## **Simulation of Laminar Pipe Flows**

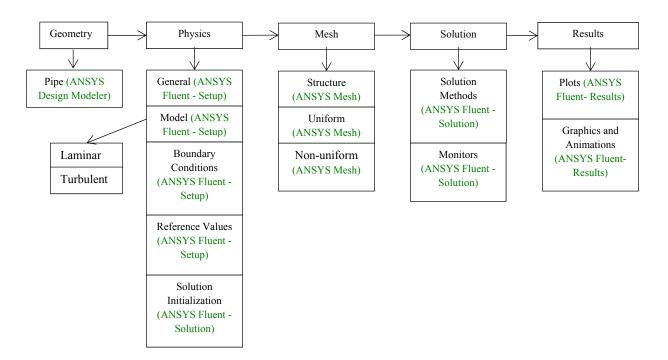
#### ENGR:2510 Mechanics of Fluids and Transport Processes CFD PRELAB 1 (ANSYS 17.1; Last Updated: Oct. 7, 2016)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Andrew Opyd, Maysam Mousaviraad, Tao Xing and Fred Stern

> IIHR-Hydroscience & Engineering The University of Iowa C. Maxwell Stanley Hydraulics Laboratory Iowa City, IA 52242-1585

### 1. Purpose

The Purpose of CFD PreLab 1 is to teach students how to use the CFD educational interface (ANSYS), be familiar with the options in each step of CFD Process, and relate simulation results to AFD concepts. Students will simulate **laminar** pipe flow following the "CFD process" by an interactive step-by-step approach. Students will have "hands-on" experiences using ANSYS to compute axial velocity profile, centerline velocity, centerline pressure, and wall shear stress. Students will compare simulation results with AFD data, analyze the differences and possible numerical errors, and present results in CFD Lab 1 report.

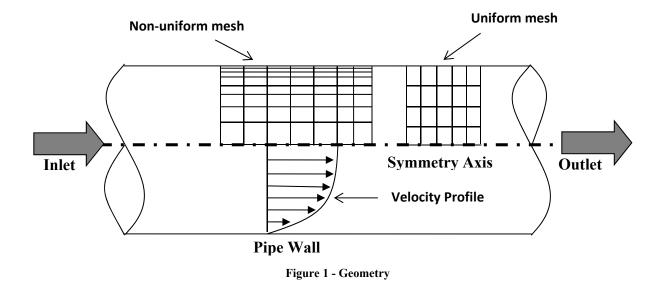


Flow chart for "CFD Process" for pipe flow

# 2. Simulation Design

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

Table 1 – Geometry dimensions								
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 in details later. Uniform flow is specified at inlet, the flow will reach the fully developed regions after a certain distance downstream. No-slip boundary condition will be used on the wall and constant pressure for the outlet. Symmetric boundary condition will be applied on the pipe axis. Since the flow is laminar, turbulence models are not necessary.

## **Navigation Tips**

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

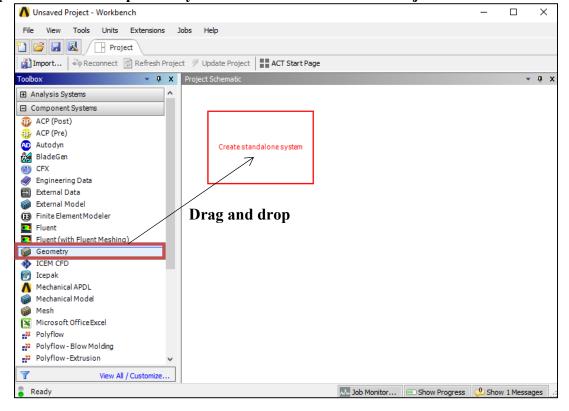
## 3. Open ANSYS Workbench

3.1. Start > All Programs > ANSYS 17.1 > Workbench 17.1

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

Alias AutoStudio 2017	^		
Altera 15.1.0.185 Lite Edition			
Anaconda3 (64-bit)			
Android SDK Tools			
ANSYS 17.1			
🔥 ANSYS AIM 17.1			
🔤 ANSYS Icepak 17.1			
🔥 Mechanical APDL 17.1			
🔥 Mechanical APDL Product Launcher 1			
🔥 SCDM 17.1			
Uninstall ANSYS 17.1			
Norkbench 17.1			
ACP			
ANSYS Client Licensing			
🔥 Aqwa			
📙 Chemkin			
📙 Fluid Dynamics			
📊 Help	¥		
◀ Back			
Search programs and files			

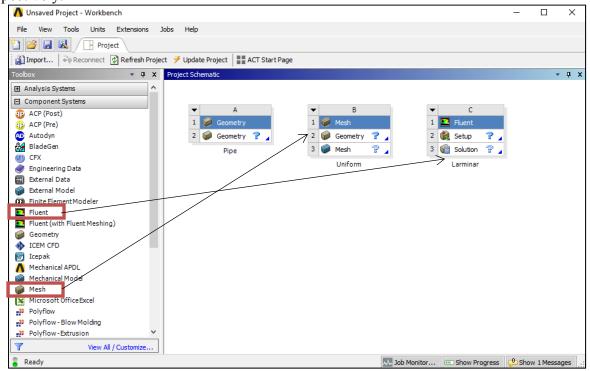
3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop the **Geometry** component for the **Component Systems** in the **Toolbox** into the **Project Schematic**.



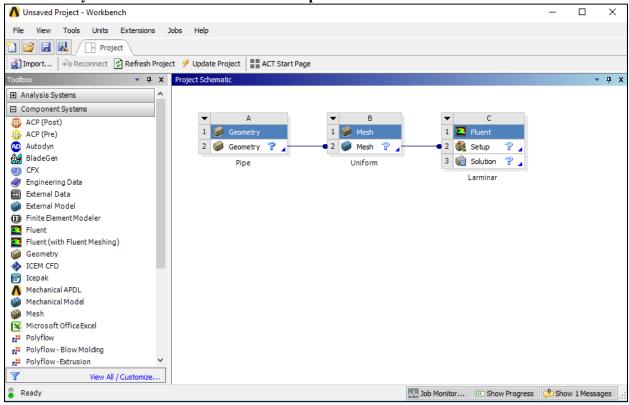
3.3. Rename the geometry "**Pipe**" by right clicking on the down arrow of the Geometry component and selecting **Rename**.

🐧 Unsaved Project - Workbench - 🗆 🗙									
File View Tools Units Extensions Jobs Help									
🚹 🚰 🛃 🔣 🕕 Project									
🕼 Import   🗞 Reconnect 👔 Refresh Project 🥖 Update Project   🔡 ACT Start Page									
Toolbox 🔹 4 X Project Schematic		Ŧ	ĻΧ						
😢 Analysis Systems									
Component Systems									
ACP (Post)									
ACP (Pre)									
🕦 Autodyn 🥖 Update									
🚵 BladeGen									
(I) CFX									
Engineering Data     Clear Generated Data									
External Data									
See External Model									
Fluent     Properties									
I Fluent (with Fluent Meshing) Add Note									
Geometry									
V ICEM CFD									
Dicepak									
A Mechanical APDL C Mechanical Model									
Mechanical Model Mesh									
Microsoft OfficeExcel									
Polyflow									
Polyflow - Blow Molding									
Polyflow-Extrusion									
View All / Customize									
🕐 Drag a Toolboxitem on top of a system to reuse components and exchange data. 🛛 🗔 Job Monitor 💷 Show Progress	🔑 Show	1 Messa	ges 🚲						

3.4. Drag and drop a **Mesh** component and a **Fluent** component into the **Project Schematic** as shown below. Rename the components as "Uniform" and "Laminar" for **Mesh** and **Fluent** components respectively.



