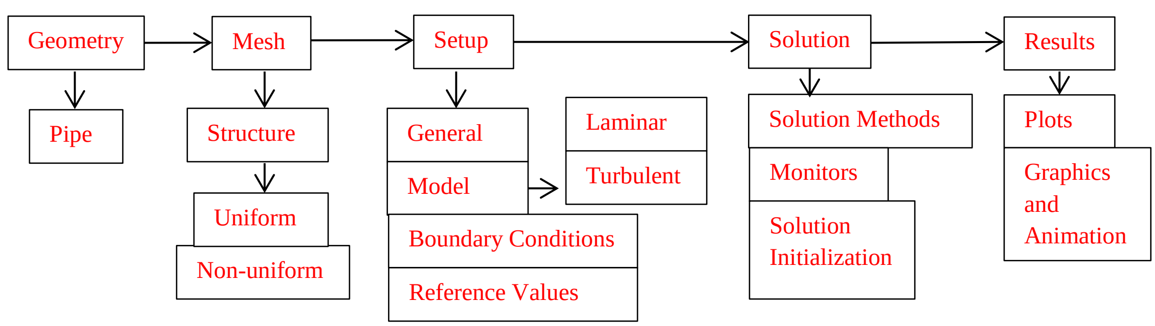
Verification of Laminar and Validation of Turbulent Pipe Flows

58:160 Intermediate Mechanics of Fluids
CFD LAB 1

By Timur Dogan, Michael Conger, Maysam Mousaviraad and Fred Stern
IIHR-Hydroscience & Engineering
The University of Iowa
C. Maxwell Stanley Hydraulics Laboratory
Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD Lab 1 is to simulate steady **laminar** and **turbulent** pipe flow following the "CFD Process" by an interactive step-by-step approach. Students will have "hands-on" experiences using ANSYS to compute axial velocity profile, centerline velocity, centerline pressure, and friction factor. Students will conduct **verification studies for friction factor and axial velocity profile** of laminar pipe flows, including iterative error and grid uncertainties and effect of refinement ratio on verification. Students will validate **turbulent pipe flow** simulation using EFD data, analyze the differences between laminar and turbulent flows, and present results in CFD Lab report.



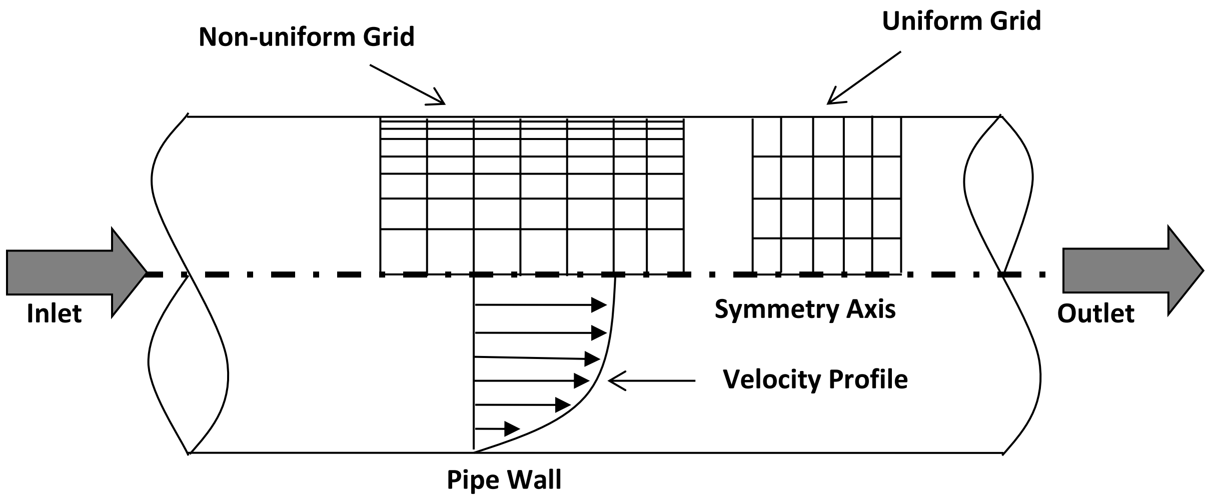
Flow Chart for ANSYS

2. Simulation Design

In CFD Lab 1, simulation will be conducted for **laminar and turbulent** pipe flows. Reynolds number is 655 for laminar flow and 111,569 for turbulent pipe flow, based on pipe diameter. The schematic of the problem and the parameters for the simulation are shown below.

Table 1 - Main particulars

Parameter	Unit	Value
Radius of Pipe	m	0.02619
Diameter of Pipe	m	0.05238
Length of the Pipe	m	7.62



Since the flow is axisymmetric we only need to solve the flow in a single plane from the centerline to the pipe wall. **Boundary conditions** need to be specified include **inlet**, **outlet**, **wall**, and **axis**, as will be described details later. Uniform flow was specified at inlet, the flow will reach the fully developed regions after a certain distance downstream. No-slip boundary condition will be used on the wall and constant pressure for outlet. Symmetric boundary condition will be applied on the pipe axis. Uniform grids will be used for the laminar flow whereas non-uniform grid will be used for the turbulent flow.

Table 2 - Grids

Grid	Grid Type	# of Divisions	
		X	R
8	Uniform	453	45
7		320	32
6		227	23
4		113	11
3		80	8
2		57	6
0		28	3
T	Non-uniform	564	15

Experimental, analytical and simulations will be compared. Additionally, detailed verification and validation study will be conducted. All the studies are detailed in the Table 3. In this manual, detailed instructions are given for the turbulent flow simulation and laminar flow simulations using non-uniform grid and uniform grid 8 respectively. Figures and data that needs to be saved are shown in Table 4.

Table 3 - Simulation matrix

Study	Grid	Model
V&V of friction factor and axial velocity profile	2,3,4	Laminar
V&V of friction factor	6,7,8	
V&V of friction factor	0,2,4	
V&V of friction factor	4,6,8	
Axial velocity, centerline velocity	8	
Axial velocity, centerline pressure, centerline velocity	T	Turbulent

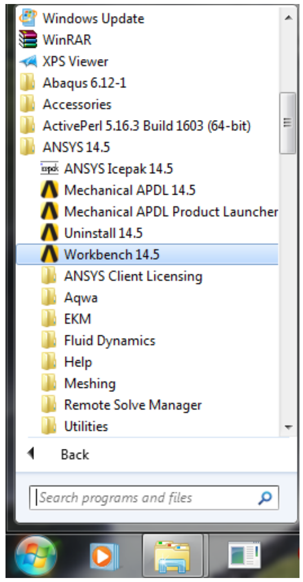
All analytical data (AFD) for Laminar Pipe Flow and EFD data for turbulent pipe flow can be downloaded from the class website (http://css.engineering.uiowa.edu/~me_160).

