To use ANSYS Fluent in your house, please use VDI (See below Link) https://etc.engineering.uiowa.edu/help-desk/how-use/vdi-how-use-virtual-windows-desktop

Verification of Laminar and Validation of Turbulent Pipe Flows

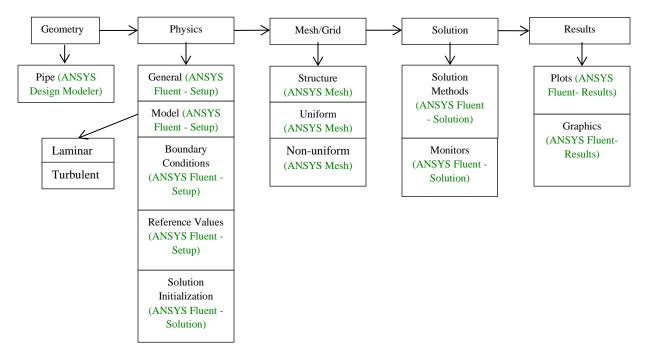
ME:5160 Intermediate Mechanics of Fluids CFD LAB 1 (ANSYS 2022R1; Last Updated: July 19, 2022)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Sung-Tek Park, Christian Milano, 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.

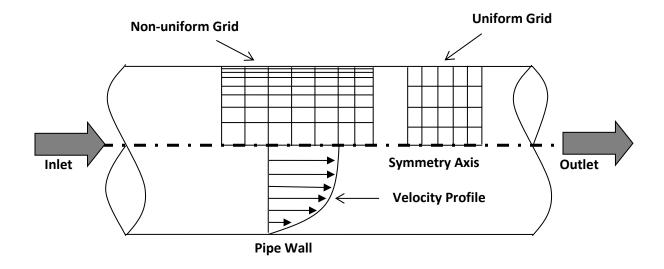


Flow Chart for "CFD Process" for pipe flow

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 Di	visions
Grid/Mesh	Туре	Х	R
8		453	45
7	Uniform	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

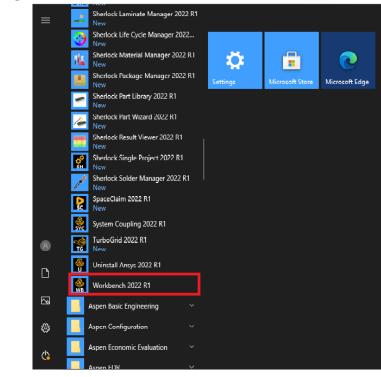
Table 3 - Simulation Matrix

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/"

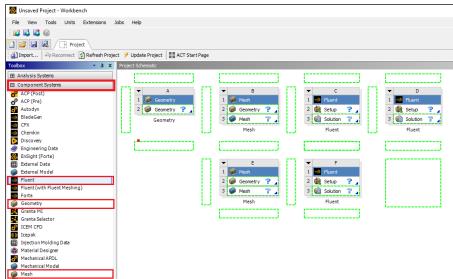
- (2) Click RMB on "axialvelocityAFD-laminar-pipe.xy" and select "Save link as..."
- (3) Click RMB on "axialvelocityEFD-turbulent-pipe.xy" and select "Save link as..."
- (4) Click RMB on "pressure-EFD-turbulent-pipe.xy" and select "Save link as..."

3. Open ANSYS Workbench Template

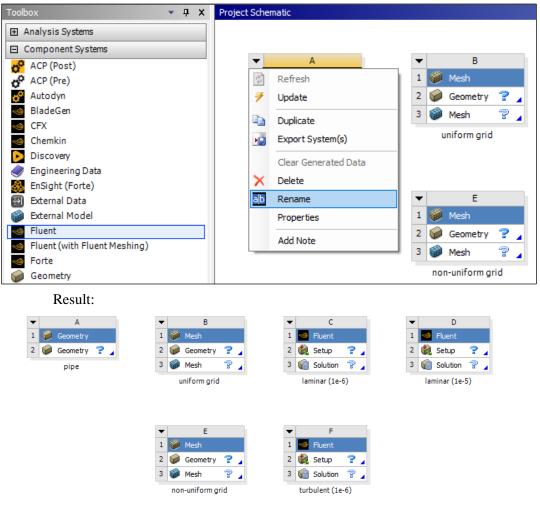


3.1. Start > All Programs > ANSYS 2022 R1 > Workbench 2022 R1

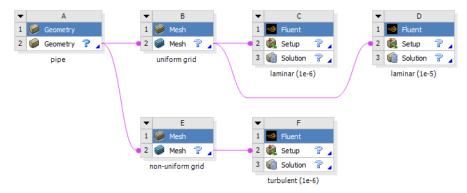
- **3.2.** You can ignore all the pop-ups by clicking "Cancel" if you see any.
- **3.3. Toolbox** > Component Systems. Click and Drag & Drop [Geometry], [Mesh] and [Fluent] components to Project Schematic as per below.



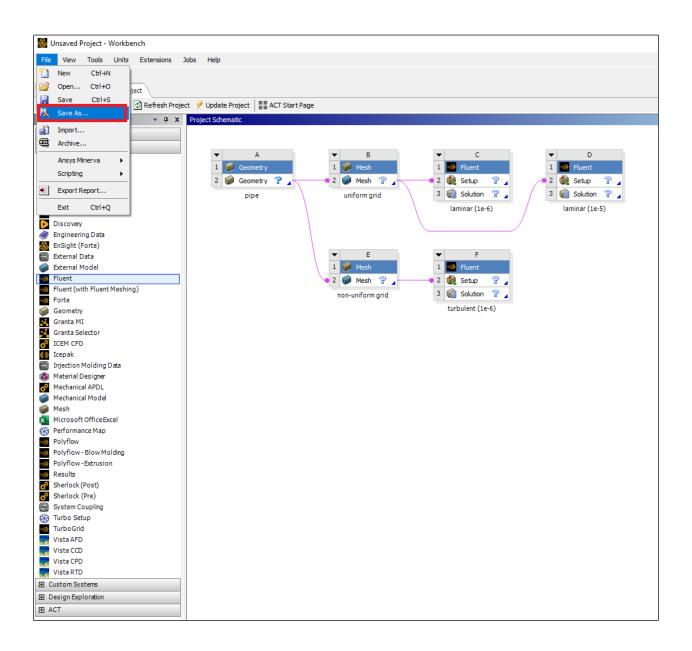
3.4. Click on the drop down arrow and select **Rename**. Change the names as per below to avoid any confusion during the work.



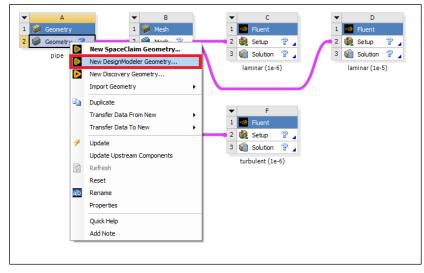
3.5. Create connections between component as per below. To make connections, click and drag the [Geometry ?] box to the [Mesh ?] box, and the [Mesh ?] box to the [Setup ?] box as per below.



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.

