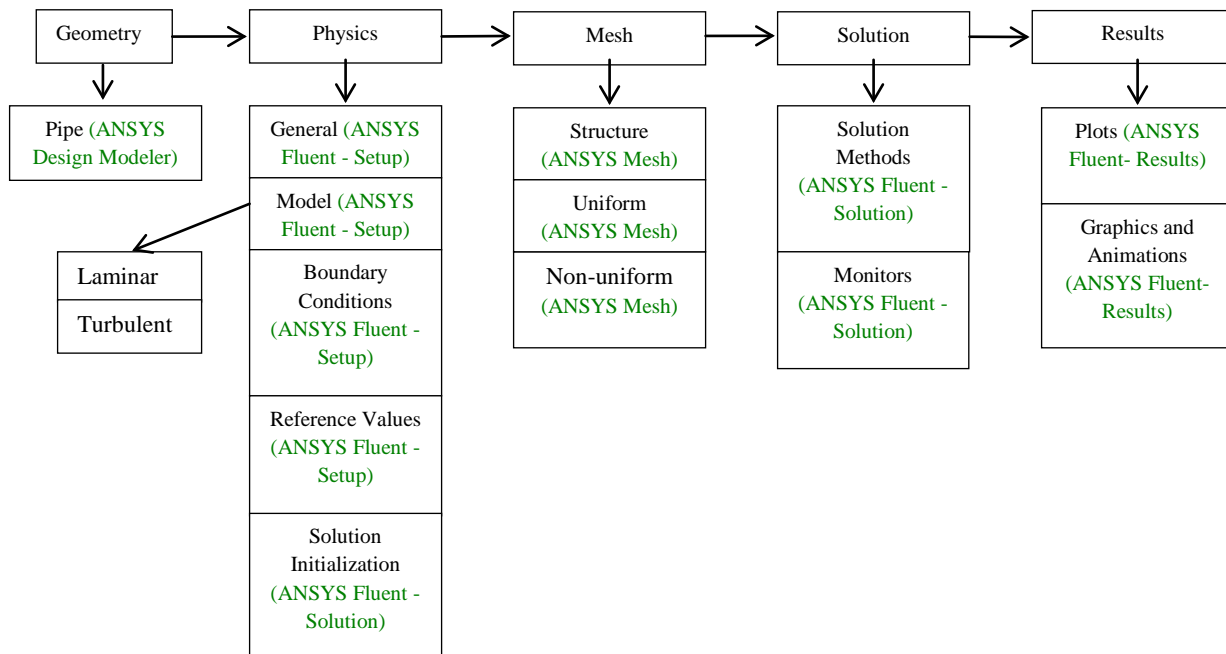# Simulation of Laminar Pipe Flows

57:020 Mechanics of Fluids and Transport Processes
CFD PRELAB 1

By Timur Dogan, Michael Conger, 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.

Geometry		Physics		Mesh		Solution		Results
Pipe (ANSYS Design Modeler)		General (ANSYS Fluent - Setup)		Structure (ANSYS Mesh)		Solution Methods (ANSYS Fluent - Solution)		Plots (ANSYS Fluent- Results)
		Model (ANSYS Fluent - Setup)		Uniform (ANSYS Mesh)				
Laminar		Boundary Conditions (ANSYS Fluent - Setup)		Non-uniform (ANSYS Mesh)		Monitors (ANSYS Fluent - Solution)		Graphics and Animations (ANSYS Fluent- Results)
Turbulent								
		Reference Values (ANSYS Fluent - Setup)						
		Solution Initialization (ANSYS Fluent - Solution)						

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
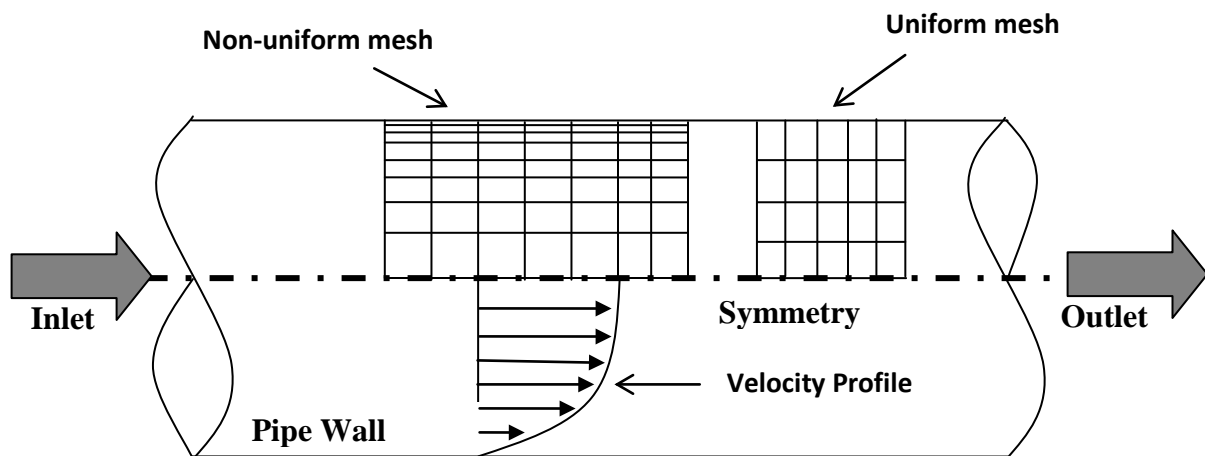
Figure 1 - Geometry

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 15** > **Workbench 15.**
(**Note:** You may ignore the firewall warnings by closing the pop-up window)
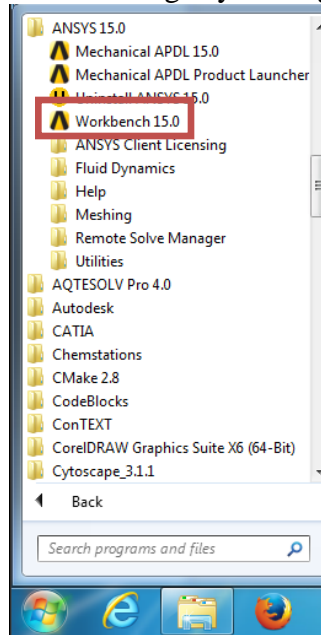
3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop the **Geometry** component for the **Component Systems** on the left side of the screen into the **Project Schematic**. Rename the geometry "Pipe" by right clicking on the down arrow of the **Geometry** component and selecting **Rename**.
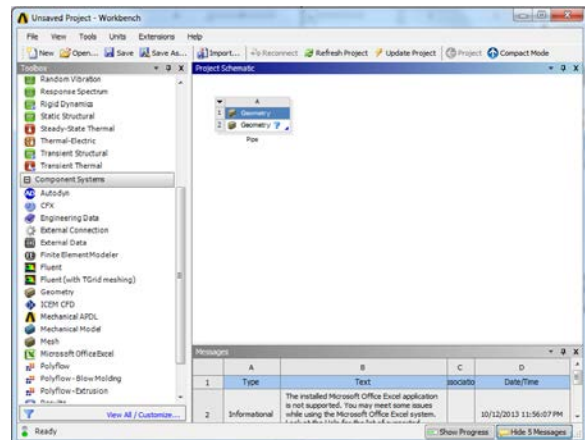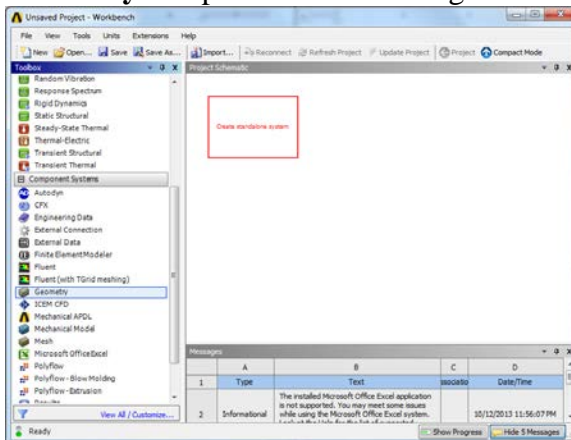
3.3. Drag and drop Mesh components and Fluent components into the schematic as shown below. Rename the components as "uniform" and laminar for Mesh and Fluent components respectively. Make the connections as per below by dragging component to component.
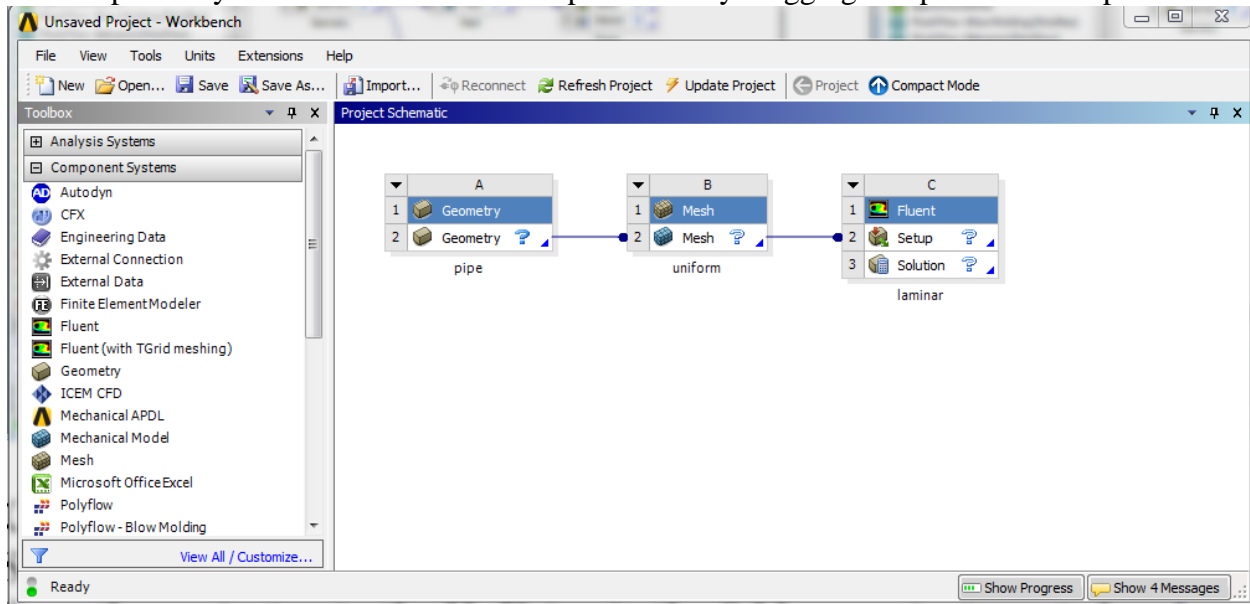
3.4. Create a Folder on the H: Drive called *CFD Pre-Lab and Lab 1*.