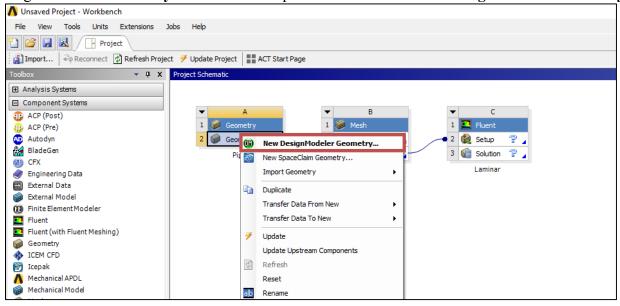
3.5. Make the connections as per below by dragging connections exactly as per below. Drag and drop "Geometry" to "Mesh" and "Mesh" to "Setup".



- 3.6. Create a folder on the network drive (home.iowa.uiowa.edu) called "CFD Pre-Lab and Lab 1".
- 3.7. Save the project file by clicking **File** > **Save As...**
- 3.8. Save the project onto the folder you just created and name it "*CFD Pre-Lab and Lab 1 Pipe Flow*". (This project file will be used for both Pre-Lab 1 and Lab 1.)

## 4. Geometry Creation

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



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

颐 A: Pipe - DesignModeler				
File Create Concept Tools	Units View Help			
] 🔄 层 🛃 👛 🗍 🏵 Und	✓ Meter	R		
	Centimeter			
XYPlane 🔻 抺 None	Millimeter			
	Micrometer	Floy		
BladeEditor: 💒 Import BGD	Foot			
] <b>这还S≡(函述</b>	Inch			
Tree Outline	Large Model Support			
🖃 🛶 😡 A: Pipe		1.		
XYPlane	✓ Degree			
ZXPlane	Radian			
↓ YZPIane ↓ YZPIane ↓ YZPIane	Model Tolerance			
1				

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

颐 A: Pipe - DesignModeler			
File Create Concept Tools Unit	s View Help		
] 🔄 层 📕 📩 🗍 🌖 Undo - 🔅	Redo 🛛 Select: 🌇 🏷 🖌 🕅	🖪 🖪   🖓 -   🗐 🕱  ] 🕻 💠 🖲	( 🕀 🍭 🔍 🔍 🎇 🌌   🔭 💽 🛝
	/x- 🗶 🖻		
XYPlane 🔻 📩 None	🕞 👻 📔 🧚 Generate 🛛 🖤 Shara	: Topology 🔀 Parameters 🗍 💽 Extruc	de 🏘 Revolve 🐁 Sweep 🚯 Skin/Loft 📋 🔳 Thin
BladeEditor: 🏰 Import BGD 🛛 🟭 Loi	ad BGD 🛛 🐼 Load NDF 🛛 😅 FlowPath	🥖 Blade   💋 Splitter 🚽 Vista TFExpo	rt 👆 ExportPoints 🎟 StageFluidZone 🔜 SectorCu
遼西昌 ( 南慶	-	- 🌳 🖋 🖨 🧭	
Tree Outline		<b>4</b>	Graphics
Contractions of the second se			

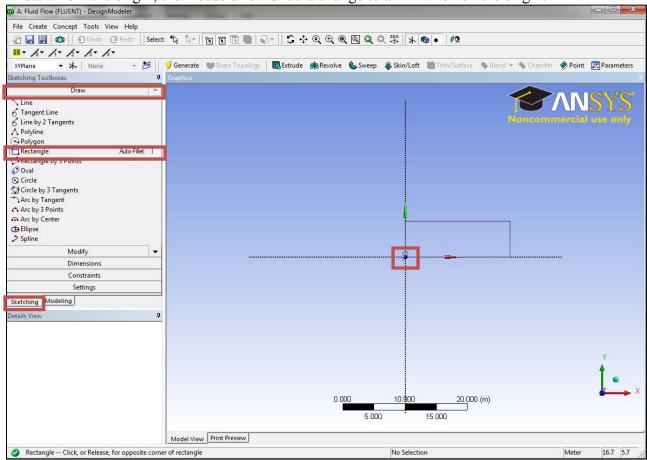
#### 4.4. Right click **XYPlane** and select **Look at**.

🚥 A: Pipe - DesignModeler				
File Create Concept Tools	Units View He	lp		
] 🔄 🔚 📑 📫 🗍 🌖 Undi	o @Redo Se	lect: 🍾 🏷		🖻   S=   🗉 🗶
<b>■</b> ▼ <b>■</b> ▼ / <sub>1</sub> ▼ / <sub>2</sub> ▼	13- 1x- x 3	<b>ਤ</b> ′		
XYPlane 🔻 🗚 Sket	ch1 🔹 💆	📙 誟 Generate	🔹 🖤 Share Top	ology 🔀 Parameters
🛛 🖪 Extrude 🙀 Revolve 🐁	Sweep 🛛 🚯 Skin/L	oft	/Surface 🛛 🔷 B	lend 🔻 🔦 Chamfer 📲
BladeEditor: 🔏 Import BGD	📑 Load BGD 🛛 🕢	Load NDF	🕏 FlowPath 🛛 🥏	🛚 Blade  Splitter 🗖
<b>&amp; 丞 S ≡ (</b> 函 <b>&amp;</b>		•		- 🖗 🖋 I
Tree Outline			₽ Gra	aphics
⊡…√🚱 A: Pipe				
E ↓ XYPlar	at	•		
🗸 🖉 Sl	Dependencies	1		
→ XXPlar → YZPlar <sup>a</sup> lo Renar	me (F2)			

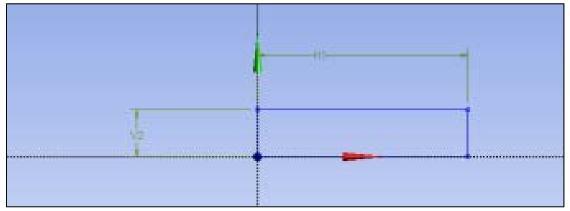
4.5. Select Sketching > Constraints > Auto Constraints. Enable the auto constraints option to pick the exact point as below.

🚥 A: Pipe - DesignModeler	
File Create Concept Tools Units View Help	
🖉 🛃 🛃 🛛 🗍 Đ Undo 📿 Redo 🗍 Select: 🆎 🏷 🖌 🕅	ⓑ ●   🚱 -   ☱ 💥  ] S 💠 Q
■ • <b>■</b> • <i>h</i> • <i>h</i> • <i>h</i> • <i>h</i> • <i>h</i> • <i>f</i>	
XYPlane 🔹 🗚 Sketch1 💌 ಶ 🗍 🧚 Generate 🖤 Shar	e Topology 🔀 Parameters
🔀 Extrude 🚓 Revolve 🐁 Sweep 🚯 Skin/Loft 📗 Thin/Surface	💊 Blend 🔻 💊 Chamfer 🐞 Slice 📋 🍕
BladeEditor: 🆓 Import BGD 🛛 Load BGD 🐵 Load NDF 🛛 😅 FlowPath	n 🥑 Blade  Splitter 🚽 VistaTFExpor
<b>废丕吕言(应陵</b> ▼	- 🗭 🖋 📾 🧭
Sketching Toolboxes 4	Graphics
Draw	
Modify	
Dimensions	
Constraints	
777 Fixed	
Vertical	
✓ Perpendicular	
A Tangent	
€¥ Coincident	
Midpoint	
্রাশ Symmetry	
// Parallel	
Oncentric	
🗡 Equal Radius	
💒 Equal Length	
¥∯ Equal Distance	
Global: 🔽 Cursor: 🔽	
Settings 👻	
Sketching Modeling	
Details View 4	
Details of Sketch1	
Sketch Sketch1	
Sketch Visibility Show Sketch	
Show Constraints? No	

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



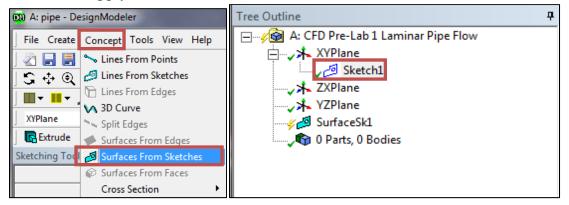
4.7. Select **Dimensions** > **General**. Click on top edge then click above the geometry to place the dimension. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.