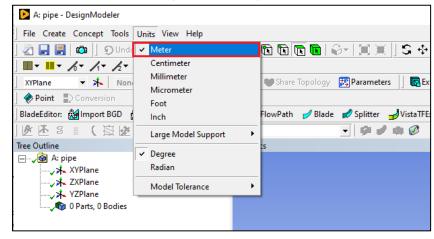


4. Geometry Creation

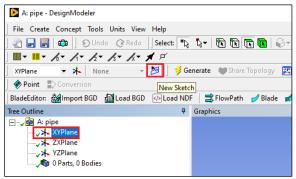


4.1. Right click **Geometry** and select **New DesignModelerGeometry...** (Since all the geometries are linked together, only one geometry creation is required)

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



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



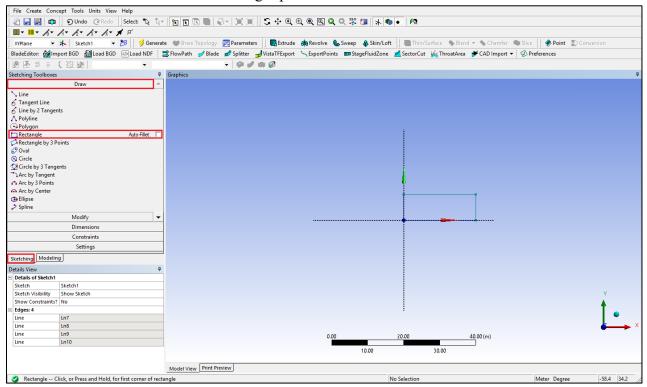
4.4. Right click Sketch1 and select Look at.

Tree Outline		џ
□····√會 A: pipe □·····/赤 XYPla □·····/示 ZXPla □····································	 Always Show Sketch Hide Sketch 	
u Part	Image: Show Dependencies ✓ Delete ✓ Generate (F5) a∏o Rename (F2)	

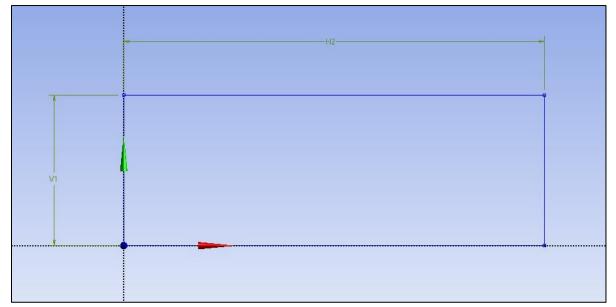
4.5. Enable the auto constraints option to pick the exact point as below. Select **Sketching** > **Constraints** > **Auto Constraints** > make sure Cursor is selected.

File Create Concept Tools Units View Help	File Create Concept Tools Units View Help
🖉 🔙 🛃 🚳 🛛 🕄 Undo 📿 Redo 🗍 Select: 🎠 🏷 🕞 💽 💽 🔍] 🖉 🚽 🛃 🔯] 🖸 Undo 🕜 Redo Select: 🌇 🖏 🕅 🔃 💽 🐚
■ • h · h · h · h · h · h ·] ■ • ∥ • / • / • / • / • /
XYPlane Sketch1 Sketch1 Generate Share Topology	
	XYPlane 🔹 📩 Sketch1 🗨 💆 🛛 🧚 Generate 🖤 Share Topology
BladeEditor: 🆓 Import BGD 🖉 Load BGD 💮 Load NDF 🛛 🚍 FlowPath 🥜 Blade	BladeEditor: 🆓 Import BGD 🖉 Load BGD ↔ Load NDF 🛛 😅 FlowPath 🥜 Blade
皮还 8 ≣ (海遼 -	皮 巫 S ≡ (冱 皮 -
Tree Outline 🛛	Sketching Toolboxes 🛛 🗜
⊟, 🔞 A: pipe	Draw
È, ★ XYPlane Sketch1	Modify
ZXPlane	Dimensions
YZPlane	Constraints
n O Parts, O Bodies	777 Fixed
	1 Vertical
	✓ Perpendicular
	→ Tangent
	€ Coincident
	Midpoint
	A Symmetry
	1/ Parallel
	© Concentric
	A Equal Radius
	v⊉ Equal Length v∯ Equal Distance
	Auto Constraints Global Cursor:
	Settings 🔍
Sketching Modeling	Sketching Modeling
Details View 7	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.



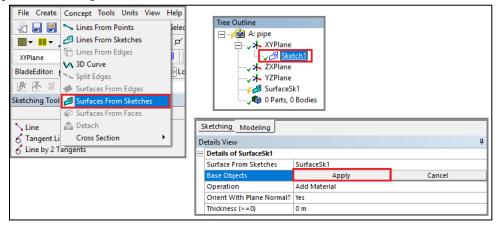
4.7. Select **Sketching** > **Dimensions** > **General**. Click on top edge then click anywhere else. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.



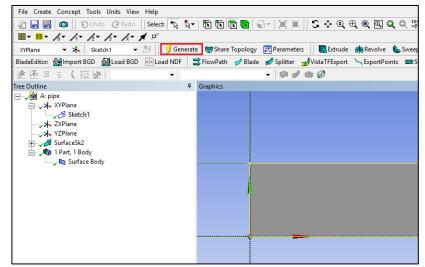
4.8. Click on **H2** under **Details View** and change it to 7.62*m*. Click on **V1** and change it to 0.02619*m*. Always omit units ("m" for this time) when you put in values.

D	etails View	t				
-	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				

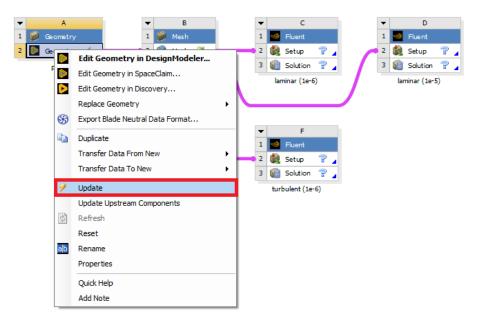
4.9. Concept > Surfaces From Sketches and select Sketch1 from the Tree Outline and hit Apply on Base Objects under Details view.



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.

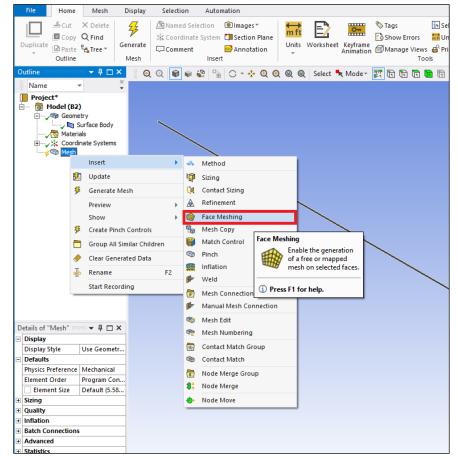


5. Mesh Generation

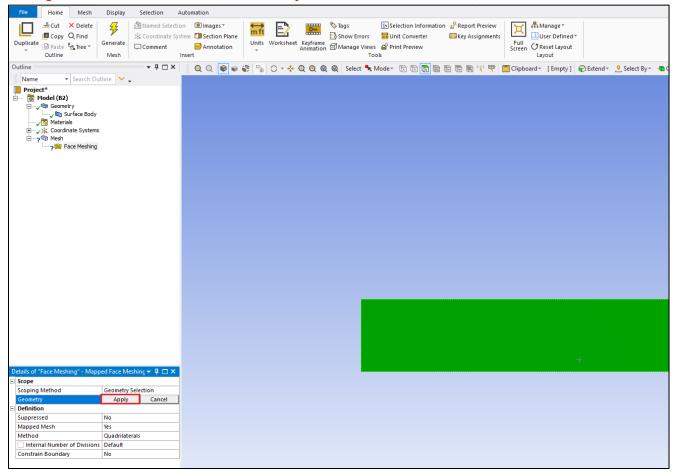
5.1. Right click on Mesh and select Edit.