3.5. Save the project file by clicking **File** > **Save As…**

3.6. Save the project onto the H: Drive in the folder you just created and name it *CFD Pre-Lab and Lab 1 Pipe Flow*. (This 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 Geometry…**
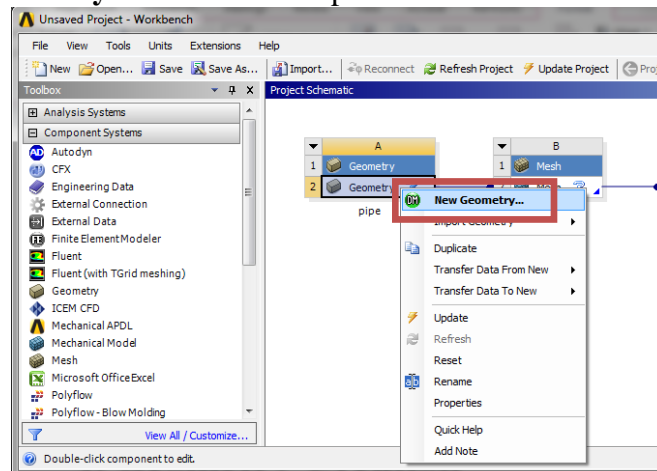
4.2. Make sure that Unit is set to Meter (default value).
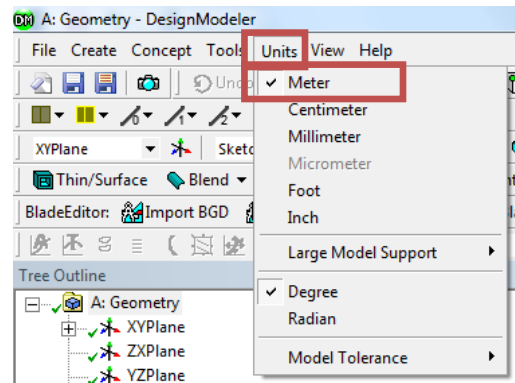
4.3. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.
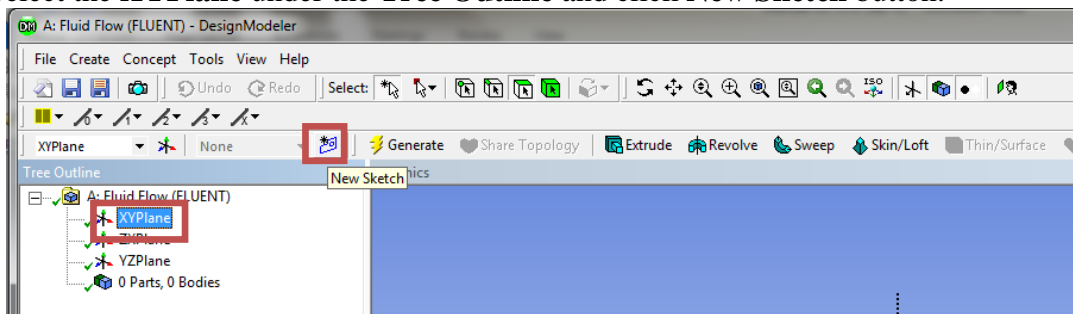
4.4. Right click **XYPlane** and select **Look at**.
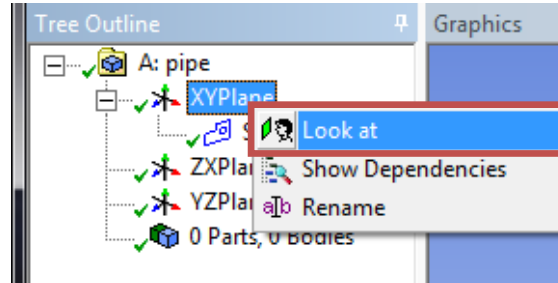
4.5. Select **Sketching** > **Constraints** > **Auto Constraints**. Enable the auto constraints option to pick the exact point as below.
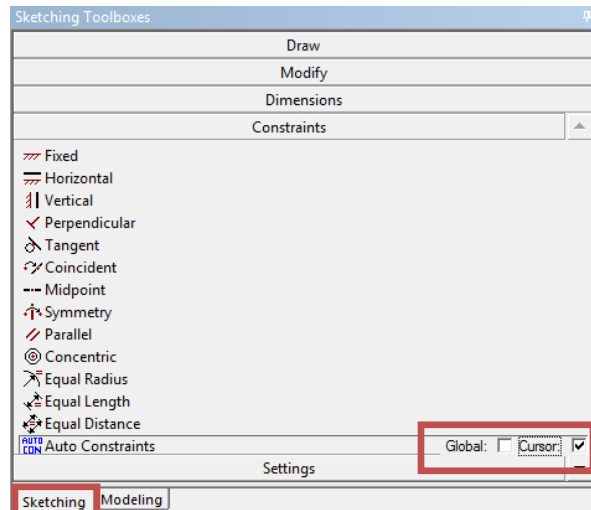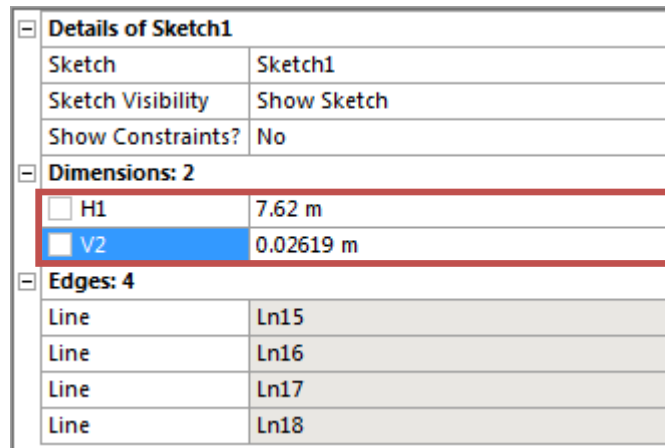
4.6. Select **Sketching** > **Rectangle**. Create a rectangle geometry as per below, make sure to start from the origin, the mouse arrow should change to a "P" when on the origin.
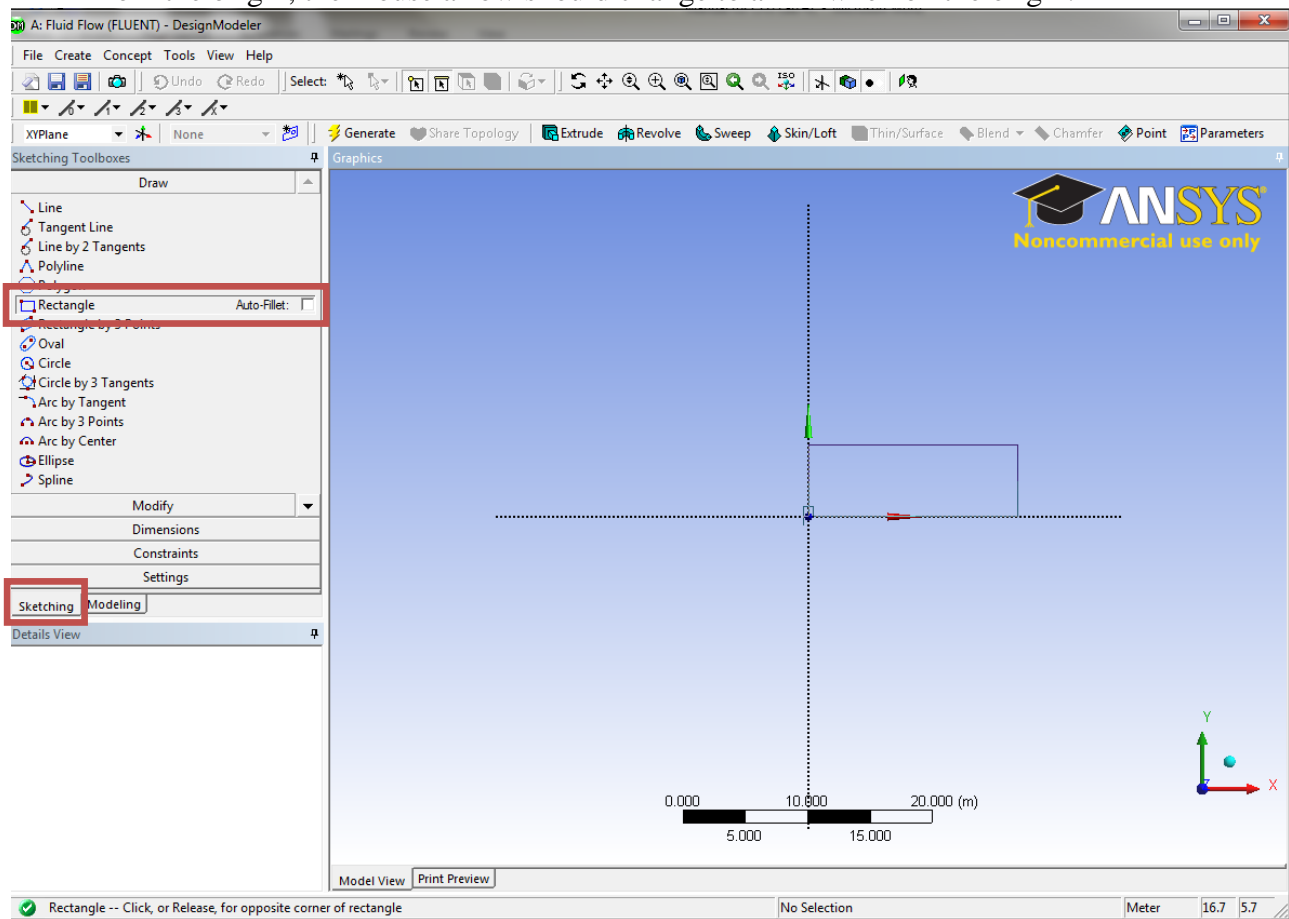
4.7. Select **Dimensions** > **General**. Click on top edge then click above the geometry to place the dimension. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.
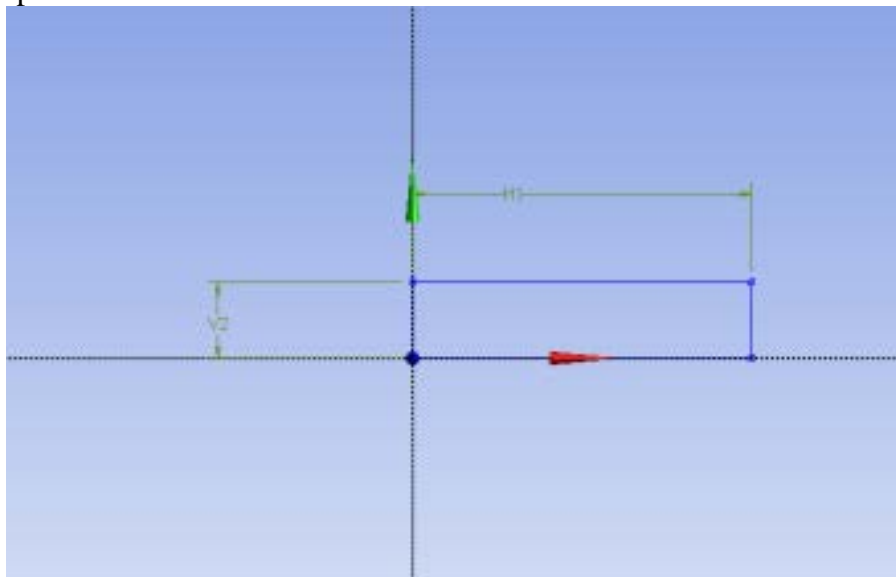
4.8. Click on **H1** under **Details View,** in the bottom left of the screen, and change **H1** to *7.62m*. Click on **V2** and change it to *0.02619m*.

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

4.9. **Concept** > **Surface From Sketches**, select the sketch by clicking on **Sketch 1** in the **Tree Outline** and hit **Apply** in the **Detatils View**.
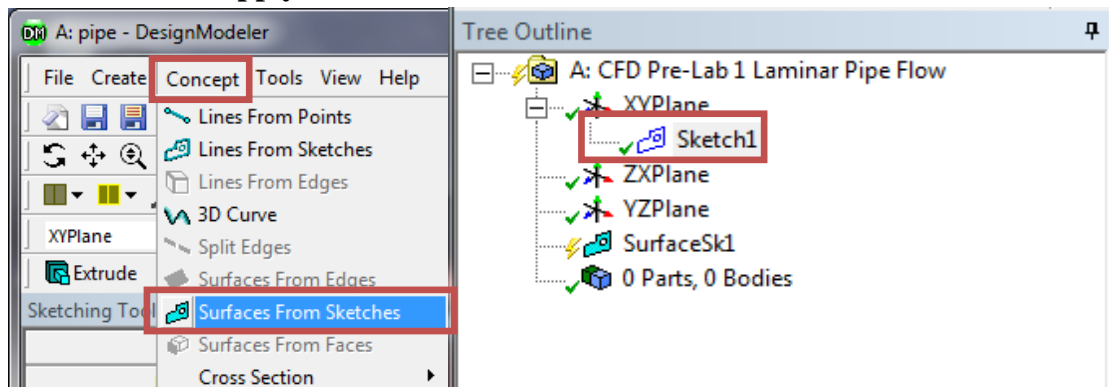
4.10. Click **Generate**. This will create a surface.
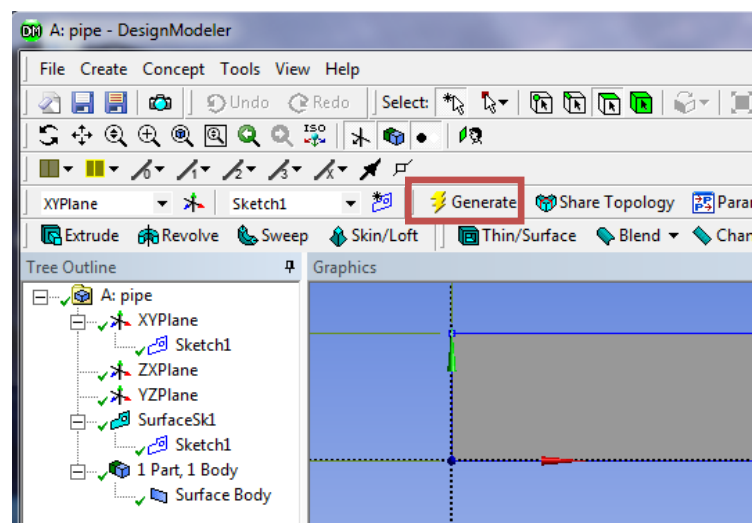
4.11. **File** >**Save Project**. Save project and close the **Design Modeler** window.

5. Mesh Generation

5.1. From the **Project Schematic** right click on **Mesh** component and select **Edit…**
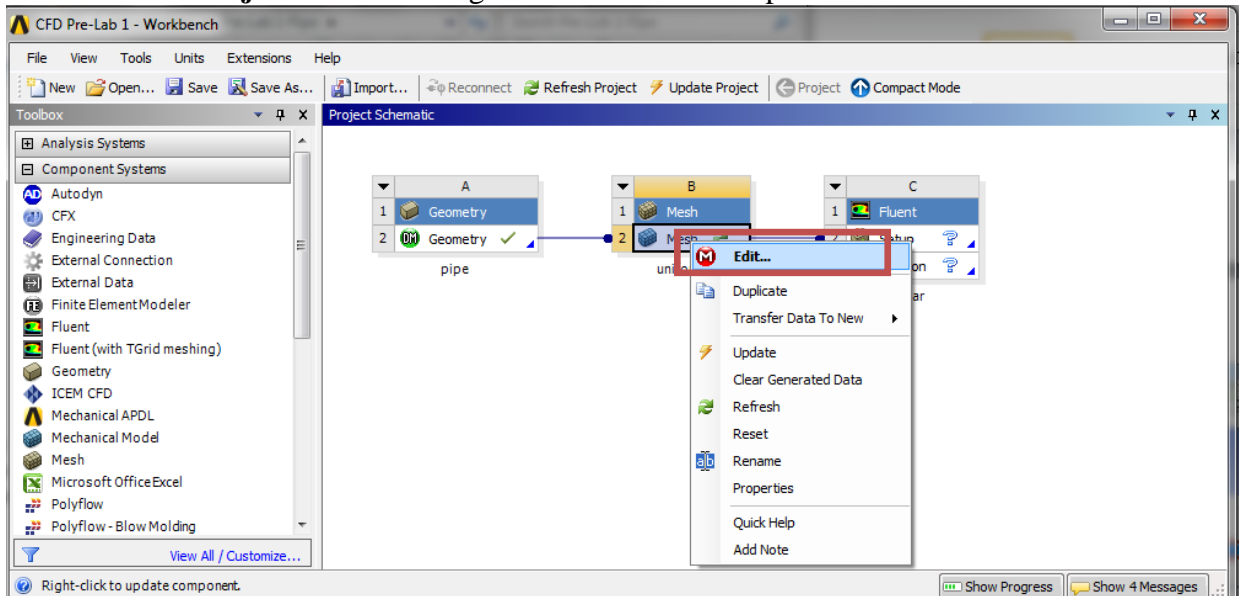
5.2. Right click on **Mesh** then select **Insert** > **Mapped Face Meshing**.
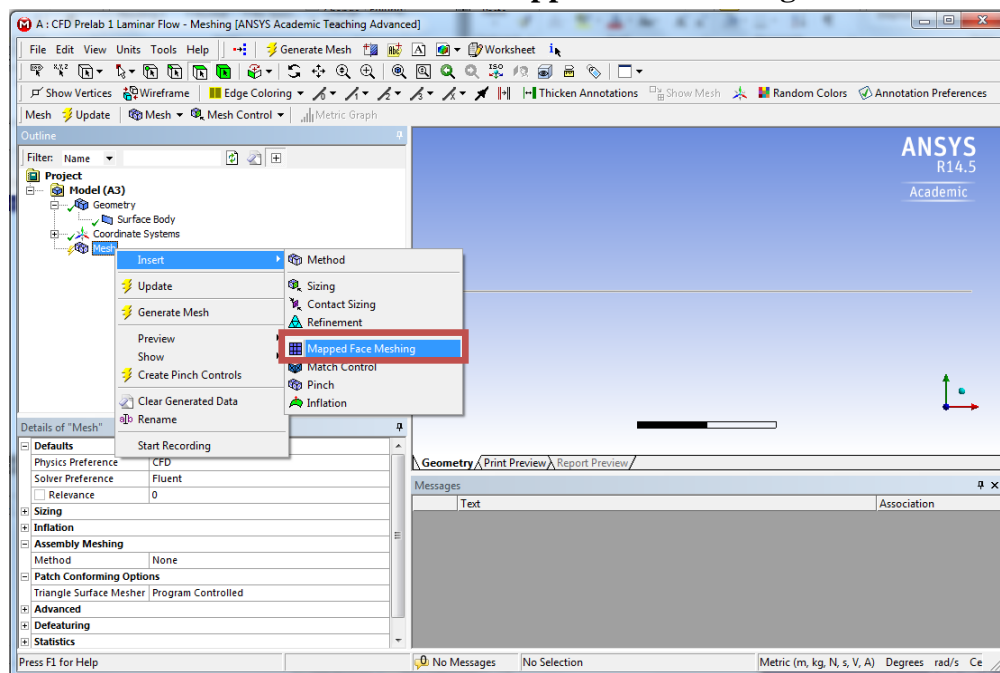
5.3. Select your geometry by clicking the yellow box which says **No Selection**, the click on the geometry surface, and click **Apply**. (Note: You can change orientation of your view by clicking the axis figure on the lower right corner. Press "F7" on your keyboard to restore to the "whole view". Zoom in by holding the right mouse button and selecting a region.)
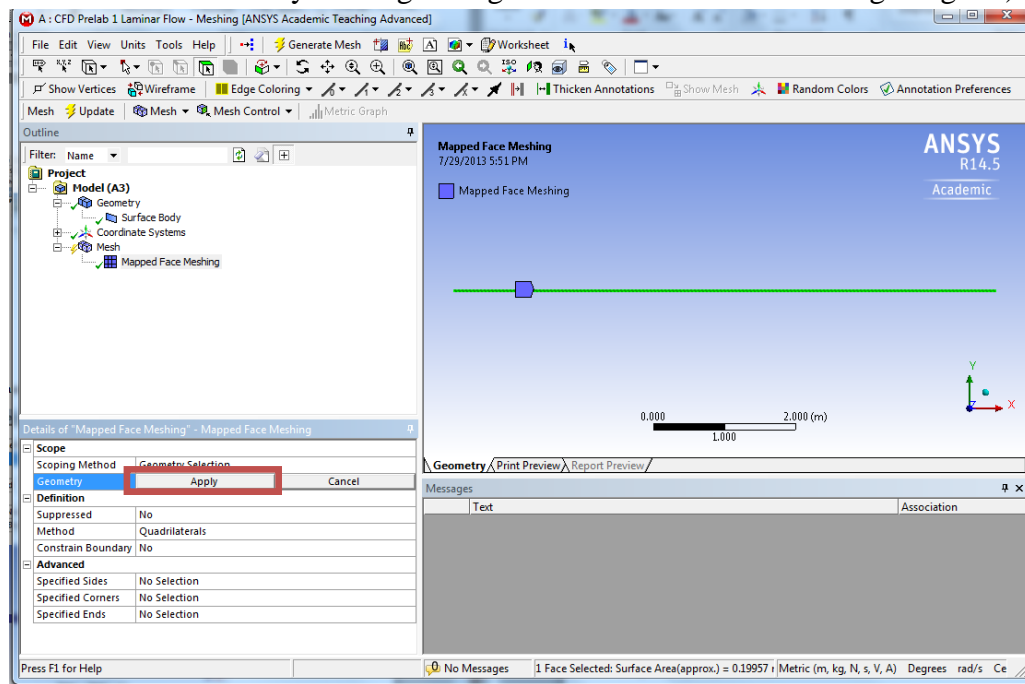
5.4. Click on the edge button. This will allow you to select edges of your geometry.
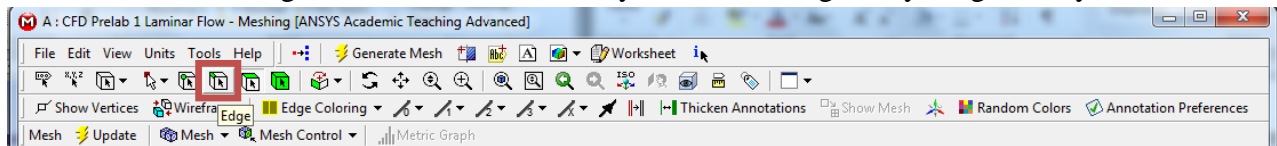
5.5. Right click on **Mesh** then select **Insert** > **Sizing**.
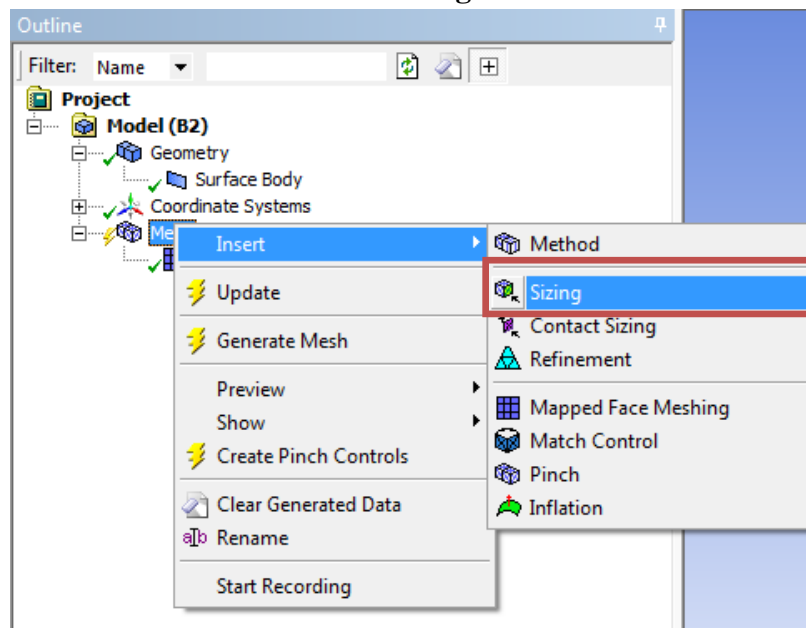
5.6. Hold **Ctrl** button and select the top and bottom edge of the rectangle then click **Apply**. Specify details of sizing as per below in the **Details of "Edge Sizing" – Sizing** window.