Table 4 - Figures and data sets needed to be saved

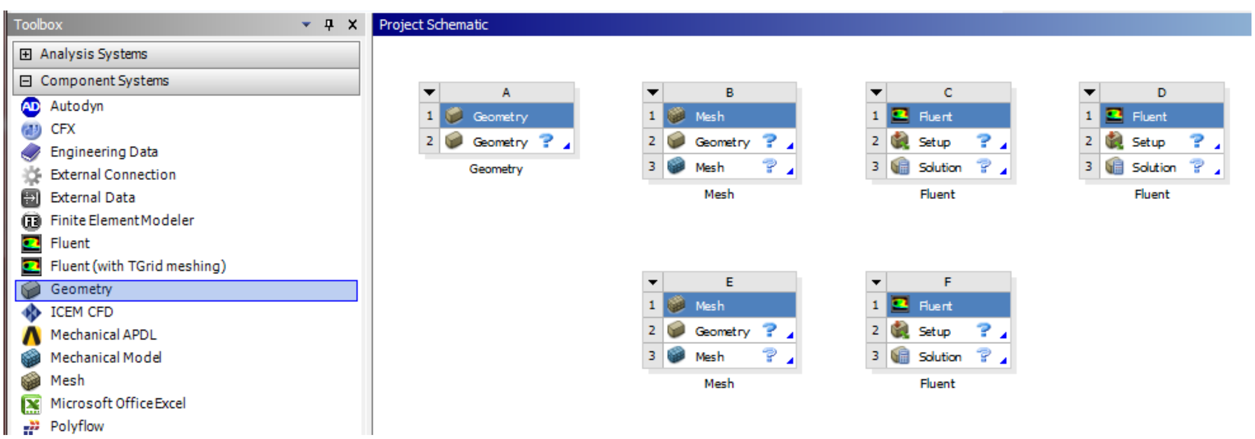
Grid	Flow	Convergence Limit	Figure	Data
T	Turbulent	1.00E-06	*	
8	Laminar	1.00E-06	Residuals	**
8	Laminar	1.00E-05	Residuals	Wall Shear Stress
7	Laminar	1.00E-06		Wall Shear Stress
6	Laminar	1.00E-06		Wall Shear Stress
4	Laminar	1.00E-06		Wall Shear Stress
4	Laminar	1.00E-05		Wall Shear Stress
3	Laminar	1.00E-06		Wall Shear Stress
2	Laminar	1.00E-06		Wall Shear Stress
0	Laminar	1.00E-06		Wall Shear Stress
* Axial velocity profile with EFD data, normalized axial velocity profile at x=100D, centerline pressure distribution with EFD data, “centerline velocity distribution”, contour of axial velocity, velocity vectors showing the developing region and developed regions.				
**Wall Shear Stress, velocity profile (10pts), centerline velocity distribution				

3. Open ANSYS Workbench Template

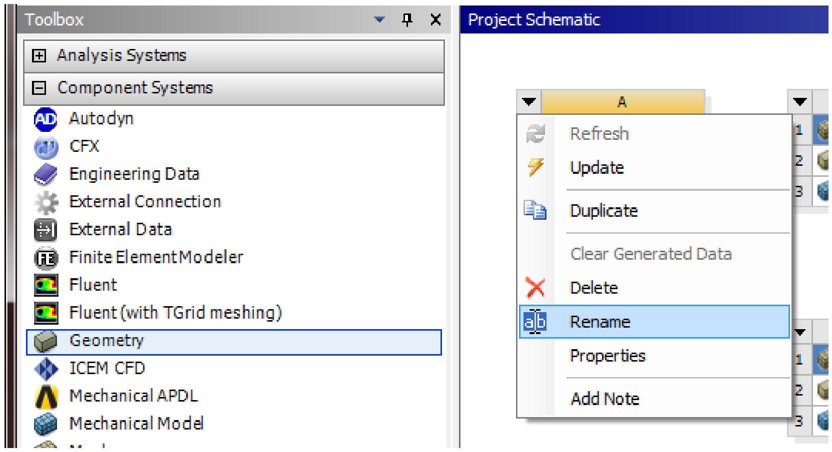
3.1. **Start** > **All Programs** > **ANSYS 14.5** > **Workbench 14.5**

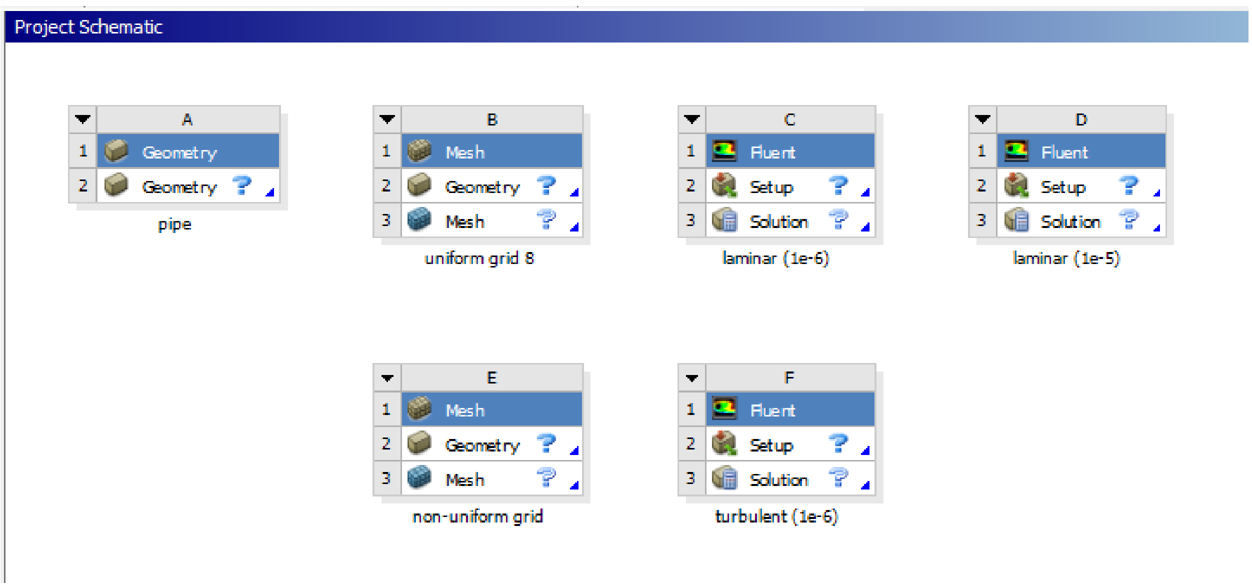


3.2. **Toolbox** > **Component Systems**. Drag and drop **Geometry**, **Mesh** and **Fluent** components to **Project Schematic** as per below.

