Simulation and Validation of Turbulent Pipe Flows

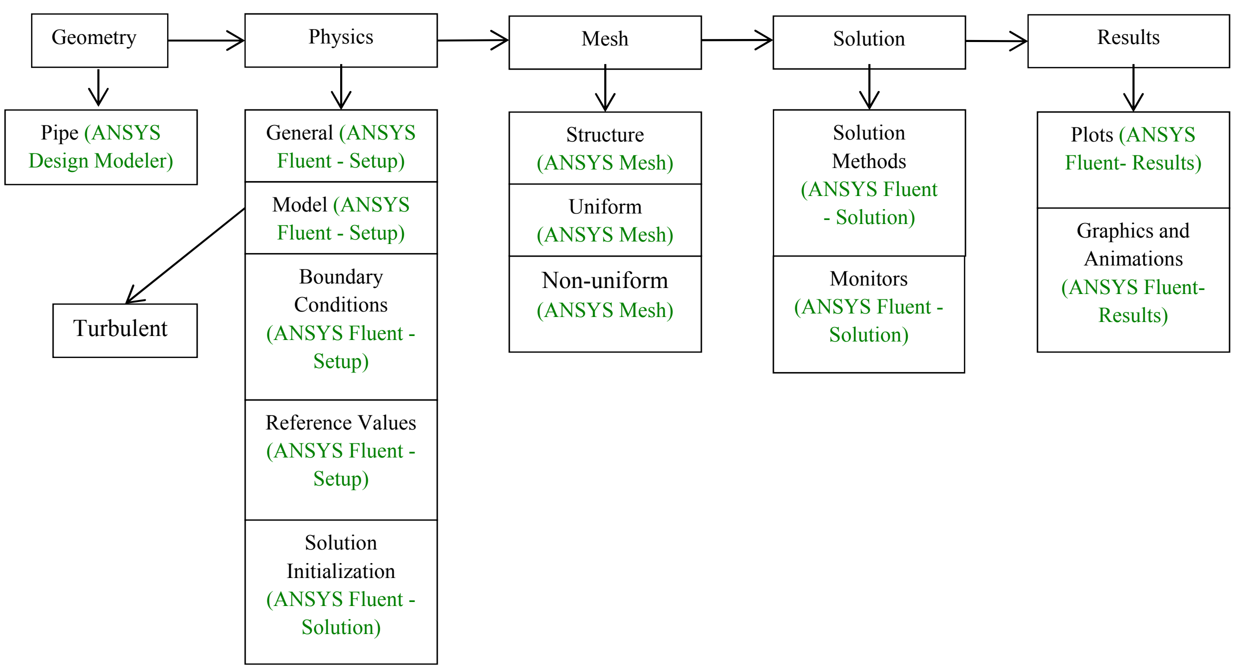
ENGR:2510 Mechanics of Fluids and Transport Processes
CFD LAB 1
(ANSYS 17.1; Last Updated: Oct. 10, 2016)

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

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

1. Purpose

The Purpose of CFD Lab 1 is to teach students how to use ANSYS, practice more options in each step of CFD Process, and relate simulation results to EFD and AFD concepts. Students will simulate **turbulent** pipe flow following the "CFD process" by an interactive step-by-step approach. The flow conditions will be the same as they used in EFD Lab2. Students will have "hands-on" experiences using ANSYS to compute axial velocity profile, centerline velocity, centerline pressure, and wall shear stress. Students will compare simulation results with their own EFD data, analyze the differences and possible numerical/experimental errors, and present results in a CFD Lab report.



Flow chart for "CFD Process" for pipe flow

2. Simulation Design

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

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

Table 1 - Main particulars

Parameter	Unit	Value
Radius of Pipe	m	0.02619
Diameter of Pipe	m	0.05238
Length of the Pipe	m	6.096

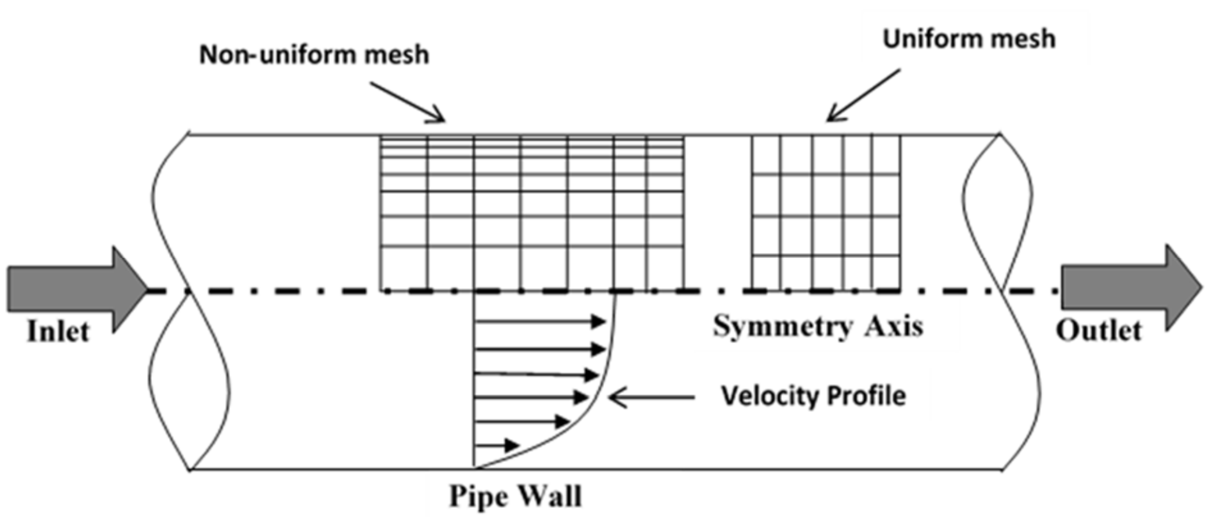


Figure 1 - Geometry

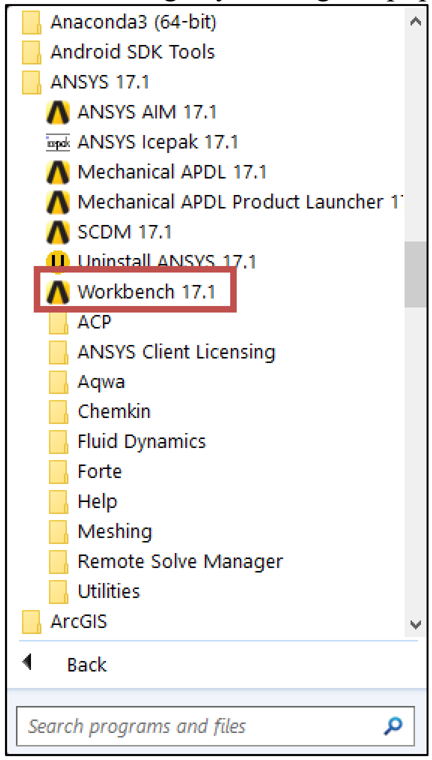
The problem formulation is similar to that in CFD PreLab1 and will not be repeated here. The Reynolds number is much higher and students need specify turbulence model, and thus more variables need to be specified in boundary conditions, as will be discussed in details later.

Navigation Tips

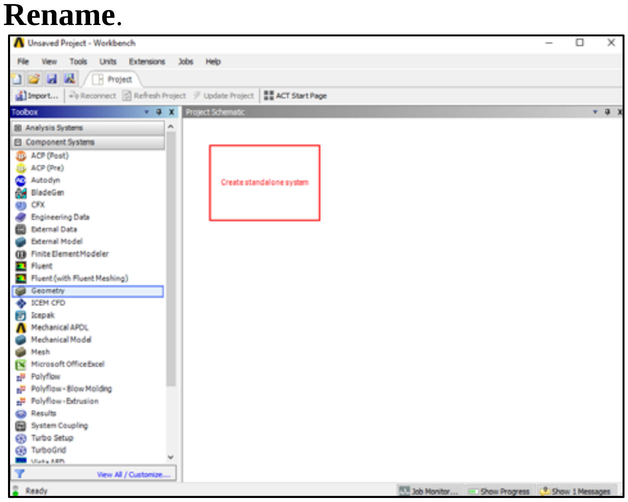
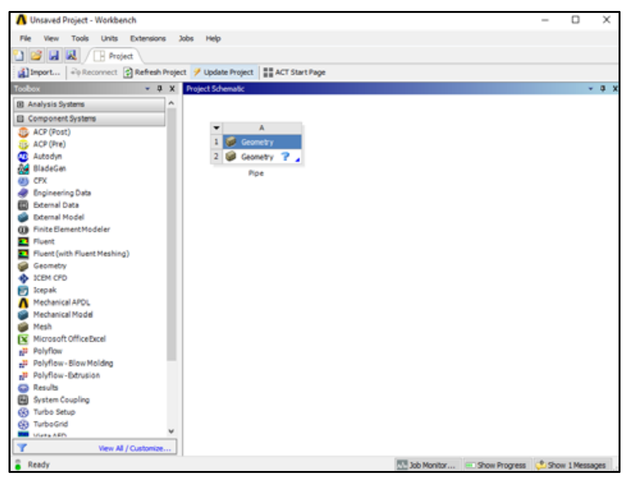
- To zoom in and out use the magnifying glass with a plus sign in it and drag, from top left to bottom right over the are you wish to zoom.
- To look at a view plane, simply click on the arrow in the coordinate system identifier in the bottom right of the screen. i.e if you wish to look at the XYplane,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

3. Open ANSYS Workbench

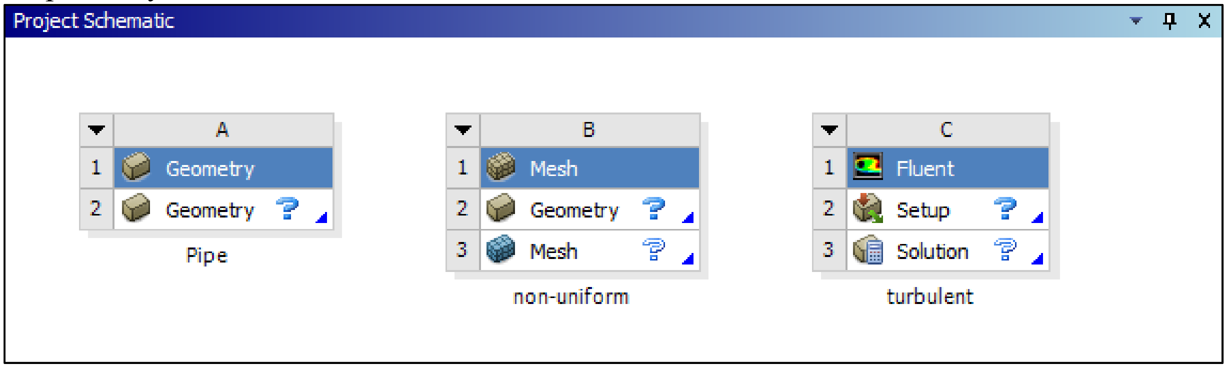
3.1. **Start** > **All Programs** > **ANSYS 17.1** > **Workbench 17.1**
(Note: You may ignore the firewall warnings by closing the pop-up window**)**



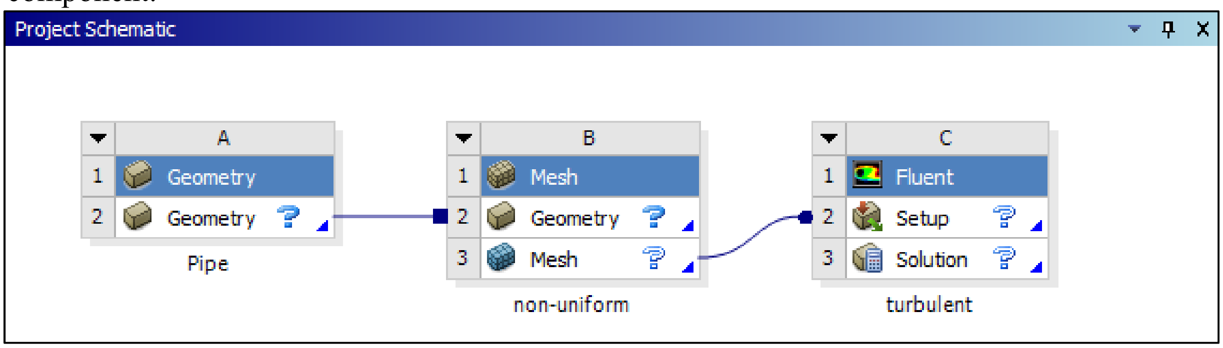
3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop the **Geometry** component for the **Component Systems** in the **Toolbox** into the **Project Schematic**. Rename the geometry "Pipe" by right clicking on the down arrow of the **Geometry** component and selecting **Rename**.

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



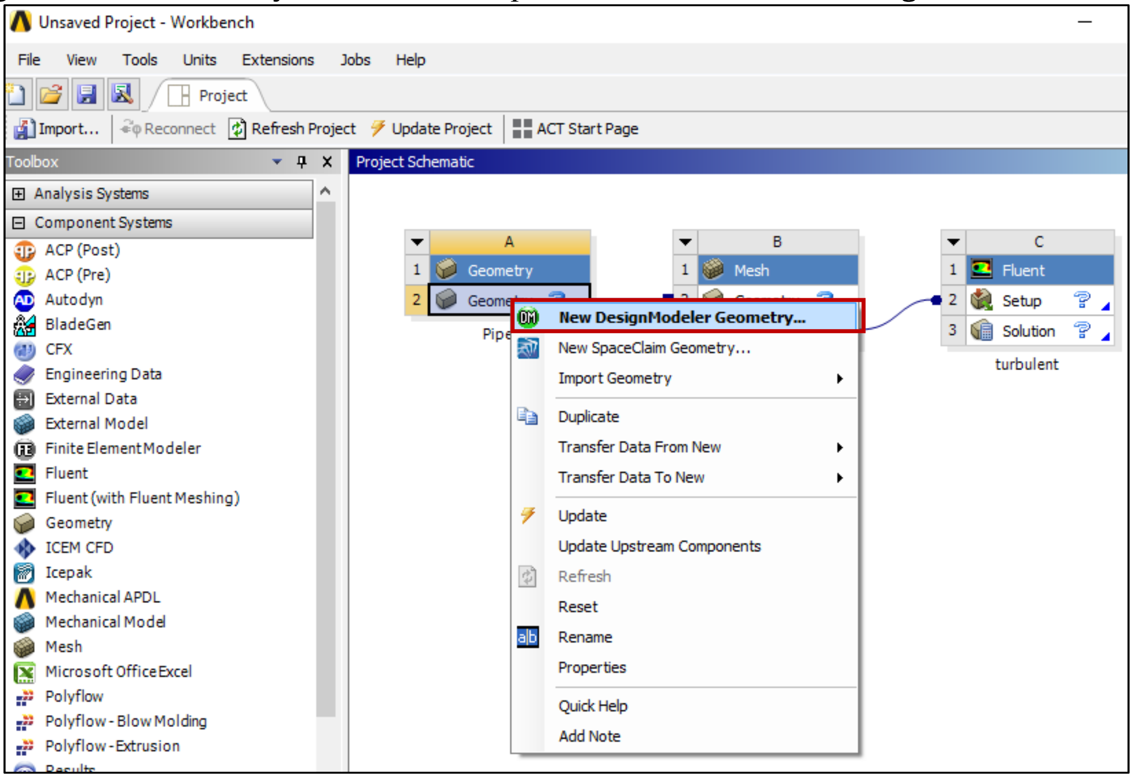
3.4. Create connection as per below by dragging and dropping each component to the corresponding component.



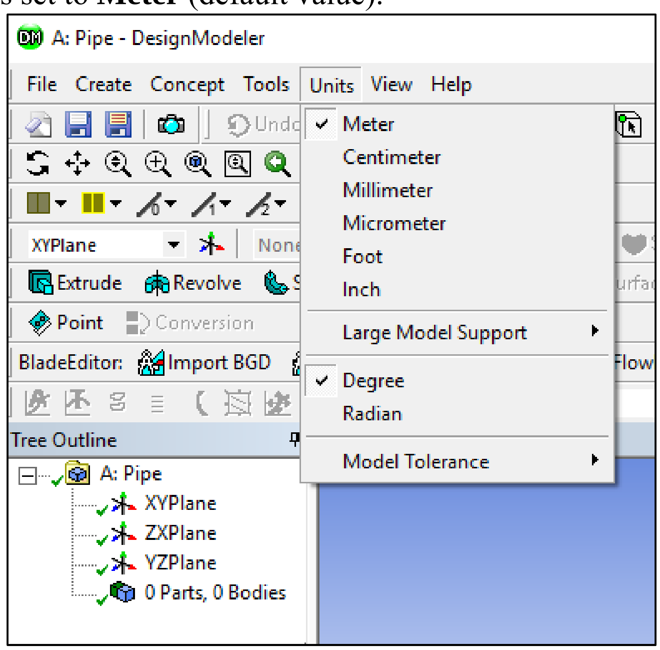
3.5. Create a Folder on the network drive called *"CFD Lab 1"*.
3.6. Save the project file by clicking **File** > **Save As…**
3.7. Save the project onto the network drive in the folder you just created and name it "*CFD Lab 1 Turbulent Flow*".