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

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

Details of "Edge Sizing 2" - Sizing		
Scope		
Scoping Method	Geometry Selection	
Geometry	2 Edges	
Definition		
Suppressed	No	
Type	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.
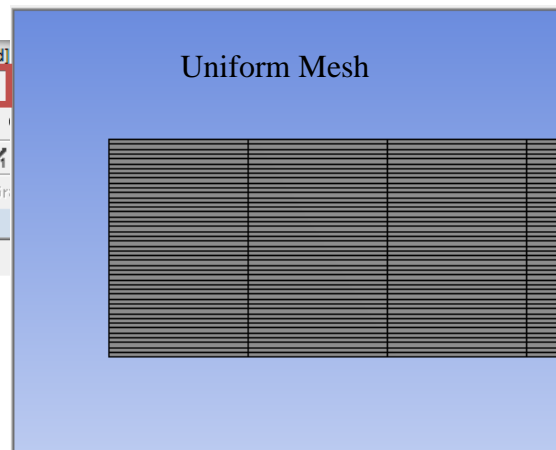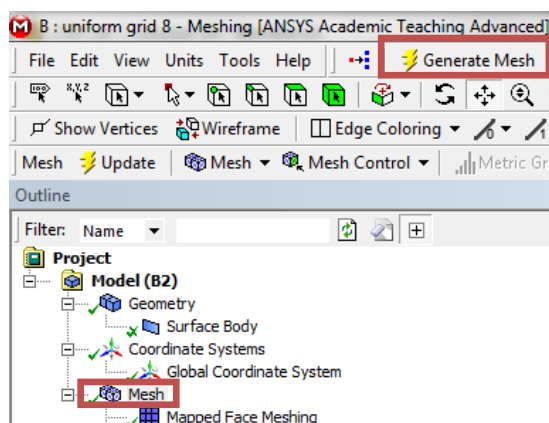
5.9. Change the edge names by selecting the edge, then right clicking on the edge and selecting **Create Named Selection**. Name left, right, bottom and top edges as *inlet*, *outlet*, *axis* and *wall* respectively. Your outline should look same as the figure below.
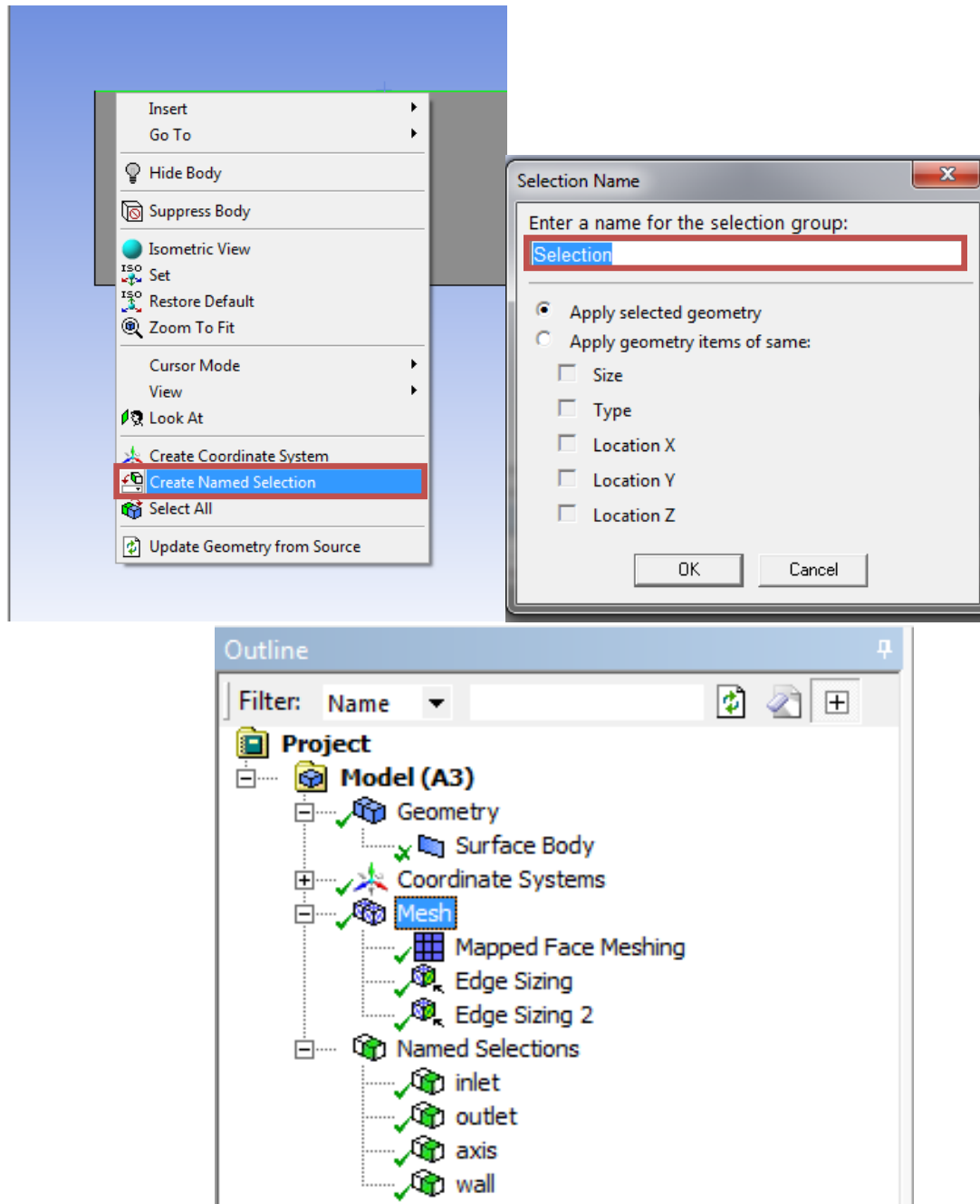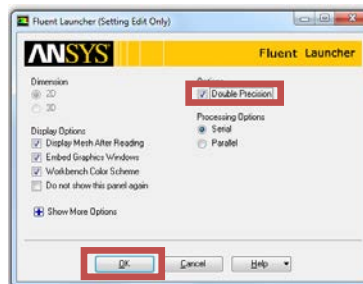

5.10.   **File** > **Save Project**. Save the project and close the window. Update mesh on **Project Schematic** by right clicking on **Mesh** and selecting **Update.**
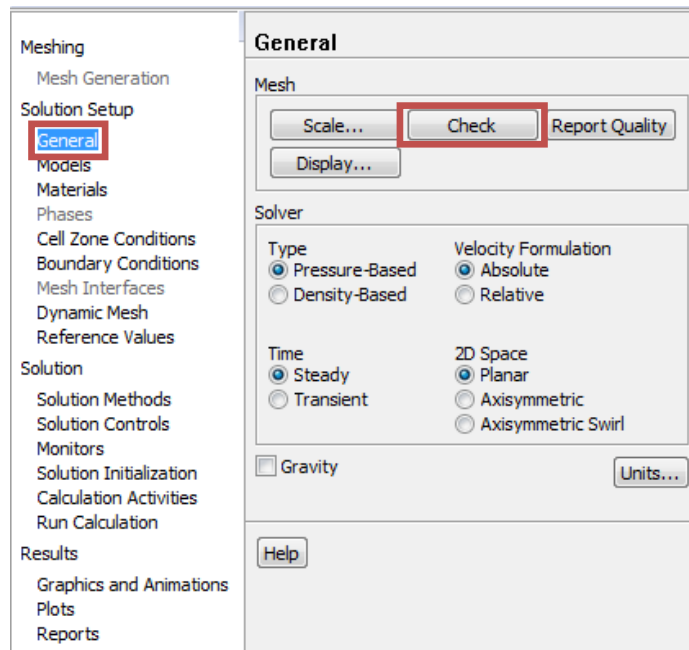
6. Setup (Physics)

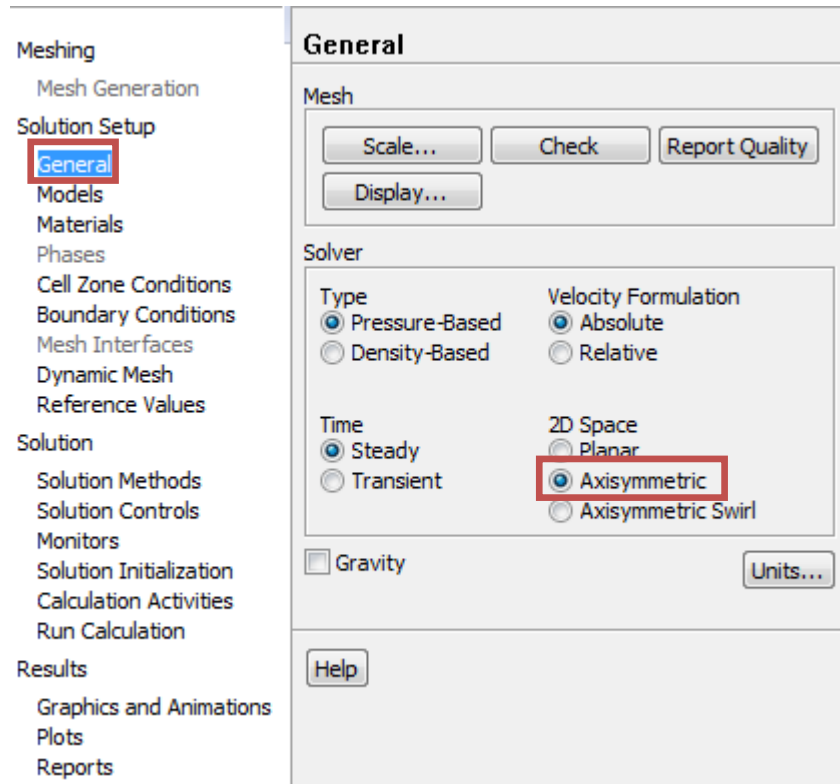6.1. Right click Setup and select **Edit…**
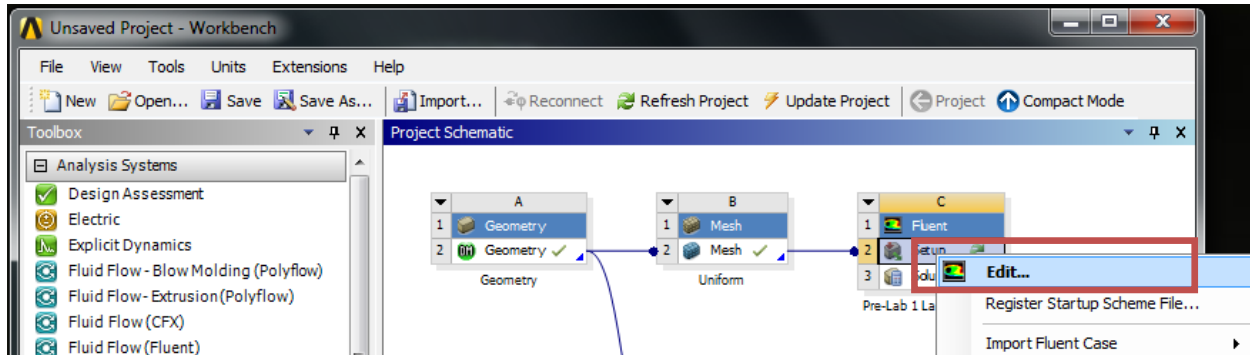
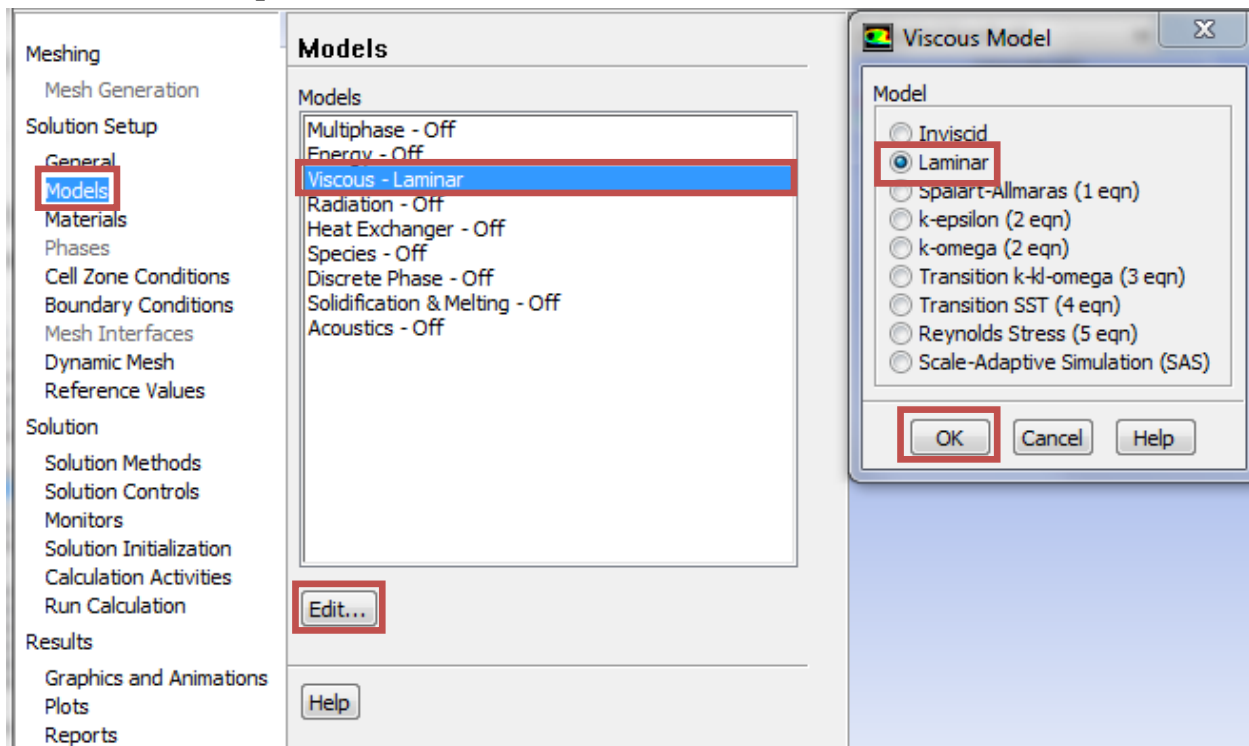6.2. Check **Double Precision** and click **OK**.