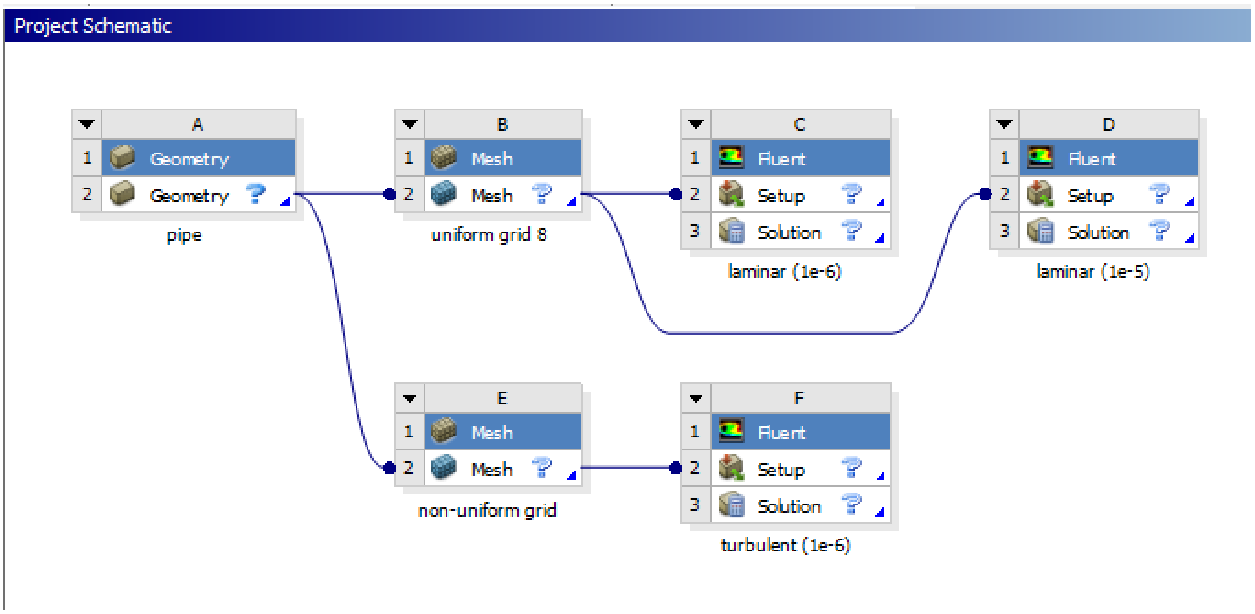


3.3. Right click on the upper corner of the components on the drop down arrow then select rename. Change the names as per below.

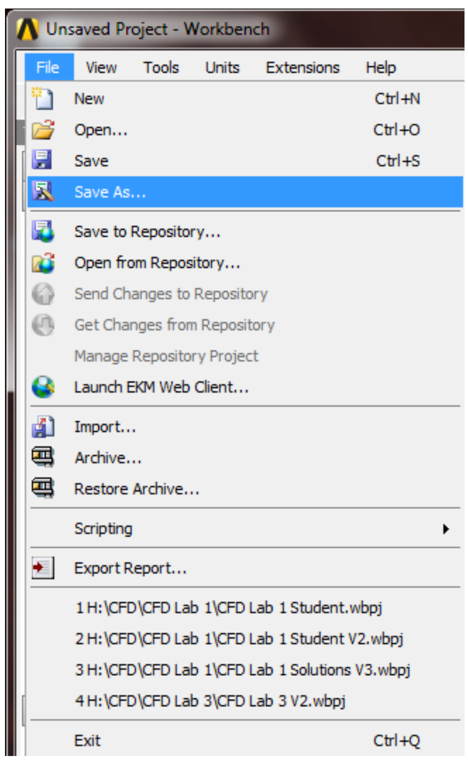




3.4. Create connections between component as per below. You can select components part and drop it onto the target component part to create connections.

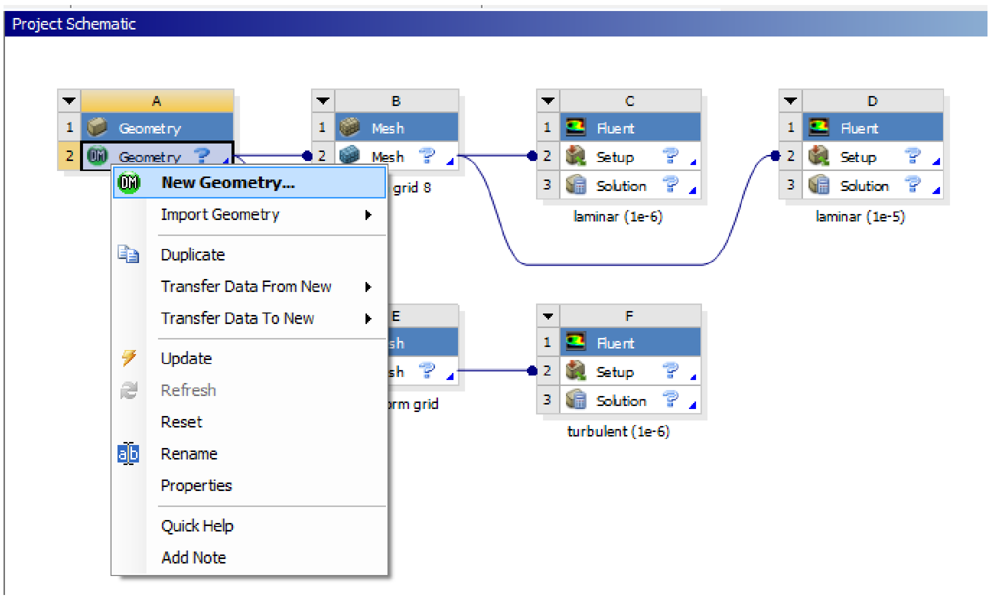


3.5. **File** > **Save As**. Save the workbench file to H drive. The H drive is shared between the computers in engineering labs.

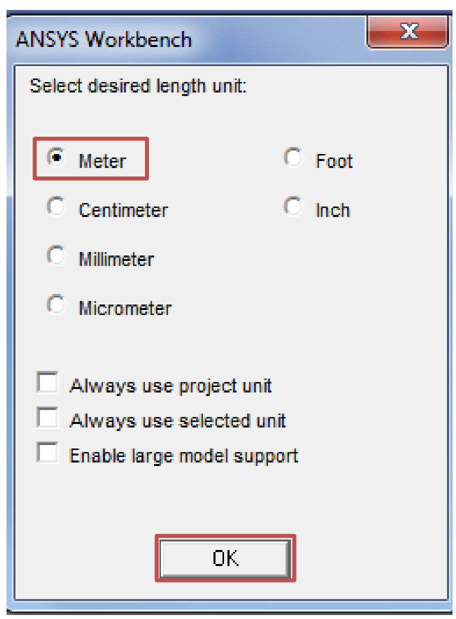


4. Geometry Creation

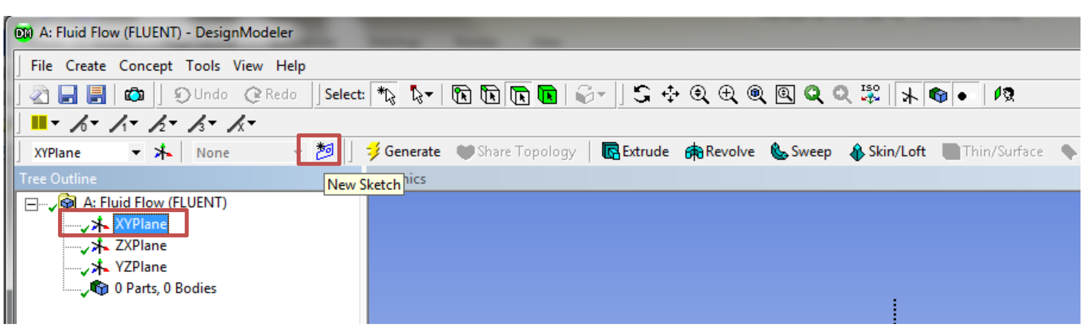
4.1. Right click **Geometry** and select **New Geometry**. (Since all the geometries are linked together, only one geometry creation is required)



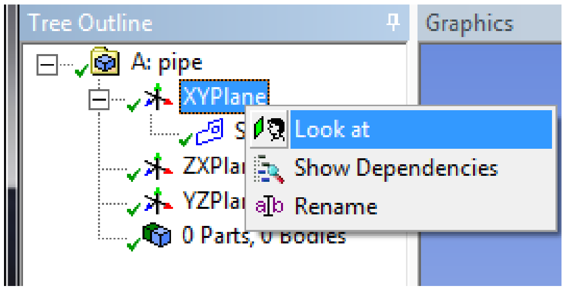
4.2. Select **Meter** for unit and click **OK**.