4. Geometry

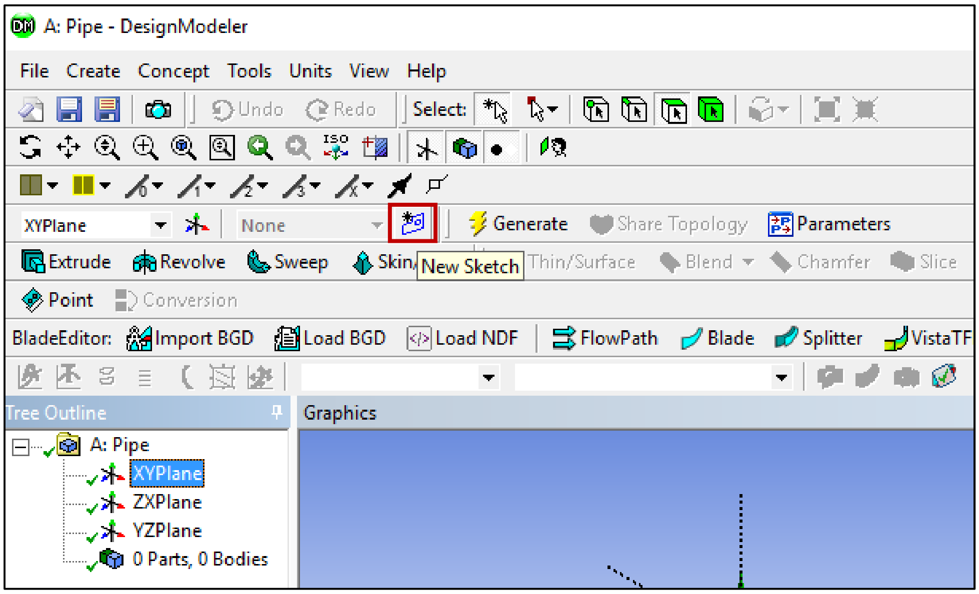
4.1. Right click on **Geometry** and from the drop down menu select **New DesignModeler Geometry…**



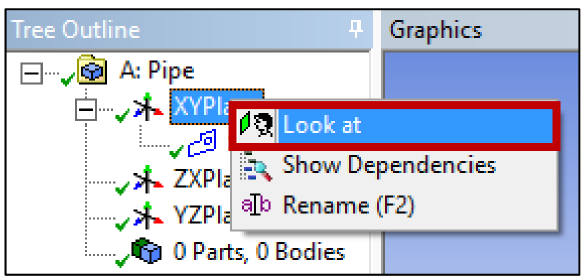
4.2. Make sure that Unit is set to **Meter** (default value).



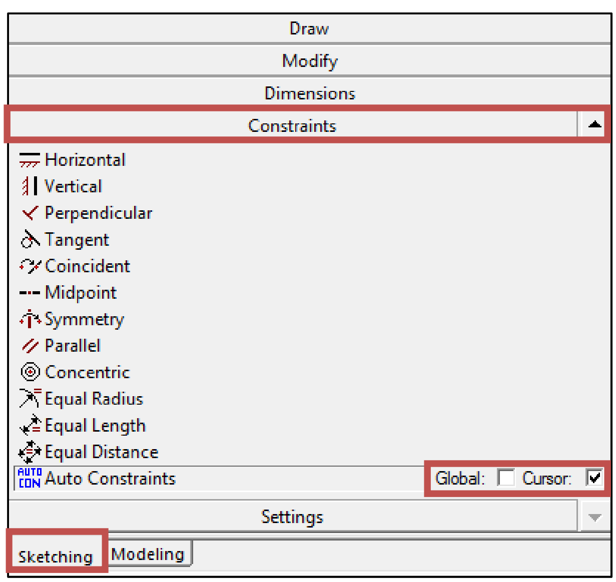
4.3. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.



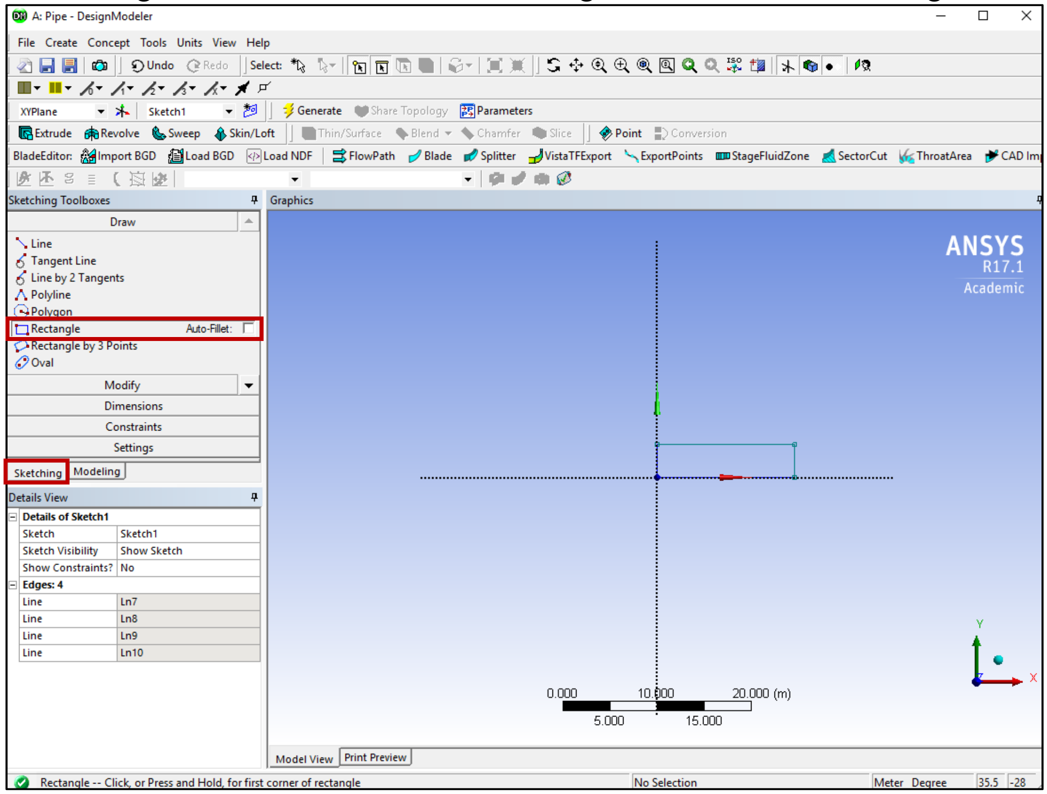
4.4. Right click **XYPlane** and select **Look at**.



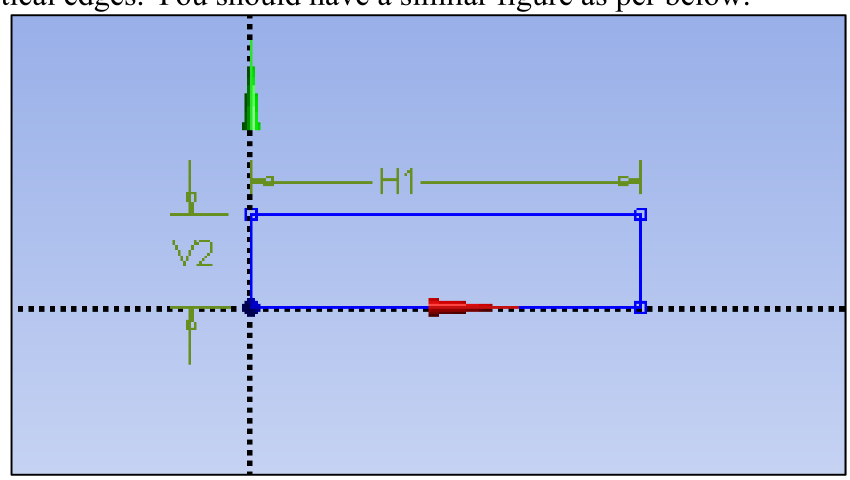
4.5. Select **Sketching** > **Constraints** > **Auto Constraints**. Enable the auto constraints option to pick the exact point as below.



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

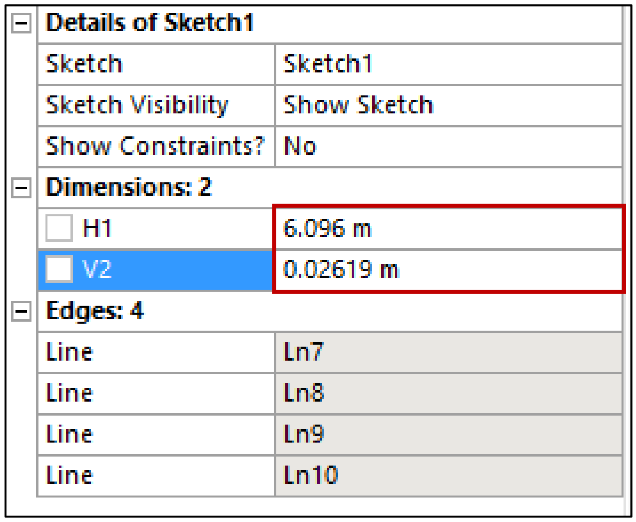


4.7. Select **Dimensions** > **General**. Click on top edge then click anywhere. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.

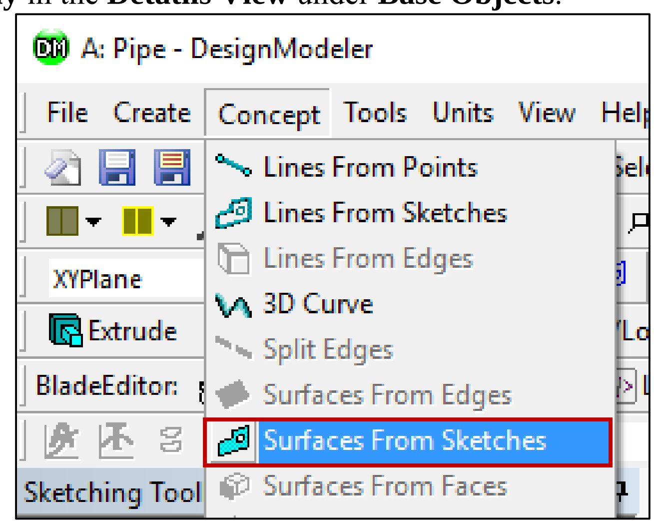


4.8. Click on **H1** under **Details View** and change it to *6.096m*. Click on **V2** and change it to *0.02619m*.

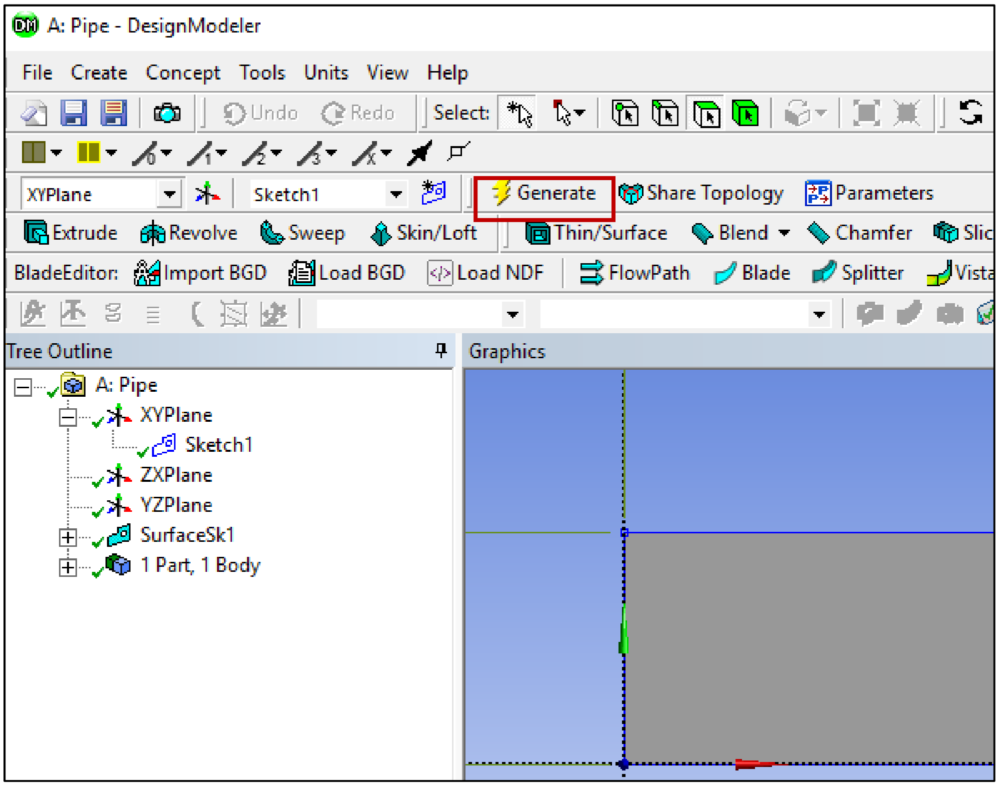
NOTE: The actual length of the pipe is 30 feet (9.144m), However, in CFD simulation, we need to specify “outlet pressure”, and we don’t have a pressure transducer at the pipe outlet. So we choose the outlet of the pipe we will simulate to be the location of the last pressure transducer, which is 6.096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



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



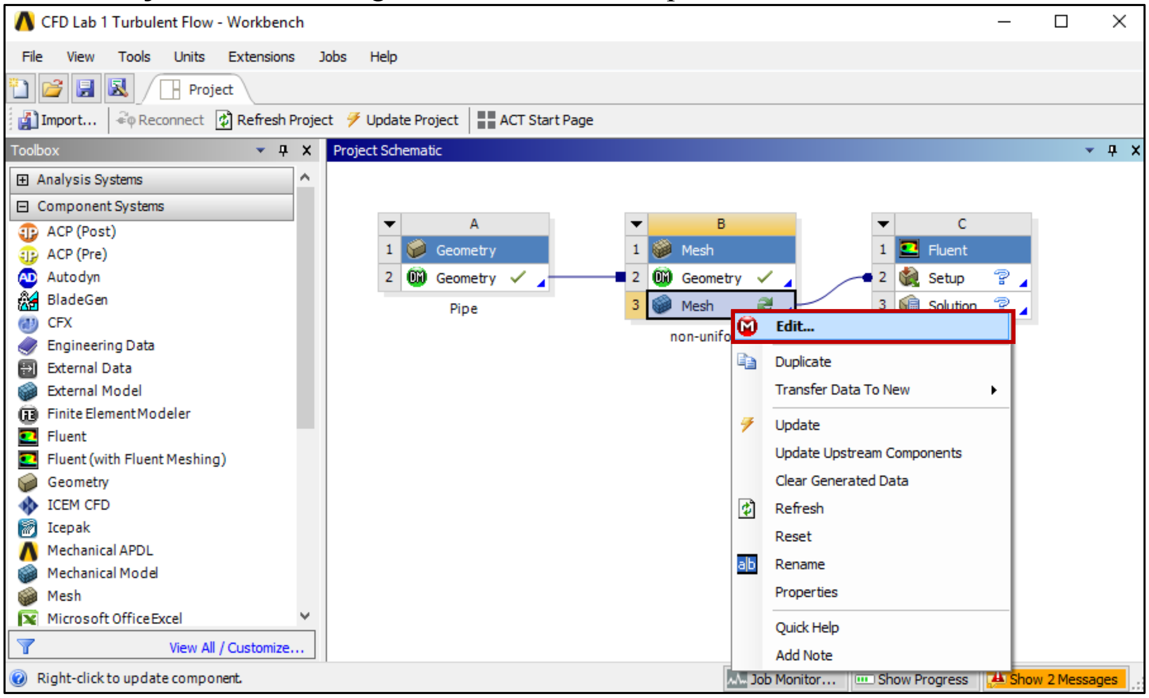
4.10. Click **Generate**. This will create a surface.



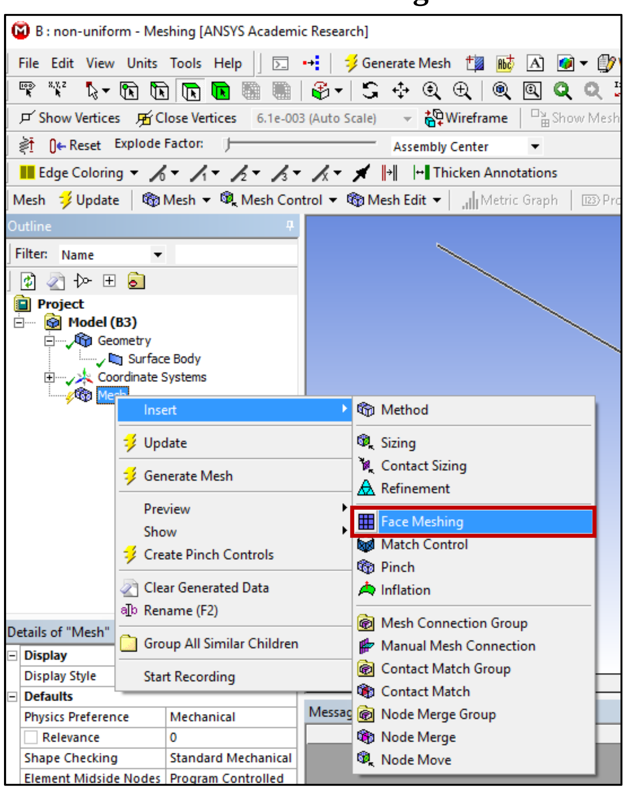
4.11. **File** >**Save Project**. Save project and close window.

