	1		Iterations Sa	mpled	0		
Data File	Quantities							
Preferer	Save	Picture	J	Number of Iterat	ions	Reporting I	nterva	al
Start Pa	de			1000	-	1		-
blancha	ge	-		Profile Update In	terval			
Close W	ithout Save			1	-			
Close Fl	uent			Data File Quant	ities	Acoustic S	ignals	
€ F	Run Calculation			<u> </u>			5	
Results	i Surfaces							
(+) 🔮 G	raphics			Calculate				



7.2. Displaying Mesh

Setting Up Domain > **Display.** Select all the surface you want to display. Lines and points you create can be displayed here as well.



*Tips

Zoom in: Click mouse wheel and create a rectangular that starts from upper left to lower right. Zoom out: Click mouse wheel and create a rectangular that starts from lower right to upper left. Move: Move the mouse with holding both LMB and RMB

7.3. Plotting Residuals



Refer 6.15. Click **Plot** next to **Ok.** Residual plot for laminar case is at below as an example.

7.4. Creating Points

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

<u>F</u> ile	Domaii	n	Physi	cs User-D	efined So	olution	Results	; Vie	w Pa	arallel	Design			
		Mesh				Zones			Interfaces	Me	sh Models	Adapt	Surface	
Display	· 👧		" 🖬	Scale	😂 Combine	→ → Delete	- - -	Append 🚽	🔆 Mesh	🛃 Dyr	namic Mesh	Refine / Coarsen	🕂 Create 🖕	
(i) Info	- 🧐		9 4	Transform 🚽	Dg Separate	▼	. "	Replace Mesh	Overset	⊃¢ Mix	ing Planes		Zone	1
🖌 Units	Check	, Quali	ity 👻 🧳	Make Polyhedra	分 Adjacency	. 🕂 Activate	-6-	Replace Zone		🗇 Tu	rbo Topology	•••• More 👻	Partition	
Outline View	w			Task Page			×						Imprint	esiduals
								Re	siduals				Point	
Filter Text				Run Calculation			\bigcirc						Line/ Point	
 Setup 				Check Case	Undate D	vnamic Mesh		contin			1e+00	7	Plane	
+ @ M	nerai odels					,		x-veic	city			1	Quadric	
🕘 🗳 м	laterials			Options				y-velo	ocity		1e-01		Iso-Surface	
(+) ⊞ Ce	ell Zone Cono	ditions		Data Sampli	ng for Steady Stat	istics						1	Iso-Clip	
	ynamic Mesh	I		Sampling Interve	al Sampling Optio	100					1e-02		Transform	
	ererence Valu	les		1	Bamping Optio	115								

Point Surface							
New Surface Nam	New Surface Name						
point-1							
Reference Frame							
global		•					
Options	Coordinates						
Point Tool	x0 (m) 7.62						
Reset	y0 (m) 0						
	z0 (m) 0						
Select Point with Mouse							
Create Close Help							

Point Name	x0	y0
point-1	7.62	0.000
point-2	7.62	0.005
point-3	7.62	0.010
point-4	7.62	0.015
point-5	7.62	0.020
point-6	7.62	0.021
point-7	7.62	0.022
point-8	7.62	0.023
point-9	7.62	0.024
point-10	7.62	0.025

7.5. Creating Lines

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

🚺 Line/Rake Su	rface		×				
New Surface Na	me						
x=10d							
Options Line Reset	Type Line	•	Number of Points				
End Points							
x0 (m) 0.5238		x1 (m) 0.5	238				
y0 (m) 0) y1 (m) 0.0	2619				
z0 (m) 0		z1 (m) 0					
Select Points with Mouse							
[Create	Close	р				

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	5.2380	0	5.2380	0.02619

7.6. Plotting Velocity Profile

Tree > Results > Plots > XY Plot (double click). Select inlet, outlet, and the lines you created and change setting as per below then click Plot.