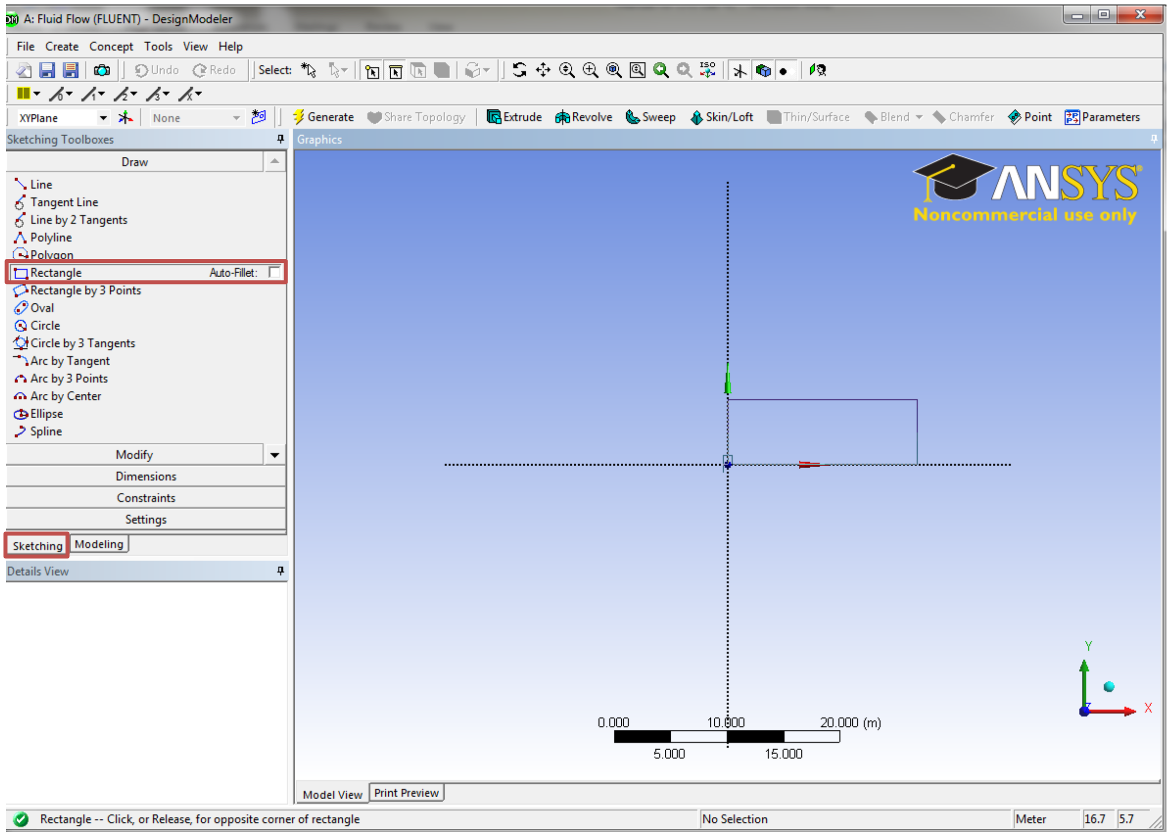
4.3. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.



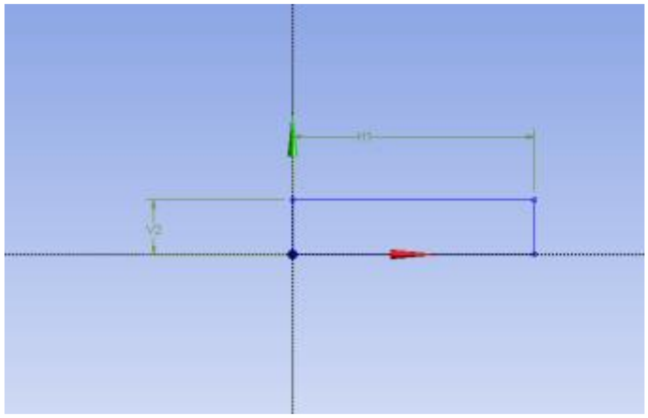
4.4. Right click **XYPlane** and select **Look at**.



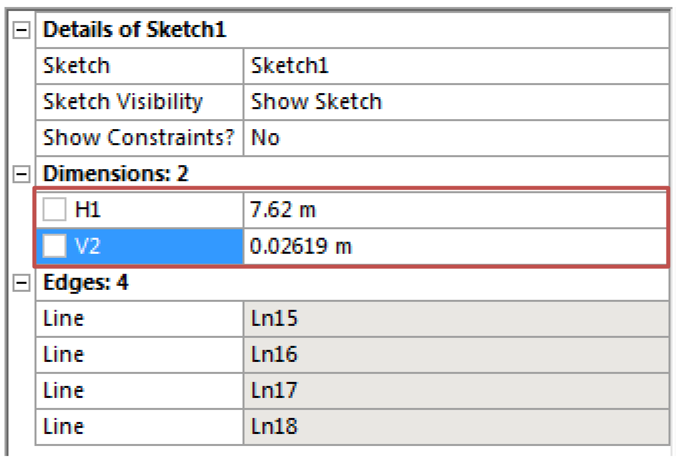
4.5. Select **Sketching** > **Rectangle**. Create a rectangle geometry as per below.



4.6. 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.

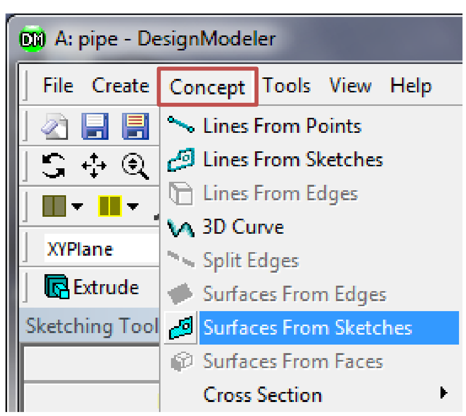


4.7. Click on **H1** under **Details View** and change it to *7.62 m*. Click on **V2** and change it to *0.02619 m*.

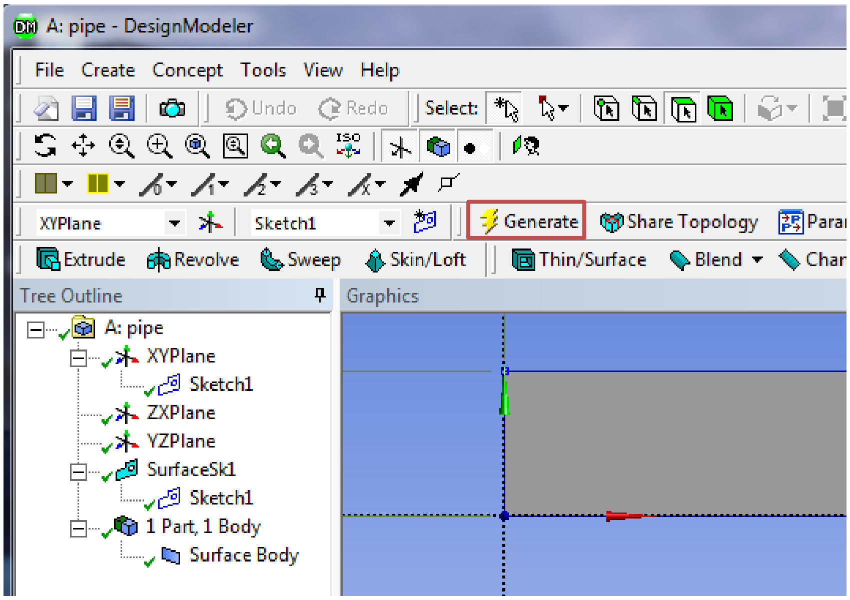


Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 2	
H1	7.62 m
V2	0.02619 m
Edges: 4	
Line	Ln15
Line	Ln16
Line	Ln17
Line	Ln18