5. Mesh Generation

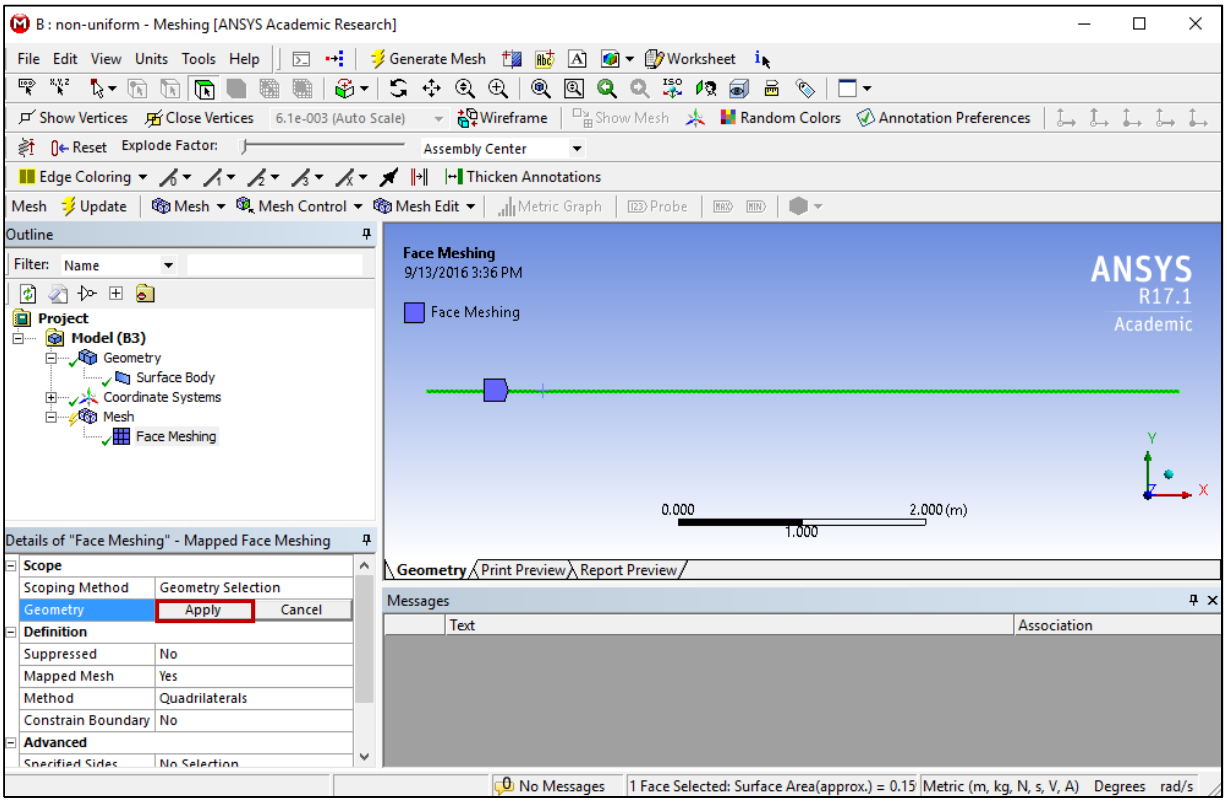
5.1. From the **Project Schematic** right click on **Mesh** component and select **Edit…**



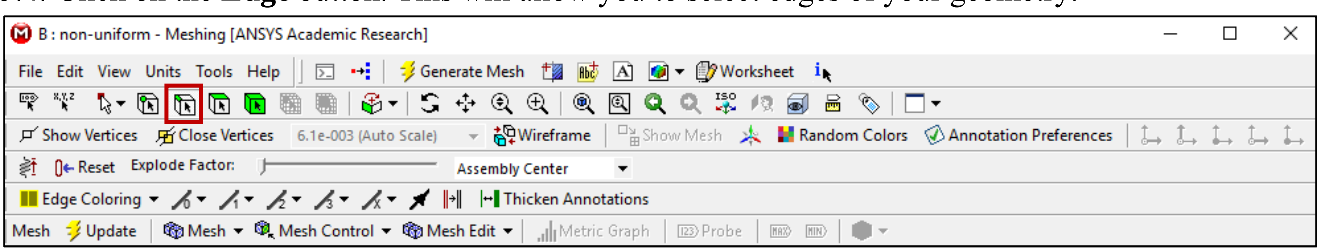
5.2. Right click on **Mesh** then select **Insert** > **Face Meshing**.



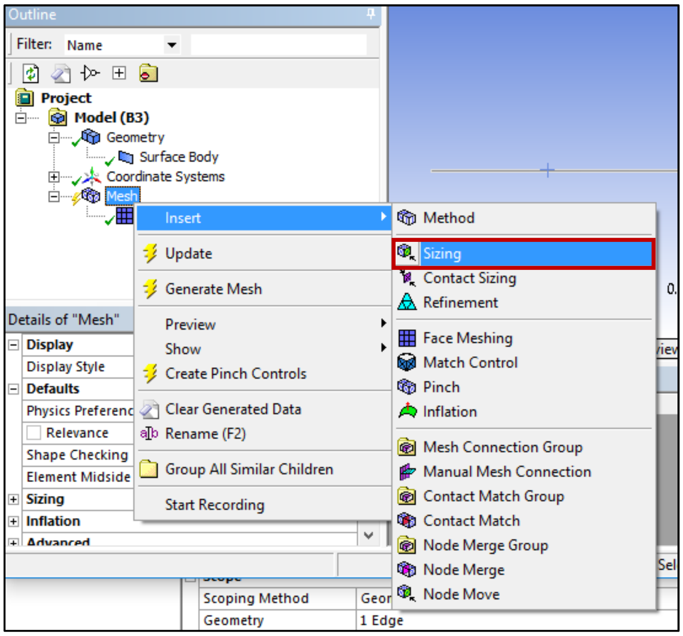
5.3. Select your geometry by clicking on the yellow box which says **No Selection**, then select the geometry surface and click **Apply**. (Note: You can change orientation of your view by clicking the axis figure on the lower right corner. Press "F7" on your keyboard to restore to the "whole view".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



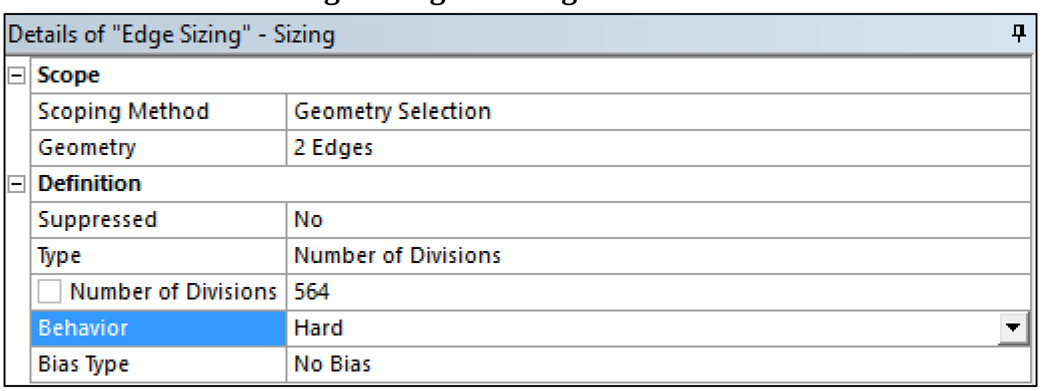
5.4. Click on the **Edge** button. This will allow you to select edges of your geometry.



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



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


Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	564
Behavior	Hard
Bias Type	No Bias

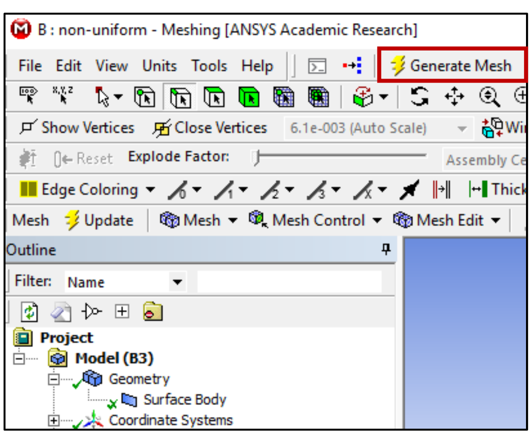
5.7. Repeat step 5.5. Select the left edge then click **Apply** and change the parameters as per below.

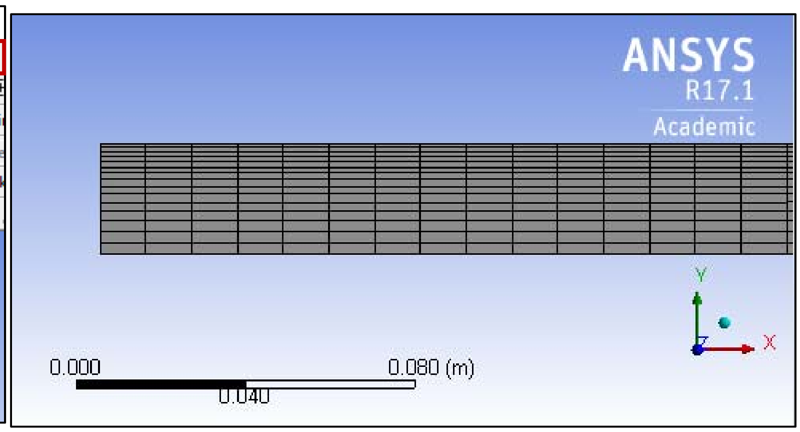
Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	15
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	3.1117
Reverse Bias	No Selection

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

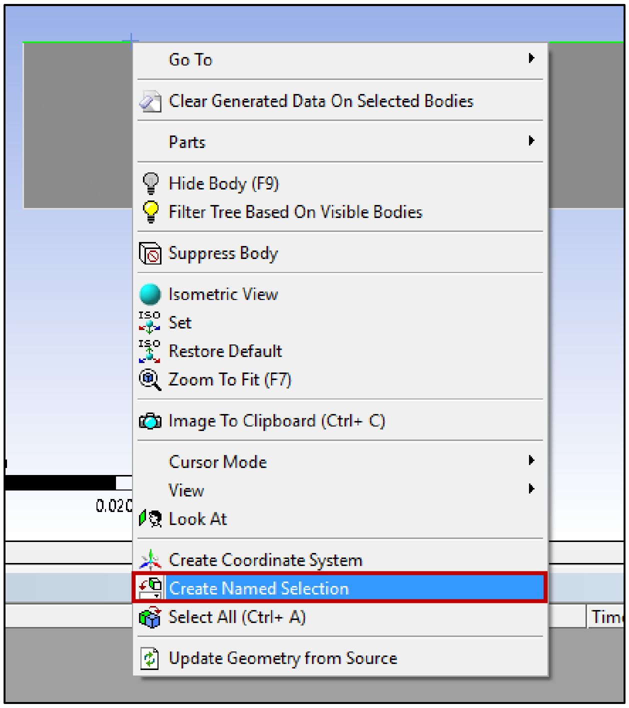
Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	15
Behavior	Hard
Bias Type	_____ ___ _ _
Bias Option	Bias Factor
Bias Factor	3.1117
Reverse Bias	No Selection

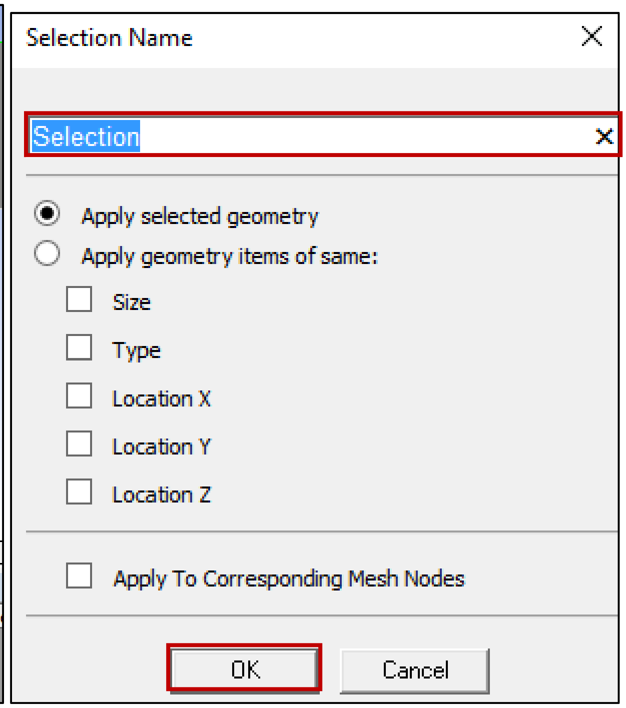
5.9. Click on **Generate Mesh** button and then select **Mesh** under **Outline**. The mesh should look like the mesh below.

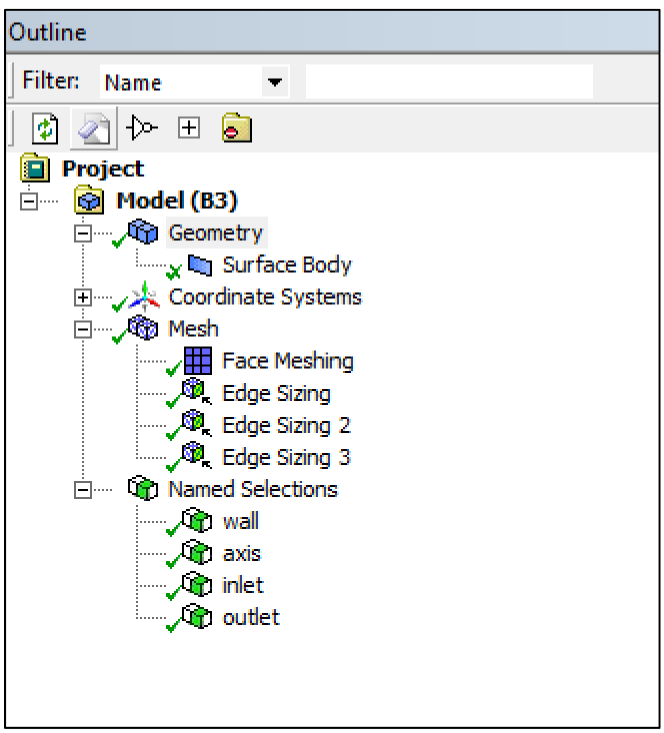




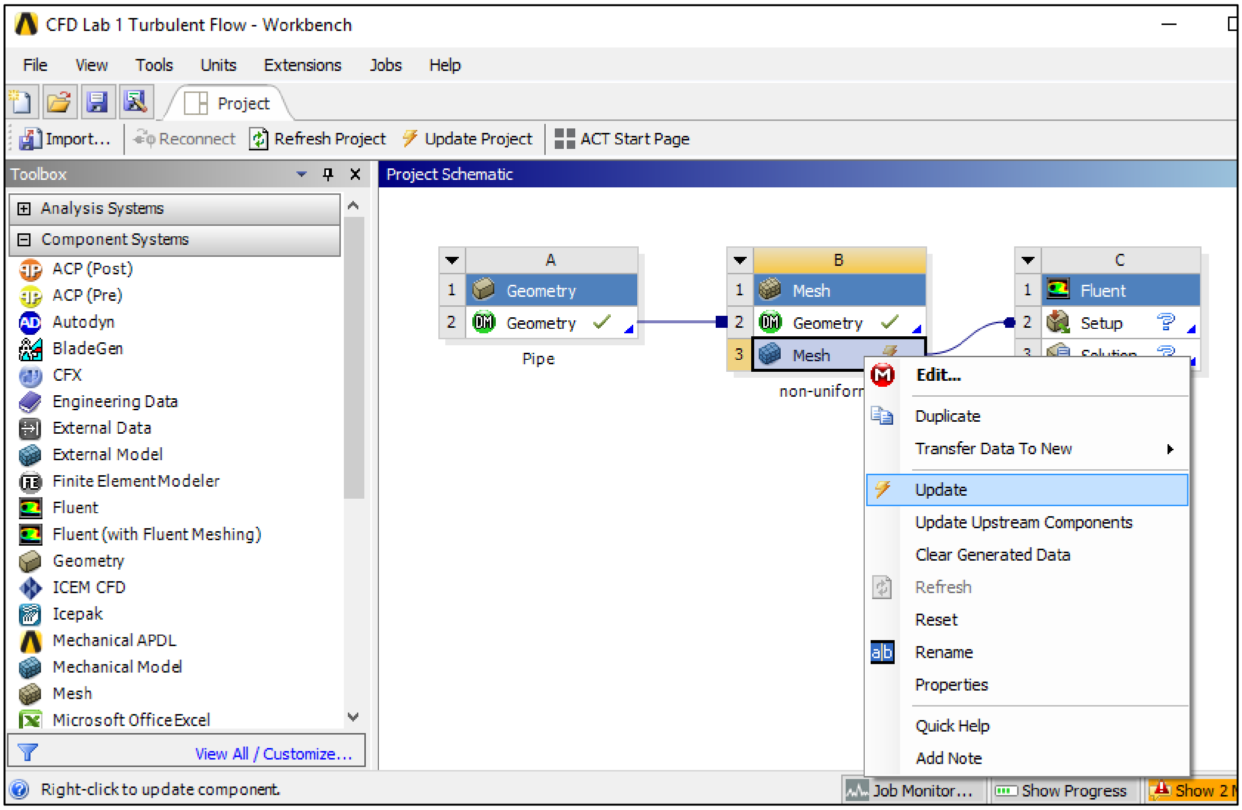
5.10. Change the edge names by selecting the edge, then right clicking on the edge and selecting **Create Named Selection** from the drop down menu. Name left, right, bottom and top edges as *inlet*, *outlet*, *axis* and *wall* respectively then click **OK**. Your outline should look same as the figure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





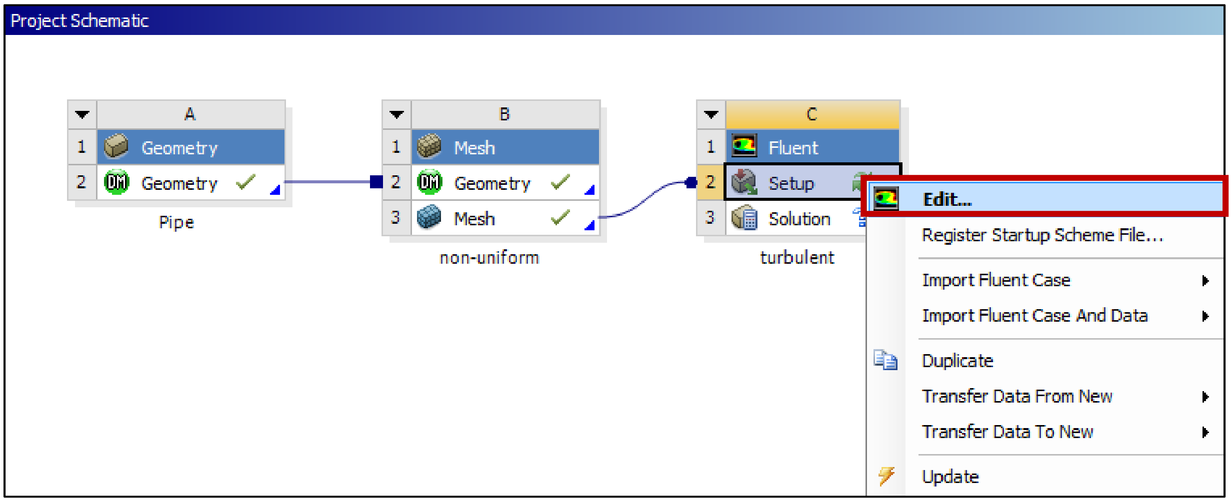


5.11. **File** > **Save Project**. Save the project and close the window. Update mesh on **Project Schematic** by right clicking on **Mesh** and selecting **Update.**