4.8. Click on H1 under **Details View**, in the bottom left of the screen, and change H1 to 7.62*m*. Click on V2 and change it to 0.02619*m*. (Don't include unit "**m**" when put in the values)

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

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



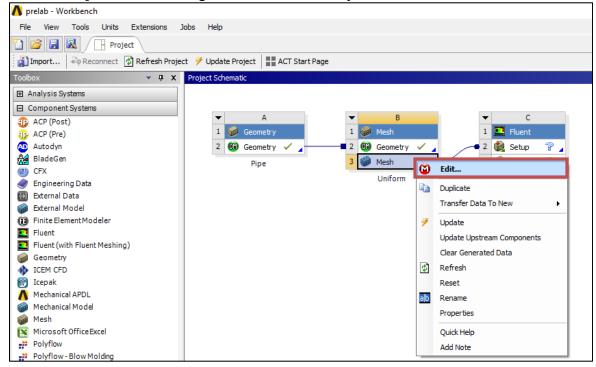
4.10. Click Generate. This will create a surface.

🕅 A: pipe - DesignModeler									
File Create Concept Tools View	File Create Concept Tools View Help								
] 🔄 🔚 📕 📫 🗍 🌖 Undo 📿	Redo 🛛 Select: 🆎 💱 🕅 💽 💽 🥪 🗍								
] S 🕂 Q 🕀 🍭 🖳 Q 🔍	器 🔭 💿 🥂								
	/k <b>- ⊀</b> ⊭								
🛛 XYPlane 🔻 📩 Sketch1	👻 📁 📝 Generate 🛛 🗑 Share Topology 🛛 🚉 Para								
🛛 💽 Extrude 🚓 Revolve 🐁 Swee	o 🚯 Skin/Loft 🛛 🛅 Thin/Surface 💊 Blend 🔻 🥎 Cha								
Tree Outline 4	Graphics								
🖃 🛶 🚱 A: pipe									
🚊 🛶 🗚 XYPlane									
∽ Sketch1									
🚊 🗤 🖉 SurfaceSk1									
∽ 🗇 Sketch1									
🖻 🗸 🖓 1 Part, 1 Body									
📖 🗸 🐚 Surface Body									

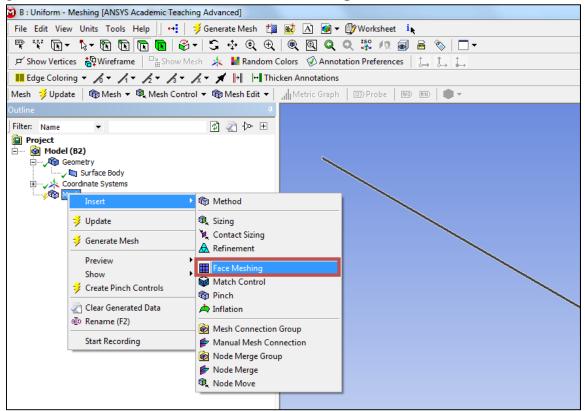
4.11. File >Save Project. Save project and close the Design Modeler window.

# 5. Mesh Generation

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



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



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

B : Uniform - Meshing [ANSYS Academic Research]		
File Edit View Units Tools Help   🗔 🕶   💋 Ger	rate Mesh  🏙 🔺 🕢 🕈 🕼 🕇 👘 🖞 🖤	👫 🔓 🔭 🕅
ダ Show Vertices 項 Close Vertices 7.6e-003 (Auto Scale)	👻 🍄 Wireframe 🛛 🖶 Show Mesh 🏻 🙏 🕌 Random Colors	s 🐼 Annotation Pre
≹Ĩ_ ()← Reset Explode Factor:		
$\blacksquare Edge Coloring \bullet /_0 \bullet /_1 \bullet /_2 \bullet /_3 \bullet /_x \bullet \not$		
Mesh <del>∛</del> Update   ∰ Mesh ▼ ♥ Mesh Control ▼ ∰ M		
Outline 7		
	Face Meshing	
Filter: Name	8/30/2016 4:16 PM	
	Face Meshing	
Project ⊡ im in item is a state of the state o		
⊡		
Surface Body		
Coordinate Systems		
⊡		
Face Meshing		
Details of "Face Meshing" - Mapped Face Meshing 🛛 📮		0.000
Scope		
Scoping Method Geometry Selection	Geometry (Print Preview) Report Preview/	
Geometry Apply Cancel	Messages	
Definition	Text	
Suppressed No		
Mapped Mesh Yes		
Method Quadrilaterals		
Constrain Boundary No		
Advanced		
Specified Sides No Selection		
Specified Corners No Selection		
Specified Ends No Selection		
	🙂 No Messages	1 Face Selecte

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

🙆 A :	CFD	Prelab 1	. Lami	nar F	low - I	Mesh	ning (	ANS	YS A	cade	mic Te	eachi	ng Ao	lvanc	ed]		L	
		View																
	X,Y,Z <b>R</b>	•	₿.	R	Ŀ			8	•	S	+‡+	٩	Ð,	0	Q	Q	Q	
∫д′	Show	Vertices	*0 6+	Wire	fra Edo	ie 📕	Ed	ge Co	olori	ng 🔻	6	- /	í- ,	1₂ ▼	13-	· /x	• •	<
		Update																

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

Outline		<b>4</b>
Filter: Name	- 🔹 🖉	
±	ometry Surface Body ordinate Systems	
Ė,∕® №	Insert	🕑 🚳 Method
	🤣 Update	🔍 Sizing
	誟 Generate Mesh	№ Contact Sizing A Refinement
	Preview Show Create Pinch Controls	Mapped Face Meshing Match Control
	省 Clear Generated Data ৰ]ি Rename	A Inflation
	Start Recording	

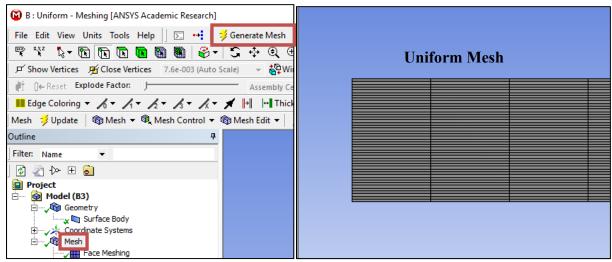
5.6. Hold **Ctrl** button and select the top and bottom edge of the rectangle then click **Apply** on **Geometry**. Specify details of sizing as per below in the **Details of "Edge Sizing" – Sizing** window.

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

5.7. Repeat step 5.5. to insert **Sizing**. Select the left and right edge of the rectangle and click **Apply** then change sizing parameters as per below (Please see **5.3.** for view restoring and zooming in).