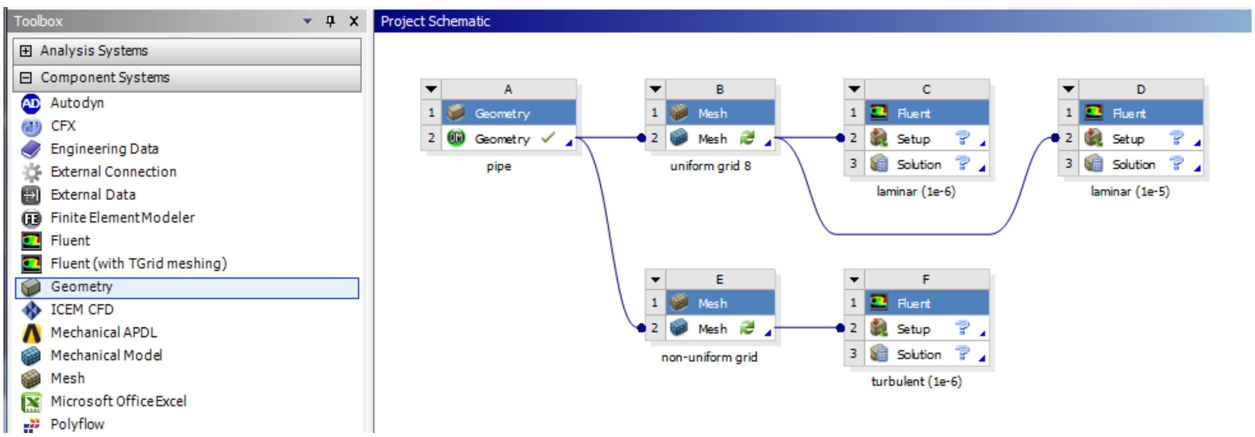
4.8. **Concept** > **Surface From Sketches** and select the sketch and hit **Apply**.



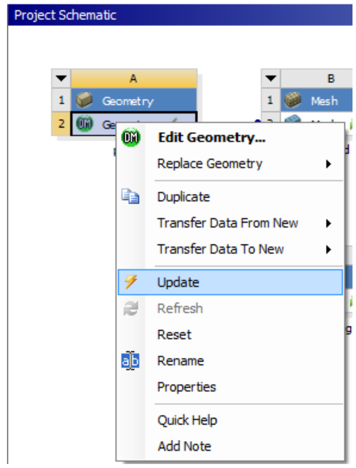
4.9. Click **Generate**. This will create a surface.



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

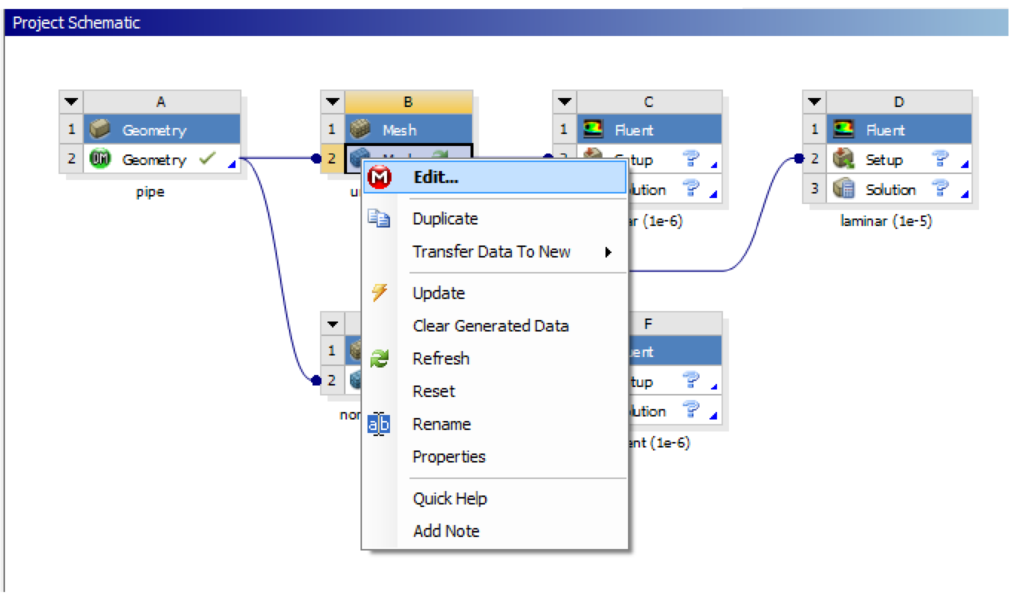


4.11. If you see the lightning sign next to **Geometry** in the workbench then right click on **Geometry** and click **Update** as shown below. If you don't see the check mark after you update then you may have made a mistake when you created the geometry.

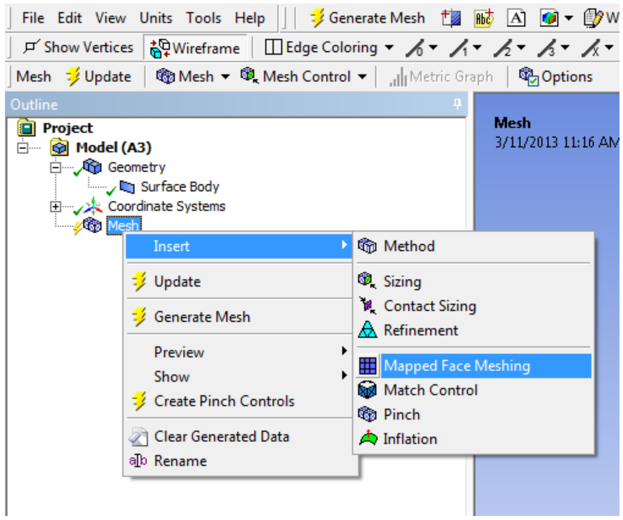


5. Mesh Generation

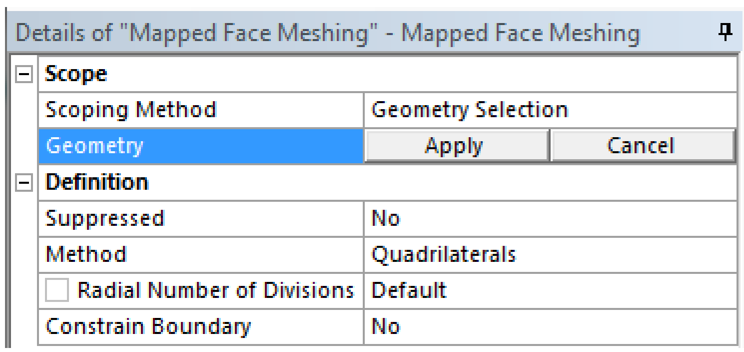
5.1. Right click **Mesh** and select **Edit**.