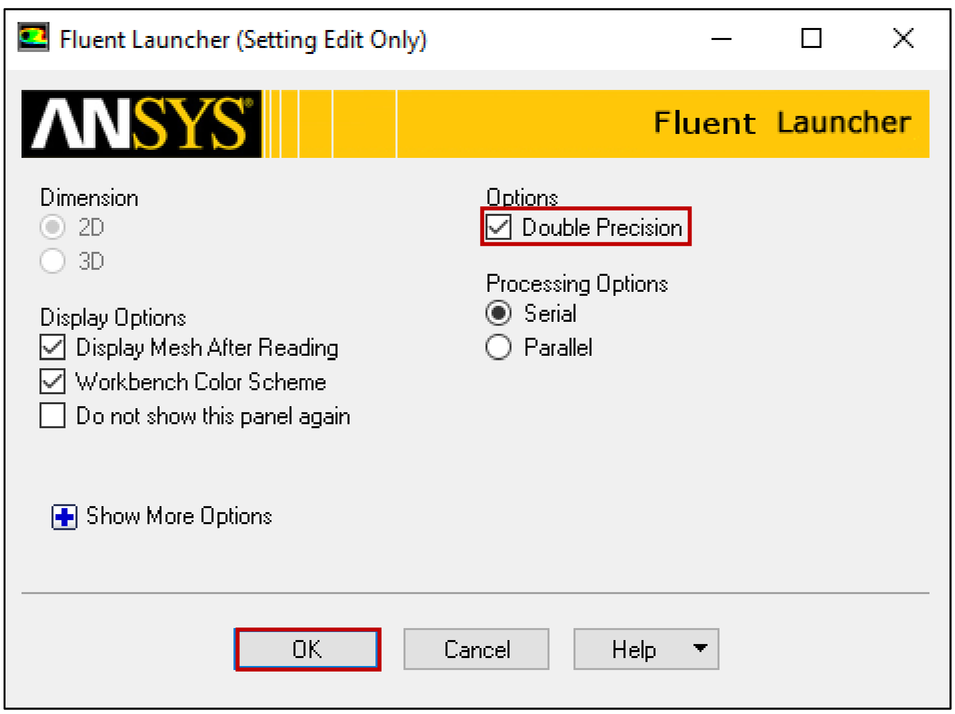


6. Setup (Physics)

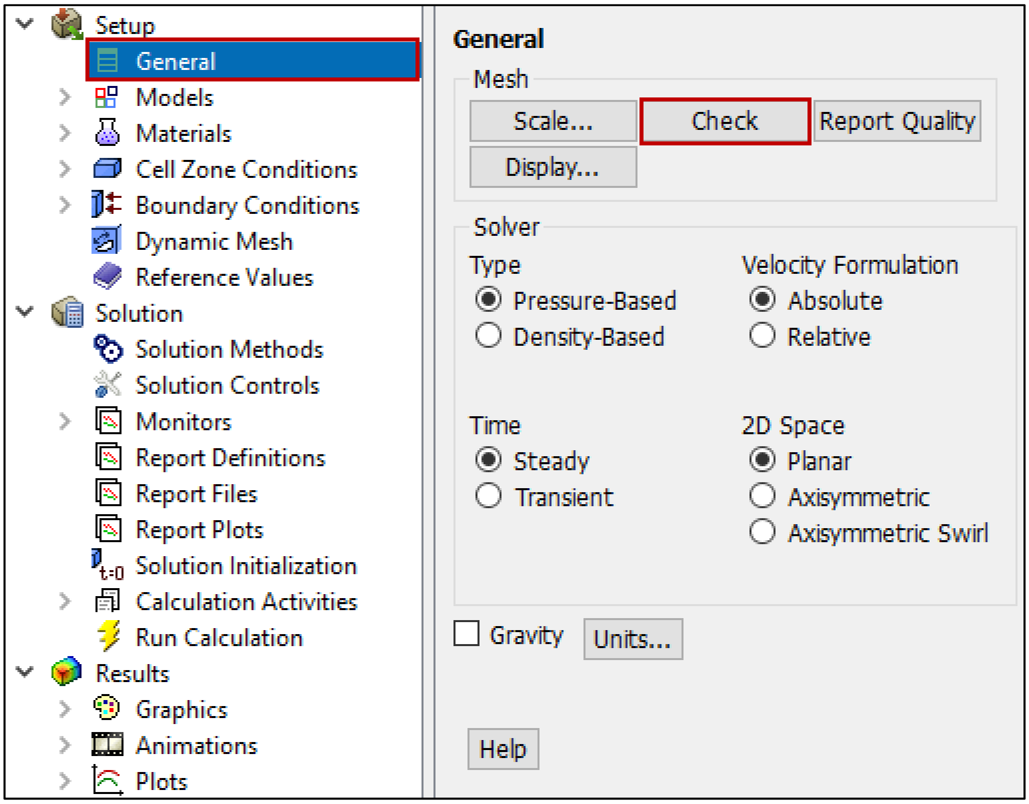
6.1. Right click **Setup** and select **Edit…**



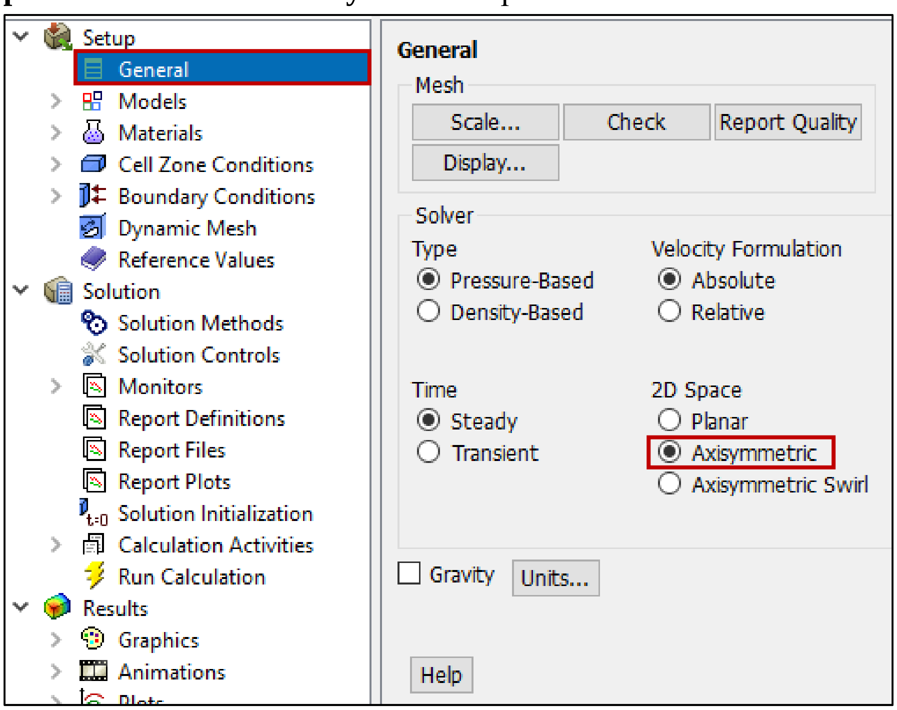
6.2. Check **Double Precision** and click **OK.**



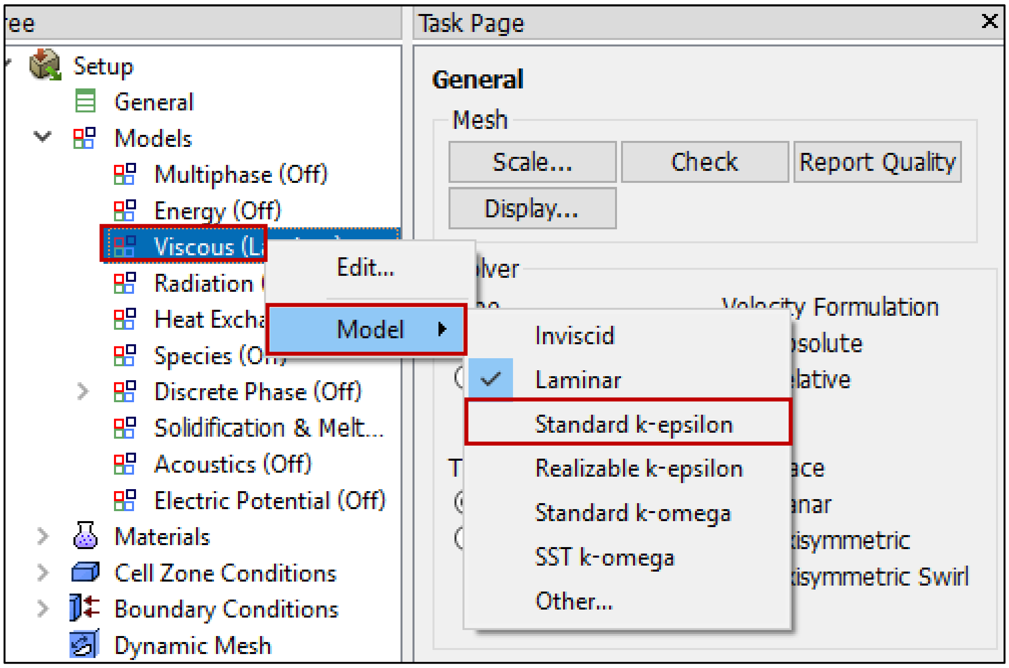
6.3. **Tree > Setup** > **General** > **Check**. (Note: If you get and error message you may have made a mistake while creating you mesh. Review steps in mesh generation and make changes.)



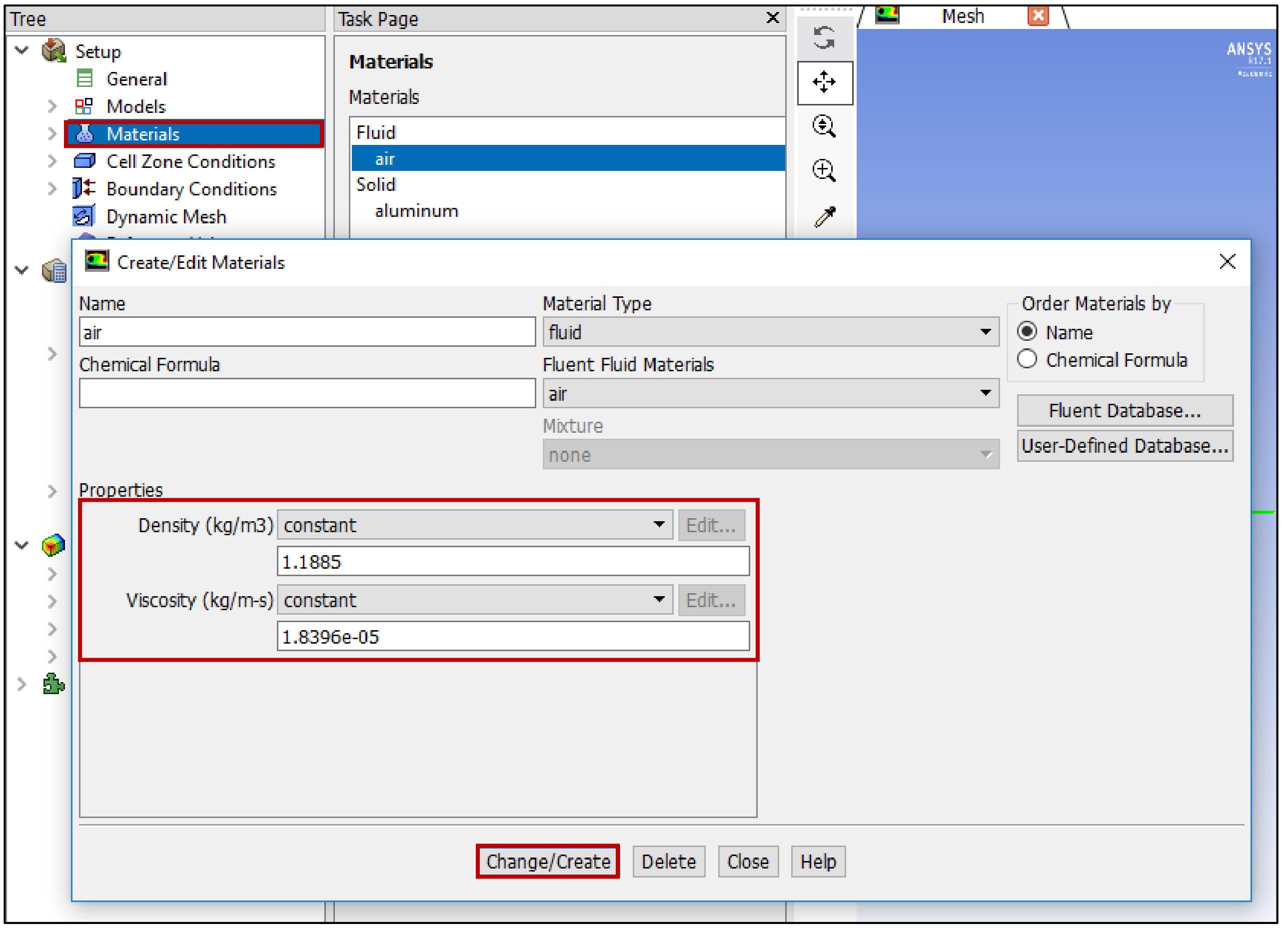
6.4. **Tree > Setup** > **General**. Choose **Axisymmetric** option shown below.



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

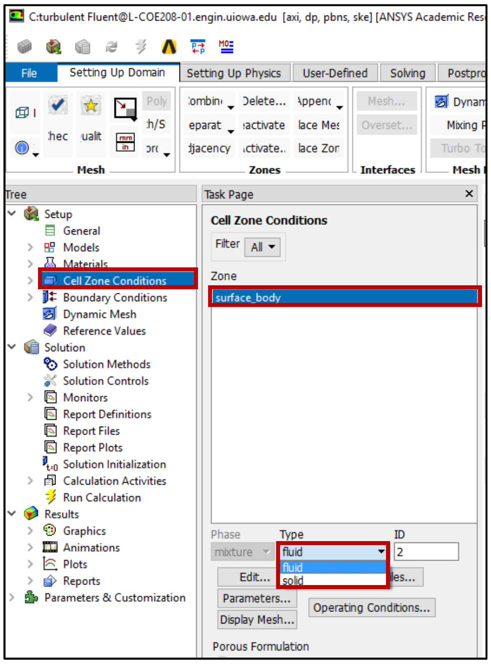


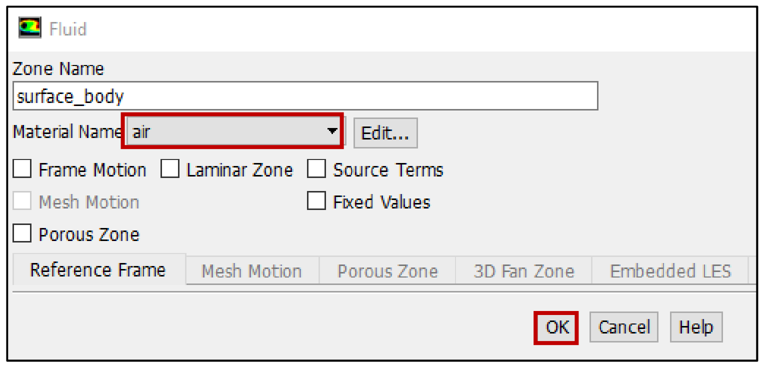
6.6. **Tree > Setup** > **Materials** > **air** > **Create/Edit**. Change the **Density** and **Viscosity** as per your experimental data. Click **Change/Create** then close the window. Use the flow property values in your EFD Lab2. If you only know the room temperature when you conducted EFD Lab 2,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



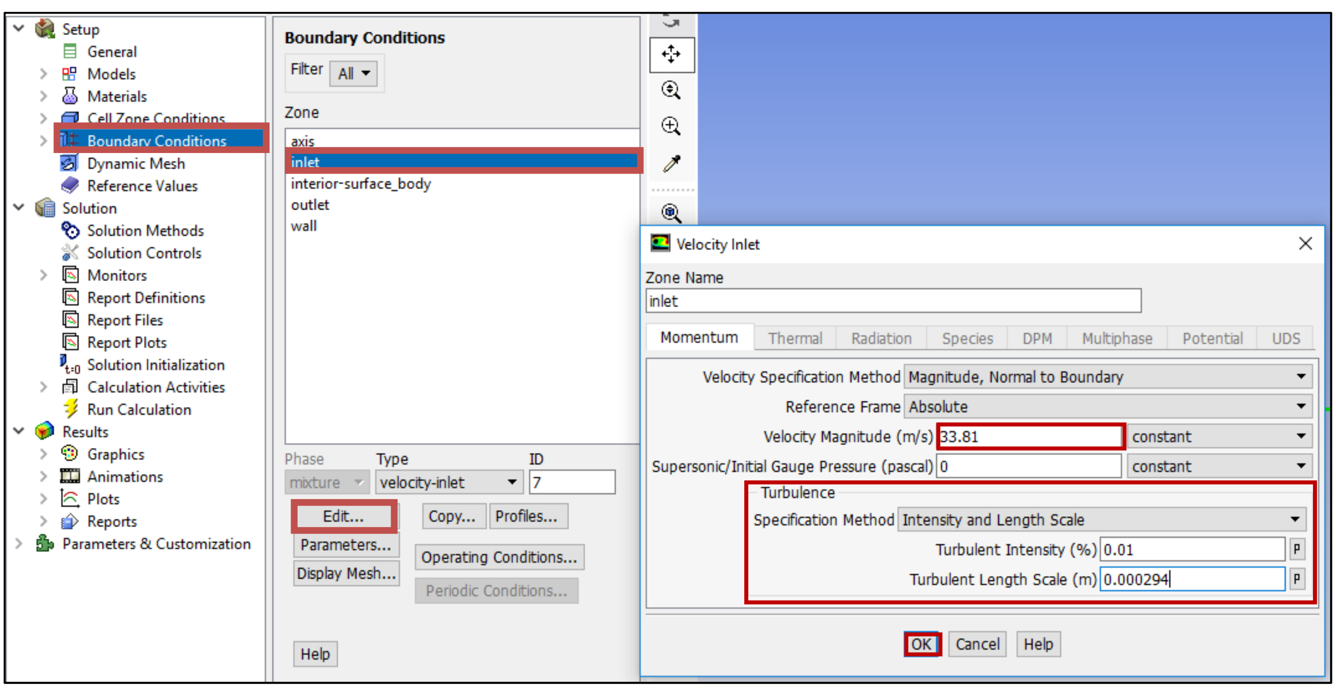
If the room temperature is 24°, the density and viscosity are the values shown in the above panel. **NOTE:** viscosity used in ANSYS is the **dynamic viscosity** ($kg/m \cdot s$), **NOT kinematic viscosity** (m^2/s)