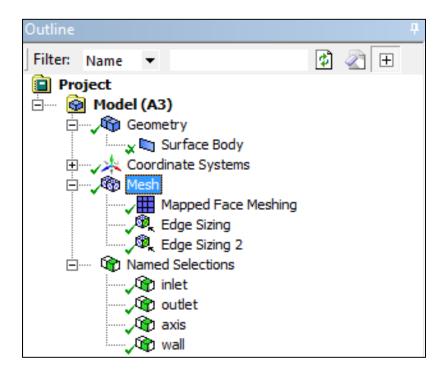
De	Details of "Edge Sizing 2" - Sizing 🗸 🖓				
-	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
-	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	45			
	Behavior	Hard 💌			
	Bias Type	No Bias			

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



5.9. Change the edge names by selecting the edge, then right click on the edge and select **Create Named Selection**. Name left, right, bottom and top edges as *inlet*, *outlet*, *axis* and *wall* respectively. Your outline should look same as the figure below (next page).

	Selection Name
Insert Go To Go To Hide Body Suppress Body Isometric View Set Set Set Zoom To Fit Cursor Mode View N Look At Look At Create Coordinate System Create Coordinate System Create Coordinate System Create Coordinate System Select All Y Update Geometry from Source	Enter a name for the selection group: Selection   Apply selected geometry  Apply geometry items of same:  Size  Type  Location X  Location Y  Location Z  OK Cancel

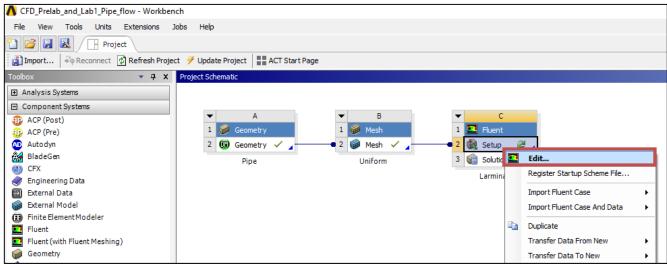


5.10. File > Save Project. Save the project and close the window. Update mesh on Project Schematic by right clicking on Mesh and selecting Update.

▼ B 1 🧼 Mesh	
▲ = <sup>2</sup> 💓 M 🔞	Edit
Unit E	Duplicate
	Transfer Data To New
7	Update
	Update Upstream Components
	Clear Generated Data
2	Refresh
	Reset
ab	Rename
	Properties
	Quick Help
	Add Note

# 6. Solve (Physics)

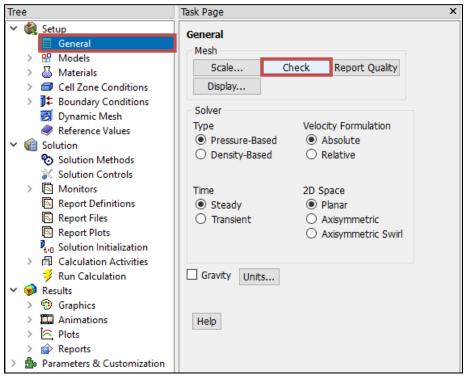
#### 6.1. Right click Setup and select Edit...



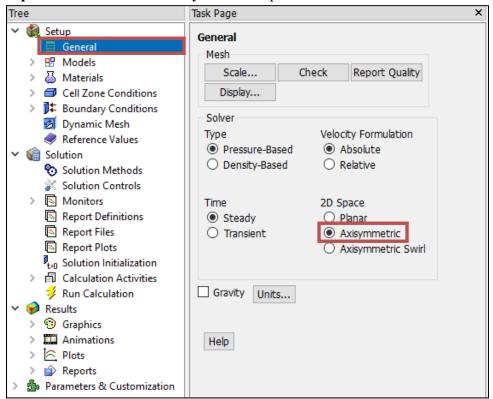
#### 6.2. Check Double Precision and click OK.

Fluent Launcher (Setting Edit Only)	_	
<b>ANSYS</b>	Fluent	Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Workbench Color Scheme Do not show this panel again	Options Double Precision Processing Options Serial Parallel	
Show More Options	ncel Help 🔻	

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



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



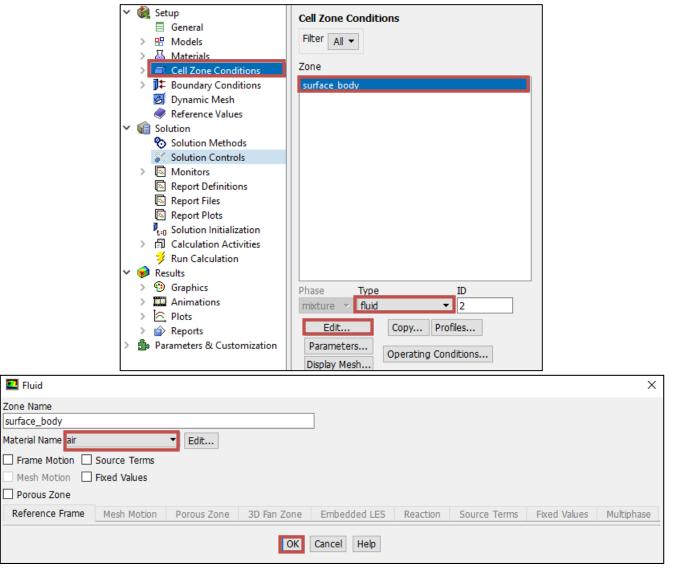
V 🍓 Setup Models	Tree	Task Page		
<ul> <li>General</li> <li>Models</li> <li>Multiphase (Off)</li> <li>Energy (Off)</li> <li>Energy (Off)</li> <li>Radiation (Of</li> <li>Heat Exchang</li> <li>Model ➤</li> <li>Inviscid</li> <li>Species (Off)</li> <li>B Discrete Phase (Off)</li> <li>Solidification &amp; Melt</li> <li>Solidification &amp; Melt</li> <li>Acoustics (Off)</li> <li>Electric Potential (Off)</li> <li>Solidifications</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>Dynamic Mesh</li> <li>Reference Values</li> </ul>	<ul> <li>Setup</li> <li>General</li> <li>Models</li> <li>Multiphase (Off)</li> <li>Energy (Off)</li> <li>Energy (Off)</li> <li>Viscous (Lan Radiation (Of Re Radiation (Of Re Reference Values     </li> </ul>	Models Models Multiphase - Off Energy - Off Viscous - Laminar ion - Off Inviscid Disci V Laminar Solic Acot Elect Standard k-epsilon Standard k-omega SST k-omega Other		

6.5. Tree > Setup > Models > Viscous (RMB click) > Model. Select Laminar.

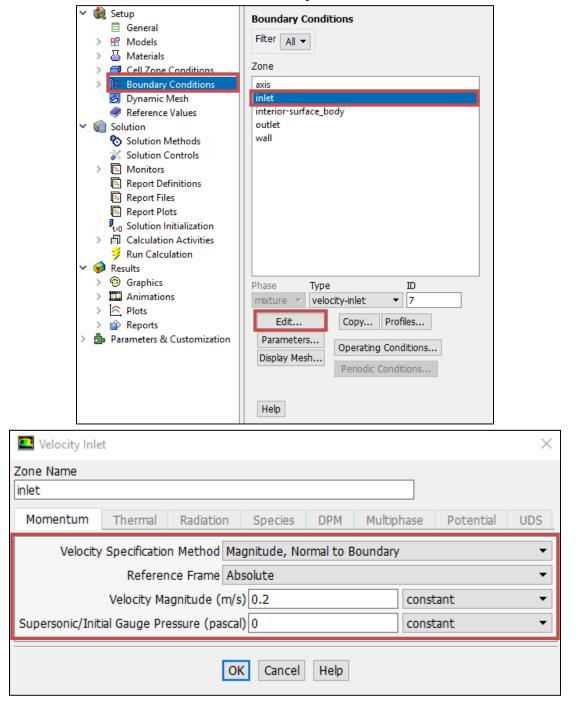
6.6. Tree > Setup > Materials. Right click on air and left click on Create/edit.... Change the Density and Viscosity as per below and click Change/Create. Close the dialog box when finished.