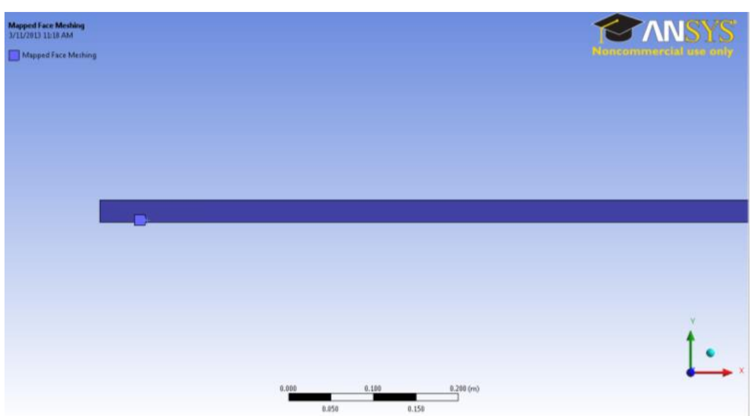


5.2. Right click on **Mesh** then select **Insert** > **Mapped Face Meshing**.

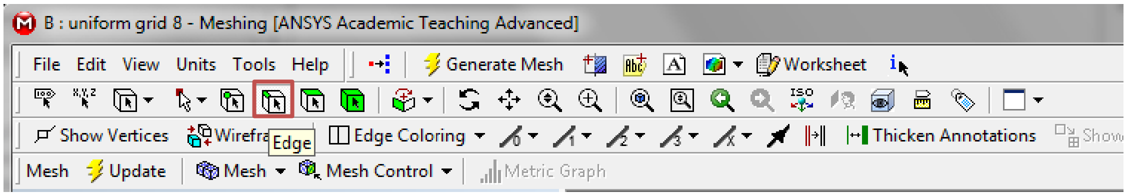


5.3. Select your geometry and click **Apply**.

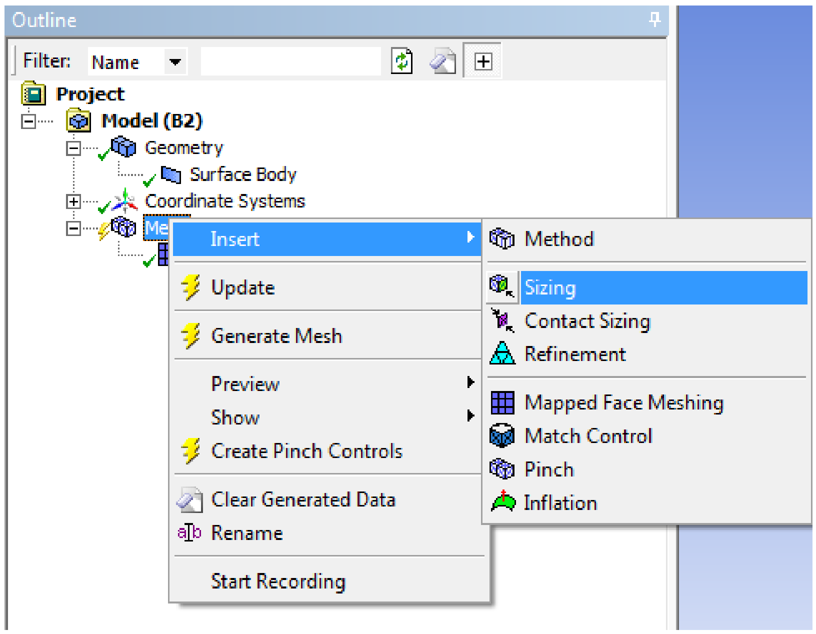




5.4. Click on the **Edge Button**. This will allow you to select edges of your geometry.



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



5.6. Hold Ctrl and select the top and bottom edge then click **Apply**. Specify details of sizing as per below.

Laminar

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

Turbulent

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. Select the left and right edge and click **Apply** for uniform grid flow and change sizing parameters as per below. Change the sizing parameters separately for non-uniform grid as per below. Make sure to select edges individually when changing sizing parameters for non-uniform grid.

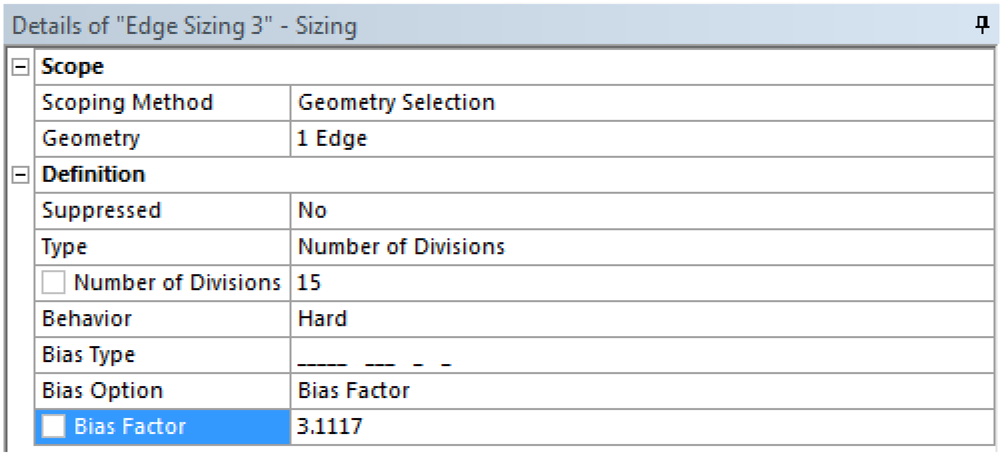
Uniform Grid 8

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

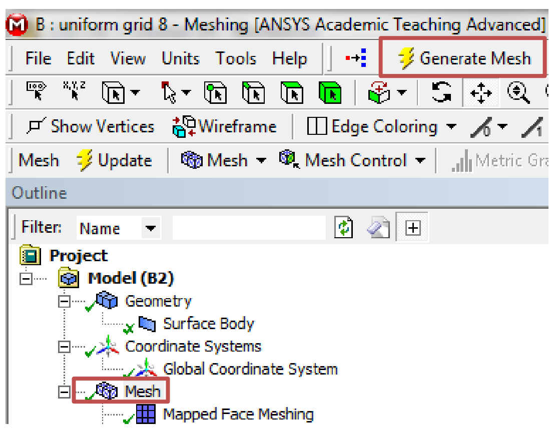
Non-uniform Grid Left Edge

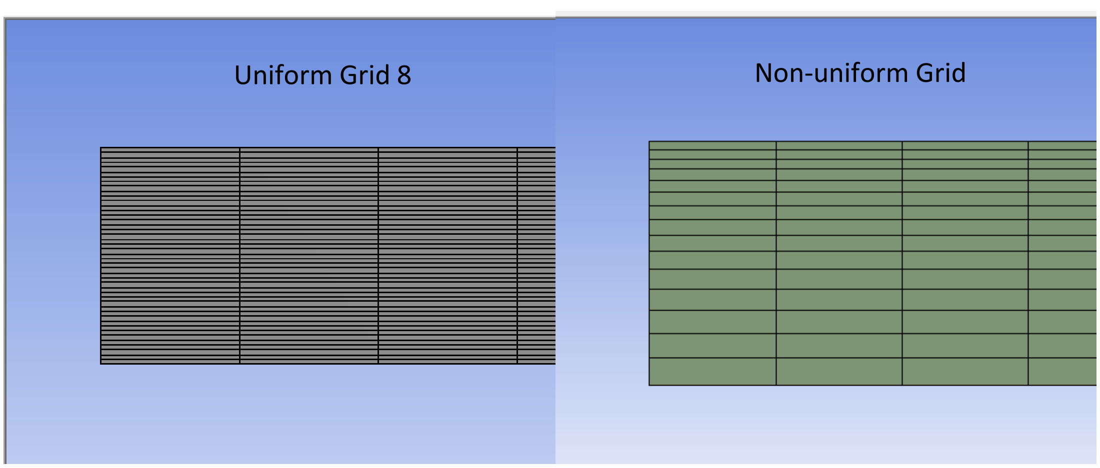
Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	15
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	3.1117