6.3. **Solution 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.)
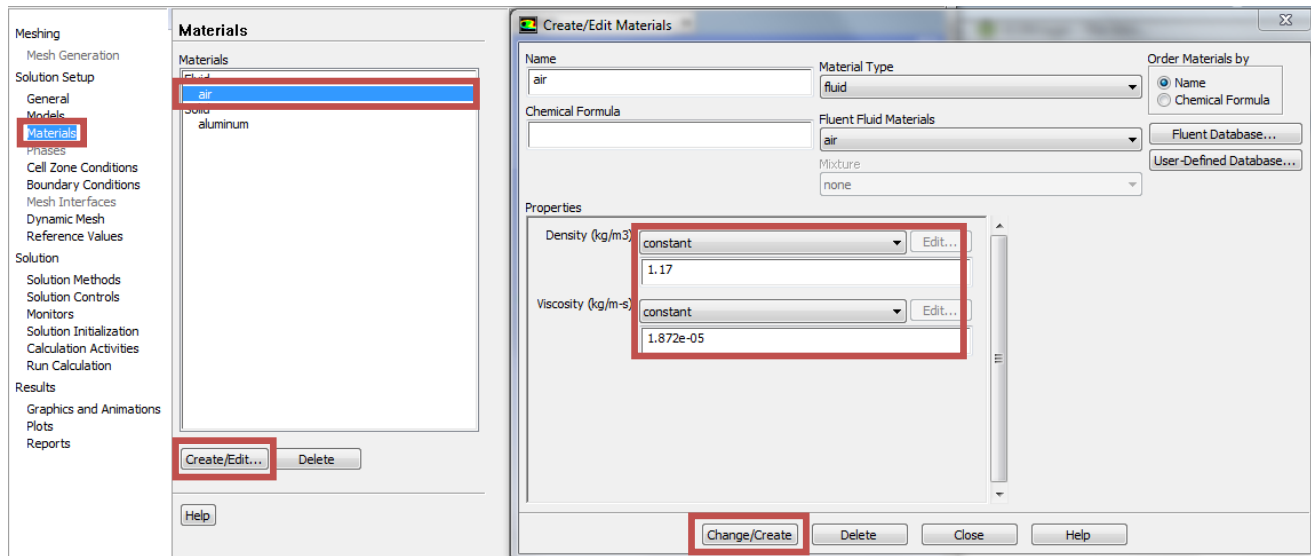
6.4. **Solution Setup** > **General** > **Solver**. Choose options shown below.

6.5. **Solution Setup** > **Models** > **Edit…** Make sure **Laminar** is selected and click **OK**.
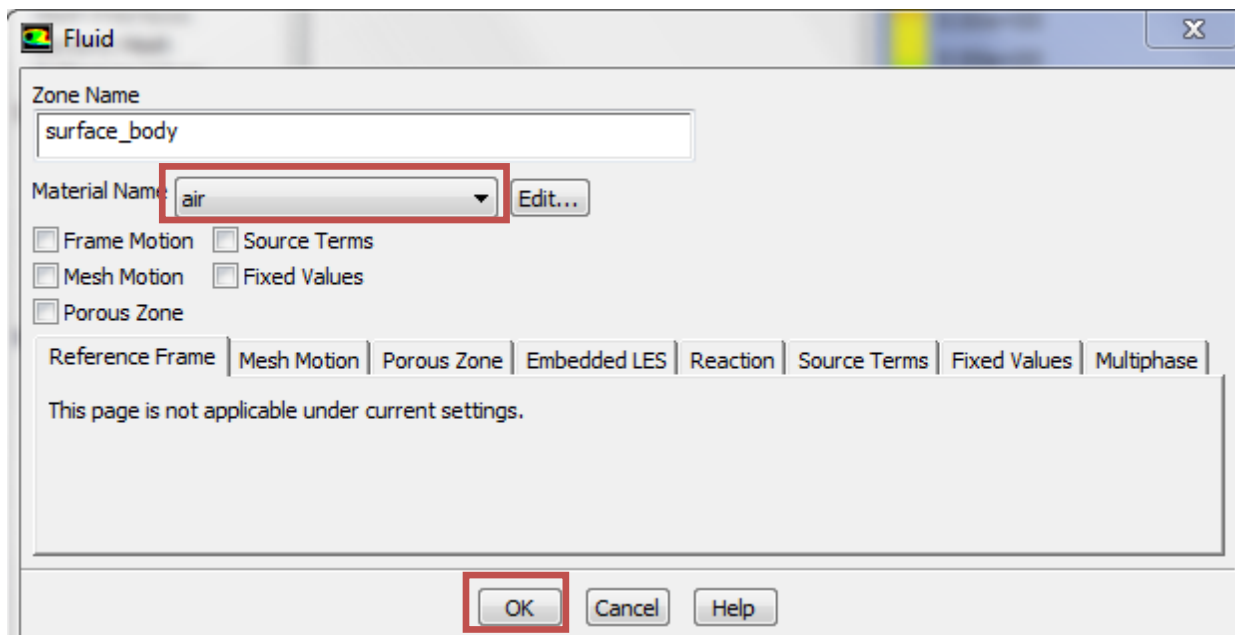
6.6. **Solution Setup** > **Materials** > **air** > **Create/Edit**. Change the **Density** and **Viscosity** as per below and click **Change/Create**. Close the dialog box when finished.


6.7. **Solution Setup** > **Cell Zone Conditions** > **Zone** > **surface_body**. Change type to **fluid** and click **OK**. Select **Material Name** as **air** and click **OK**. This should be defaulted to fluid.
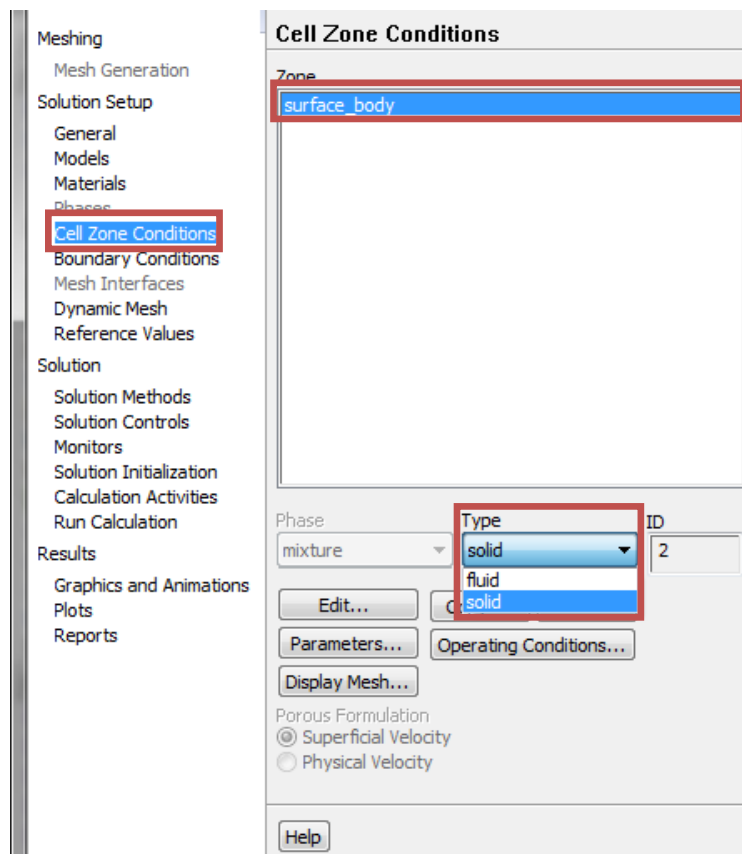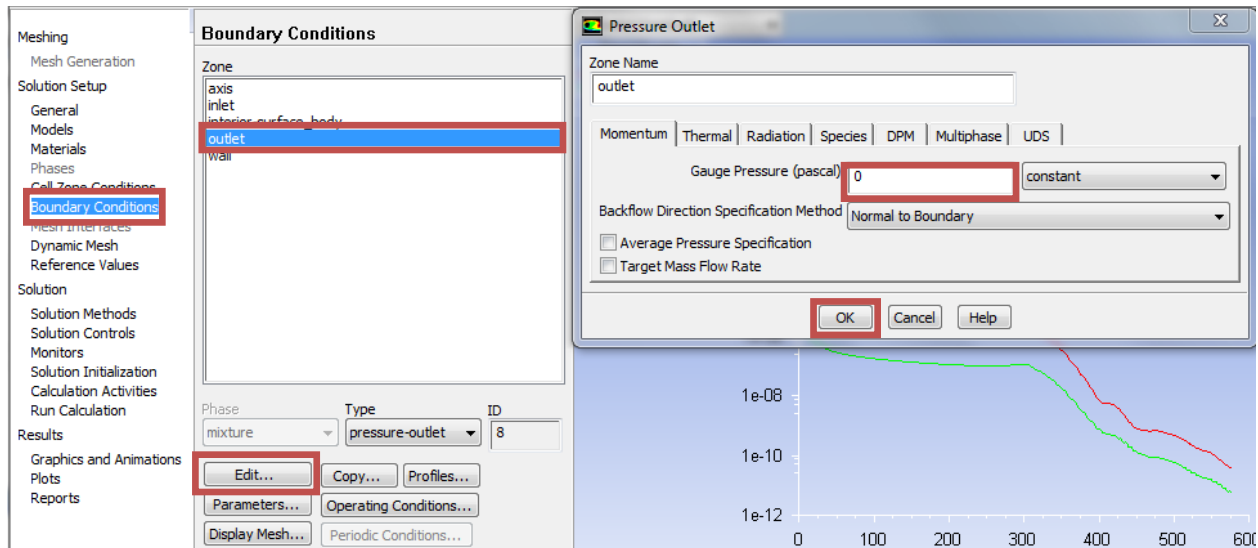
6.8. **Solution Setup** > **Boundary Conditions** > **inlet** > **Edit…** Change parameters as per below and click **OK**. Table below shows the summary of

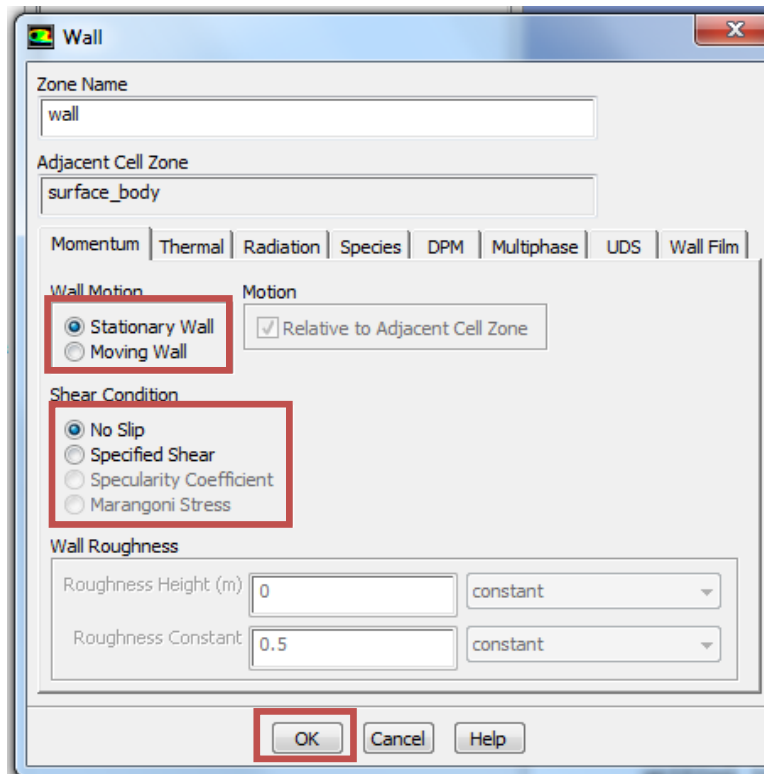| Inlet Boundary Condition			
Variable	u (m/s)	v (m/s)	P (Pa)
Magnitude	0.2	0	-
Zero Gradient	N	N	Y
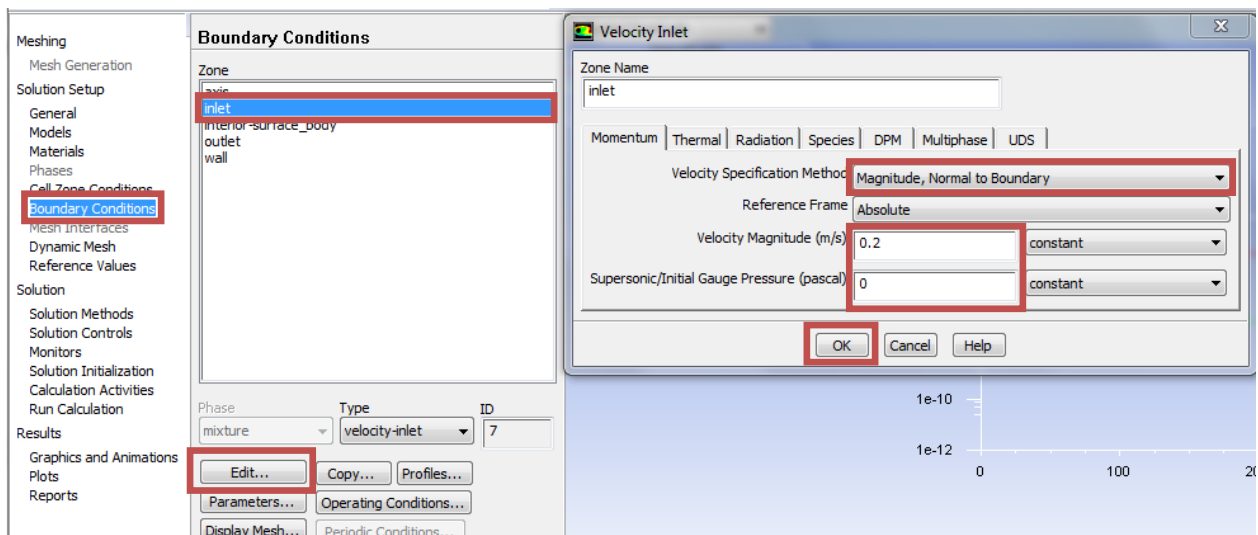
6.9. **Solution Setup** > **Boundary Conditions** > **outlet** > **Edit…** Change parameters as per below and click **OK**.