Tree	Task Page	× 🖳 / 🖬	Mesh 🛛 🛛
✓ in Setup ☐ General	Materials		
> B Models	Materials	Create/Edit Materials	
✓ ▲ Materials	Fluid	Name	Material Type
> 👗 Fluid	air	air	fluid
> 🐱 Solid	Solid		
>	aluminum	Chemical Formula	Fluent Fluid Materials
> 🕽 🛱 Boundary Conditions			air
🛃 Dynamic Mesh			Mixture
Reference Values			none
✓ i Solution		Properties	
🇞 Solution Methods		Density (kg/m3) constant	← Edit
😹 Solution Controls			Edition
> 🖎 Monitors		1.17	
Report Definitions		Viscosity (kg/m-s) constant	▼ Edit
🔊 Report Files		1.872e-5	
Report Plots		10/200	
₱ <sub>t=0</sub> Solution Initialization			
Calculation Activities			
🤣 Run Calculation			
✓			
> (9) Graphics			
> Animations	Create/Edit Delete		
> 🦳 Plots		Chan	ge/Create Delete Close Help
> 😰 Reports		Chang	Jefereate Close help
> Derameters & Customization	Help		
	T		

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

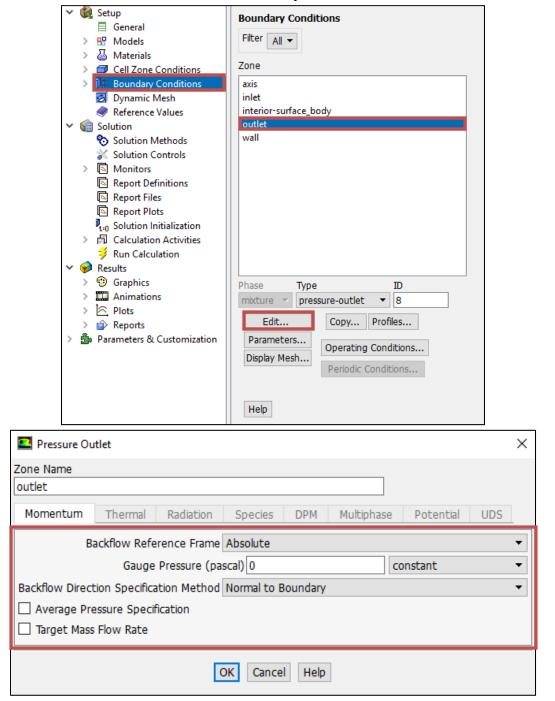


6.8. Tree > Setup > Boundary Conditions > inlet > Edit... Change parameters as per below and click OK. Table below shows the details of the boundary conditions.



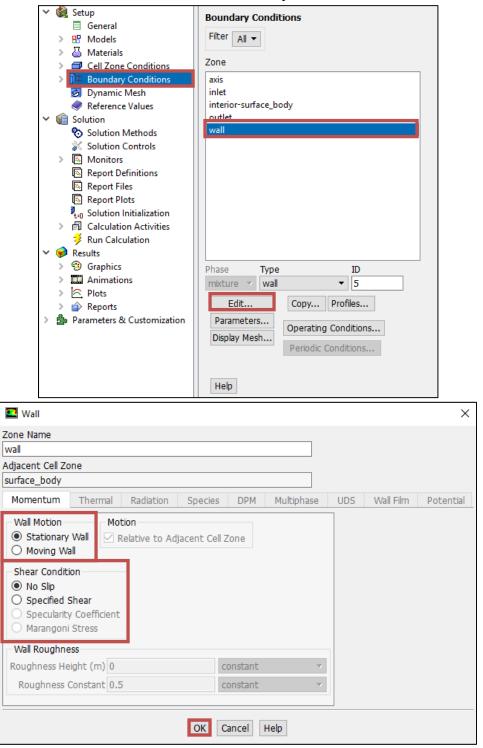
Inlet Boundary Condition				
Variable u (m/s) v (m/s) P (Pa)				
Magnitude	0.2	0	-	
Zero Gradient	N	N	Y	

6.9. Tree > Setup > Boundary Conditions > outlet > Edit... Change parameters as per below and click OK. Table below shows the details of the boundary conditions.



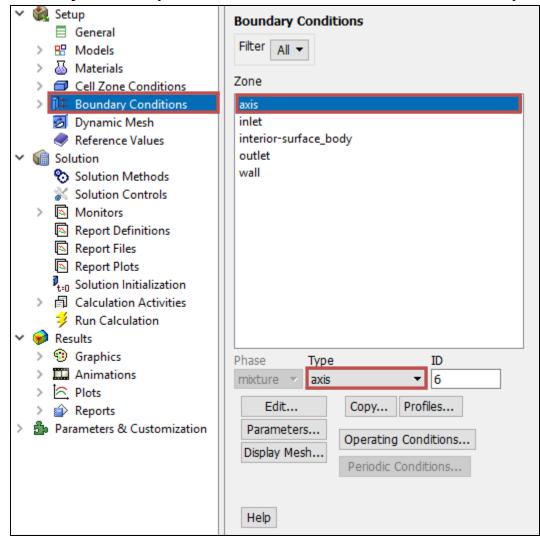
Outlet Boundary Condition			
Variable u (m/s) v (m/s) P (Pa			P (Pa)
Magnitude	-	-	0
Zero Gradient	Y	Y	Ν

6.10. Tree > Setup > Boundary Conditions > wall > Edit... Change parameters as per below and click OK. Table below shows the details of the boundary conditions.



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

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



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

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

<ul> <li>Setup         <ul> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> </ul> </li> <li>Cell Zone Conditions</li> <li>Dynamic Mesh         <ul> <li>Population</li> <li>Solution</li> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> <li>Monitors</li> </ul> </li> </ul>	Boundary Conditions Filter All  Zone axis inlet interior-surface_body outlet wall		AN œ
<ul> <li>Report Definitions</li> <li>Report Files</li> <li>Report Plots</li> <li>Solution Initialization</li> <li>Calculation Activities</li> <li>Results</li> <li>Graphics</li> <li>Minimations</li> <li>Plots</li> <li>Plots</li> <li>Parameters &amp; Customization</li> </ul>	Phase       Type       ID         mixture       wall       5         Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions	Operating Conditions     Pressure     Operating Pressure (pascal)     97225.9     P     Reference Pressure Location     X (m) 0     P     Y (m) 0     P     Z (m) 0     P     OK Cancel Help	X

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

~	Setup	Reference Values
	> 🗄 Models	Compute from
	> 🔠 Models	•
	>  Cell Zone Conditions	Reference Values
	> 🕽 🗱 Boundary Conditions	Area (m2) 0.002154869
	🗭 Dynamic Mesh	Density (kg/m3) 1.17
~	<ul> <li>Reference Values</li> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> <li>Monitors</li> <li>Report Definitions</li> <li>Report Files</li> <li>Report Plots</li> </ul>	Enthalpy (j/kg) 0 Length (m) 0.05238 Pressure (pascal) 0 Temperature (k) 298.16 Velocity (m/s) 0.2 Viscosity (kg/m-s) 1.872e-05
* >	<ul> <li>Image: Solution Initialization</li> <li>Image: Calculation Activities</li> <li>Image: Results</li> <li>Image: Graphics</li> <li>Ima</li></ul>	Ratio of Specific Heats 1.4 Reference Zone surface_body  v