•	Α	-		в	-	C			•	D	
1	🥩 Geometry	1	🧭 м	1esh	1	🧐 Fluent			1	🧐 Fluent	
2	🝺 Geometry 🗸 🖌	2	🍘 м	1est 🗇		🚵 Caking			2	🍓 Setup 🛛 👕 🧧	1
	pipe		unifor	rm 🤷	Edit			/	з	👔 Solution 💡	1
					Duplicate					laminar (1e-5)	
					Transfer Data F	rom New	•				
					Transfer Data T	o New	•	r -			
				Е 🥖	Update						
		1	🧼 м	1esł	Update Upstrea	m Components					
		2	🧼 м	1esł	Clear Generated	Data					
			non-uni	ifori 🧳	Refresh						
					Reset						
				ab	Rename						
					Properties						
					Quick Help						
					Add Note						

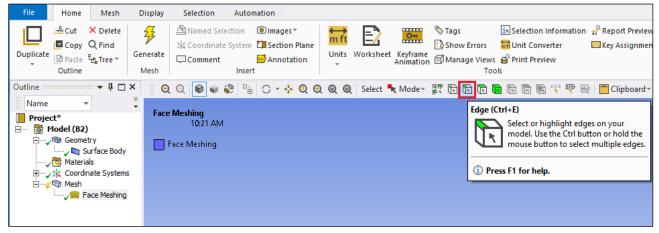
5.2. Right click on **Mesh** then select **Insert** > **Face Meshing**.



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	select	Insert >	Sizing.

					0					
File Ho	ome	Mesh	Display	Selec	tion	Auto	omation			
Duplicate	opy aste	X Delete Q Find La Tree T	Generate Mesh	Anna ∦ Coo ♥ Com	rdinat					<mark>← ft</mark> Units
Outline			े 🕶 🕂 🗖 🗸	(Ə 🖸		📦 🍪		0 - 🔅	Q
	eome E S lateria	try urface Body	5							
ė % ® [eshi	Insert		•	sA.	Metho	d			T.
	羟	Update			i 👔	Sizing				
	۶	Generate M	lesh		Ų.	Conta	Sizing			
		Preview		•	æ	Refine	I		trol size-	
		Show		►		Face N	1		number ere or bo	
	₽	Create Pin	ch Controls		9	Mesh		-		
		Group All	Similar Childr	en		Match	Pre	ss F1 f	or help.	
	۲	Clear Gene	rated Data			Pinch				
	alb	Rename		F2		Inflatio				
		Start Reco	rding				Connecti I Mesh (

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.

T		•	
La	m	In	ar

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			
-	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.

De	Details of "Edge Sizing 2" - Sizing 4									
-	Scope									
	Scoping Method	Geometry Selection								
	Geometry	2 Edges								
-	Definition									
	Suppressed	No								
	Туре	Number of Divisions								
	Number of Divisions	45								
Ξ	Advanced									
	Behavior	Hard								
	Capture Curvature	No								
	Capture Proximity	No								
	Bias Type	No Bias								

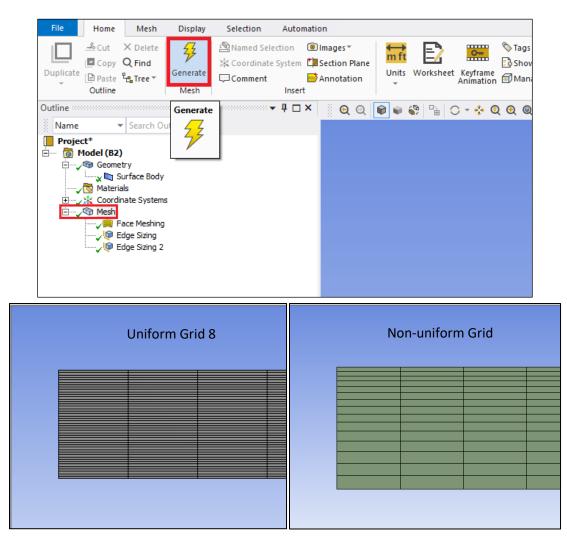
Uniform Grid 8

		ě					
De		Sizing 📍					
-	Scope						
	Scoping Method	Geometry Selection					
	Geometry	1 Edge					
Ξ	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					

Non-uniform Grid Left Edge

Non-uniform Grid Right Edge

De	Details of "Edge Sizing 3" - Sizing 4									
Ξ	Scope									
	Scoping Method	Geometry Selection								
	Geometry	1 Edge								
Ξ	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								



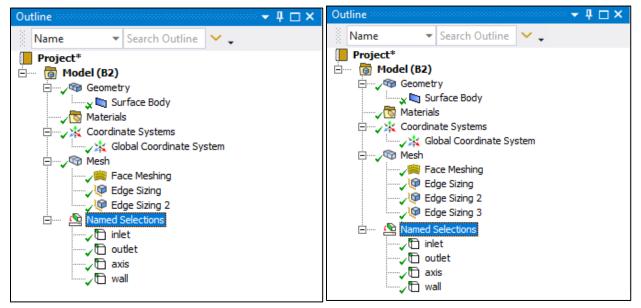
5.8. Click on Generate Mesh button and click Mesh under Outline to show mesh.

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.

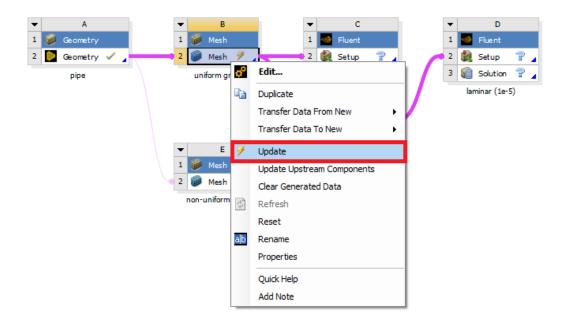
_				
	Insert	•		
	Go To	· · · ·	Selection	on Name X
Ŷ	Hide Body	F9	beleend	
	Filter Tree Based On Vis	ible Bodies		
G	Suppress Body		Select	tion ×
	Isometric View			
*	Set Restore Default	н		
Q	Zoom To Fit	F7		pply selected geometry
Q	Zoom To Selection	z		pply geometry items of same:
()	Image To Clipboard	- Ctrl+C		Size
C.	Cursor Mode	cuir c		Туре
	View			
49.	Look At			Location X
*	Create Coordinate Syste	em		Location Y
	Create Named Selection	N		Location Z
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	graphical inter selection and	ice (bodies, faces, etc.). You can specify a name for the u can specify criteria based on the selected geometry.	Apply 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

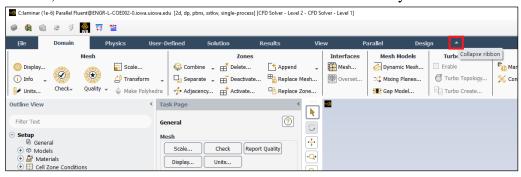
6.1. Right click Setup and select Edit.