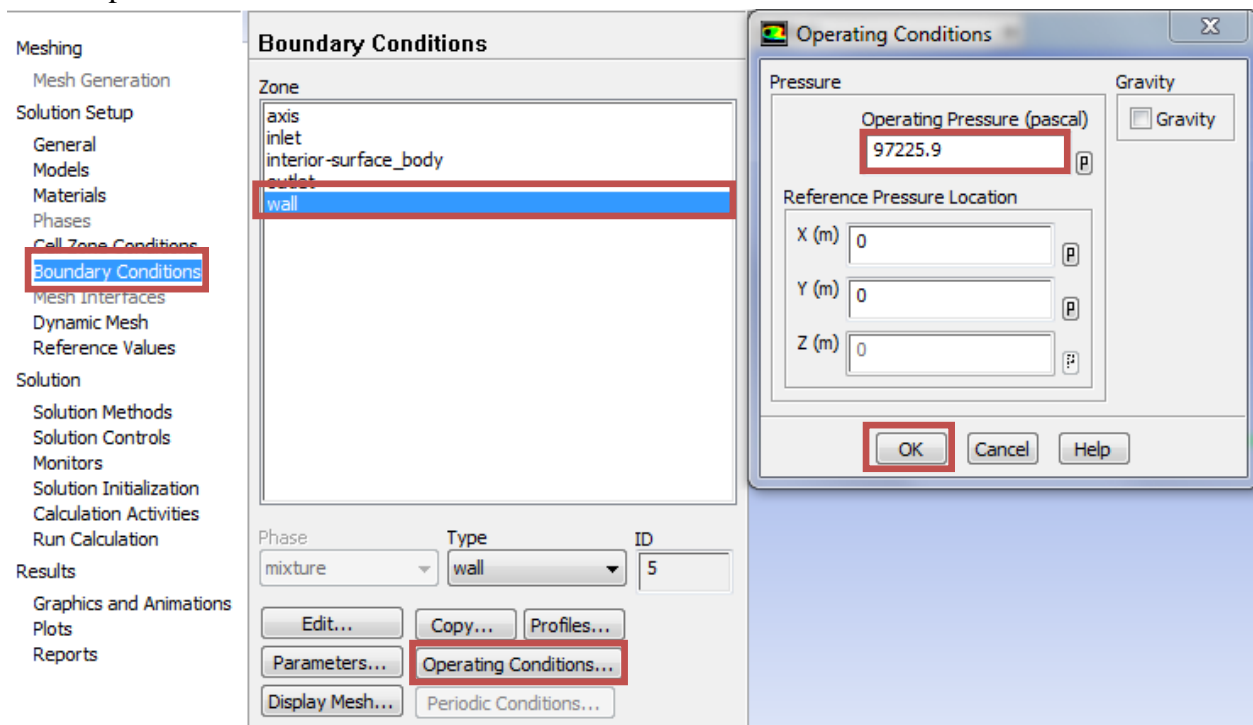| Outlet Boundary Condition			
Variable	u (m/s)	v (m/s)	P (Pa)
Magnitude	-	-	0
Zero Gradient	Y	Y	N

6.10. **Solution Setup** > **Boundary Conditions** > **wall** > **Edit…** Change parameters as per below and click **OK**.

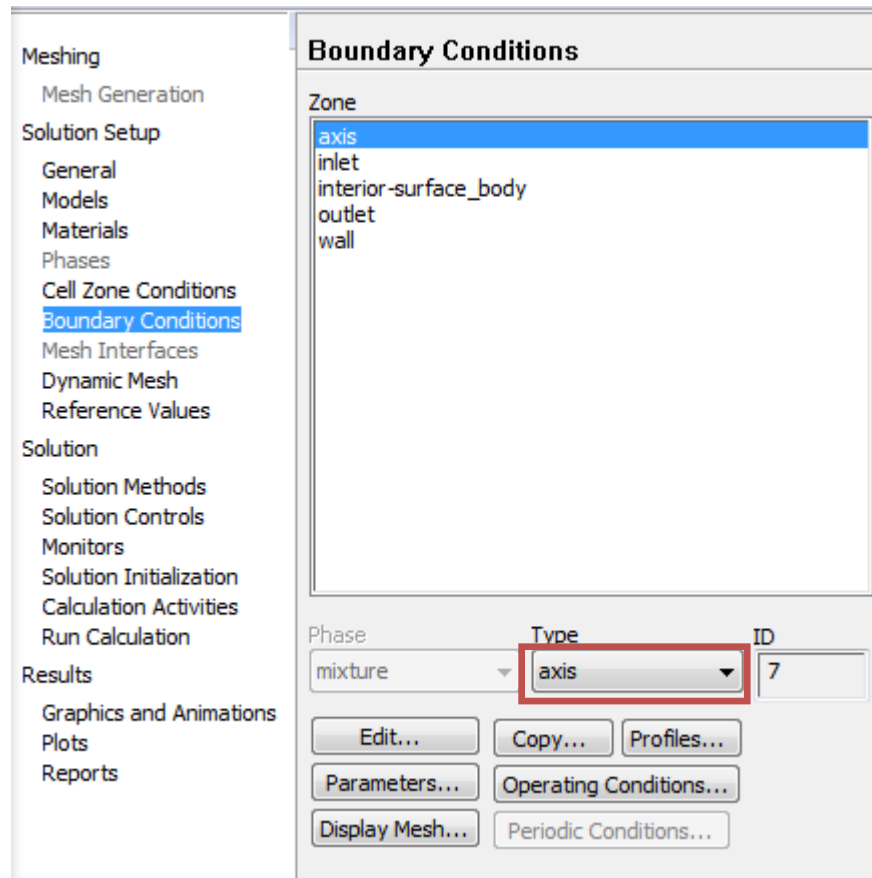| Wall Boundary Condition			
Variable	u (m/s)	v (m/s)	P (Pa)
Magnitude	0	0	-
Zero Gradient	N	N	Y

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

6.12.   **Solution 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	N	Y

6.13.   **Solution Setup** > **Reference Values**. Change parameters as per below.

7. Solution

7.1. **Solution** > **Solution Methods**. Change parameters as per below.

7.2. **Solution** > **Monitors** > **Residuals –Print, Plot** > **Edit…** Change convergence criterion to 1e-06 for all three equations as per below and click **OK**.
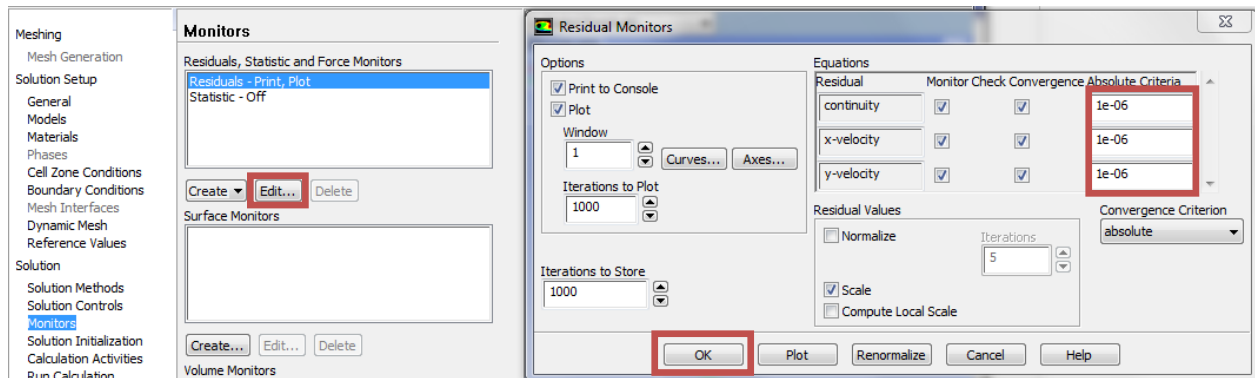
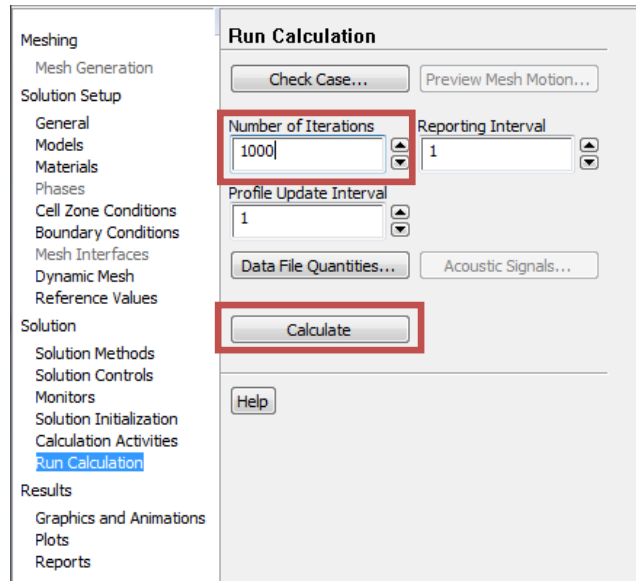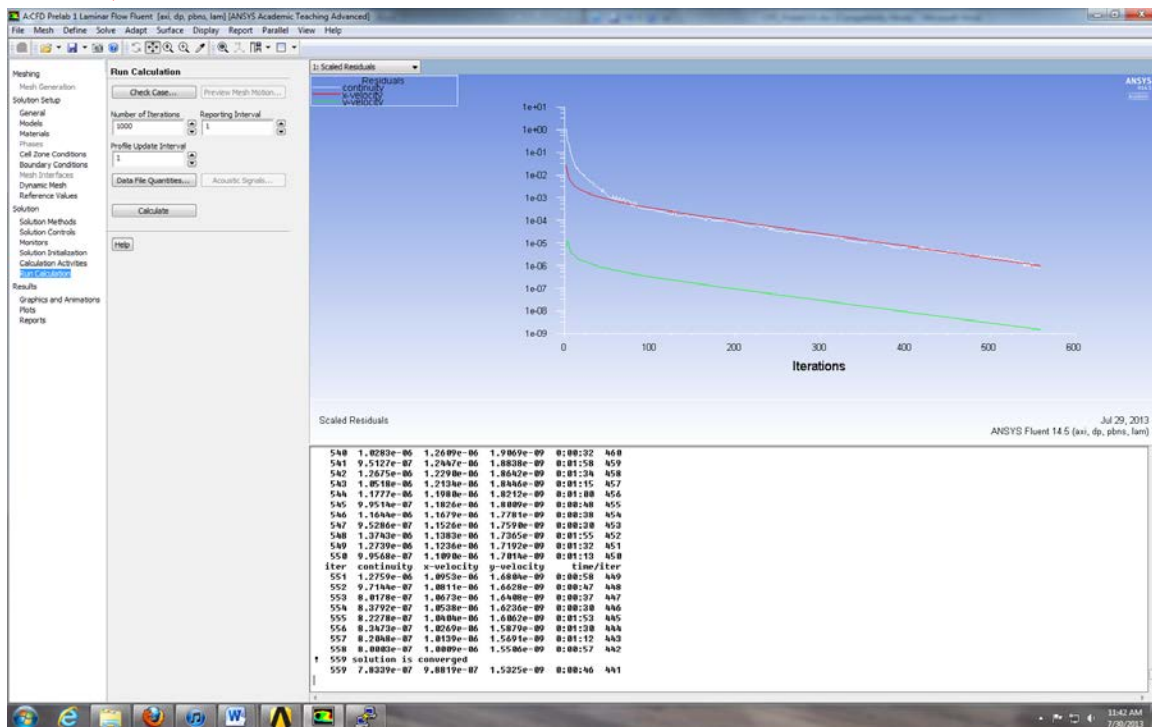7.3. **Solution** > **Solution Initialization**. Change parameters as per below and click **Initialize**.
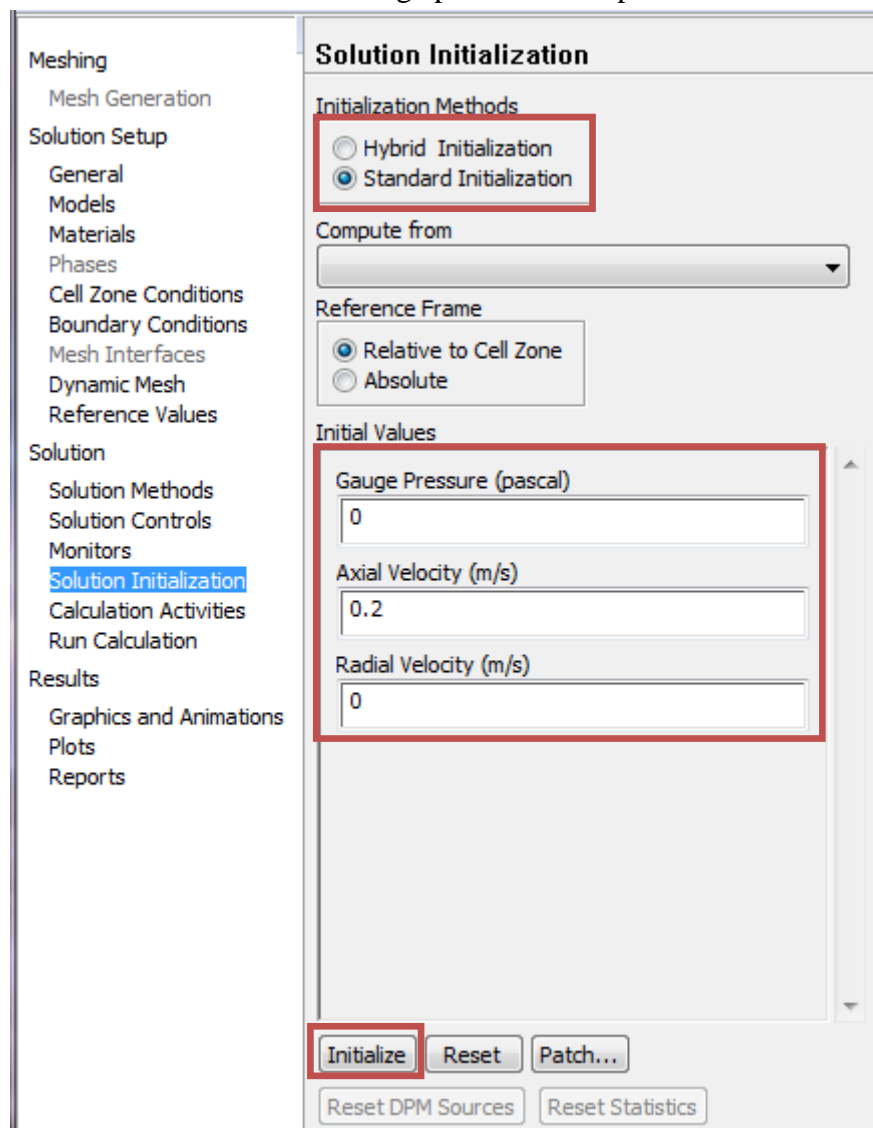
7.4. **Solution** > **Run Calculation**. Change **Number of Iterations** to 1000 and click **Calculate**.