Solution XY Plot				×
XY Plot Name				
xy-plot-3				
Options		Plot Direction	Y Axis Function	
Node Values Position on X Axis Position on Y Axis Write to File Order Points		X 0 Y 1 Z 0	Velocity Axial Velocity X Axis Function Direction Vector	
File Data [0/0]		Load File Free Data	Surfaces Filter Text axis inlet interior-surface_body outlet point-1 wall x=100d x=10d x=20d x=40d x=60d	×
	Save/Plot	Axes) Curv	New Surface 🖕 es) Close Help	

Tree > Results > Plots > XY Plot (double click) > **Curves.** For Curve # 0 select the Line Style **Pattern, Line Style Color** as per below and click **Apply**. Repeat this for all the curves 1 through 7.

xy-plot-3								
Options		Plo	t Direction		Y Axis Function			
✓ Node Values		x	0		Velocity		-	
Position on X Axis	s	Y	1	\Box	Axial Velocity		-	
Position on Write to File	Curves - Solution XY Plo	t	0		· · · · · · · · · · · · · · · · · · ·	×		
Order Point Curv	e # Line Style			Mai	ker Style			
0	Pattern			Syr	nbol			/][=,
File Data [0/0] Sam	ple	▼ (*) ▼			•			
	Color			Col	or			
	orange		-	or	ange	•		
	Weight			Siz	e			
	1			0.:	3			
				_				
		\pph	Close	He	lφ			
					x=60d			
					New Surface			

Download the experimental data for the simulation from the class website:

 $(\underline{http://user.engineering.uiowa.edu/~me_160/CFD\%20Labs/Lab1/axialvelocityAFD-laminar-pipe.xy})$

 $(http://user.engineering.uiowa.edu/~me_160/CFD\% 20 Labs/Lab1/axialvelocity EFD-turbulent-pipe.xy)$

Tree > Results > Plots > XY Plot (double click) > **Load File**. Select "axialvelocityAFDlaminar-pipe.xy" (if laminar) or "axialvelocityEFD-turbulent-pipe.xy" (if turbulent) downloaded and click **Plot**.

Solution XY Plot				\times			
XY Plot Name						X	
xy-plot-3							
Options	Plot Direction	Y Axis Function				••••	
✓ Node Values	X 0	Velocity		•			
Position on X Axis	Y 1	Axial Velocity	Select File				? ×
Position on Y Axis	20	X Axis Function	Look in:	H:\9. 1	TA	- 000	🤼 🔃 🗏
Order Points		Direction Vector	S My Com	nputer	axialvelocityAFD-laminar-pipe.xy		
File Data [1/1]		Surfaces Filter Text	a sungtpa	rk			
	Load File	axis					
Velocity Magnitude		inlet					
	Free Data	interior-surface_body					
		point=1					
		wall	XY File	axialvelocit	vAED-laminar-nine.xv		ок
		x=100d		axiarreioen	gra o laminar piperxy		
		x=10d	Files of type:	XY Files (*	*.xy)		 Cancel
		x=20d					
		x=40d	Filter String				Filter
		x=000					
		Harry Cruefer an					Remove
		new surface _	H:/9. TA/axial	velocityAFC)-laminar-pipe xv		
	Save/Plot Axes Curve	s Close Help	Thy 3. Thy datur	relocityAre			
	453 2D wall faces, zon	e 8, binary.					
	20884 nodes, binary.		L				
Dot	2000% noue flags, binary ne.						
201							
Wr:	iting " gzip -2cf > SYS-	1-00559.dat.gz"					
Re-	iting temporary file C+\U	eerel SUNCTE-1 Apple	te\locel\Tem	n\flntaz	-61564		



Result for laminar flow is presented as an example below.

7.7. Plotting Static Pressure Profile at Centerline

Tree > Results > Plots > XY Plot (double click). Change Y function to Pressure... and select axis then click Plot.

Solution XY Plot			×			
XY Plot Name						
xy-plot-5						
Options	Plot Direction	Y Axis Function	_			
✓ Node Values	XI	Pressure	*			
Position on X Axis	YO	Static Pressure	•			
Position on Y Axis	Ζ [0	X Axis Function				
Write to File		Direction Vector	-			
	For turbulent	Surfaces Filter Text	-			
	Load File	axis	~			
Pressure Profile (H:/9. TA/pressure-EFD	-tur	inlet	_			
For turbulent	Free Data	outlet				
		point-1				
		wall				
		x=100d				
	•	New Surface 🚽				
Save/P	Plot Axes Curve	es) Close Help				

For the turbulent case, download the experimental data for the simulation from the class website: <u>http://user.engineering.uiowa.edu/~me_160/CFD%20Labs/Lab1/pressure-EFD-turbulent-pipe.xy</u>

(Turbulent case continued) Tree > Results > Plots > XY Plot (double click) > Load File. Select "pressure-EFD-turbulent-pipe.xy" downloaded and click Plot.