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

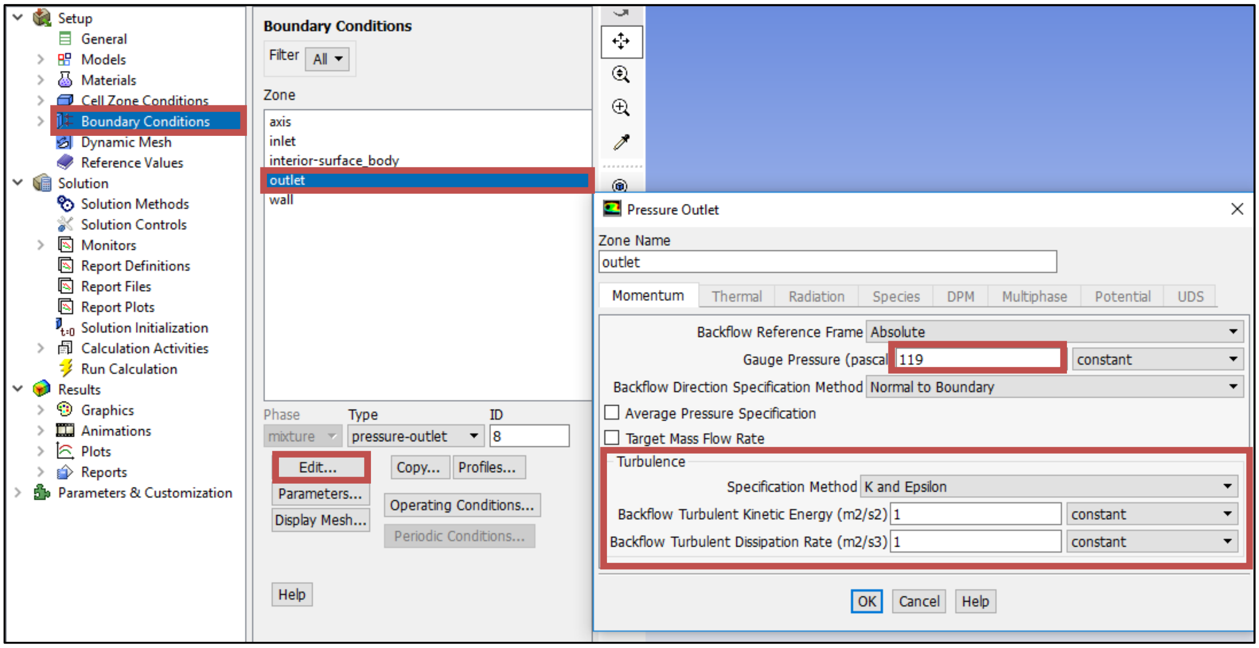




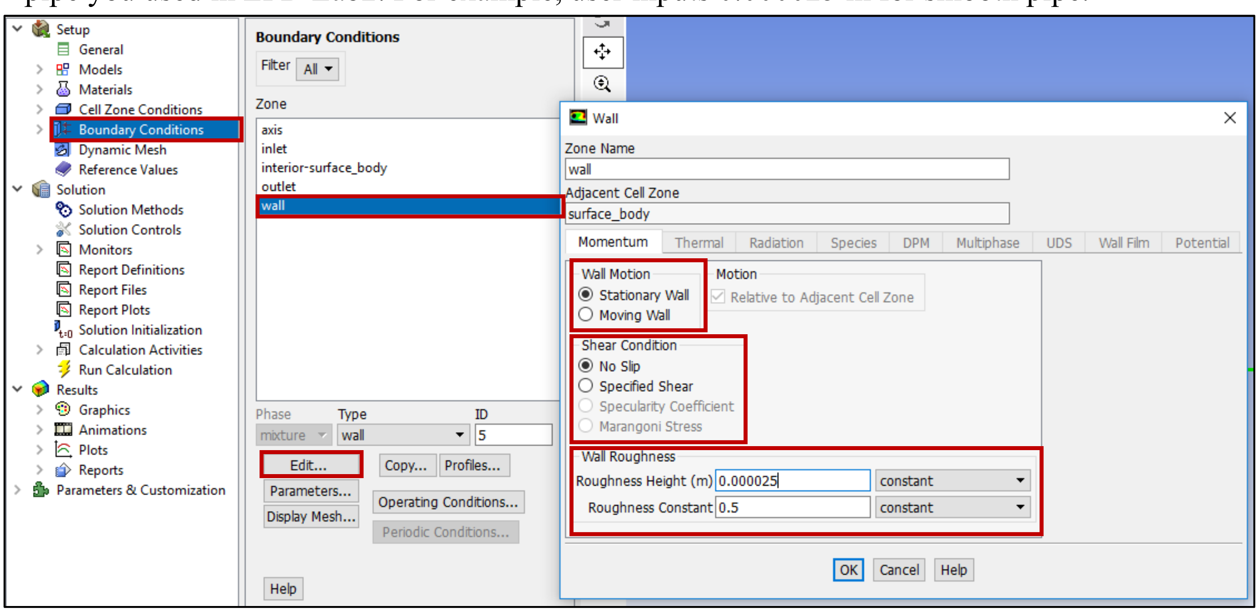
6.8. **Tree** > **Setup** > **Boundary Conditions** > **inlet** > **Edit…** Change velocity magnitude as per your experimental data. Change other parameters as below and click **OK**. You must calculate the inlet velocity u, which is uniform, based on the flow rate Q (m^3/s) you computed in EFD Lab2, and cross section area of the pipe $\pi D^2/4$, i.e. $u = 4Q/\pi D^2$.



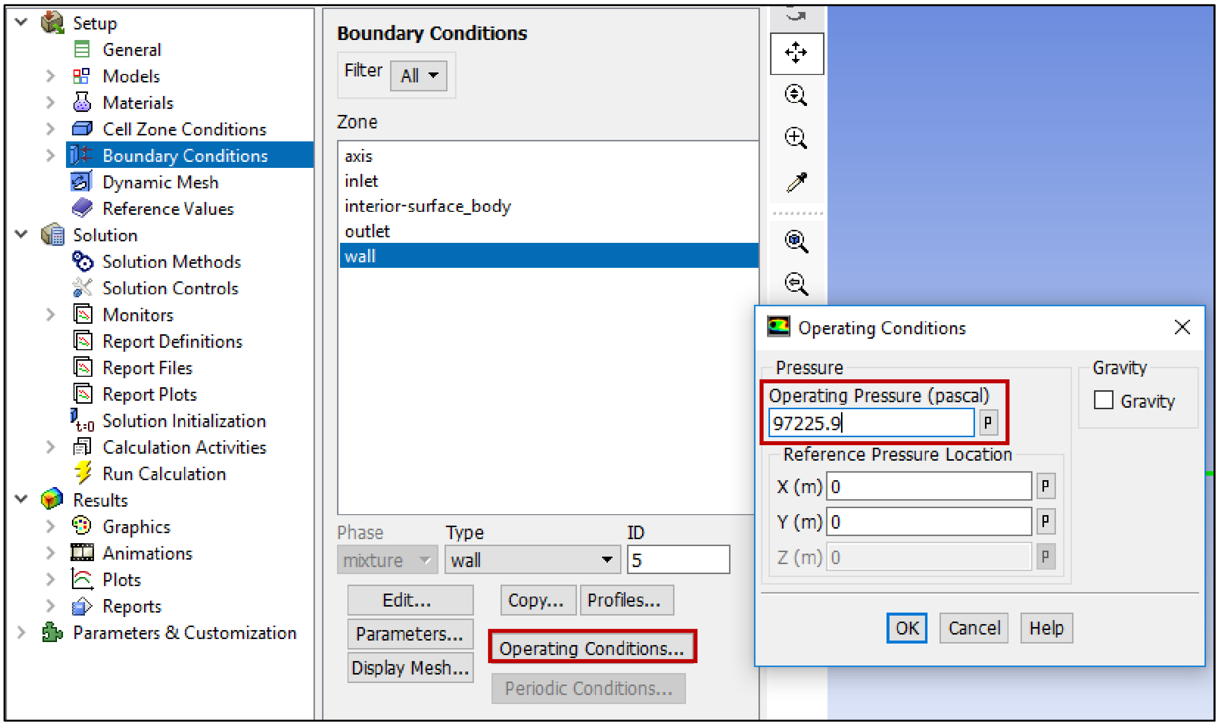
6.9. **Tree > Setup > Boundary Conditions > outlet > Edit…** Change gauge pressure as per experimental data. Change other parameters as below and click OK. You need to transform the four pressure tap pressure values from “feet water” to “Pascal” and input pressure tap #4 value as the “outlet pressure”.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



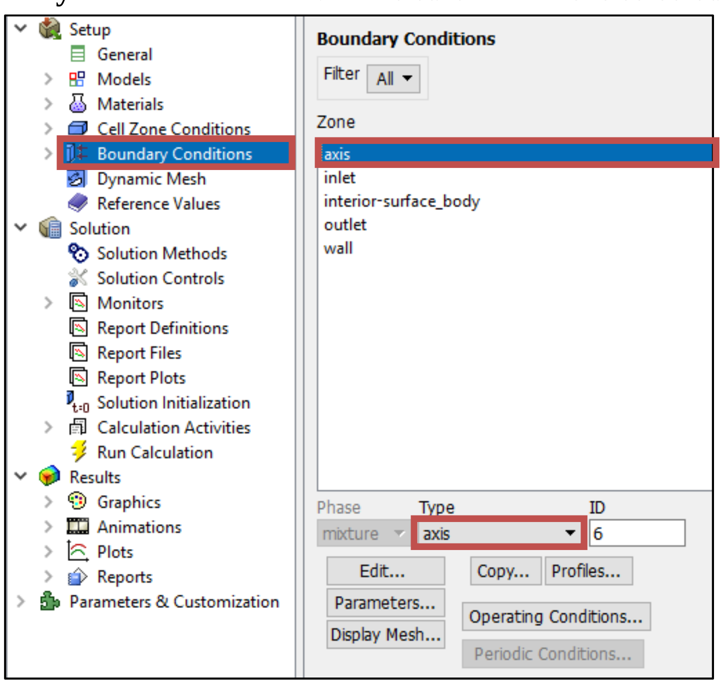
6.10. **Tree > Setup > Boundary Conditions > wall > Edit…** Change roughness height as per experimental data. Change other parameters as below and click **OK**. Input the pipe roughness of the pipe you used in EFD Lab2. For example, user inputs 0.000025 m for smooth pipe.



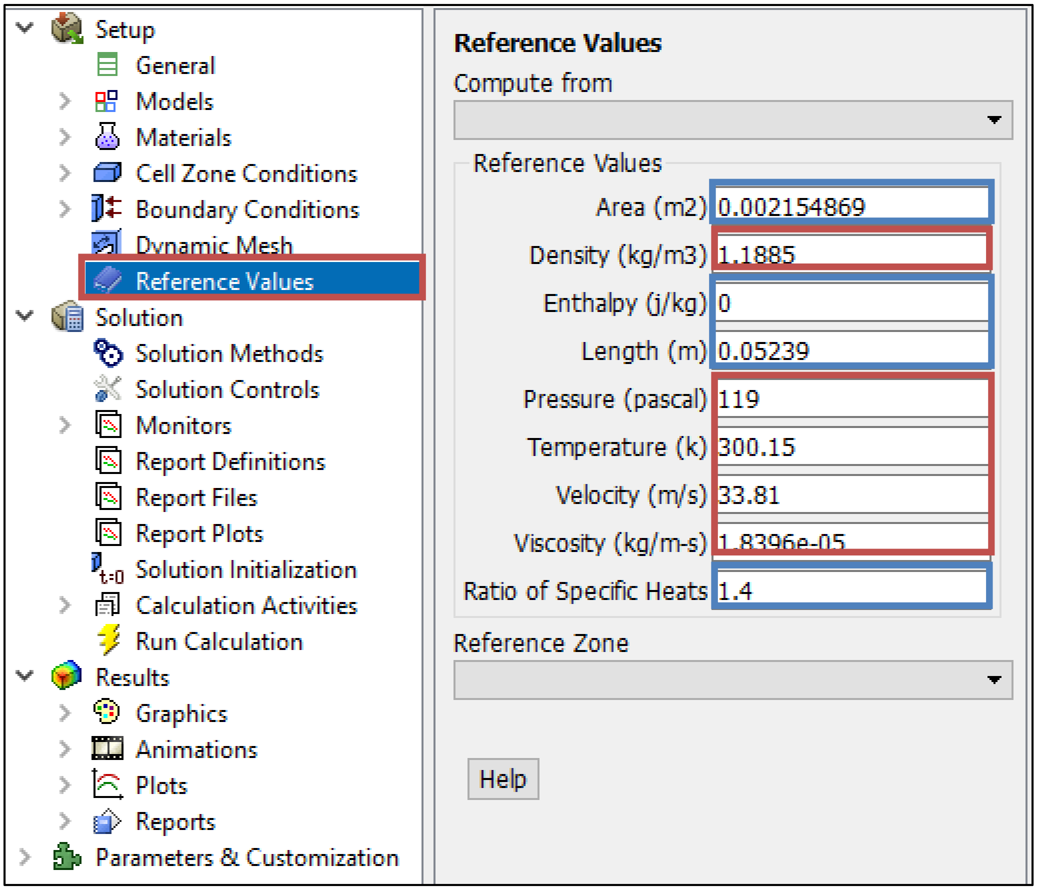
6.11. **Tree > Setup** > **Boundary Conditions** > **Operating Condition**. Change parameters as per below and click **OK**.



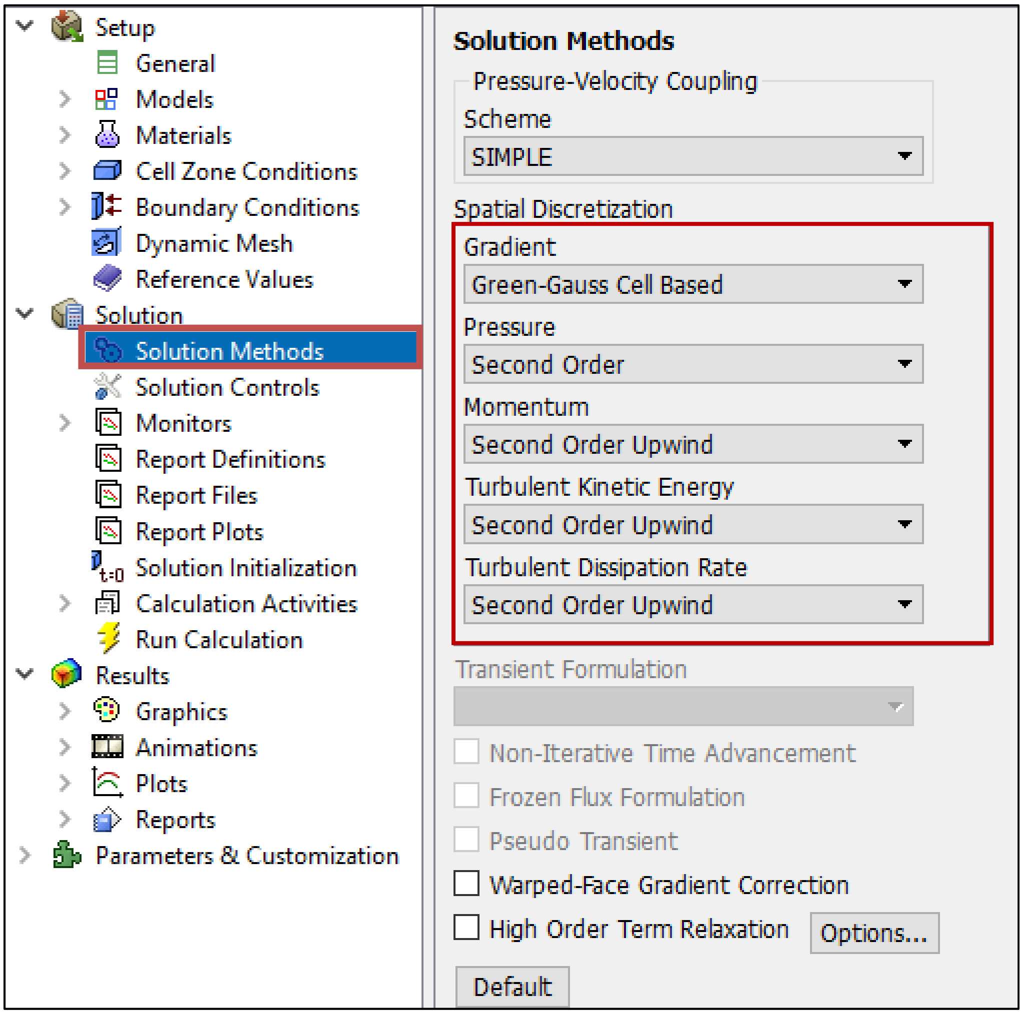
6.12. **Tree > Setup** > **Boundary Conditions** > **axis**. Make sure that axis is selected as per below.