Non-uniform Grid Right Edge

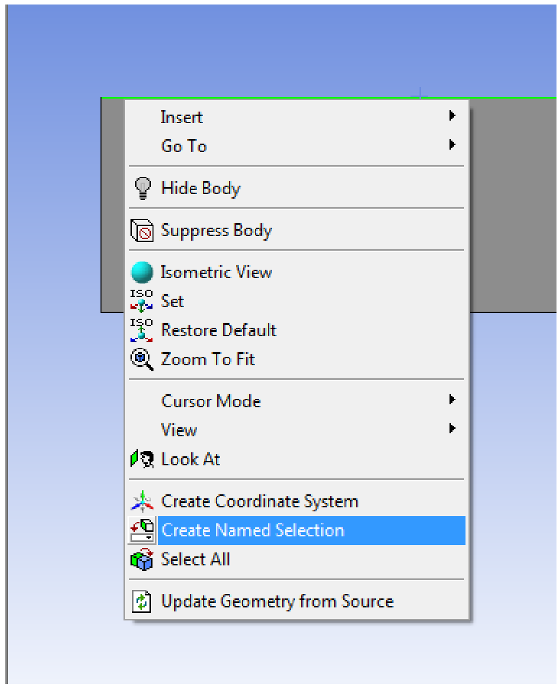


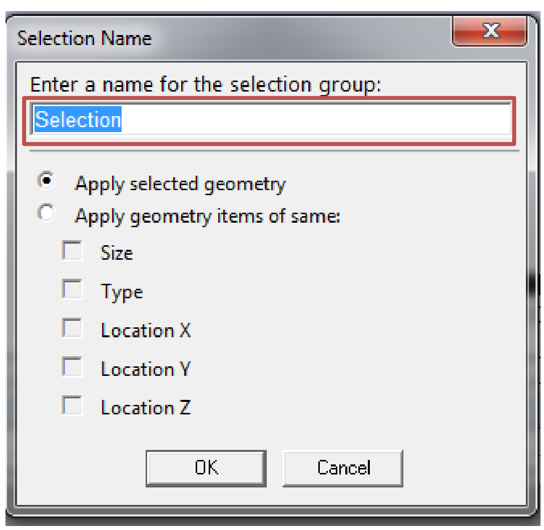
5.8. Click on **Generate Mesh** button and select **Mesh** under **Outline**.



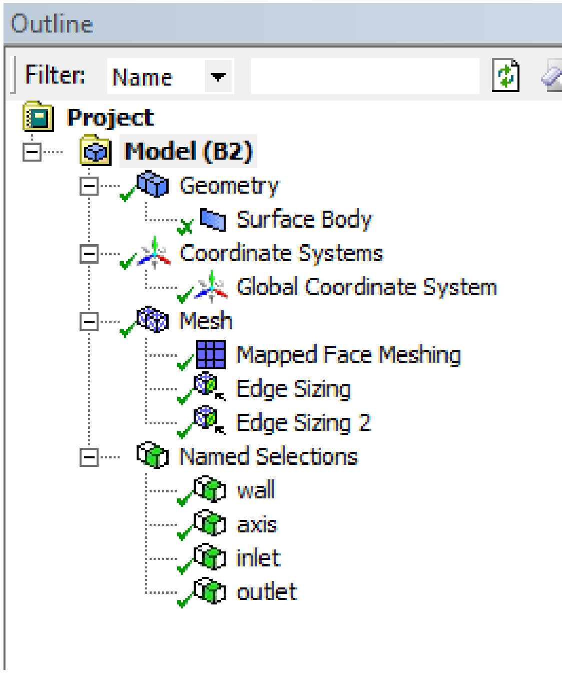


5.9. Change the edge names by right clicking and selecting **Create Named Selection**. Name left, right, bottom and top edges as inlet, outlet, axis and wall respectively. Your outline should look same as the figure below.