Result for laminar flow is presented as an example.

7.8. Plotting Velocity at Centerline

Tree > Results > Plots > XY Plot (double click). Change Y function to Velocity... and Axial Velocity. Select axis then click Plot. Change Plot Direction as below if necessary.

Solution XY Plot					×
XY Plot Name					
xy-plot-5					
Options		Plot Direction	Y Axis Function		
✓ Node Values		X 1	Velocity	•	
Position on X Axis		YO	Axial Velocity	•	
Position on Y Axis		20	X Axis Function		
Order Points			Direction Vector	•	
File Data [0/0]]	Surfaces Filter Text		
		Load File	axis		
			inlet		
		Free Data	interior-surface_body		
			point-1		
			wall		
			x=100d		-
			New Surface 🚽		
	Save/Plot	Axes) Curve	es Close Help		

Example for the laminar case is presented.



7.9. Exporting Wall Shear Stress Values

Tree > Results > Plots > XY Plot (double click). Change Y function to **Wall Fluxes...** and **Wall Shear Stress**. Select **wall** then click **Write to File** to enable **Write**. Click **Write** to 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.

Solution XY Plot					×
XY Plot Name					
xy-plot-6					
Options		Plot Direction	Y Axis Function		
✓ Node Values		X 1	Wall Fluxes	-	
Position on X Axis		Y 0	Wall Shear Stress	*	
Position on Y Axis		20	X Axis Function		
Write to File Order Points			Direction Vector	•	
File Data [0/0]			Surfaces Filter Text		=
		Load File	inlet		-
			interior-surface_body		
		Free Data	point-1		
			wall		
			x=100d		-
			New Surface 🚽		
	Write	Axes Curve	s) Close Help		

7.10. Plotting Velocity Vectors

Tree > Results > Graphics > Vectors (double click). Change the vector parameters as per below and click **Display**.

Vectors	;	×
Vector Name		
vector-1		
Options	Vectors of	
✓ Global Range	Velocity	۳.
✓ Auto Range	Color by	
Clip to Range	Velocity	٢
Auto Scale Draw Mesh	Axial Velocity	-
	Min (m/s) Max (m/s)	
Style	0.00887987 0.3994213	
3d arrow		
Scale Skip		×
0.2 0 Vector Options Custom Vectors Colormap Options	axis inlet interior-surface_body outlet point-1 surface_body wall x=100d x=100d x=20d x=40d x=60d	
	New Surface	
Sav	ve/Display Compute Close Help	



Result of laminar flow is presented as an example.

7.11. Plotting Velocity Contours

Tree > Results > Graphics > Contours (double click). Change the parameters as per below and click **Display**.

Contours	×
Contour Name	
contour-1	
Options	Contours of
✓ Filled	Velocity
✓ Node Values	Axial Velocity
Contour Lines	Min (m/s) Max (m/s)
Global Range	0 0.3994114
Auto Range	
Clip to Range	Surfaces Filter Text
Draw Profiles	axis
Draw Mesh	inlet
	interior-surface_body
Coloring	point-1
Banded	surface_body
O Smooth	wall
	x=100
Colormap Options	x=20d
	x=40d
	x=60d
	New Surface 👻
Sa	ve/Display Compute Close Help

Result of laminar flow is presented as an example.



8. V&V Instructions

8.1. V&V Instructions for Velocity Profile

Download CFD Lab 1 Workbench file from class website (http://user.engineering.uiowa.edu/~me_160/)

CFD Lab1: Pipe flow CFD Lab1 Concepts CFD Lab1 Manual (PDF, DOC) CFD Lab1 Workbench (Download) CFD Lab1 V&V Excel sheet (Download)

Click update project button. This will run all the simulation on the workbench file and it may take few minutes.



Right click Solution > Select Edit...



Create reference points by following 7.4.

Tree > Results > Plots > XY Plot (double click). Change parameters as per below and click **Write...** Make sure to select points 1 through 10.

Solution XY Plot				>
XY Plot Name				
xy-plot-7				
Options		Plot Direction	Y Axis Function	
✓ Node Values		X 0	Velocity	•
Position on X Axis		Y 1	Axial Velocity	-
Position on Y Axis		ZO	X Axis Function	
Vrite to File			Direction Vector	•
File Data [0/0]		Load File Free Data	Surfaces Filter Text	
	Write	Axes) Curve	New Surface -	

Name file according to which grid solution you are using.