7.5. 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.
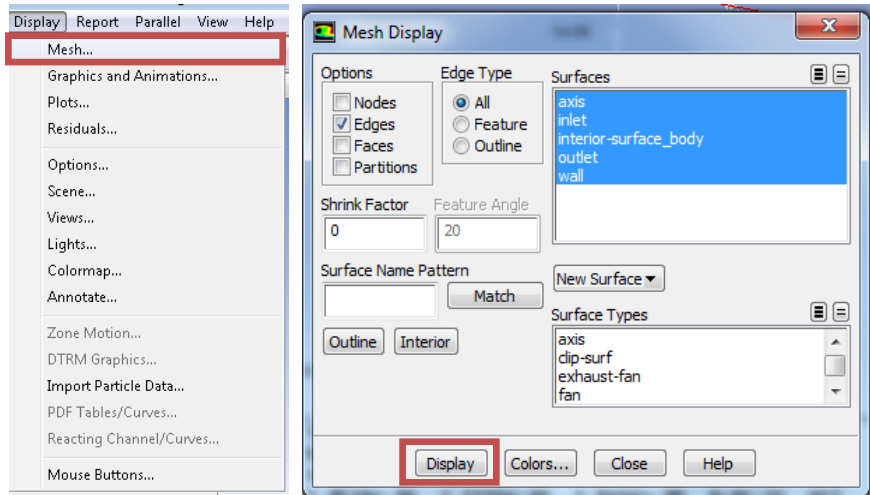
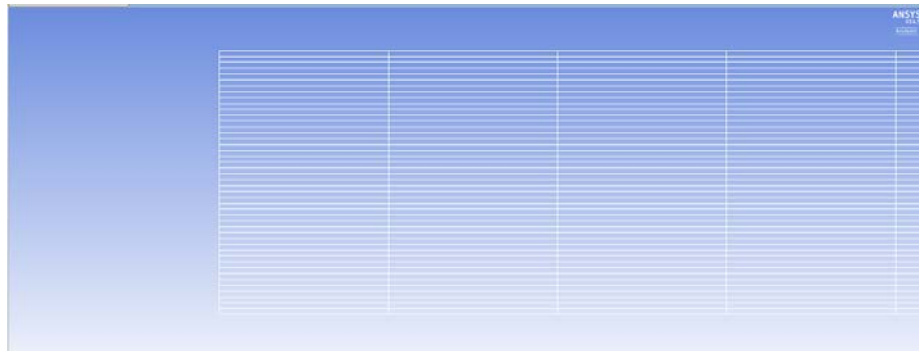7.6. **File** > **Save Project**. Save the project

8. Results

Please read exercises before continuing.

8.1. Displaying Mesh

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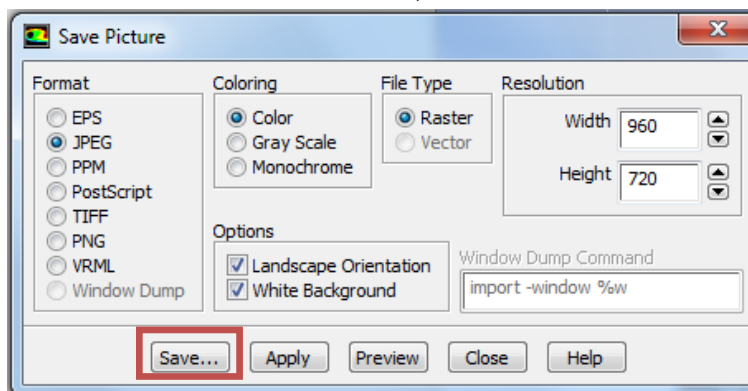

Zoom in to the inlet by using the magnifying glass with a plus sign in the middle of it. The mesh should look like the one below.


8.2. 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)

Name the File as *CFD Pre-Lab1 Laminar Pipe Flow Mesh*, navigate to the CFD Pre-Lab 1 file you created and save it in that file. Then close the **Save Picture** window.
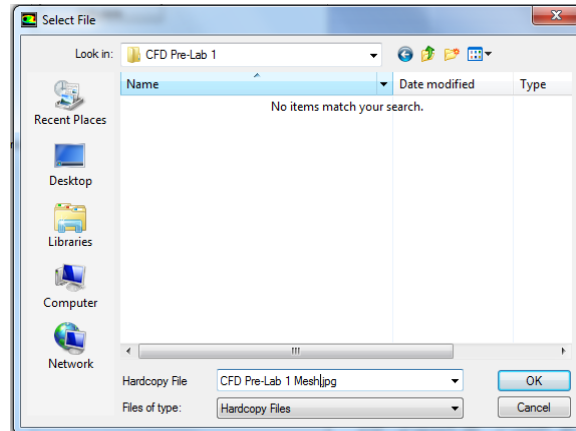
8.3. Displaying and Saving Residuals

To display the residuals click **Solution** > **Monitors** > **Residuals – Print, Plot** > **Edit…** > **Plot** then click **Cancel.**
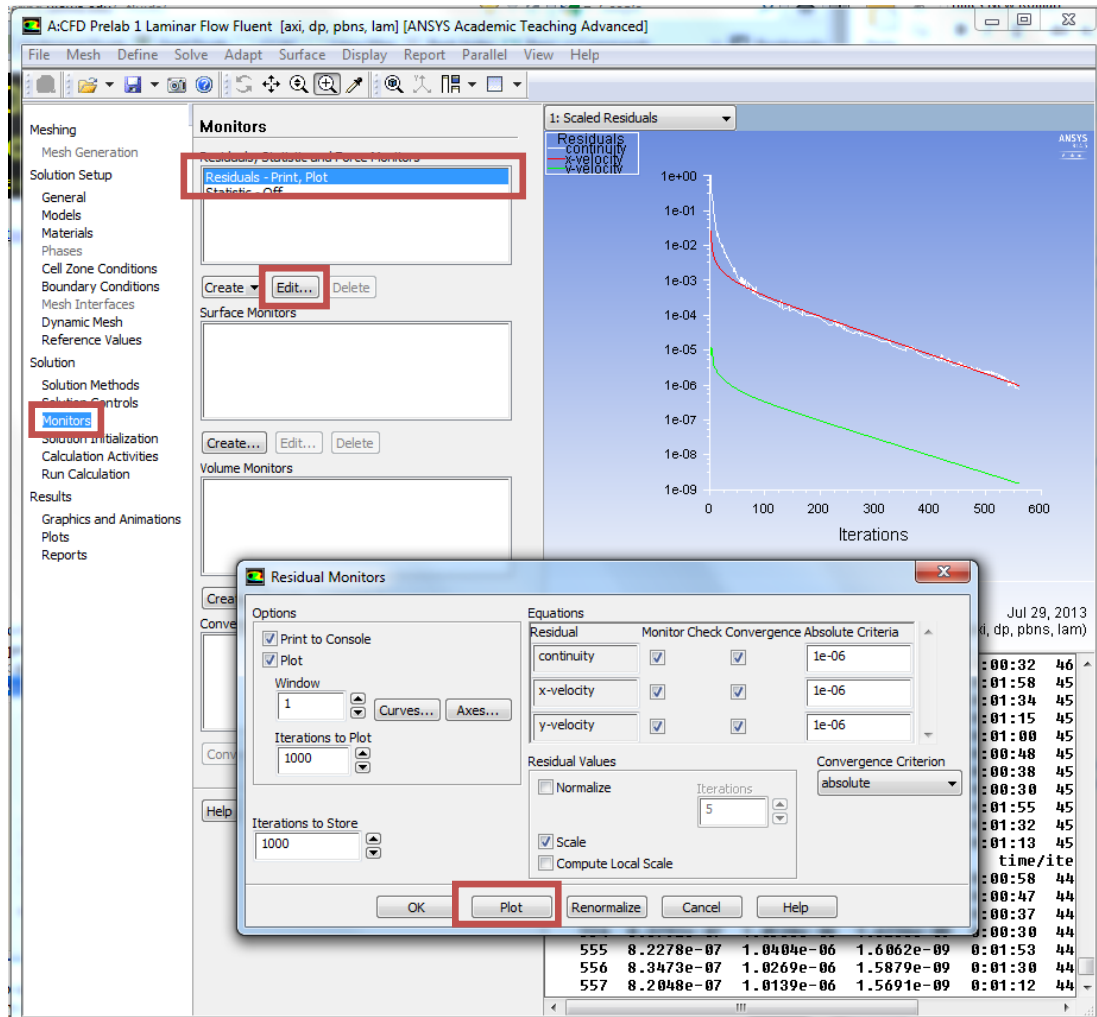
You can save this picture the same way you saved the mesh. Name it *CFD Pre-Lab 1 Laminar Pipe Flow Residuals History* and save it to the folder you created on the H: Drive.

8.4. Plotting and Saving Results

To plot results, click **Results** > **Plots** > **XY Plot** > **Set Up…**

To plot the Centerline Pressure Distribution, copy the parameters as per below and click **Plot**.
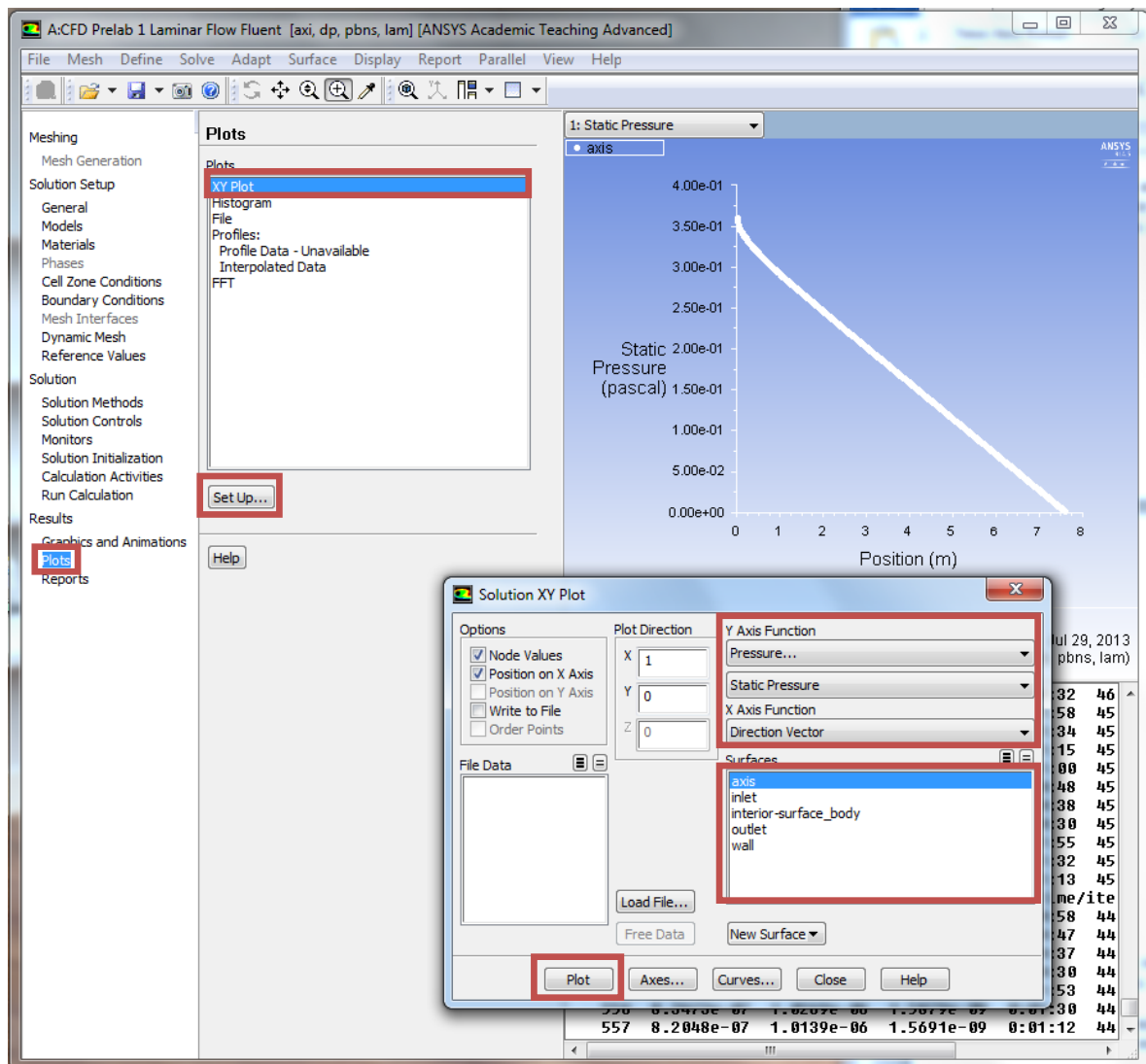
Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Centerline Pressure Distribution* and save it in the folder you created.

.

To plot Centerline Velocity Distribution, copy the parameters as per below and click **Plot**.
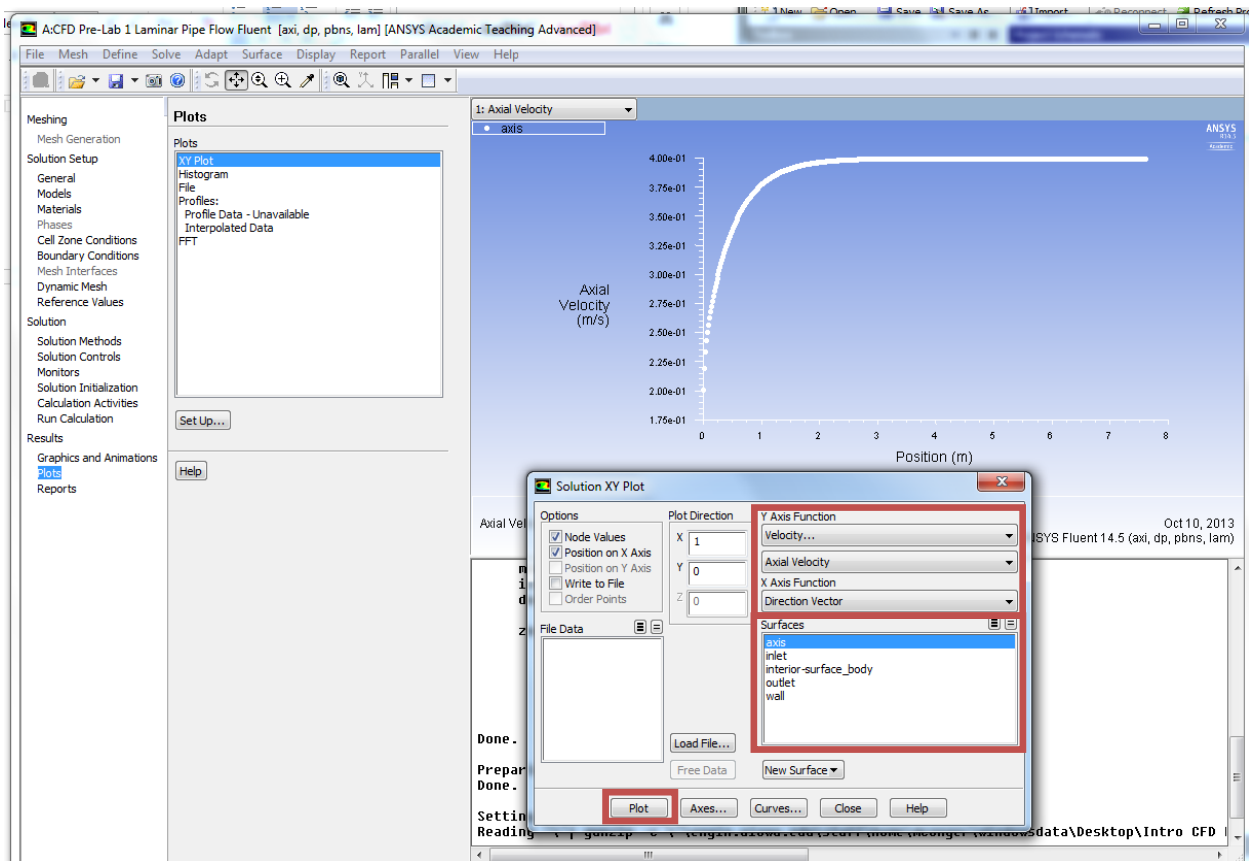
Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Centerline Velocity Distribution* and save it in the folder you created.

To plot the Wall Shear Stress Distribution, copy the parameters as per below and click **Plot**.