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

🗸 🎆 Setup	Solution Methods
General	Pressure-Velocity Coupling
> 🗄 Models	Scheme
> 🔠 Materials	SIMPLE
> 🗇 Cell Zone Conditions	
> J Boundary Conditions	Spatial Discretization
🛃 Dynamic Mesh	Gradient
Reference Values	Green-Gauss Cell Based 🔻
V Solution	Pressure
Solution Methods	Second Order 🔹
Solution Controls	Momentum
> 🖸 Monitors	Second Order Upwind 👻
Report Definitions	
Report Files	
Report Plots	
L <sub>t=0</sub> Solution Initialization	
Calculation Activities	
Run Calculation	Transient Formulation
✓ ♥ Results	
<ul> <li>Graphics</li> <li>Animations</li> </ul>	
-	Non-Iterative Time Advancement
> 🔄 Plots	Frozen Flux Formulation
<ul> <li>Reports</li> <li>Burameters &amp; Customization</li> </ul>	Pseudo Transient
> 🖆 Parameters & Customization	Warped-Face Gradient Correction
	High Order Term Relaxation Options
	Default
	Help

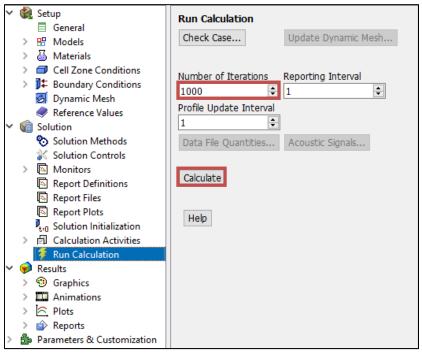
6.15. Tree > Solution > Monitors > Residual. Right click Residual and select Edit... Change convergence criterion to 1e-06 for all three equations as per below and click OK.

	<ul> <li>Setup         <ul> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>Eoundary Conditions</li> <li>Dynamic Mesh</li> <li>Reference Values</li> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> <li>Solution Controls</li> <li>Drag</li> <li>Lift</li> <li>Moment</li> <li>Surface</li> <li>Volume</li> </ul> </li> </ul>	
Residual Monitors		×
Options	Equations	
Print to Console		
Plot Plot	Residual Monitor Check	Convergence Absolute Criteria
Window	continuity	✓ 1e-06
1 Curves Axes	x-velocity	✓ 1e-06
Iterations to Plot	y-velocity	✓ 1e-06
1000 ≑	Residual Values	Convergence Criterion
	Normalize Iterati	
Iterations to Store	5	
1000 🖨	🗹 Scale	
	Compute Local Scale	
ОК	Plot Renormalize Cancel H	elp

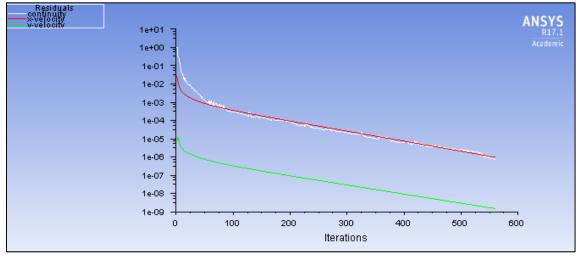
6.16. **Tree > Solution > Solution Initialization**. Change parameters as per below and click **Initialize**.

<ul> <li>Setup         <ul> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>Dynamic Mesh</li> <li>Performance Values</li> </ul> </li> <li>Solution         <ul> <li>Solution</li> <li>Solution Methods</li> <li>Solution Controls</li> <li>Monitors</li> <li>Report Definitions</li> <li>Report Files</li> <li>Report Plots</li> </ul> </li> <li>Calculation Activities</li> <li>Run Calculation</li> <li>Signaphics</li> <li>Signaphics</li> <li>Plots</li> <li>Plots</li> <li>Parameters &amp; Customization</li> </ul>	Solution Initialization Initialization Methods Hybrid Initialization Standard Initialization Compute from Reference Frame Reference Frame Reference Frame Absolute Initial Values Gauge Pressure (pascal) O Axial Velocity (m/s) O.2 Radial Velocity (m/s) O
	Initialize Reset Patch Reset DPM Sources Reset Statistics Help

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



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



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

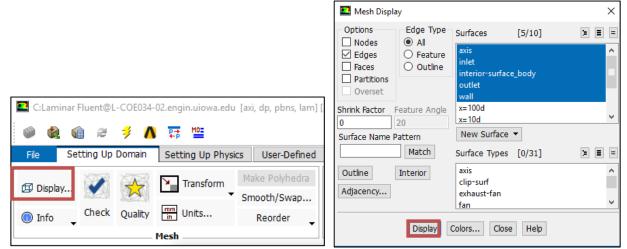
6.19. File > Save Project. Save the project

# 7. Results

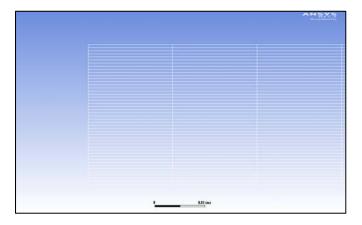
Please read exercises before continuing. This section tells you how to post-process the results.

#### 7.1. Displaying Mesh

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



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



#### 7.2. Saving Pictures

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

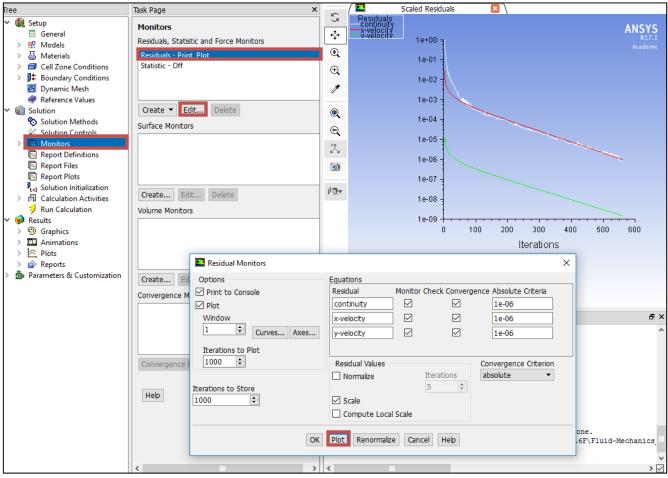
Save Picture				Х		
Format C EPS IPEG PPM	Coloring File Typ Color  Color  Gray Scale  Vector Monochrome		ter Width 960 🖨			
<ul> <li>PostScript</li> <li>TIFF</li> <li>PNG</li> <li>HSF</li> </ul>	Options Landscape Orie White Backgrou	ntation i	Vindow Dump Command import -window %w			
O VRML O Window Dump						
Save Apply Preview Close Help						

Name the File, navigate to the CFD Pre-Lab 1 file you created and save it in that file. Then close the Save Picture window.

Select File							
Look in: H:\Intro Fluids\CFD Pre Lab 1							
Name Size Type Date Modified							
Hardcopy File CFD Pre-Lab 1 Laminar Pipe Flow Mesh, jpg		ОК					
Files of type: Hardcopy Files (*.jpg)	C	ance					
Filter String		Filter					

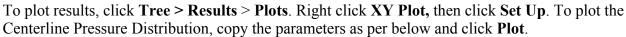
#### 7.3. Plotting Residuals

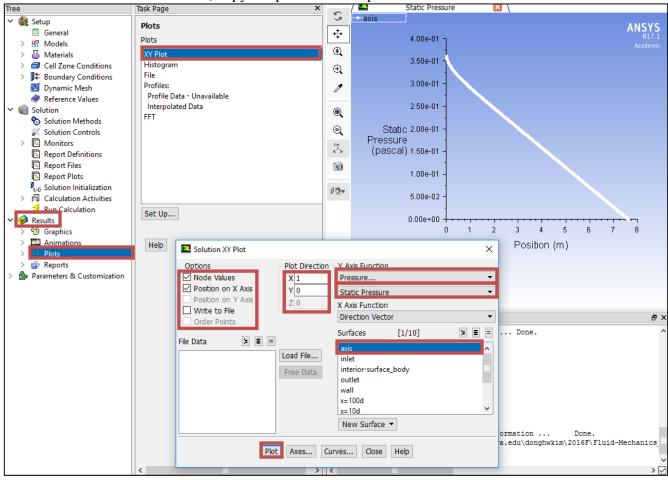
To display the residuals click **Tree > Solution > Monitors**. Right click **Residuals** select **Edit**, click **Plot** then click **Cancel**.