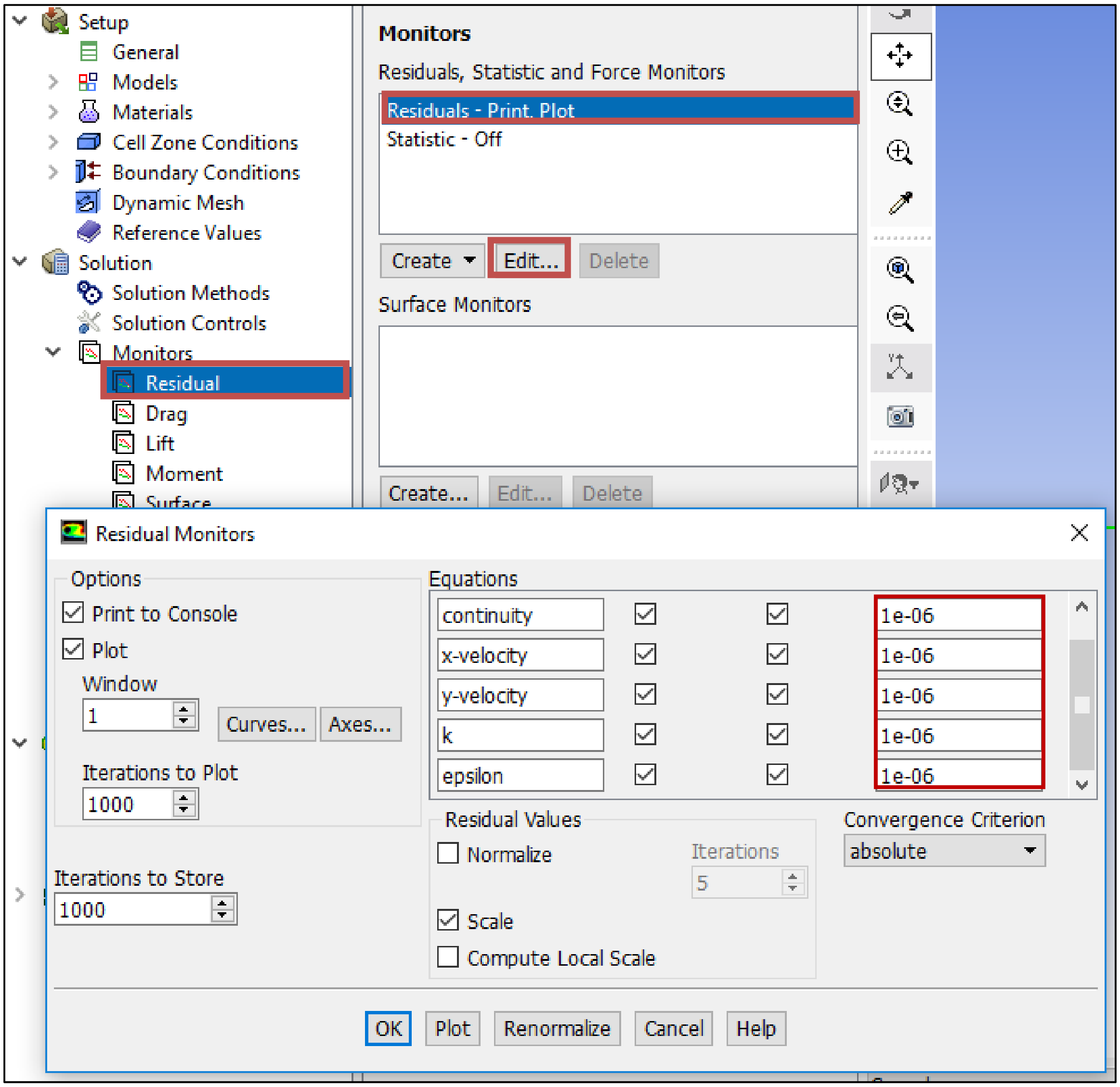
6.13. **Tree** > **Setup** > **Reference Values**. Change parameters as per experimental data. Parameters in blue are constant and should be entered as seen below. Parameters in red are from experimental data.



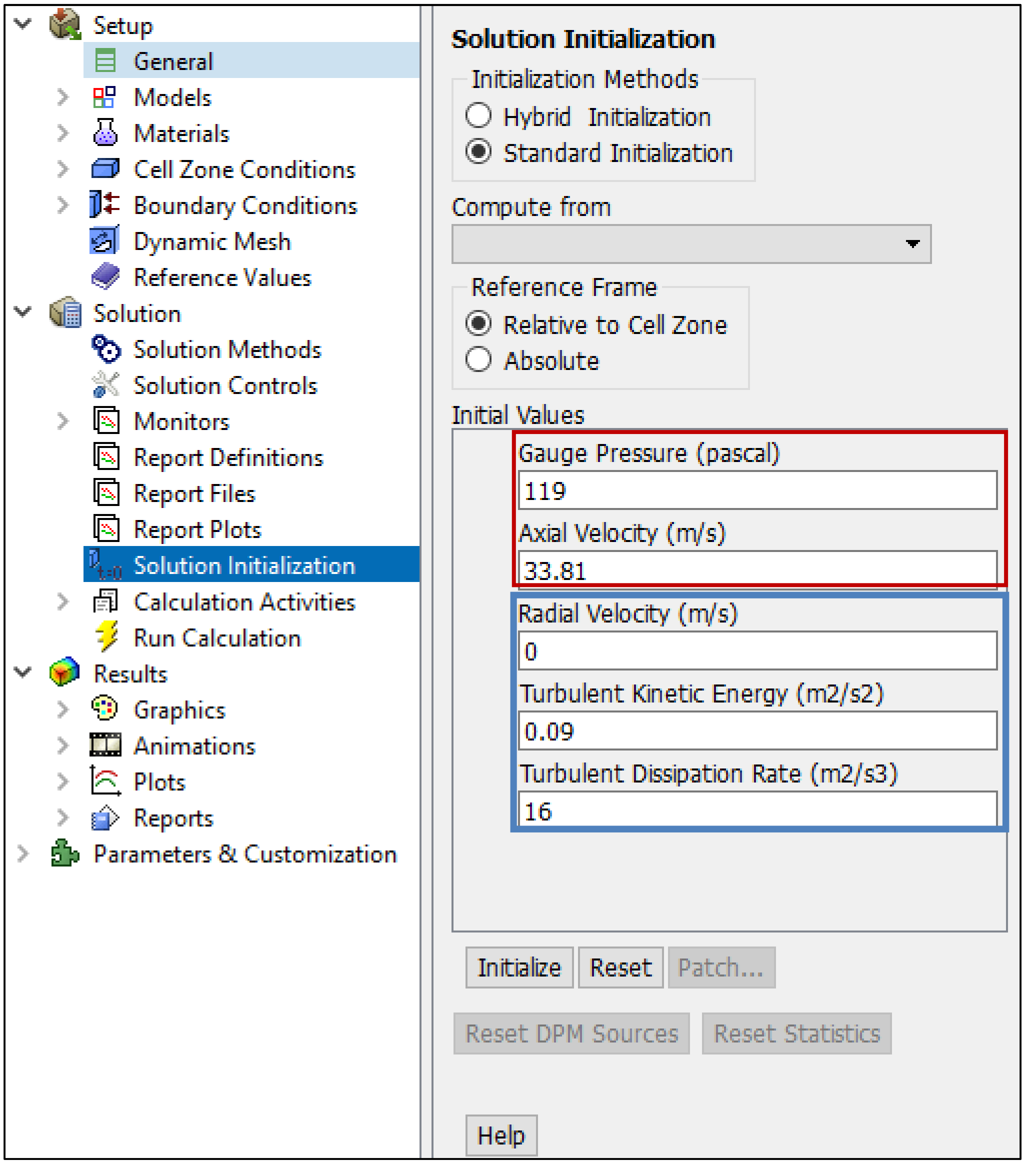
6.14. **Tree > Solution > Solution Methods**. Change parameters as per below.



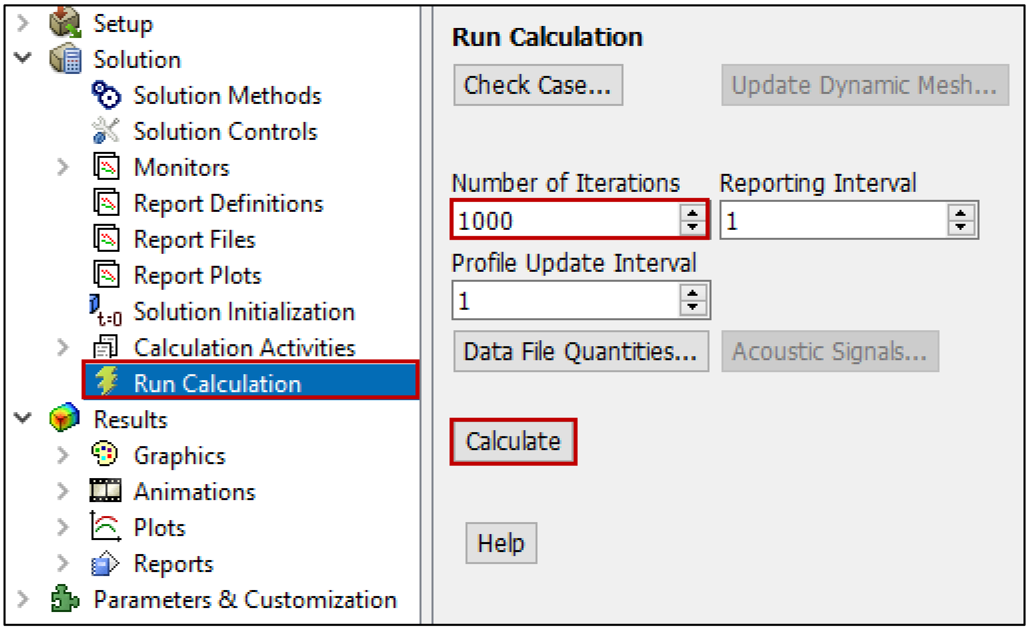
6.15. **Tree** > **Solution** > **Monitors** > **Residuals.** Right click **Residual** and select **Edit…** Change convergence criterion to 1e-06 for all five equations as per below and click **OK**.



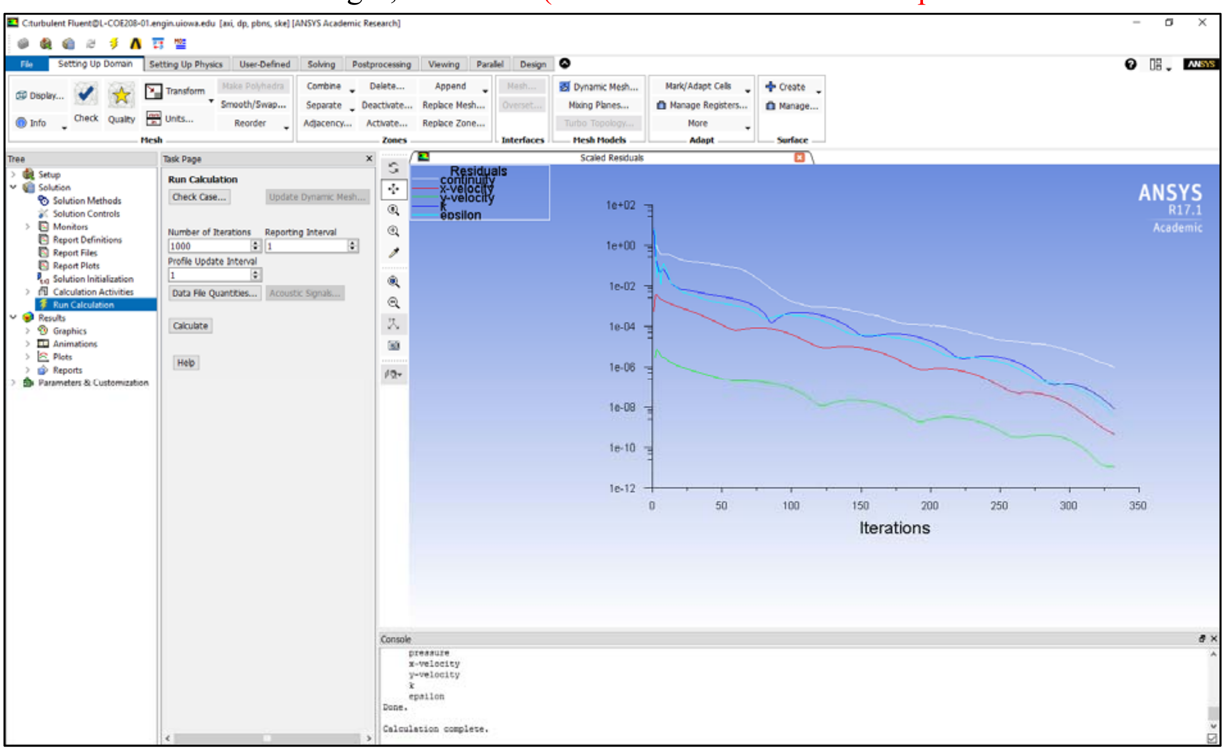
6.16. **Tree** > **Solution** > **Solution Initialization**. Change gauge pressure and axial velocity as per experimental data and click **Initialize**. Parameters in blue are constant and should be entered as seen below. Parameters in red are from experimental data.



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.)



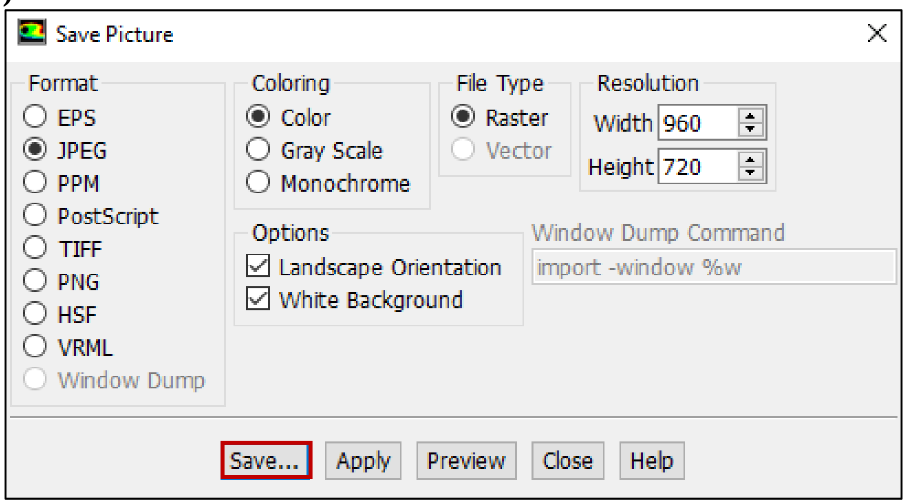
6.19. **File** > **Save Project**. Save the project.

7. Results

Please read exercises before continuing.