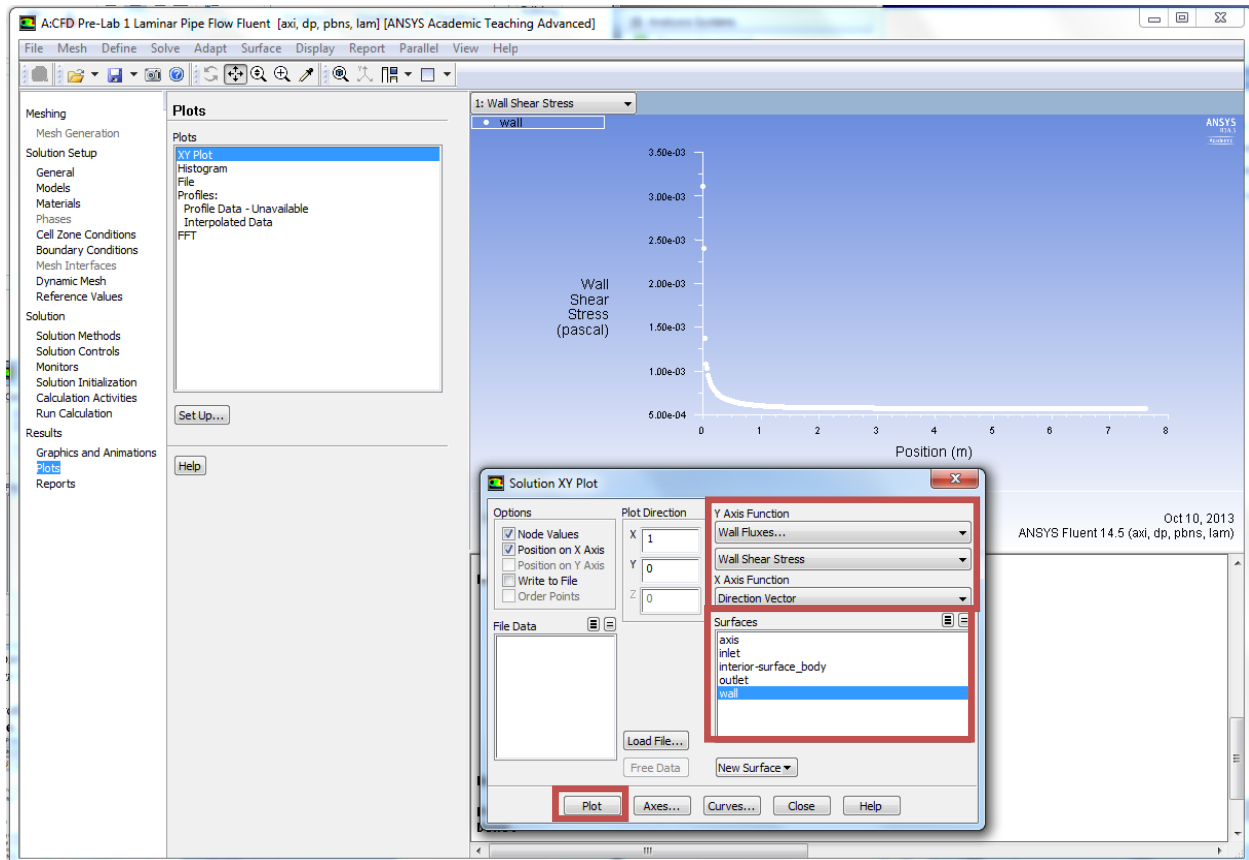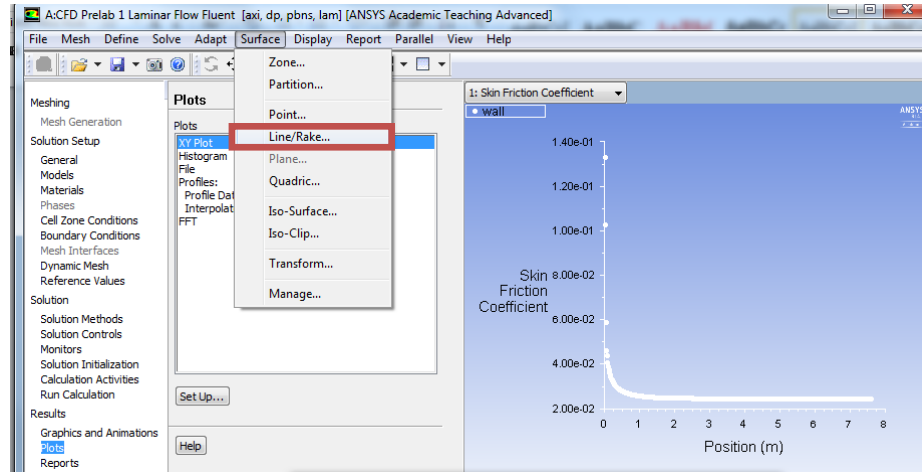Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Wall Shear Stress Distribution* and save it in the folder you created.

To plot Profiles of Axial Velocity at All Axial Locations with AFD Data, click **Surface** > **Line/Rake…**

Change x and y values as per below, name the surface, and click **Create**. Repeat this for all lines shown in the table.
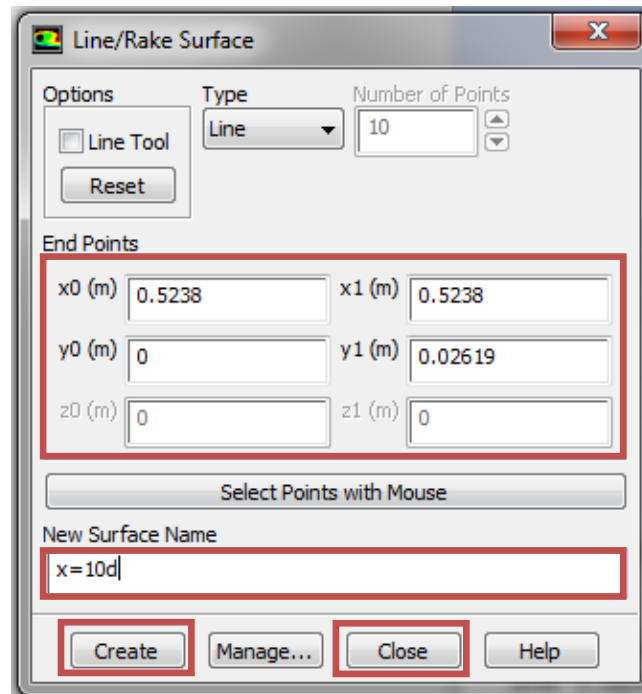
Surface Name	X0	Y0	X1	Y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.238	0	5.238	0.02619

When all lines are created, click **Close**.

Click **Results** > **Plots** > **XY Plot** > **Set Up…** Click **Load File…** and select axialvelocityAFD-laminar-pipe.xy, which can be found from the class website. Click **OK**.
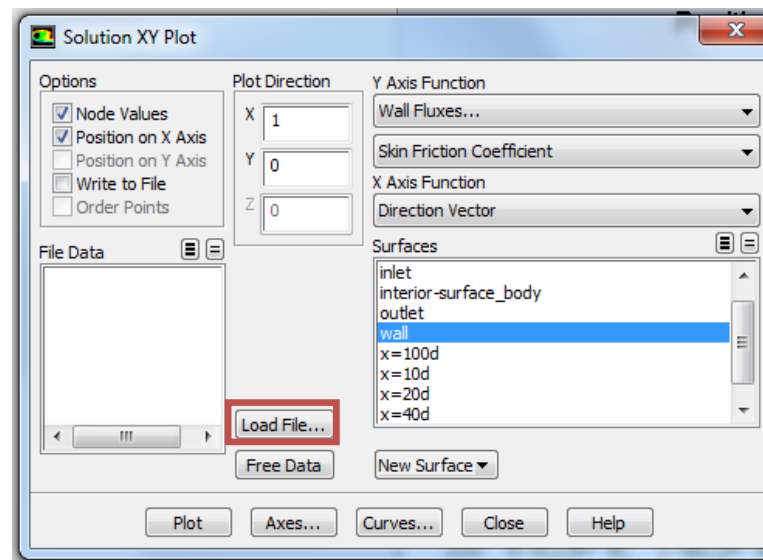
Change Parameters as per below. Make sure to select inlet as well.
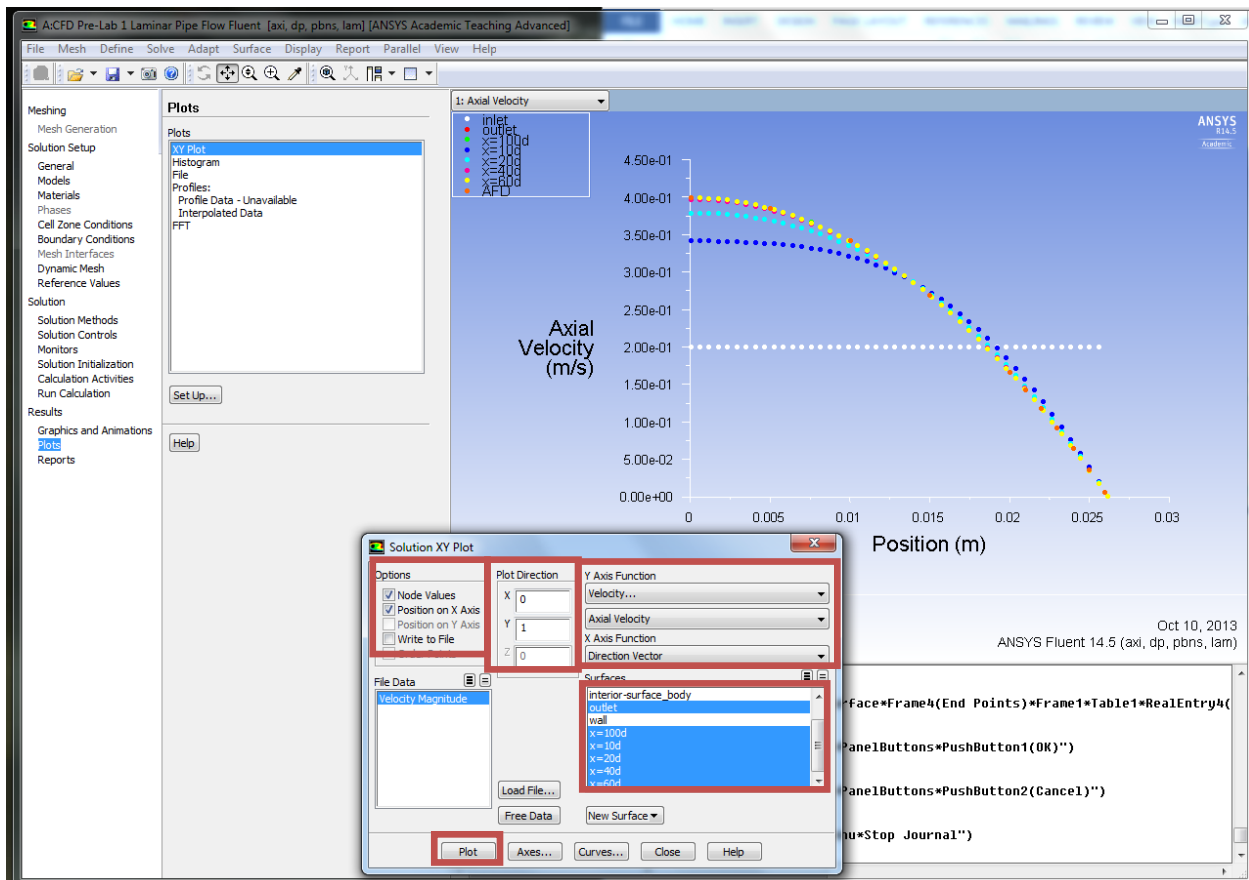
Click **Curves…** > Change the **Pattern** to the pattern seen below and click **Apply**. Incriment the **Curve #** by one and repeat. Do this for curves 0 through 7 then click **Close**.
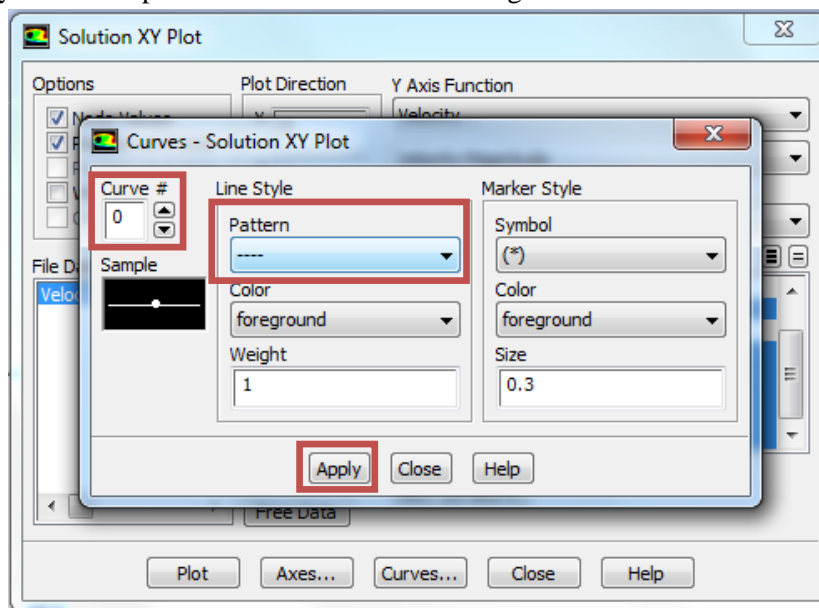
Click **Plot.**
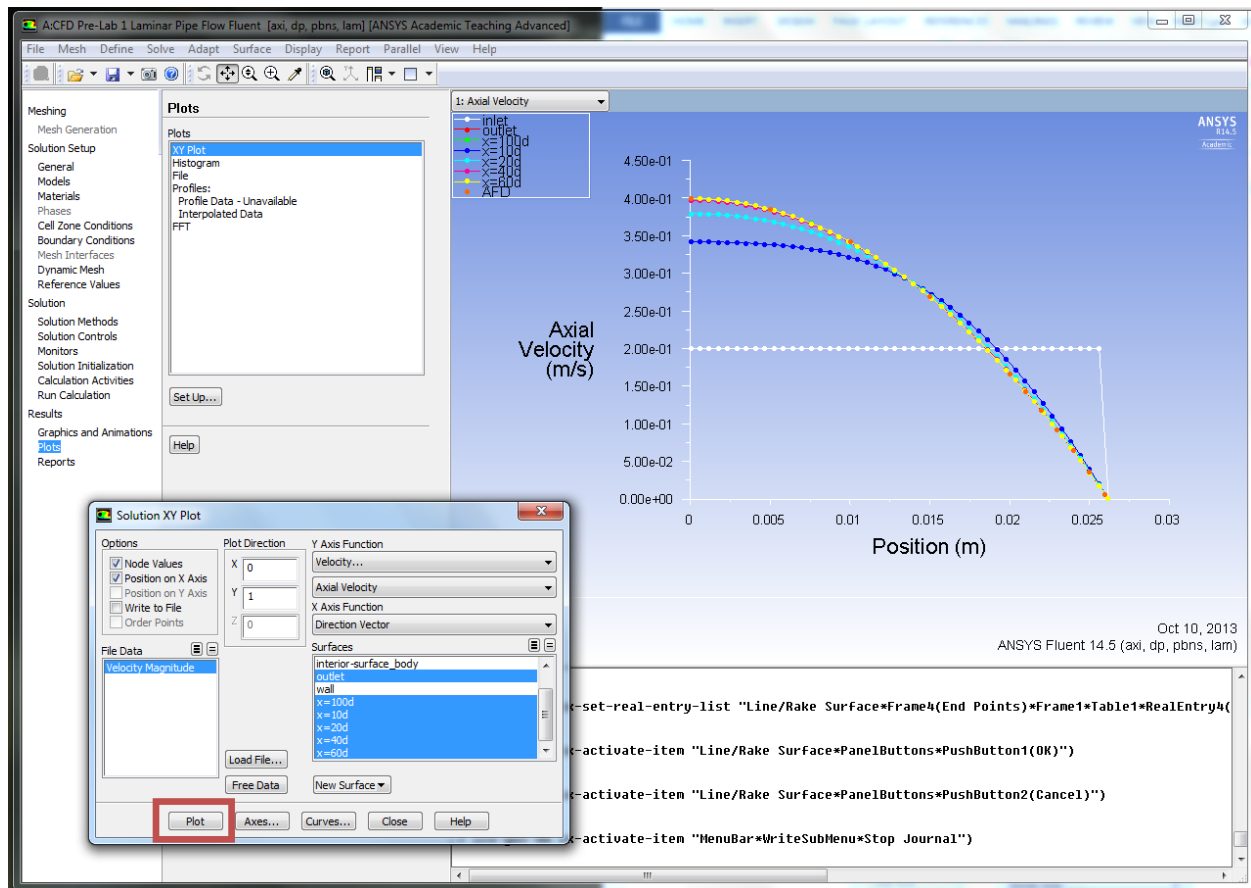
Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Axial Velocity at All Axial Locations with AFD Data* and save it in the folder you created. Close the **Solution XY Plot** window.