•	Α		-	В		-	С		-	D	
1	🥏 Geometry		1	🍘 Mesh		1	Fluent		1	🥌 Fluent	
2	Geometry	× .	2	🍘 Mesh	 	 2	Setup	2	2	🧌 Setup	2
	pipe			uniform g	rid	3 💼	Soli 🥯	Edit			
			▼ 1 2	uniform gi E Mesh	rid Rəf 🖌	lar 1 2 2	Soli Soli minar Set Soli buler	Registe Import Import Duplica Transf Update	er Startup Sch Fluent Case : Fluent Case : ROM ate ier Data From ier Data To Ne e e Upstream Co Generated Da sh	neme File And Data New :w))))
							ab	Renam			
								Proper	ties		
								Quick H	Help		
								Add No	ote		

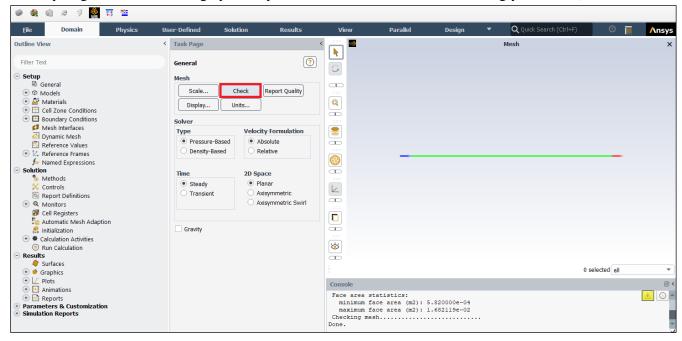
6.2. Under options check Double Precision and click START.

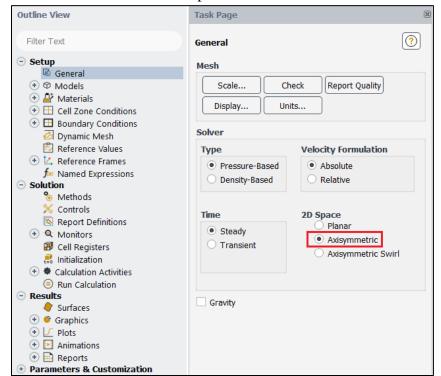
🛸 Fluent Launcher 20	022 R1 (Setting	Edit Only)	—		\times
Fluent Lau	ncher			<mark>/\n</mark>	sys
general-purpose set	ip, solve, and	nd transient industrial aj post-processing capabil s for multiphase, combus	ities of A	NSYS F	luent
		Dimension			
		2D			
		🔾 3D			
		Options			
		Double Precision	ı		
		🕑 Display Mesh Af	ter Read	ing	
		Do not show this	s panel a	igain	
		Load ACT			
		Parallel (Local Mac	chine)		
		Solver Processes		1	\$
		Solver GPGPUs per M	Machine	0	\$
➤ Show More Opt	ions 🗡 Sh	ow Learning Resources			
	Start	Cancel 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)



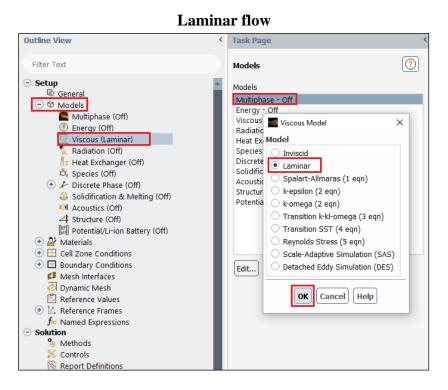


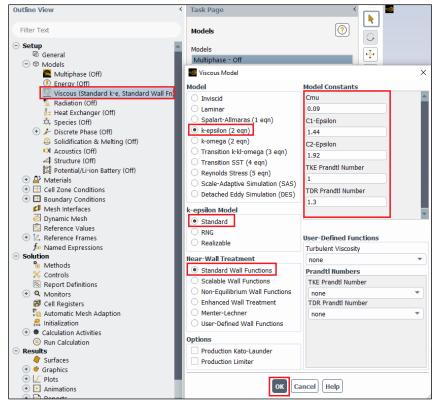
6.5. Setup > General > Solver. Choose an option shown below.

Axis Boundary Condition											
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	N	Y	Y	N	Y	Y	Y			

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

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





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	<		Mesh
Filter Text	Materials	(?)		
 Setup 	Materials			
C General	Fluid			
	air			
Multiphase (Off)	Solid			
🕑 Energy (Off)	aluminum		Q	
Radiation (Off)				
Heat Exchanger (Off)			$\overline{\mathbf{O}}$	
🛱 Species (Off)				
📀 🥕 Discrete Phase (Off)				
🚨 Solidification & Melt 🔤 Create/Edit Materia	ls			×
Acoustics (Off) Acoustics (Off) Name		Material Type		Order Materials by
Potential/Li-ion Batte air		fluid		Name
Arrian Sector State Chemical Formula		Eluent Eluid Materials		Chemical Formula
Chemical Portidia		air	-	
🔐 air				Fluent Database
💿 🛃 Solid		Mixture		GRANTA MDS Database
💿 🖽 Cell Zone Conditions		none	~	GRANTA MDS Database
🛞 🖽 Boundary Conditions				User-Defined Database
Mesh Interfaces	roperties			
Dynamic Mesh Reference Values				
Reference Values	Density [kg/m ³]	onstant		Edit
f∞ Named Expressions	1	17		
 Solution 				
Methods	Viscosity [kg/(m s)]	onstant		Edit
🄀 Controls	1	872e-05		
🔊 Report Definitions	<u> </u>	0720 00		
Monitors				
😰 Cell Registers				
Se Automatic Mesh Adapti Initialization				
Calculation Activities				
Calculation Activities				
Results	ch	ange/Create Delete Clos	e Help	
Surfaces	Ch	ange/ create Delete 0.05	Help	

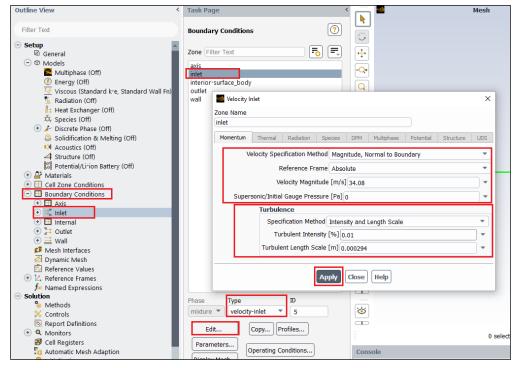
6.8. Tree > Setup > Cell Zone Conditions(Double click) > **Zone > surface_body**. Change type to **fluid**, make sure **air** is selected and click **Apply**.

Outline View	Task Page <		Mesh
Filter Text	Cell Zone Conditions		
○ Setup Image: Beneral ○ Models Image: Beneral Image: Beneral	Zone Filter Text	÷	
🕖 Energy (Off)	I Fluid		×
Viscous (Laminar)	Zone Name		
Heat Exchanger (Off)	surface_body		
🛱 Species (Off)	Material Name air 🔹 Edit		
 	Frame Motion Source Terms		
Solidification & Melting (Off) Acoustics (Off)	Mesh Motion Fixed Values		
∠4 Structure (Off)	Porous Zone		
Potential/Li-ion Battery (Off)	· · · · · · · · · · · · · · · · · · ·		
Aterials Cell Zone Conditions	Reference Frame Mesh Motion Porous Zone 3D Fan Zo	ne Embedded LES Reaction Source Terr	ms Fixed Values Multiphase
Cell Zone Conditions (*) Fluid		ly Close Help	
🛞 🖽 Boundary Conditions	Арг	Close Help	
💋 Mesh Interfaces			
🖉 Dynamic Mesh	Phase Type ID		
Reference Values	mixture 🔻 fluid 🔻 2		
Keference Frames		`	
Solution	Edit Copy Profiles		
% Methods	Parameters		
X Controls	Operating Conditions		0 selected all
📉 Report Definitions	Display Mesh	Console	
Q Monitors	Porous Formulation		
🐻 Cell Registers		Adjusting the following settin	ig:
Mutomatic Mesh Adaption	Superficial Velocity Divergent Velocity	Changing Discretization Scheme	
🔒 Initialization	 Physical Velocity 	Energy: from: First Order Up	wind to: Second Orde
💿 🏶 Calculation Activities		Upwind	