E Select File	?		×
Look in: 📃 H:\9. TA 🔹 🔾 🗿 🗿	G	::	≡
Ny Computer 📋 axialvelocityAFD-laminar-pipe.xy			
sungtpark			
XY File Velocity Profile Grid 4		OK	
Files of type: XY Files (*.xy)	•	Cano	el
Filter String	5	Filte	



Download V&V excel sheet for CFD Lab 1 from class website (http://user.engineering.uiowa.edu/~me_160/)

Open file using Textpad/Wordpad/Notepad, 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	1.4142136							
	Grid 4	Grid 3	Grid 2					
y (m)	Sg1 (FINE)	Sg2 (MEDIUM)	Sg3 (COURSE)	Α	E	ε21	ε32	Rg
0				0.400000	1.000000	0.000000	0.000000	#DIV/0!
0.005				0.385000	0.962500	0.000000	0.000000	#DIV/0!
0.01				0.342000	0.855000	0.000000	0.000000	#DIV/0!
0.015				0.269000	0.672500	0.000000	0.000000	#DIV/0!
0.02				0.167000	0.417500	0.000000	0.000000	#DIV/0!
0.021				0.143000	0.357500	0.000000	0.000000	#DIV/0!
0.022				0.118000	0.295000	0.000000	0.000000	#DIV/0!
0.023				0.092000	0.230000	0.000000	0.000000	#DIV/0!
0.024				0.064000	0.160000	0.000000	0.000000	#DIV/0!
0.025				0.036000	0.090000	0.000000	0.000000	#DIV/0!

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

8.2. V&V Instructions for the Friction Coefficient

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



Tree > Results > Plots > XY Plot (double click). Change parameters as per below and click **Write...**

Solution XY Plot					×
XY Plot Name					
xy-plot-8					
Options		Plot Direction	Y Axis Function		
✓ Node Values		X [1	Wall Fluxes	.	
Position on X Axis		Y 0	Wall Shear Stress	•	
Position on Y Axis		Ζ []	X Axis Function		
Order Points			Direction Vector	•	
File Data [0/0]	FFF		Surfaces Filter Text		=
		Load File	axis		
		Free Data	interior-surface_body		
		The Data	outlet		
			wall		
			New Surface		
	Write	Axes Curves	s Close Help		

			?		\times
G	0	0	ß	::	≣
				0	۲
				Can Filt	cel er
	0			?	?

Name the file according to grid number and save to project folder.

Open file with a text editor such as Textpad/Wordpad/Notepad and copy wall shear stress at the x location of approximately 7m.



	Brost	2	1											
	rgest	1 414213562												
Grids 2.3.4	-8 Sσ1 (EINE)	Sg2 (MEDIUM)	Sg3 (COURSE)	Δ	F	£21	£32	Rσ	Convergence	Ρσ	δ	р	+L10	-110
01100 2,0,1	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
Grids 6,7,8	Sg1 (FINE)	Sg2 (MEDIUM)	Sg3 (COURSE)	А	E	ε21	ε32	Rg	Convergence	Pg	δ	Р	+Ug	-Ug
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
	Pgest	2												
	rg	2												
Grids 0,2,4	Sg1 (FINE)	Sg2 (MEDIUM)	Sg3 (COURSE)	A	E	ε21	ε32	Rg	Convergence	Pg	δ	Р	+Ug	-Ug
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
Grids 4,6,8	Sg1 (FINE)	Sg2 (MEDIUM)	Sg3 (COURSE)	A	E	ε21	ε32	Rg	Convergence	Pg	δ	Р	+Ug	-Ug
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
	Wall						_							
Grid	Shear	c					4							
	Stress													
0		0	-											
2		0												
3		0												
		0												
7		0												
8		0												
c=8*t/(r*U	^2).													

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.

9. Data Analysis and Discussion

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

* 9.1.-9.4. and 9.6. are for laminar flows, 9.5. is for turbulent flows

9.1. Iterative error studies (+6)

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 (grid 4 is given on workbench file which can be found on the class website). 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 to 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, τ 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.	Friction Factor	Relative Error	Friction Factor	Relative Error
	with Convergence	with Convergence	with Convergence	with Convergence
	Limit 1e-5	Limit 1e-5	Limit 1e-6	Limit 1e-6
4				
8				

- Figure need to be reported: residuals history for mesh 8 for two convergent limits.
- Data need to be reported: the above table with values for friction factor and relative error.