8.5. Plotting and Saving Graphics

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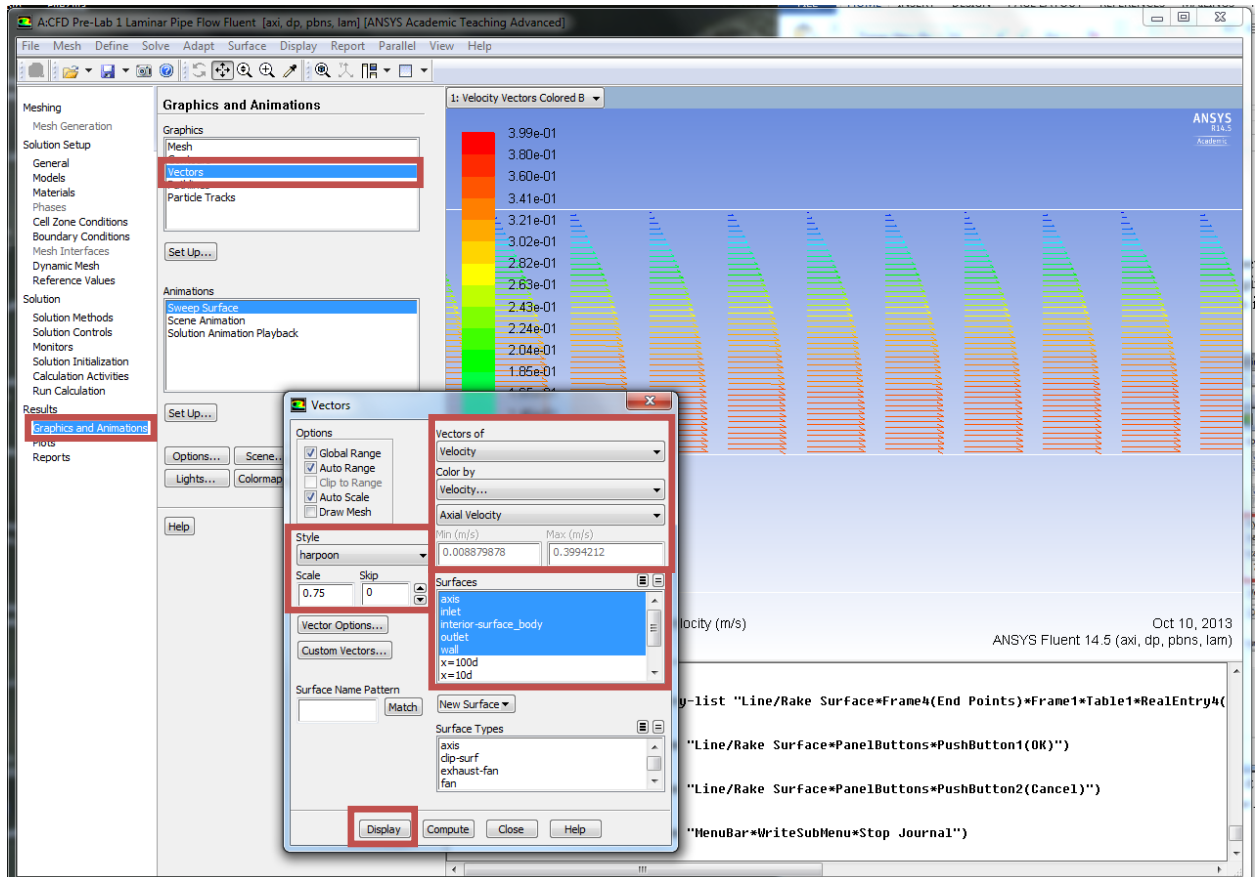
Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Laminar Pipe Flow Velocity Vectors at the Region Flow Begins to Become Fully Developed* and save it in the folder you created. Close the **Vectors** window.

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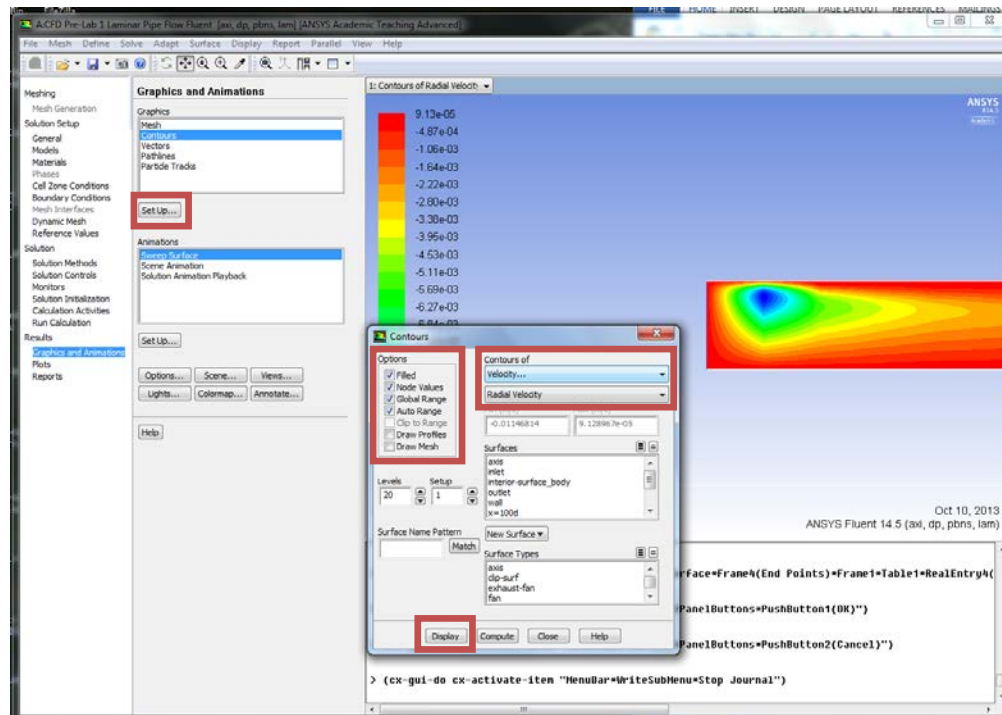**Display**. Zoom in to the pipe inlet to see the contours of radial velocity.
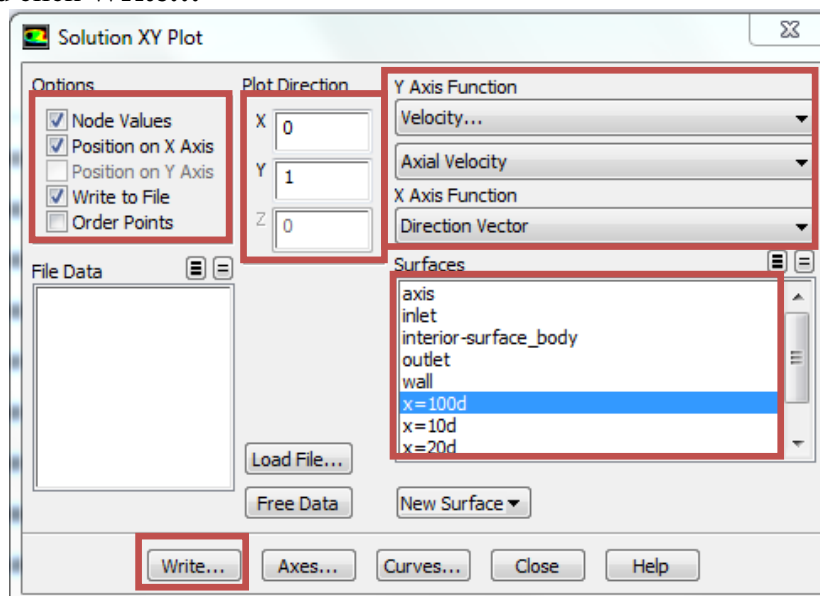

Save the picture as you did for the mesh and call it *CFD Pre-Lab 1 Contours of Radial Velocity* and save it in the folder you created. Close the **Contours** window.
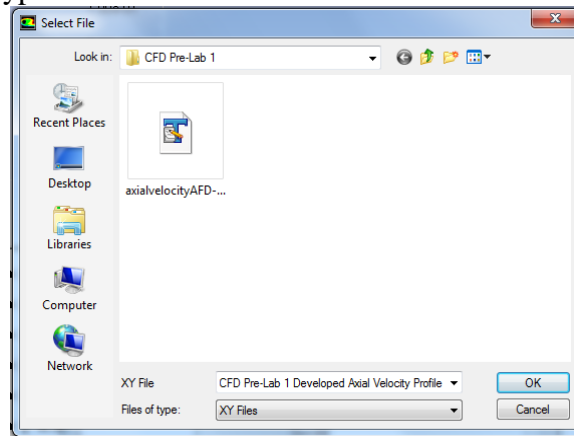
8.6. Exporting Results

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

To export the wall shear stress distribution, copy the parameters as per below and click **Write…**
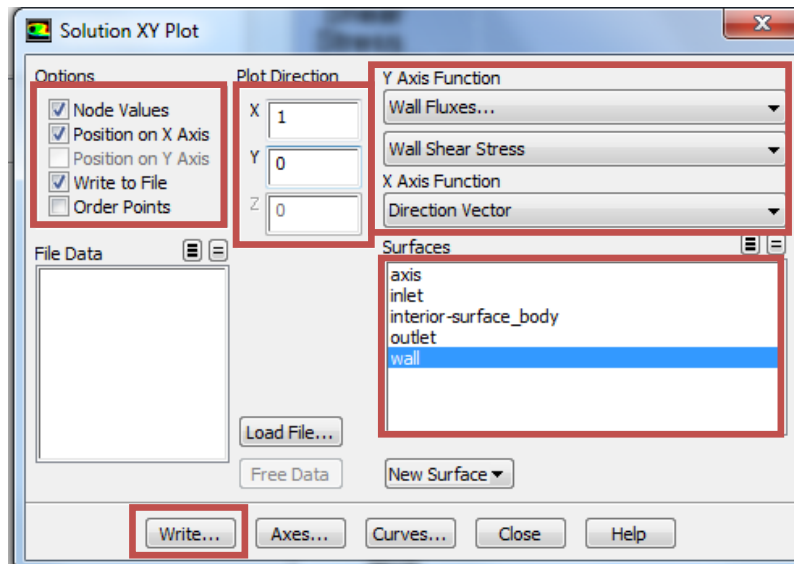
Name the file *CFD Pre-Lab 1 Laminar Pipe Flow Wall Shear Stress Distribution* and leave the Files of Type: as XY Files. Click **OK.** Close the Solution XY Plot.

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

8.7. Normalizing Velocity Profile
- Open excel from Start Menu.
- Click **File** > **Open**, navigate to your folder you created on the H: 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.
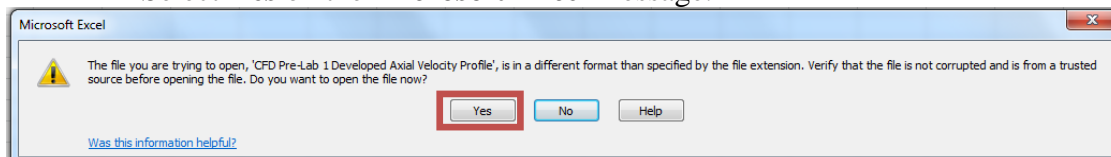
- Make sure delimited is selected and click **Next >.**
- Make sure that **Tab** and **Space Delimiters** are checked and hit **Finish**.

- 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.

- Insert a **Scatter Plot With Smooth Lines and Markers.**
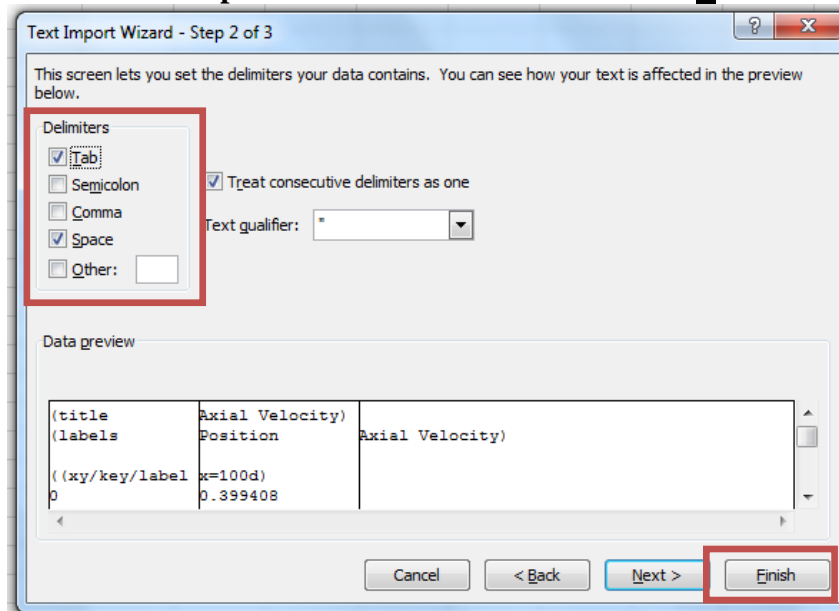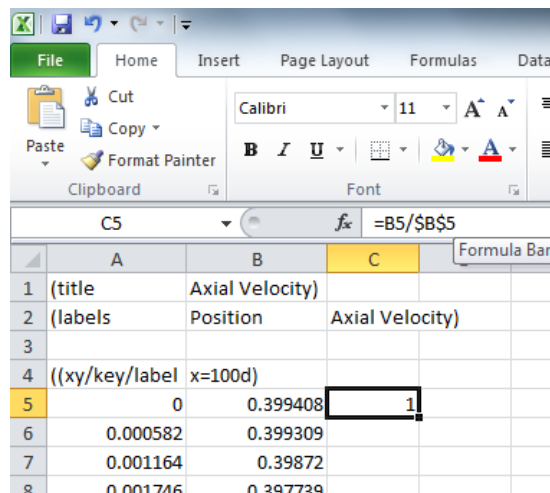- For the x-axis use the radial position, and for the y-axis use the normalized velocity.
- Name it CFD Velocity Profile (Laminar).
- You can move this plot to a new tab by clicking on the chart **Chart Tools** > **Design** > **Move Chart Location** > **New Sheet** > **OK**
- Next open the file Normalized-velocity-AFD-laminar-pipe.xy 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 Veloctity [-]* and *Radial Position [m]*.

- Save this Sheet by selecting File > Save As, name it *CFD Pre-Lab 1 Developed Axial Velocity Profile*.

9. 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 use CFD Lab1 Report Template.doc to save all the figures and data**

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 (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*τ/(ρ*U^2) to solve for the Friction Factor. Where C is the friction factor, τ is wall shear stress, ρ 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 (x/D=10,20,40, 60,100) with AFD data, 6. contour of radial velocity, and 7. velocity vectors (pick up the region where flow begins to become fully developed).
- **Data need to be saved:** shear stress in the developed region, developing length. Here, developing length is defined as the length from pipe inlet to the axial location where the centerline velocity does not change any more.

2. Normalized developed axial velocity profile

2.1. Export the axial velocity profile data at x=100d following the instructions in Step 8.6.
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 WORD, together with other figures you made in Exercise 1.

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

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