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

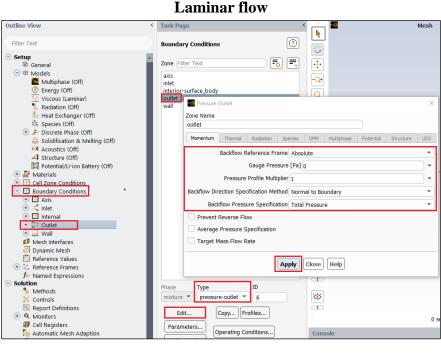
Laminar flow

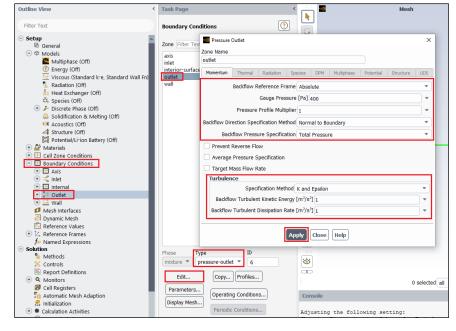
Outline View <	Task Page Mesl	h
Filter Text	Boundary Conditions	
 Setup General Models Multiphase (Off) <u>G</u> Multiphase (Off) <u>G</u> Security (Laminar) <u>S</u> Sadiation (Off) 	Zone Filter Text	2
Acutation (Cif) A = text Exchanger (Off) X, Species (Off) ✓ Discrete Phase (Off) Solidification & Melting (Off) Solidification & Melting (Off)	Zone Name Image: Constraint of the second seco	
	Velocity Specification Method Magnitude, Normal to Boundary	
	Supersonic/Initial Gauge Pressure [Pa] 0	
Mesh Interfaces Dynamic Mesh Reference Values K. Reference Frames	Phase Type ID S mixture velocity-inlet s Edit Copy Profiles	0 selected all
Named Expressions Solution Methods Add Second Se	Parameters Display Mesh Display Mesh Display Mesh Display Mesh	



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

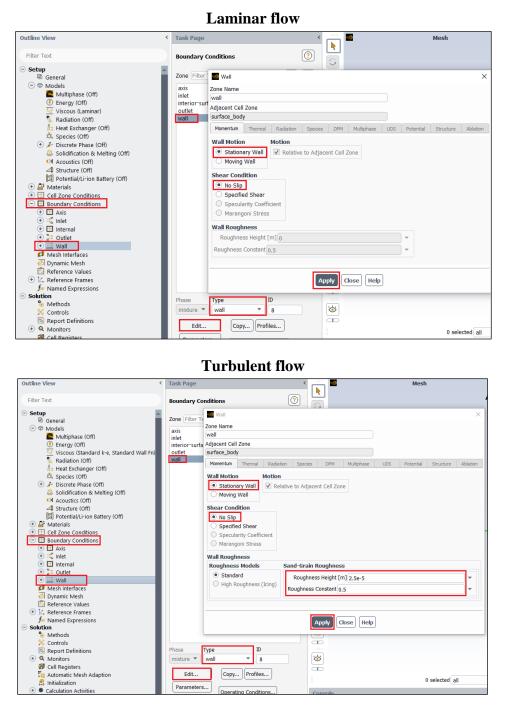
6.10. Tree > Setup > Boundary Conditions > outlet (double click) or click Edit.... Change parameters as per below and click Apply.





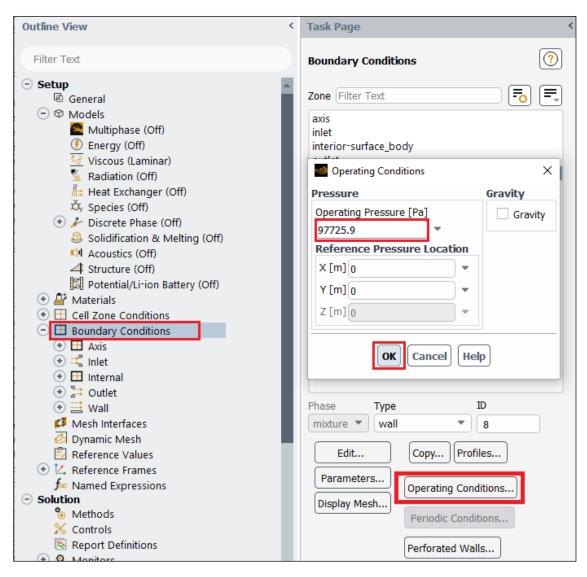
Outlet Boundary Condition											
Model Laminar Turbulent											
Variable u [m/s] v [m/s] P [Pa]				u [m/s]	v [m/s]	P [Pa]	k [m ² /s ²]	e [m ² /s ³]			
Magnitude	-	-	0	-	-	400	1	1			
Zero Gradient	Y	Y	Ν	Y	Y	Ν	Y	Y			

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



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

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



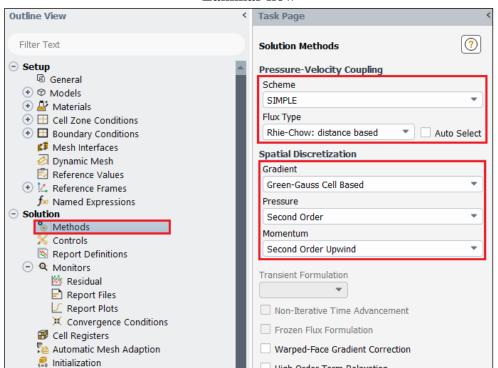
6.13. Tree > Setup > Reference Values. Change parameters as per below.

Outline View <	Task Page				
Filter Text	Reference Values	(?)			
 Setup General Models Multiphase (Off) Energy (Off) Viscous (Laminar) Radiation (Off) Heat Exchanger (Off) Species (Off) Solidification & Melting (Of ✓ Discrete Phase (Off) Solidification & Melting (Of ✓ Acoustics (Off) Structure (Off) Structure (Off) Structure (Off) Cell Zone Conditions Mesh Interfaces Dynamic Mesh Reference Values 	Compute from Reference Values Area (m2) 0 Density (kg/m3) 1 Enthalpy (j/kg) 0 Length (m) 0 Pressure (pascal) 0 Temperature (k) 2 Velocity (m/s) 0 Viscosity (kg/m-s) 1 Ratio of Specific Heats 1 Yplus for Heat Tran. Coef. 3 Reference Zone	.17 .05238 .88.16 .2 .872e-05 .4			
 A Reference Frames Mamed Expressions 					

Laminar flow

Outline View <	Task Page	<
Filter Text	Reference Values	(?)
 General ♥ Models 	Compute from	•
Multiphase (Off)	Reference Values	
Interprete (Off)	Area (m2)	0.002154869
🛂 Viscous (Standard k-e, Star	Density (kg/m3)	1.17
Radiation (Off)	Enthalpy (j/kg)	0
着: Heat Exchanger (Off) 英: Species (Off)	Length (m)	0.05238
	Pressure (pascal)	0
🚨 Solidification & Melting (Of	Temperature (k)	288.16
Acoustics (Off)	Velocity (m/s)	34.08
4 Structure (Off)	Viscosity (kg/m-s)	1.872e-05
🕑 🚑 Materials	Ratio of Specific Heats	1.4
🕑 🖽 Cell Zone Conditions	Yplus for Heat Tran. Coef.	300
 Boundary Conditions Mesh Interfaces 	Reference Zone	
Dynamic Mesh		•
🖾 Reference Values		
Keference Frames		
🏂 Named Expressions		