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

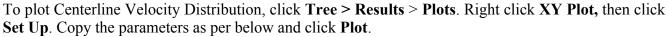
#### 7.4. Plotting Centerline Pressure Distribution

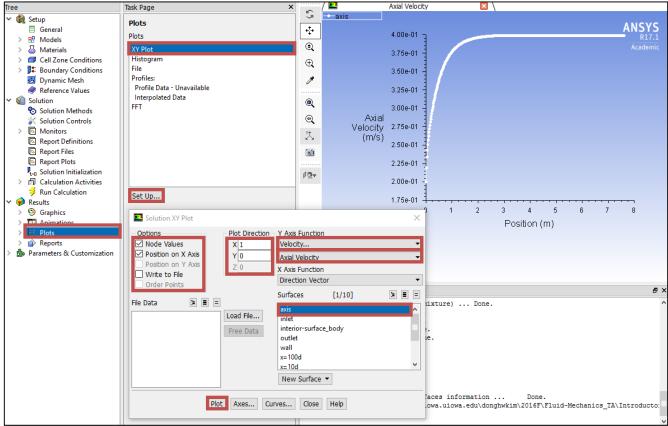




Save the picture as you did for the mesh and call it "*CFD Pre-Lab 1 Laminar Pipe Flow Centerline Pressure Distribution*" and save it in the folder you created.

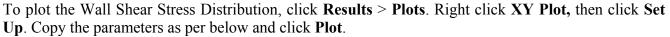
#### 7.5. Plotting Centerline Velocity Distribution

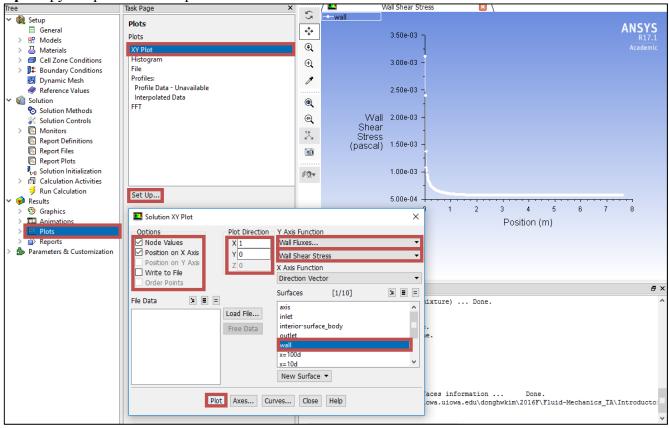




Save the picture as you did for the mesh and call it "*CFD Pre-Lab 1 Laminar Pipe Flow Centerline Velocity Distribution*" and save it in the folder you created.

#### 7.6. Plotting Wall Shear Stress Distribution





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

#### 7.7. Plotting Profiles of Axial Velocity at All Axial Locations

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

File	Setting Up	Domain	Settir	ng Up Physic	us User-Defined	Solving	Postpro	cessing	Viewing	Paralle	l Design	8		
Display	y Check	Quality		ransform ▼ Inits	Make Polyhedra Smooth/Swap Reorder		• Deact	ete ivate rate	Append Replace Mes Replace Zon	_	Mesh Overset	<ul> <li>Dynamic Mesh</li> <li>Mixing Planes</li> <li>Turbo Topology</li> </ul>	Mark/Adapt Cells Manage Registers More	Partition
			Mesh –				7	ones			Interfaces	Mesh Models	Adapt	Imprint
Tree			Та	isk Page			×				Window 1			Point
> 🔮 Setu > 續 Solu > 🥩 Resu	ition	stomizati	on s	Solution Se The task pag allow you to setup tasks.	etup Jes accessed under perform the most of Additional problem issed through the m	ommon prob etup tasks	Jp lem	5 ◆ ● ● ● ● ● ● ● ● ● ● ● ● ●			Line/Rake     Options     Line Tool     Reset     End Points     x0 (m) 0.522     y0 (m) 0     z0 (m) 0     New Surface     x=10d	2 Surface Type Line • 38 x1 (m) 38 x1 (m) 21 (m) 21 (m) Select Points with I Name		Point Plane Quadric Iso-Surface Iso-Clip Transform
												0	2 (m)	

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

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

When all lines are created, click Close.

Click **Tree > Results > Plots**. Right click **XY Plot** and click **edit...**.Click **Load File** and select *"axialvelocityAFD-laminar-pipe.xy"*, which is downloadable from the class website. Click **OK**.

Solution XY Plot				$\times$
Options Vode Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 0 Y 1 Z 0	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vect	1	
File Data [1/1] E =	Load File Free Data	Surfaces outlet wall x=100d x=10d x=20d x=40d x=60d New Surface	[6/10] •	
Plot	t Axes Cur	Close	Help	

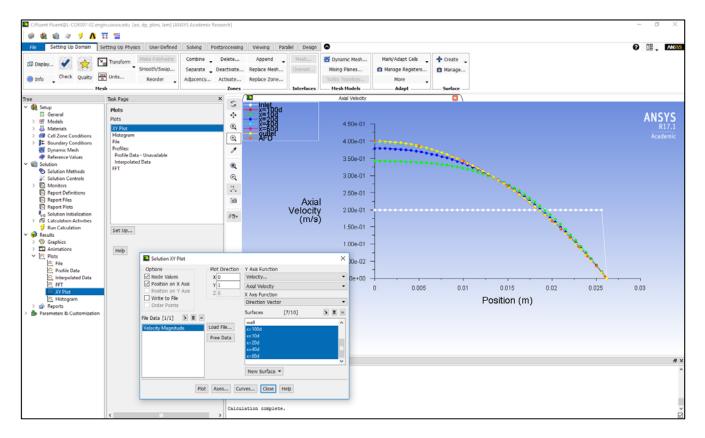
Change Parameters as per below. Make sure to select the inlet and outlet as well.

Solution XY Plot		_		$\times$			
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points File Data [1/1]	Plot Direction X 0 Y 1 Z 0	Y Axis Function Velocity Axial Velocity X Axis Function Direction Vector Surfaces	, DN				
Velocity Magnitude	Load File Free Data	outlet wall x=100d x=10d x=20d x=40d x=40d New Surface	e <b>v</b>	^ ~			
Plot Axes Curves Close Help							

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

🛄 Curv	Curves - Solution XY Plot								
Curve # 0 🜩 Sample		Line Style Pattern Color foreground Weight 1	•	Marker Style Symbol (*) • Color foreground Size 0.3	•				
	Apply Close Help								

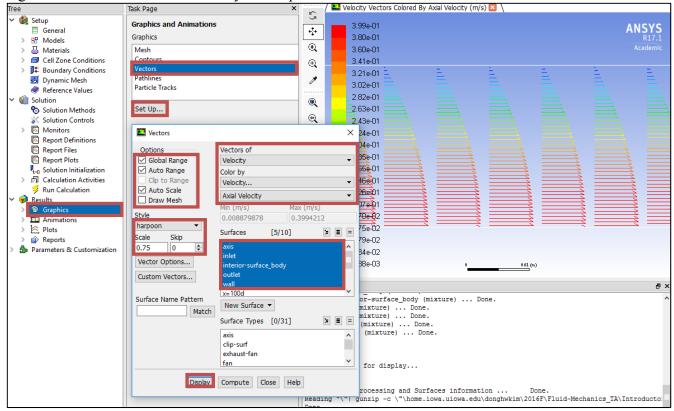
#### Click Plot.