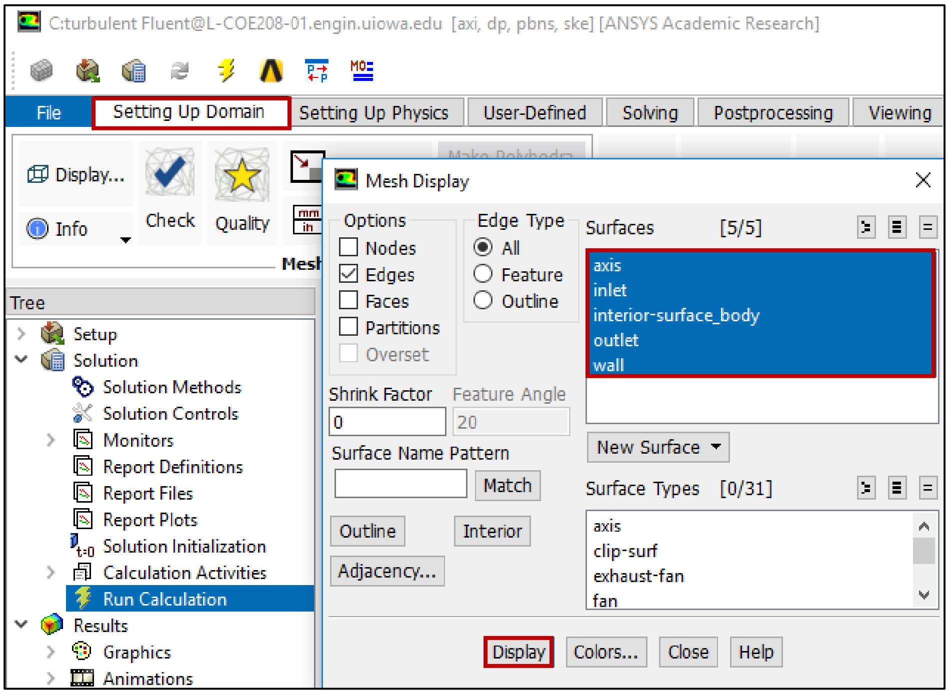
7.1. Saving Pictures

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

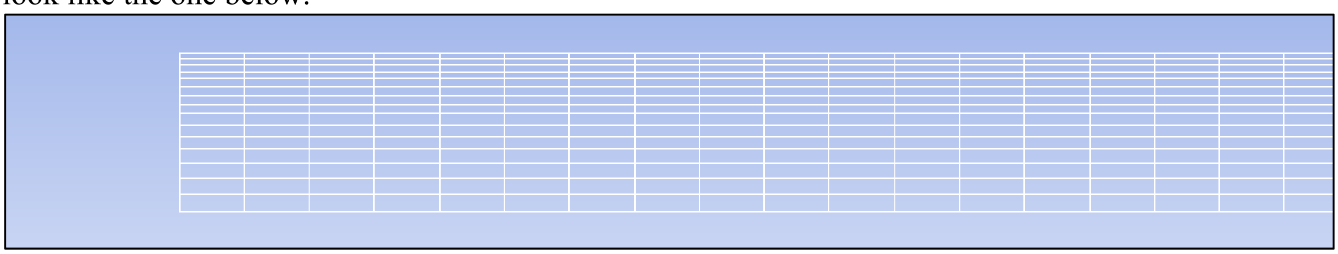


7.2. Displaying Mesh

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



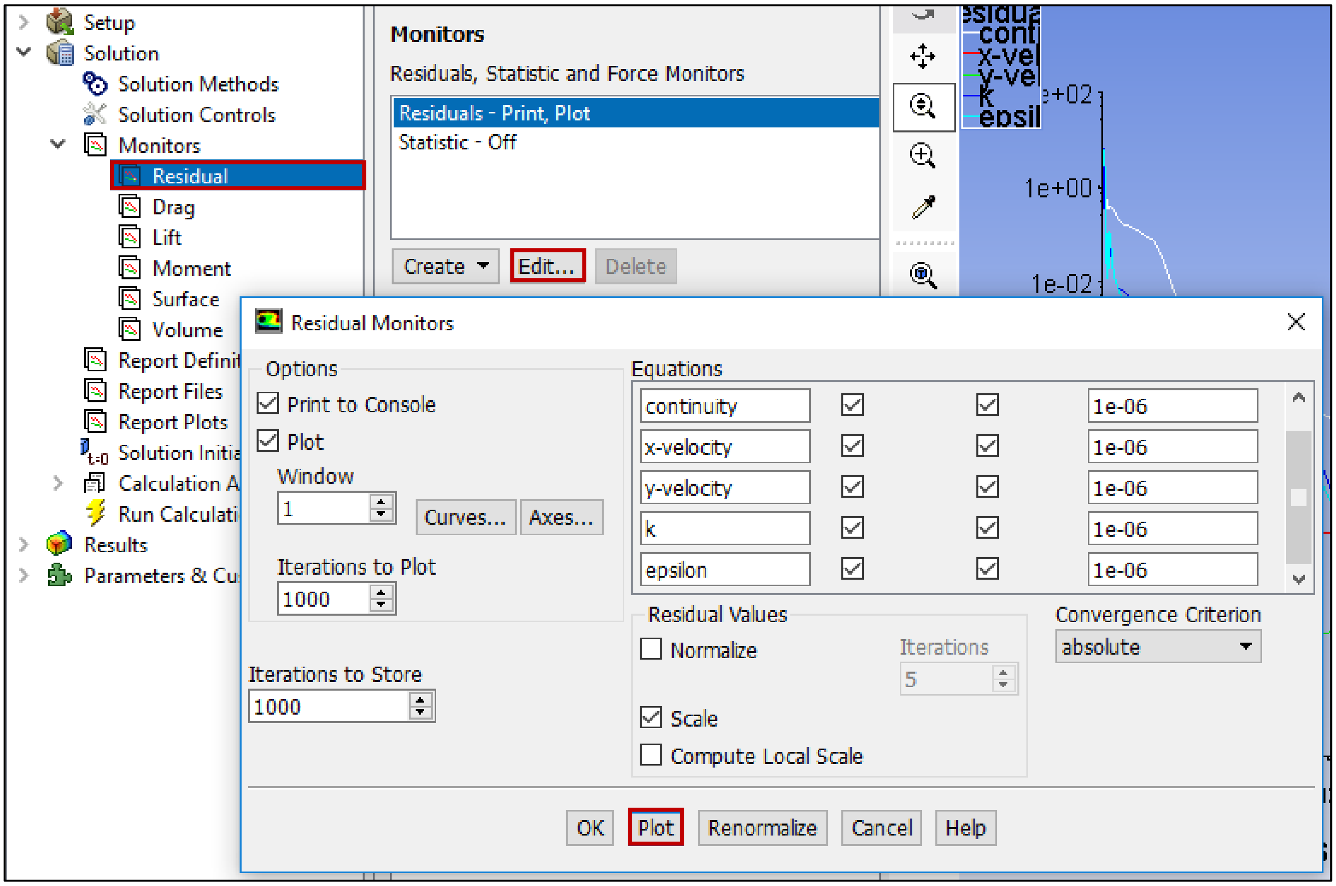
Zoom in to the inlet by using the magnifying glass with a plus sign in the middle of it. The mesh should look like the one below.



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

7.3. Displaying and Saving Residual History

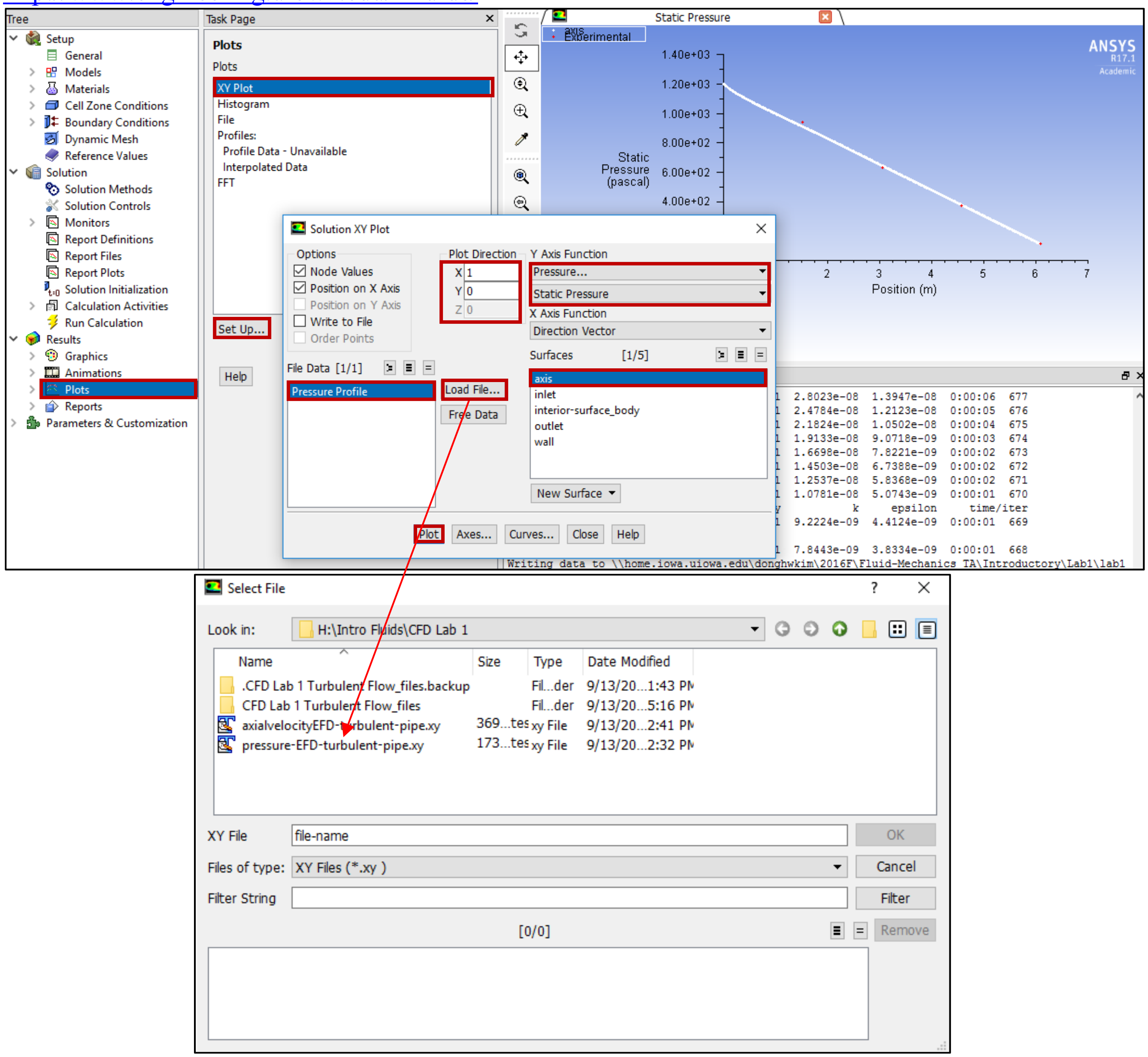
To display the residuals click **Tree > Solution** > **Monitors**. Right click **Residuals** select **Edit…**, click **Plot** then click **Cancel.**



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

7.4. Plotting Centerline Pressure Distribution

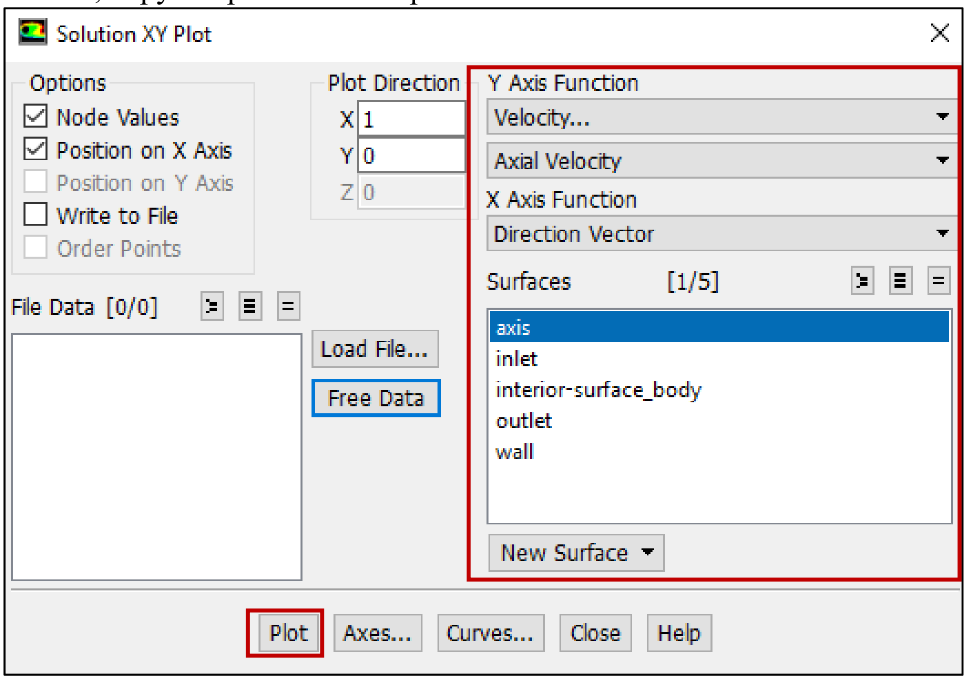
To plot results, click **Results** > **Plots**. Right click **XY Plot,** then click **edit…** To plot the Centerline Pressure Distribution, copy the parameters as per below. Next click **Load File…**, select the file named ***"pressure-EFD-turbulent-pipe.xy"*** and click **OK**. Then click **Plot.**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



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

7.5. Plotting Centerline Velocity Distribution

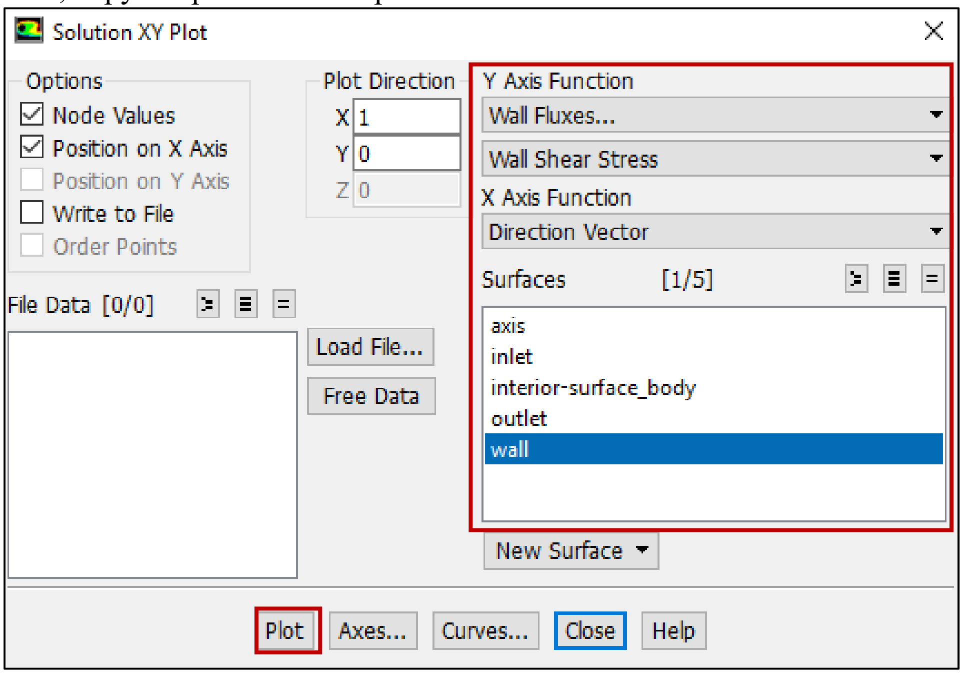
To plot results, click **Results** > **Plots**. Right click **XY Plot,** then click **edit…** To plot Centerline Velocity Distribution, copy the parameters as per below and click **Plot**.



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

7.6. Plotting Wall Shear Stress Distribution

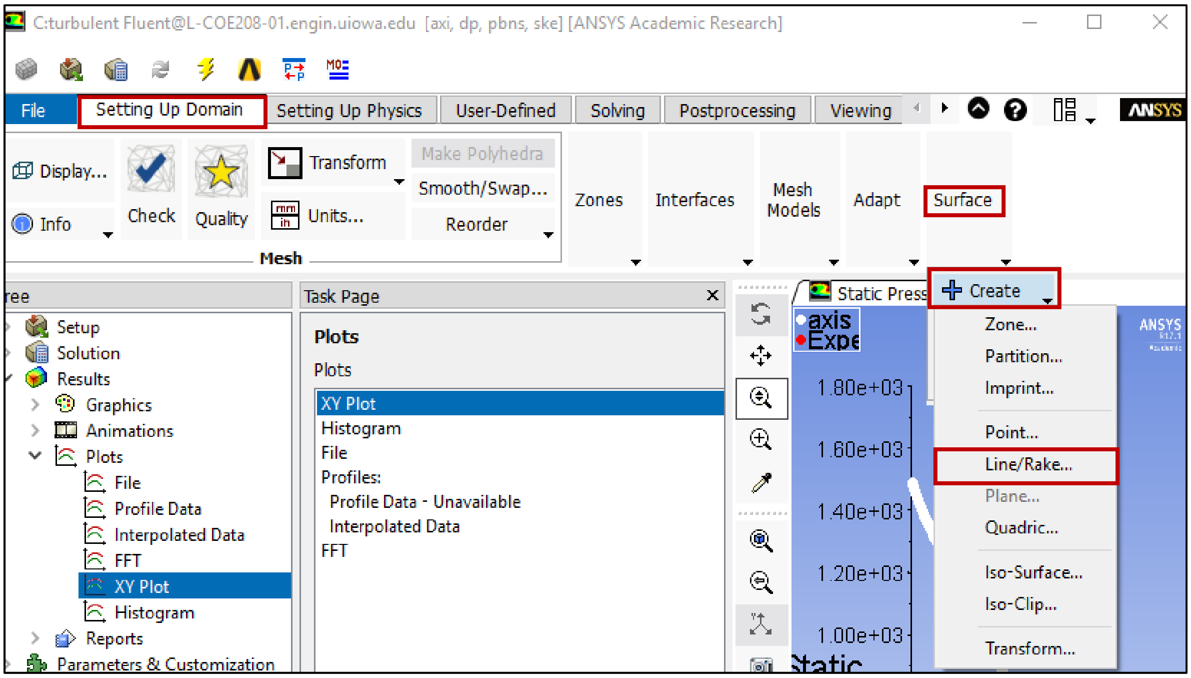
To plot results, click **Results** > **Plots**. Right click **XY Plot,** then click **edit…** To plot the Wall Shear Stress Distribution, copy the parameters as per below and click **Plot**.



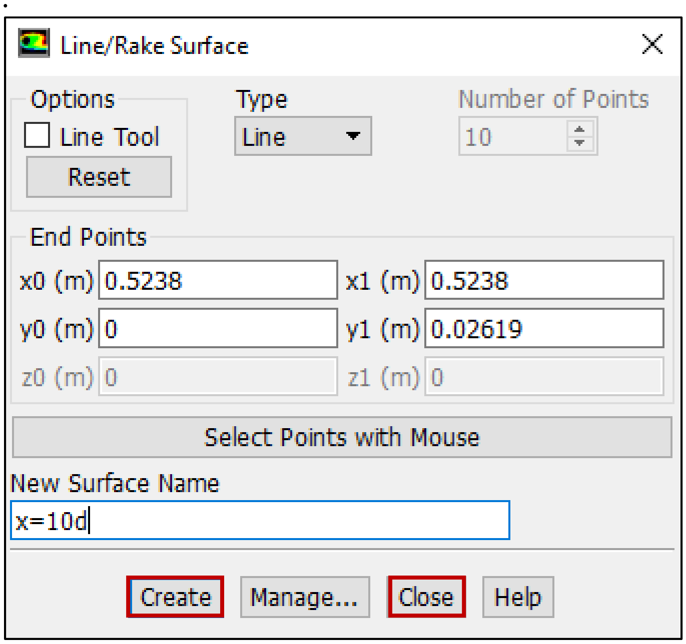
Save the picture as you did for the mesh and call it *"CFD Lab 1 Wall Shear Stress Distribution"* and save it in the folder you created.