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



Laminar flow

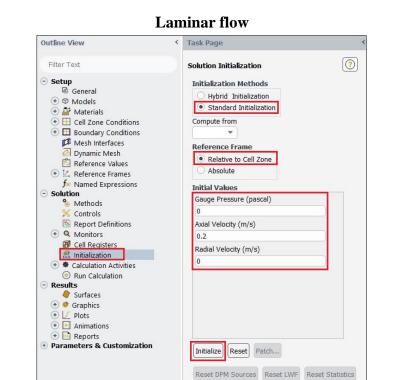
Outline View <	Task Page
Filter Text	Solution Methods
 Setup General Models Materials Cell Zone Conditions Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values X. Reference Frames Named Expressions Solution Methods Controls Report Definitions Cell Registers Automatic Mesh Adaption Initialization Calculation Activities 	Pressure-Velocity Coupling Scheme SIMPLE Flux Type Rhie-Chow: distance based Auto Select Spatial Discretization Gradient Green-Gauss Cell Based Pressure Second Order Momentum Second Order Upwind Turbulent Kinetic Energy Second Order Upwind Turbulent Dissipation Rate Second Order Upwind

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)

Outline View	Task Page			
Outline View Filter Text [‡] : Heat Exchanger (Off) [★] , Species (O	Task Page Monitors Report Definition quantiti during solution when the Files or Report Plots. Residual Monitors Options Print to Console Plot Curves Iterations to Plot 1000 \$	y are included in Report Equations Residual Monito continuity v-velocity v-velocity Convergence Condition Show Advanced Option	ns	× ence Absolute Criteria 1e-06 1e-06 1e-06
Image: Second Secon		Residual Values Normalize ✓ Scale Compute Local Scale Renormalize OK Plot Can	Iterations abs	vergence Criterion solute

Laminar flow

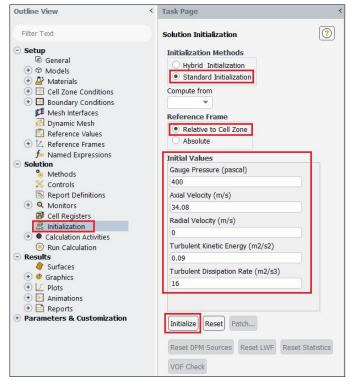
Outline View	<	Task Page		<	
Filter Text Setup Ø Ø General		Monitors Report Definition quantitie during solution when they Files or Report Plots.			
Let General	l	Residual Monitors Options	Equations Residual continuity x-velocity y-velocity k epsilon Convergence Co V Show Advance		rgence Absolute Criteria 1e-06 1e-06 1e-06 1e-06 1e-06
			Residual Values OK Plot	Iter 5 al Scale	Convergence Criterion



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

Turbulent flow

VOF Check



- **Outline View** < Task Page ? Filter Text **Run Calculation** Setup Update Dynamic Mesh... C General Check Case ... Parameters Number of Iterations Reporting Interval \$ 📀 🖽 Boundary Conditions 1000 -1 Mesh Interfaces Profile Update Interval 🙆 Dynamic Mesh * 1 🗐 Reference Values 💿 🖾 Reference Frames Solution Processing for Named Expressions Statistics Solution Data Sampling for Steady Statistics ⁰ Methods ℅ Controls Data File Quantities... 🖄 Report Definitions 🕂 🍳 Monitors Solution Advancement 🗃 Cell Registers 🛃 Initialization Calculate 📀 🏶 Calculation Activities Run Calculation Results Surfaces 📀 🔮 Graphics Scene 📀 💽 Animations 📀 📑 Reports Parameters & Customization
- 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

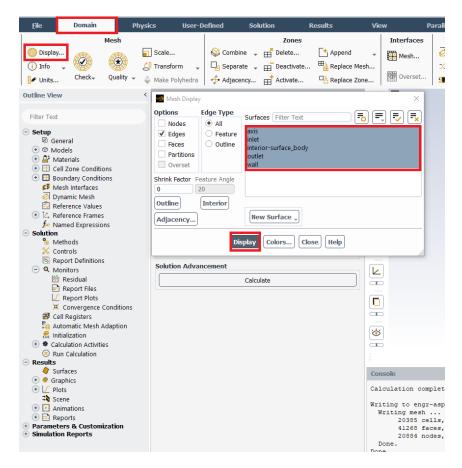
File > Save Picture. Your current display can be saved as a picture file by adjusting formats or resolutions like below and by clicking **Save**. Use this function whenever you need to save pictures for the report.

<u>F</u> ile Domain	Physics	User-Defined	Solution F
Life Domain Refresh Input Data Recorded Mesh Operations: Save Project Reload Sync Workbench Read Write Import Export Table File Manager Solution Files Interpolate EM Mapping FSI Mapping Save Picture Data File Quantities Idle Timeout Preferences Start Page Close Without Save Close Fluent * Catculation Activities @ Rucalculation		ale ansform ansform ansf	Zones mbine Delete parate Deactivate acency Activate (7) Update Dynamic Mesh Reporting Interval 1

Format	Coloring	File Type	Resolution					
⊖ EPS	Color	Raster	Use Window Resolution					
IPEG	Gray Scale	Vector	Carla					
О РРМ	O Monochrome		Scale					
O PostScript			100%					
			Select Resolution					
			custom 💌					
			Width 960 🗘					
O Window Dump			Height 720 🤤					
	Options							
	✓ Landscape Ori	entation 🛛 👝	ndow Dump Command					
	✓ White Backgro	und	port -window %w					
Save Apply Preview Close Help								

7.2. Displaying Mesh

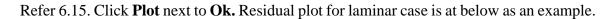
Setting Up Domain > **Display.** Select all the surface you want to display. Lines and points you create can be displayed here as well.

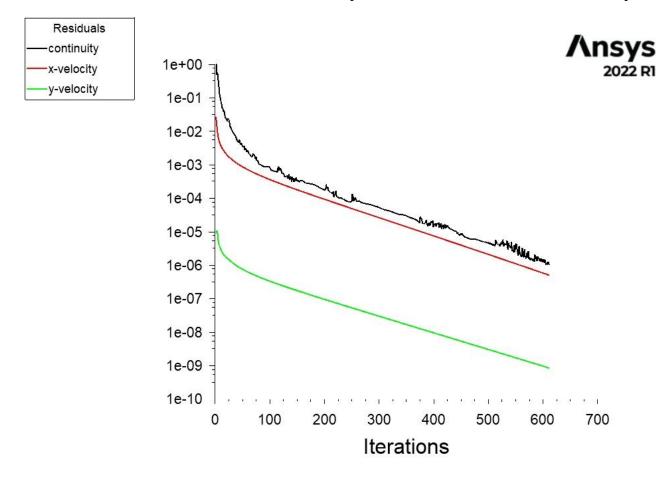


*Tips

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





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 Domain Physi	ics User-Defined Solution	Results	View Pa	rallel Design	^	
(i) Info 🖕 🤡 🦉] Scale Scale ③ Scale ④ Combine → ☆ Delete ④ Transform → □ Separate → ☆ Delactivate ▲ Make Polyhedra ☆ Adjacency ☆ Adjacency	[+ Append ate [⊞] } Replace Mesh		Mesh Models Dynamic Mesh Mixing Planes Turbo Topology	Adapt Refine / Coarsen	Surface + Create Zone Partition
Outline View Filter Text General Models Materials Cell Zone Conditions Boundary Conditions Dynamic Mesh Reference Values 	Task Page Run Cakulation Check Case Update Dynamic Mesh Options Data Sampling for Steady Statistics Sampling Interval 1 \$ Sampling Options	x-v	Residuals ntinuity relocity relocity	1e+00 1e-01 1e-02		Imprint Point Plane Quadric Iso-Surface Iso-Clip Transform

Point Surface	×
Name	
point-1]
Reference Frame	
global	-
Coordinates	
x (m) 7.62	
y (m) 0	Center
z (m) -0	
Create	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 Sur		×	
New Surface Nar	ne		
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	
z0 (m) 0		z1 (m) 0	
5	elect Point	s with Mou	se
[Create	lose	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**.