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

#### 7.8. Plotting Velocity Vectors

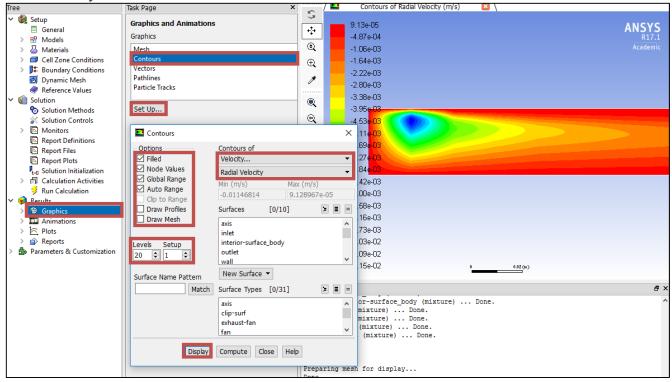
Click **Tree > Results > Graphics > Vectors > Set Up.** To plot the velocity vectors at the region flow begin to becomes fully developed, copy the parameters as per below and click **Display**. Zoom into the region where the flow is almost fully developed.



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

#### 7.9. Plotting Contours of Radial Velocity

Click **Tree > Results > Graphics > Contours > Set Up...** To plot the Contours of Radial Velocity, copy the parameters as per below and click **Display**. Zoom in to the pipe inlet to see the contours of radial velocity.



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

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

To export solution value, click **Tree > Results > Plots.** Double click **XY Plot**. To export the Developed Axial Velocity Profile at x=100d, copy the parameters as per below and click **Write...** 

Solution XY Plot				×		
Options Options Node Values Position on X Axis Position on Y Axis Vrite to File Order Points	Plot Direction X 0 Y 1 Z 0	Y Axis Function Velocity Axial Velocity X Axis Function Direction Velocity	/ on	•		
File Data [0/1] 🕨 🔳	E Load File	Surfaces axis inlet interior-surfa outlet wall x=100d x=10d				
Write Axes Curves Close Help						

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

🞴 Select File									?		Х
Look in: H:\Intro Fluids\CFD Pre-Lab 1										::	
Name	~	Size	Туре	Date Modified							
.CFD Pre	-Lab 1_files.backup		Filder	9/11/201:35 PN							
	Lab 1_files		9/11/209 <b>:</b> 14 P№								
	prelab_files Fil…der 8/30/20…8:18 PN										
🛛 💁 axialvelo	cityAFD-laminar-pipe.xy	290te	<sup>s</sup> xy File	8/30/204:38 AN							
XY File CFD Pre-Lab 1 Laminar Pipe Flow Developed Axial Velocity Profile							ОК				
Files of type: XY Files (*.xy)							C	ance			
Filter String								1	Filter		

#### 7.11. Exporting Wall Shear Stress Distribution

To export the wall shear stress distribution, click **Tree > Results > Plots.** Double click **XY Plot**. Copy the parameters as per below and click **Write...** 

Solution XY Plot			×			
Options	- Plot Direction	Y Axis Function				
✓ Node Values	X 1	Wall Fluxes  Wall Shear Stress				
Position on X Axis	Y 0					
Position on Y Axis Write to File	Z 0	X Axis Function				
Order Points		Direction Vector				
		Surfaces [1/10]	3 2 -			
File Data [0/1]		axis  inlet interior-surface_body				
	Load File					
	Free Data					
		outlet				
		wall				
		x=100d	~			
		x=10d				
		New Surface 🔻				
Write Axes Curves Close Help						

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

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

- 7.12. Normalizing Velocity Profile
  - Open excel from Start Menu.
  - Click File > Open, navigate to your folder you created on the network drive.
  - Change the file type to all files.
  - Select the file CFD Pre-Lab 1 Laminar Pipe Flow Developed Axial Velocity Profile and click **Open**.
  - Select **Yes** on the **Microsoft Excel** message. (You might not see this pop-up depending on the version of Excel being used, if not then please proceed to next step)

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

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

Text Import Wizard - S	Step 2 of 3					?	×
This screen lets you set preview below. Delimiters Jab Semicolon Comma Space Qther: Data greview	t the delimiters your d ✓ Treat consecutive Text <u>q</u> ualifier:			see how yo	ur text is affected	l in the	
(title	Axial Velocity)						<b>^</b>
(labels	Position	Axial Velo	city)				
( (m) (hen () ehe)	-1003						
((xy/key/label) 0	0.399408						<b>v</b>
<	1	I				3	
		Cancel		< <u>B</u> ack	<u>N</u> ext >	<u>F</u> ini:	sh

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

⊟ 5 • ੋ ∗ 🖆 👳											
F	ile	Home	Inse	rt Pag	ge Layout	Formulas	Da				
Pa	te C	opy 🔹 ormat Pa	iinter G	Calibri B I		11 - A					
C5	C5 ▼ : × ✓ f <sub>x</sub> =B5/\$B\$5										
	А		в	С	D	E	F				
1	(title	Axia	l Veloc	ity)							
2	(labels	Posit	tion A	Axial Vel	ocity)						
3											
4	((xy/ke	y/l x=10	0d)								
5		0 0.39	9408	1	L						
6	0.0005	82 0.39	9309								
7	0.0011	64 0.3	89872								
8											

- Insert a Scatter Plot With Smooth Lines and Markers.
- For the x-axis use the radial position, and for the y-axis use the normalized velocity.
- Name it CFD Velocity Profile (Laminar).
- You can move this plot to a new tab by clicking on the chart **Chart Tools** > **Design** > **Move Chart Location** > **New Sheet** > **OK**
- Next open the file "*Normalized-velocity-AFD-laminar-pipe.xy*" in TextPad, highlight the data and paste it into your Excel spread sheet next to the CFD velocity profile data.
- Plot this in the same way as the other set on the existing plot and call this "*AFD Velocity Profile* (*Laminar*)".
- Create axis titles and make sure the legend is shown. You should move the legend to the bottom of the chart. Call the axes *Normalized Velocity* [-] and *Radial Position* [m].
- Save this Sheet by selecting File > Save As, name it "*CFD Pre-Lab 1 Developed Axial Velocity Profile*".

### 8. Exercises

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

### **Simulation of Laminar Pipe Flow**

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

#### 8.1. Compare CFD with AFD on friction factor

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

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

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

**Figures need to be saved:** (1) Residual History (2) Centerline Pressure Distribution (3) Centerline Velocity Distribution (4) Wall Shear Stress Distribution (5) Profiles of Axial Velocity at all Streamwise Locations (inlet, outlet, x/D=10,20,40,60,100) with AFD data (6) Contour of Radial Velocity (7) Velocity Vectors in Fully Developed Region.

**Data need to be saved:** (1) Shear Stress in the Developed Region (2) Developing Length (=Length from the pipe inlet to the start of developed region. Use figure of centerline velocity distribution).

#### 8.2. Normalized developed axial velocity profile

- 8.2.1. Export the axial velocity profile data at x=100d following the instructions in Step 7.10.
- 8.2.2. Use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude, which is the maximum value on that profile. Plot the normalized velocity profile in EXCEL and paste the figure into your report together with other figures you made in Exercise 8.1.

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

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