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

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



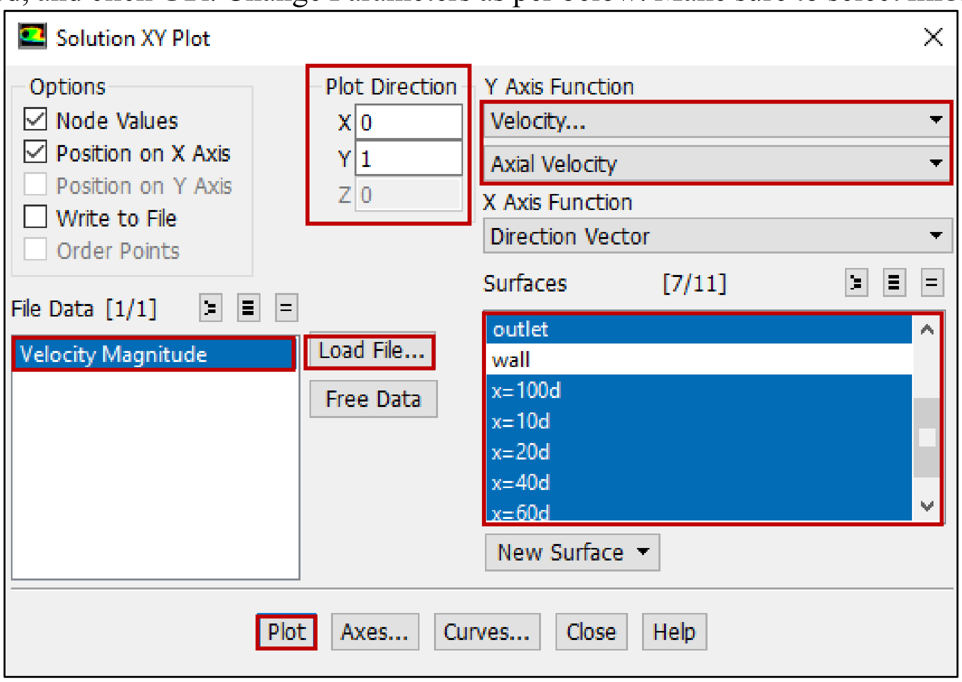
Change x and y values as per below, name the surface, and click **Create**. Repeat this for all the lines shown in the table below.



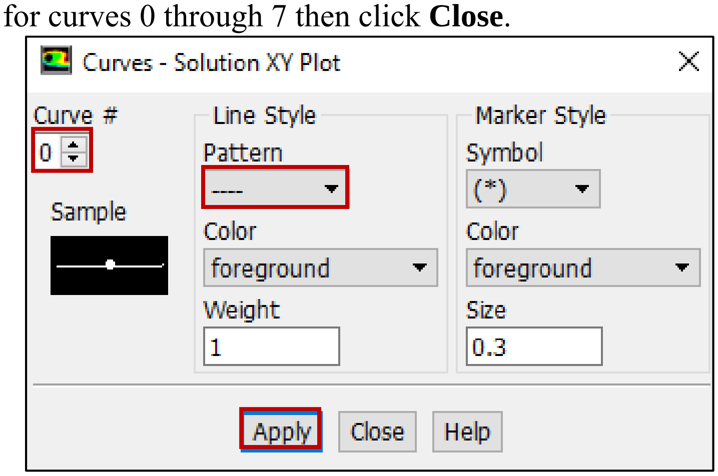
Surface Name	X0	Y0	X1	Y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.238	0	5.238	0.02619

When all lines are created, click **Close**.
Next you should open the file **“axialvelocityEFD-turbulent-pipe.xy”** with a text editor and replace the velocity values with the values you recorded experimentally**.** Save the file**.**

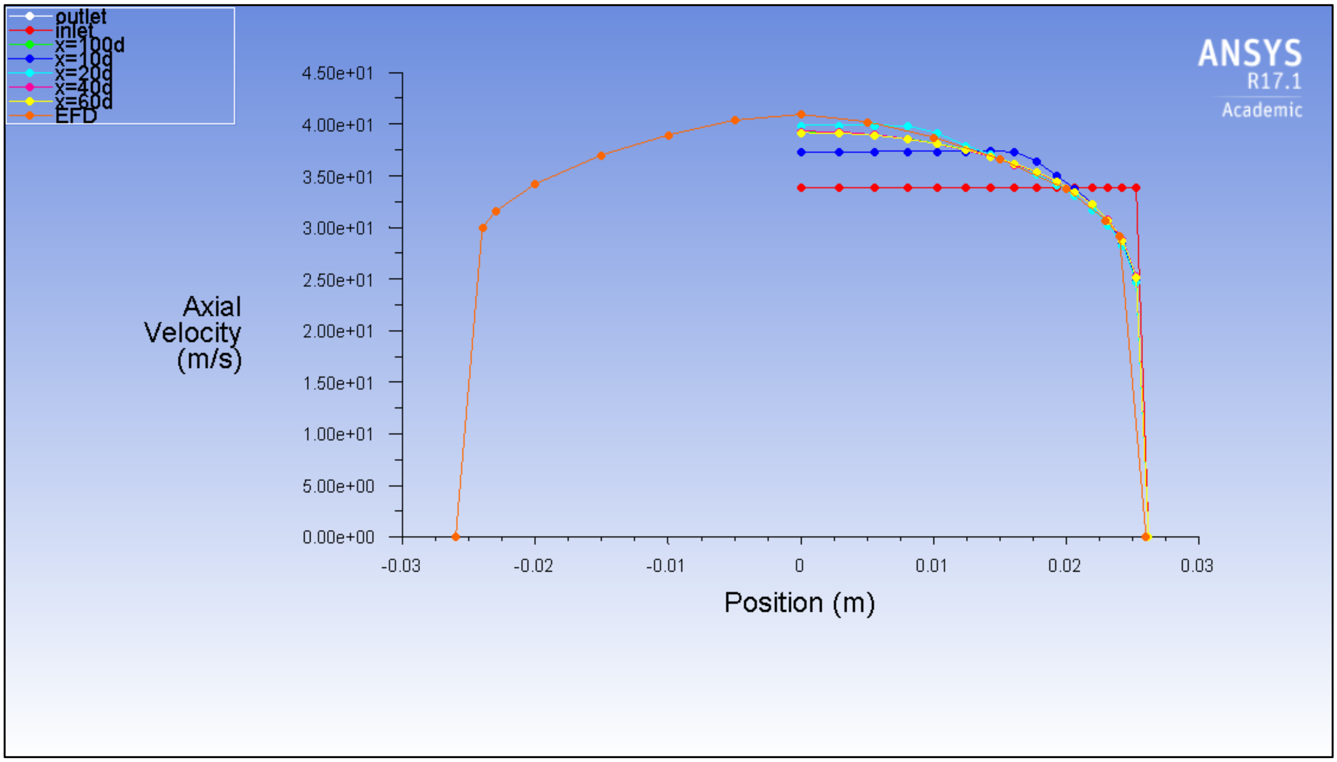
Go back to XY plot. Click **Load File…** and select “**axialvelocityEFD-turbulent-pipe.xy**”, which is the file you just modified, and click **OK**. Change Parameters as per below. Make sure to select **inlet** as well.



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**.



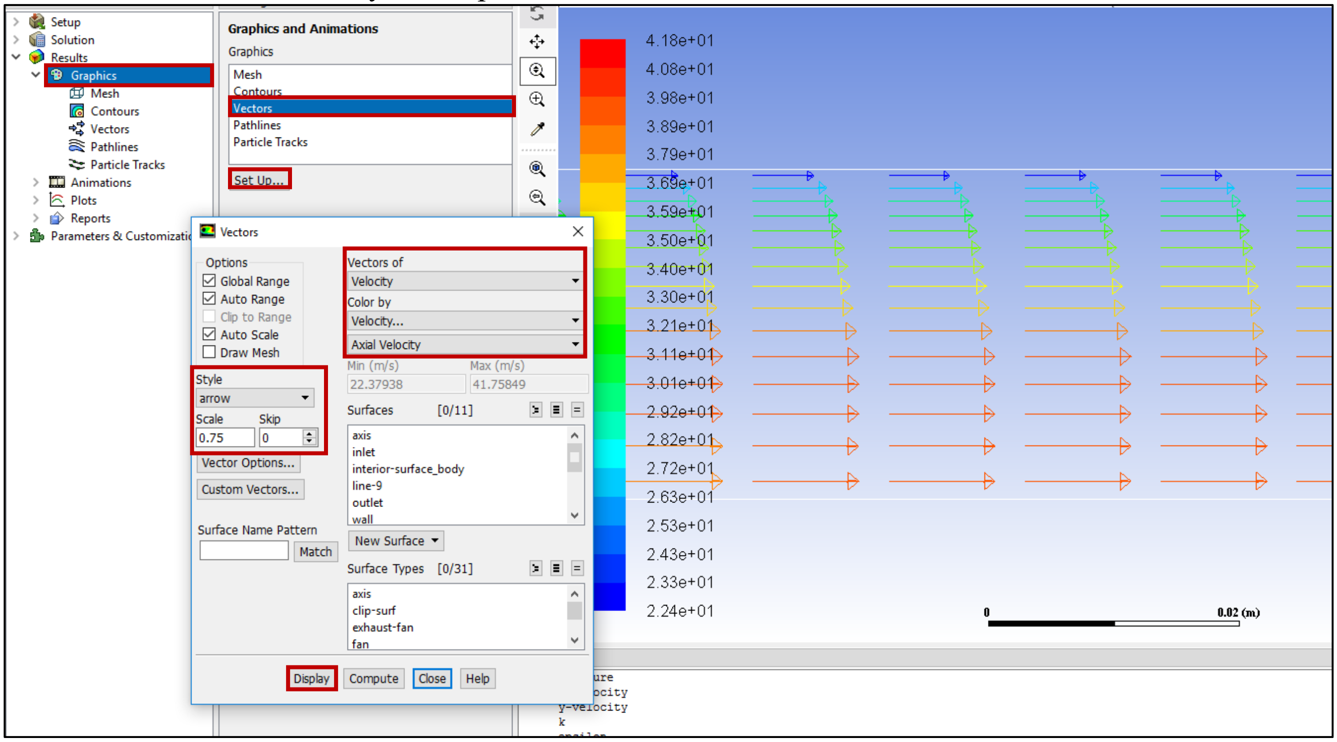
Click **Plot.**



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

7.8. Plotting Velocity Vectors

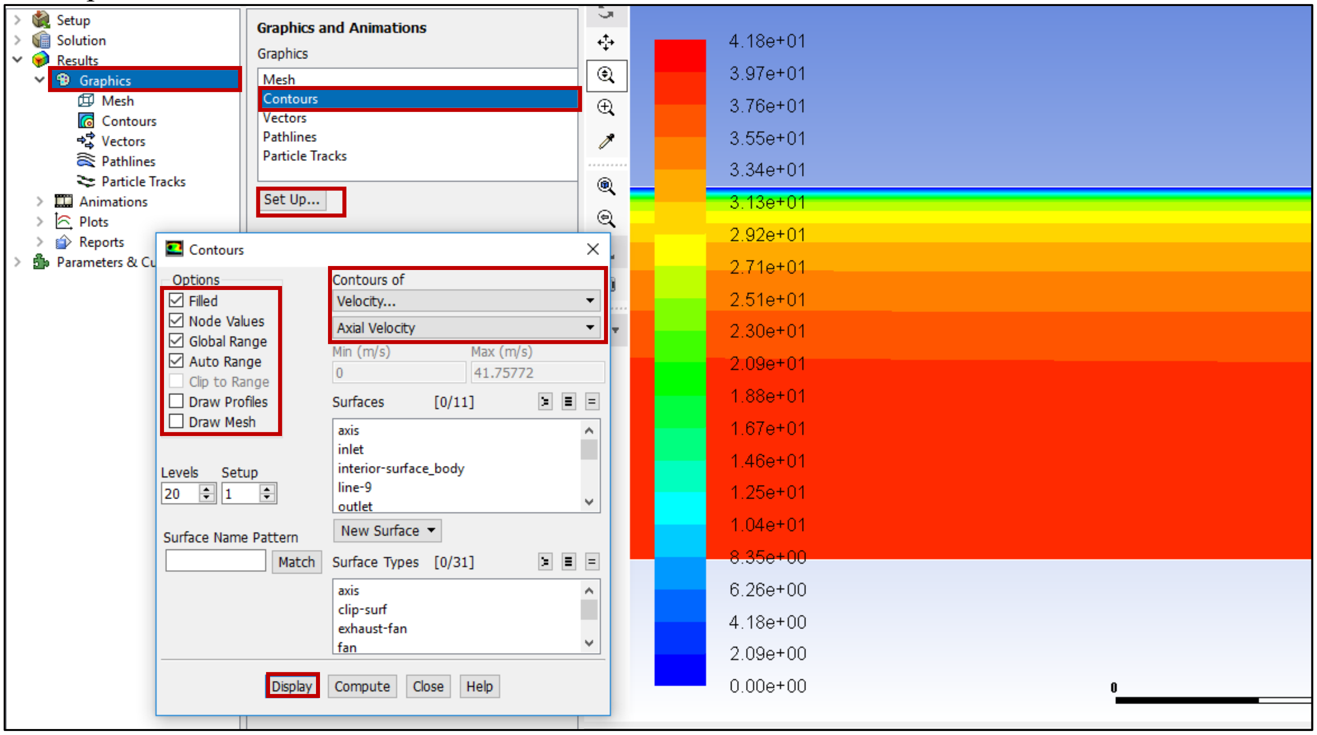
Click **Results** > **Graphics** > **Vectors** > **Set Up…** To plot the velocity vectors at the region flow begin to become fully developed, copy the parameters as per below and click **Display**. Zoom into the region where the flow is almost fully developed.



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

7.9. Plotting Axial Velocity Contours

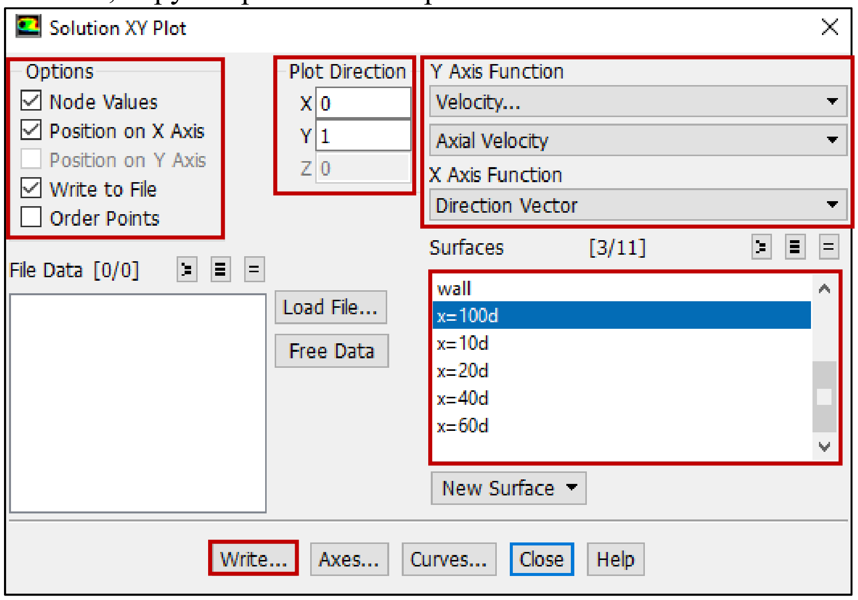
Click **Results** > **Graphics** > **Contours** > **Set Up…** To plot the Contours of Axial Velocity, copy the parameters as per below and click **Display**. Zoom into the region where the flow is almost fully developed.



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

7.10. Exporting Axial Velocity Profile at x=100d

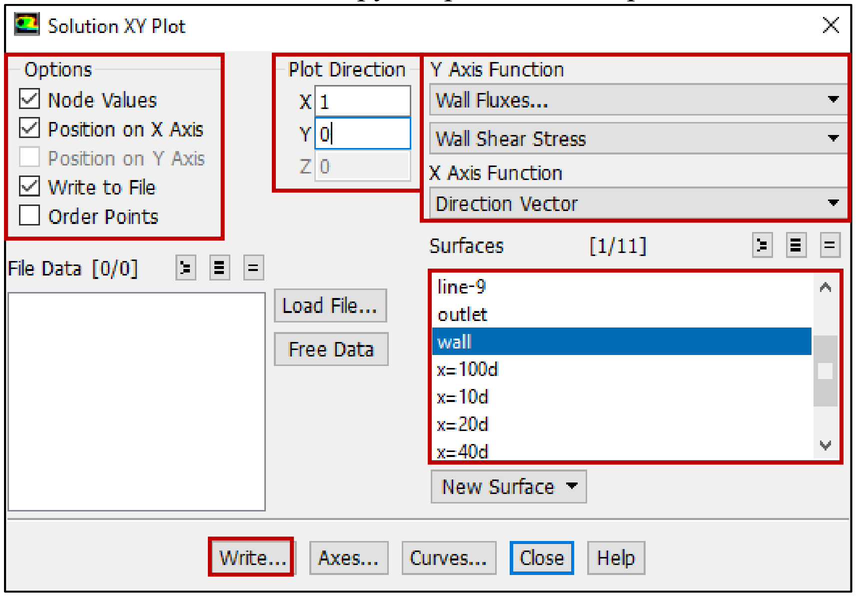
To export Results, click **Results** > **Plots** > **XY Plot** > **Set Up…** To export the Developed Axial Velocity Profile at x=100d, copy the parameters as per below and click **Write…**



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

7.11. Exporting Wall Shear Stress Distribution

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



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

7.12. Normalizing Velocity Profile

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

8. Exercises

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

Simulation and Validation of Turbulent Pipe Flow

8.1. Validation of CFD using EFD data

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

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

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

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

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

8.2. Normalized developed axial velocity profile

8.2.1. Export the axial velocity profile data at x=100d only.

8.2.2. Use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude, which is the maximum value of the profile. Plot the normalized turbulent and laminar flow

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

8.3.1. Where is the mesh clustered? Near the wall or near the axis? Why?

8.3.2. In the developed region, is the pressure constant? Where are the minimum and maximum pressure locations? What does the axial velocity profile look like? Where does it reach maximum and minimum?

8.3.3. Where is the developing region? How can you determine the length of the developing region? (hint: centerline velocity). What is the difference for axial velocity profile between "developing" region and "developed" region?

8.3.4. Compare the normalized axial velocity profiles for laminar and turbulent pipe flows at x=100d, discuss the difference of axial velocity gradient (du/dy) on the wall, which is larger?

8.3.5. What are the correct boundary conditions (velocities u, v and pressure) on the pipe wall and on the pipe centerline?

8.3.6. Summarize your findings and use the knowledge you learned from classroom lectures, textbooks and ANSYS figures to answer questions.