XY Plot Name					
xy-plot-1					
		Plot Direction X 0 Y 1 Z 0 Load File	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vector Surfaces Filter Text Axis Axis Intet	• • •	R V .
		Free Data	inlet		
	Save/Plot Axe	es) Curves.	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 Name							
xy-plot-2							
Options			Plot Direction	n Y Axis Function	n		
✓ Node Values			X 0	Velocity		-	
Position on X	Axis		Y 1	Axial Velocity	,		
Position on Y	Axis		Z 0				
Write to File				X Axis Function			
Order Points	Curves	- Solution XY Plot		Direction Vect		 =] [] []	=_
File Data [0/0]	Curve #	Line Style	Marker Style	2	TEAL		<u> </u>
	0	Pattern	Symbol				<u> </u>
	Sample	•	(*)	-			
	l mine	Color	Color				Т.
		orange	orange	-	surface_body		
		Weight	Size		e		
		1	0.3				
		[Apply] Clos	e Help				I
				eutlet Point-surf Wall New Surface	_		¥
		Save/Plot	axes Curves	Close H	ielp		

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

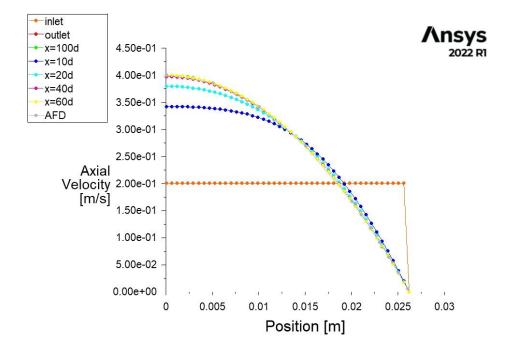
[Right mouse button-> Save link as..]

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

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

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									
xy-plot-3					N				
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 Write to File	Z 0	X Axis Function	Look in:	H:\9. T	A	- G	0	0 👩	:
Order Points		Direction Vector	Ny Comp	outer	axialvelocityAFD-laminar-pipe.xy				
File Data [1/1]	₹,	Surfaces Filter Text	a sungtpark	k					
	Load File	axis							
Velocity Magnitude		inlet interior-surface_body							
	Free Data	outlet							
		point-1							
		wall	XY File a	xialvelocit	yAFD-laminar-pipe.xy				ок
		x=100d							
		x=10d x=20d	Files of type: >	XY Files (*	.xy)			•	Cancel
		x=20d x=40d							
		x=60d	Filter String						Filter
		x-000	_					G	Remove
		New Surface 🚽	Lh/0_TA (oviehu	olo cituA FD	-laminar-pipe.xy				teniove
	ave/Plot Axes Curve		n./9. IA/dxidive	elocityArD	таплінаї фірежу				
	453 2D wall faces, zon	e 8, binary.							1
	884 nodes, binary. 884 node flags, binary								////
Done.									
Doniel									
	ng " gzip -2cf > SYS-								
Writi	ng temporary file C+\U	eare\SUNCTP~1\Ann	etellocellTemp	flataz.	-61564				



Result for laminar flow is presented as an example below.

7.7. Plotting Static Pressure Profile at Centerline

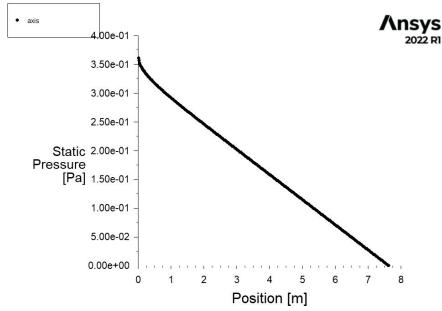
Tree > Results > Plots > XY Plot (double click). Change Y function to **Pressure...** and select **axis** then click **Plot**.

XY Plot Name	
xy-plot-4	
Options	Plot Direction Y Axis Function
✓ Node Values	X 1 Pressure 🔻
Position on X Axis	Y 0 Static Pressure
Position on Y Axis	ZO
Write to File	X Axis Function
Order Points	Direction Vector
File Data [1/1]	Load File Free Data Internal
	 ◆ Line-surface ◆ Outlet ◆ Point-surface ◆ Wall
<	New Surface 💂
Save/Plot	Axes) Curves) 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.

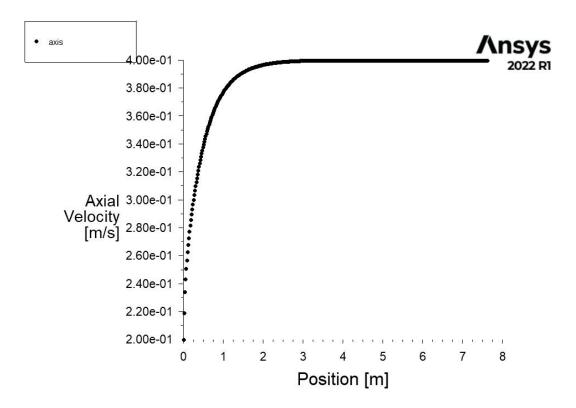


7.8. Plotting Velocity at Centerline

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		ZO	X Axis Function	
Write to File			Direction Vector	•
File Data [0/0]		Load File Free Data	Surfaces Filter Text axis inlet interior-surface_body outlet point-1 wall x=100d New Surface	
	Save/Plot	Axes Curv	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.

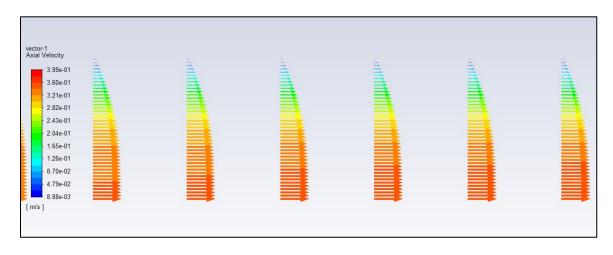
XY Plot Name			
xy-plot-4			
Options		Plot Direction	Y Axis Function
 ✓ Node Values ✓ Position on X Axis Position on Y Axis ✓ Write to File Order Points File Data [0/0]		X 1 Y 0 Z 0	Wall Fluxes Wall Shear Stress X Axis Function Direction Vector Surfaces Filter Text Axis Axis Axis Axis Axis Axis Axis Axis
	Write	es) Curves	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			*
✓ Auto Range	Color by			
Clip to Range	Velocity			*
 Auto Scale Draw Mesh 	Axial Velocity			-
	Min (m/s)	Max (m/s)		
Style	0.00887987	0.3994213		
3d arrow 💌 Scale Skip	Surfaces Filter Text		- -) 🛃 🖡
0.2 0 Cector Options	axis inlet interior-surface_bod outlet point-1 surface_body wall x=100d x=100d x=20d x=40d x=60d	ly		
Sav	ve/Display Compu	Ite	Help	

Result of laminar flow is presented as an example.