9.2. Verification study for friction factor of laminar pipe with refinement ratio $\sqrt{2}$ (+7)

Use the simulations with the meshes for grid 0, 2, 3, 4, 6, 7, and 8 with convergence limit 10^{-6} (Except for mesh 8 other meshes and their setup is provided on the workbench file in the class website). Export friction factor and insert the values into V&V excel sheet (Refer to V&V instructions for friction factor). For each parameter, refer to 'Nomenclature' sheet in V&V excel sheet.

Which set of meshes is closer to the asymptotic range and why (refer to CFD Lecture 1 on class website)? 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 8.1) with the grid uncertainty for 6,7,8, which is higher and what does that mean for mesh 8?

• Figure need to be reported: Table from V&V spread sheet.

9.3. Verification study for friction factor of laminar pipe with refinement ratio 2 (+5)

Use the simulation for the meshes 0, 2, 4, 6 and 8 with convergence limit 1e-6. Results should already be included in V&V spread sheet from previous exercise (Refer to V&V instructions for friction factor). Compared to results in 9.2, which set of meshes is sensitive to grid refinement ratio? Why?

• Figures need to be reported: Table from V&V spread sheet.

9.4. Verification study of axial velocity profile (+7)

Use mesh 4 as the "fine mesh", use grid refinement ratio 1.414 and convergence limit 10⁻⁶. Follow the V&V for axial velocity profile in the results section. Save the figures and discuss if the simulation has been verified. Discuss which mesh solution is closest to the AFD data, give an explanation of why this is the case?

• Figures need to be reported: Figures and tables in the V&V excel sheet.

9.5. Simulation of turbulent pipe flow using Grid T (+9)

Use 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 (Non-dimensionalize the profile by dividing with the reference value (For this exercise, reference value is the centerline velocity (=max. velocity)). Plot the normalized velocity profile in EXCEL and paste the figure into WORD.

- Figures need to be reported: Axial velocity profile with EFD data, normalized axial velocity profile at x=100D with EFD data, 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 reported: Developing length and compare it with that using formula in textbook.

9.6. Comparison between laminar and turbulent pipe flow (+9)

Compare the results of laminar pipe flow using mesh 8 in exercise 9.1 (convergent limit 10⁻⁶) with results of turbulent pipe flow in exercise 9.5. Analyze the difference in normalized axial velocity profile and developing length for laminar and turbulent pipe flows.

- Figures need to be reported: Axial velocity profile with AFD data, normalized axial velocity profile at x=100D with AFD data, normalized axial velocity profile at x=100D comparing laminar and turbulent CFD results, centerline velocity distribution for laminar flow.
- Data need to be reported: Developing length for laminar pipe flow and compared it with that using formula in textbook.

9.7. Questions need to be answered in CFD Lab report

- 9.7.1. Answer all the questions in exercises 9.1 to 9.6
- 9.7.2. Analyze the difference between CFD/AFD and CFD/EFD and possible error sources (+2)

10. Grading scheme for CFD Lab Report

(Applied to all CFD Lab reports)

Section

1	Title Page		5
	1.1 Course Name		
	1.2 Title of report		
	1.3 Submitted to "Instructor's name"		
	1.4 Your name (with email address)		
	1.5 Your affiliation (group, section, department)		
	1.6 Date and time lab conducted		
2	Test and Simulation Design		10
	Purpose of CFD simulation		
3	CFD Process		20
	Describe in your own words how you implemented CFD process		
	(Hint: CFD process block diagram)		
4	Data Analysis and Discussion <a>Fection 9 (Page# 51) for CFD Lab 1		45
	Answer questions given in Exercises of the CFD lab handouts		
5	Conclusions		20
	Conclusions regarding achieving purpose of simulation		
	Describe what you learned from CFD		
	Describe the "hands-on" part		
	Describe future work and any improvements		
		Total	100

Additional Instructions:

- 1. Each student is required to hand in individual lab report.
- 2. Conventions for graphical presentation (CFD):
 - * Color print of figures recommended but not required
- 3. Reports will not be graded unless section 1 is included and complete