7.11. Plotting Velocity Contours

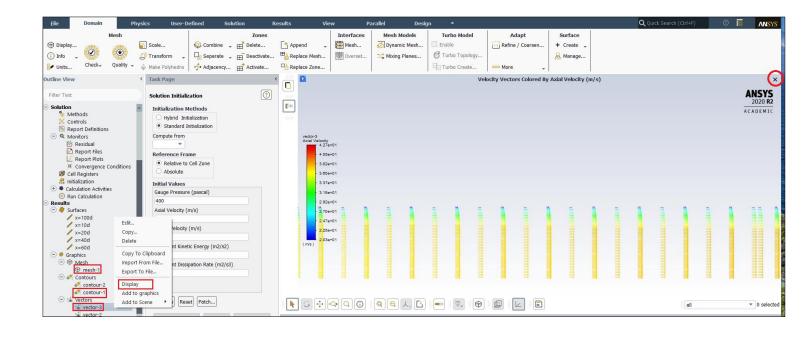
Tree > Results > Graphics > Contours (double click). Change the parameters as per below and click **Display**.

Contours	×
Contour Name	
contour-1)
Options	Contours of
✓ Filled	Velocity 🔻
✓ Node Values	Axial Velocity
Boundary Values	Min Max
Contour Lines	0 0
✓ Global Range	
✓ Auto Range	Surfaces Filter Text
Clip to Range Draw Profiles	axis
Draw Mesh	inlet interior-surface_body
	outlet
Coloring	surface_body
Banded	wall x=100d
Smooth	x=10d
Sinoda	x=20d
Colormap Options	x=40d x=60d
	New Surface
Sav	ve/Display Compute Close Help

Result of laminar flow is presented as an example.

contour-1 Axial Velocity			
3.99e-01			
3.20e-01			
2.80e-01			
2.40e-01			
2.00e-01			
1.60e-01			
1.20e-01			
7.99e-02			
- 3.99e-02			
0.00e+00			
[m/s]			

If the plot does not show up well, remove the plot window, and use a 'Display' button as below.



8. V&V Instructions

8.1. V&V Instructions for Velocity Profile

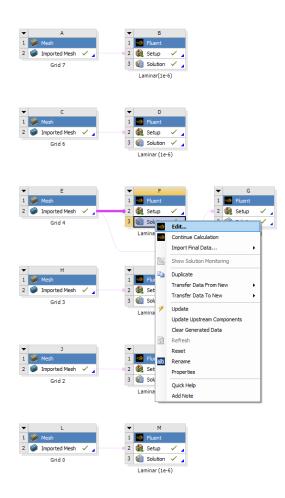
Download CFD Lab 1 Workbench file from class website (http://user.engineering.uiowa.edu/~me_160/)

CF	D Lab1: Pipe flow
CF	D Lab1 Concepts
CF	D Lab1 Manual (<u>PDF</u> , <u>DOC</u>)
CF	D Lab1 Workbench (Download) D Lab 1 V&V Excel sheet
	ownload)

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

File	View	Tools	Units	Extensior	ns Jo	obs	Help	
1	🔒	3	- Proj	ect				
👔 Im	port	∉φ Rec	onnect	🕸 Refresh	n Projec	t 🕖	Update Project	ACT Start Page
Toolbox				र प	μx	Proj	ect Sche <mark>matic</mark>	
🗆 Ana	alysis Sy	stems			^		Read m	odified inputs and generate outputs
🔽 D	esion As	sessment	t					

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

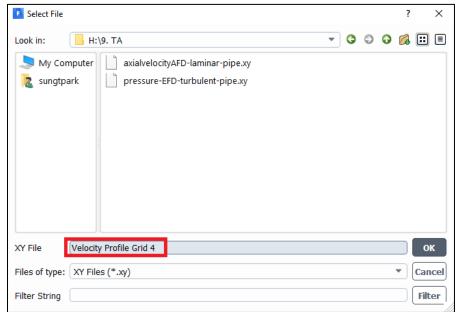


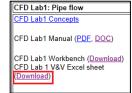
Create reference points by following 7.4.

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

XY Plot Name				
xy-plot-2				
Options	Pl	ot Direction Y	Axis Function	
 ✓ Node Values ✓ Position on X Axis Position on Y Axis ✓ Write to File Order Points 		K 0 K 1 Z 0 X D Su Su Free Data	/elocity Axial Velocity Axis Function Direction Vector	
		N	lew Surface 🚽	
	Write	es) Curves	. Close Help	

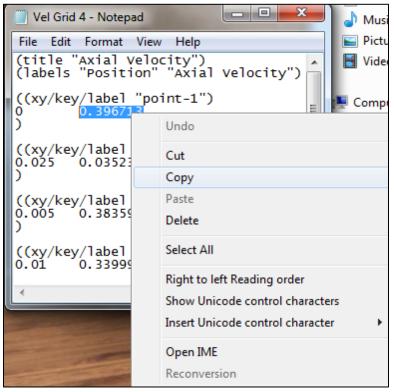
Name file according to which grid solution you are using.





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.



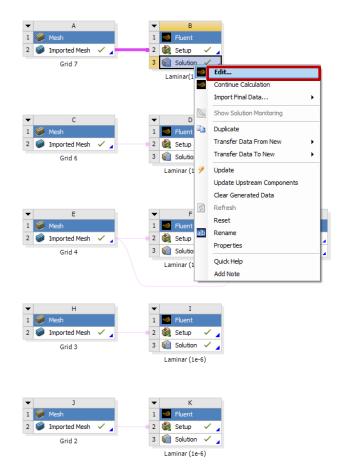
Paste value into V&V Excel file according to its y position and its grid number. Use the Keep Text Only paste function by right clicking in the cell and selecting it from the paste options.

									ŗ
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...**

XY Plot Name			
xy-plot-2			
Options		Plot Direction	Y Axis Function
✓ Node Values		X	Wall Fluxes 👻
 Position on X Axis 		Y 0 Z 0	Wall Shear Stress
 Position on Y Axis Write to File 		20	X Axis Function
Order Points			Direction Vector
File Data [0/0]		Load File Free Data	Surfaces Filter Text Fo F F F
			New Surface
	Write	Axes) Curve	s Close Help

			?		\times
G	0	0	ß	::	≣
				0	۲
				Can Filt	=
	0				Can

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.

6.55899 0.0005603 6.65544 0.0005603 6.7519 0.0005603 6.84835 0.0005603 <u>6.94481 0.0005603</u>	189 184 195 171		
7.13772 0.0005603 7.23418 0.0005605		Properties	
7.33063 0.0005603		Cut	
7.52354 0.0005599		Сору	
)		Paste	
		Cut Other	•
		Copy Other	•
		Insert	•
		Delete	•
Clipboard		Change Case	

	Pgest	2												
	rg	1.414213562												
		Sg2 (MEDIUM)			E	ε21	ε32	Rg	Convergence	Pg	δ	Р	+Ug	-Ug
	0.000E+00	0.000E+00			1.000E+02			#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0
		Sg2 (MEDIUM)		Α	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												
		Sg2 (MEDIUM)		A	E	ε21	ε32	Rg	Convergence	Pg	δ	Р	+Ug	-Ug
	0.000E+00	0.000E+00			1.000E+02			#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0
		Sg2 (MEDIUM)		Α	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					¢							
	Stress													
0		0												
2		0												
3		0												
4		0												
6		0												
7		0												
8		0												
c=8*τ/(r*U	<u>^2).</u>													

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.

Grid	Wall Shear Stress	с
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 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 ←Section 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