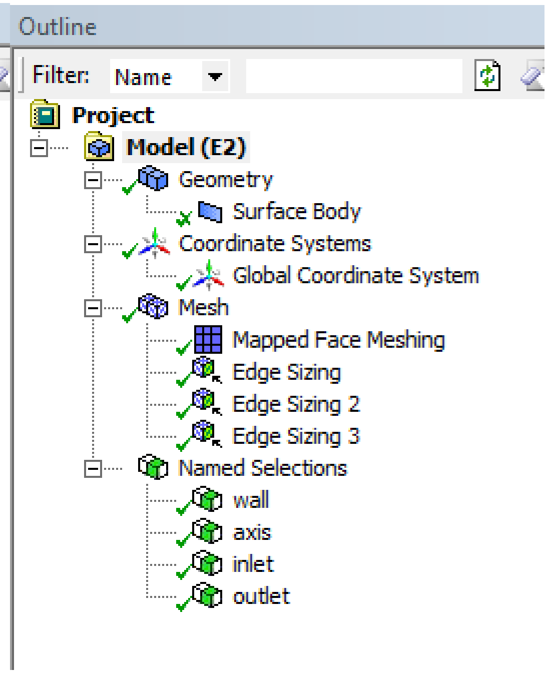




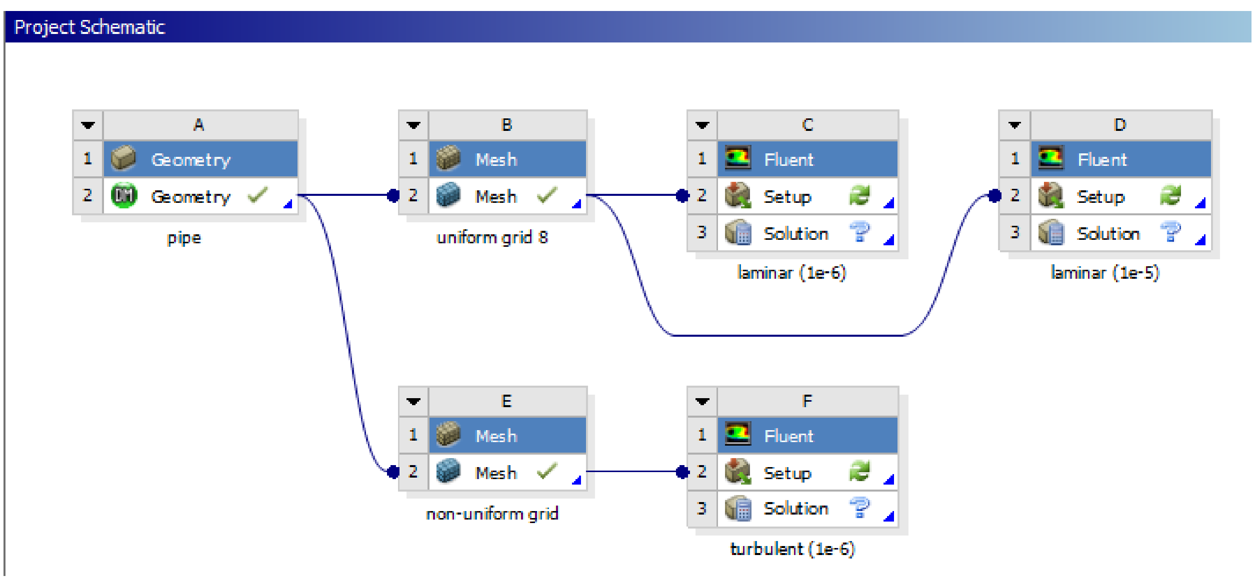
Uniform Grid 8



Non-uniform Grid

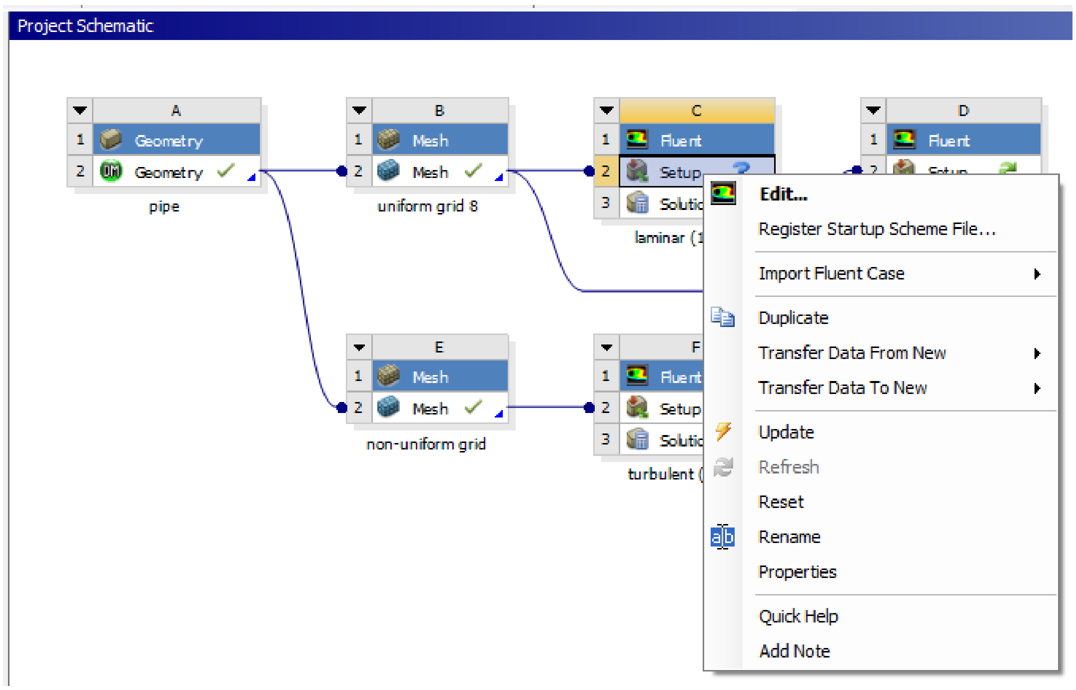


5.10. **File** > **Save Project**. Save the project and close the window. Update **Mesh** on Workbench if necessary.

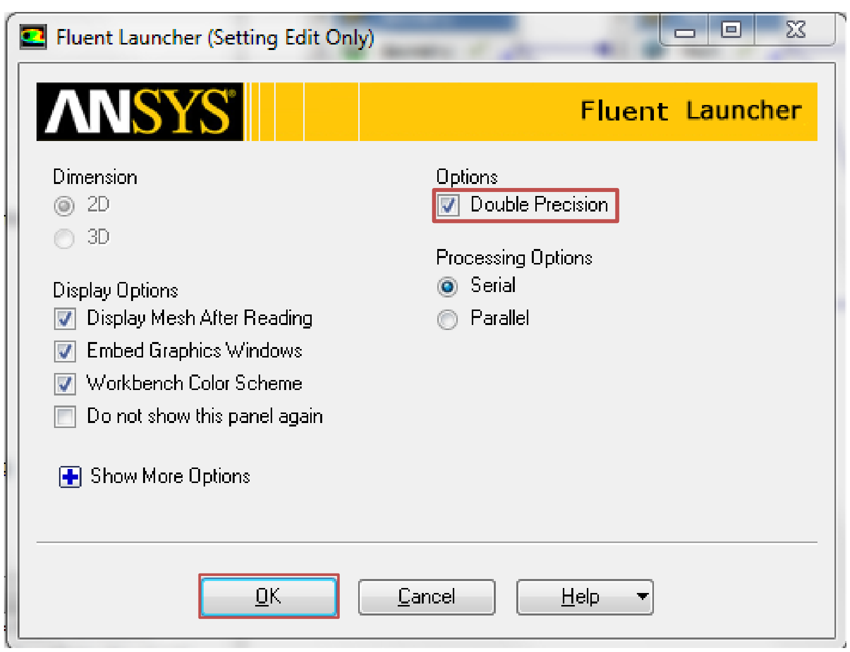


6. Solve

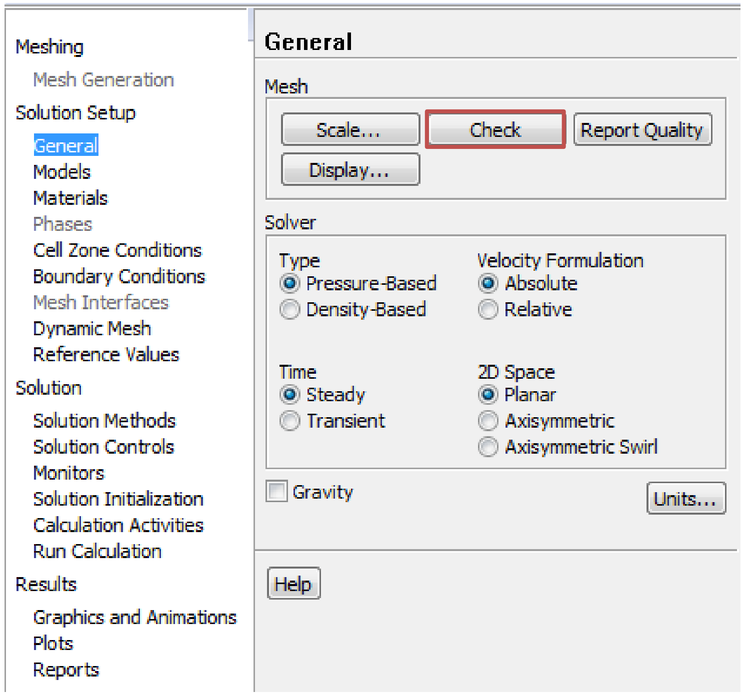
6.1. Right click **Setup** and select **Edit**.



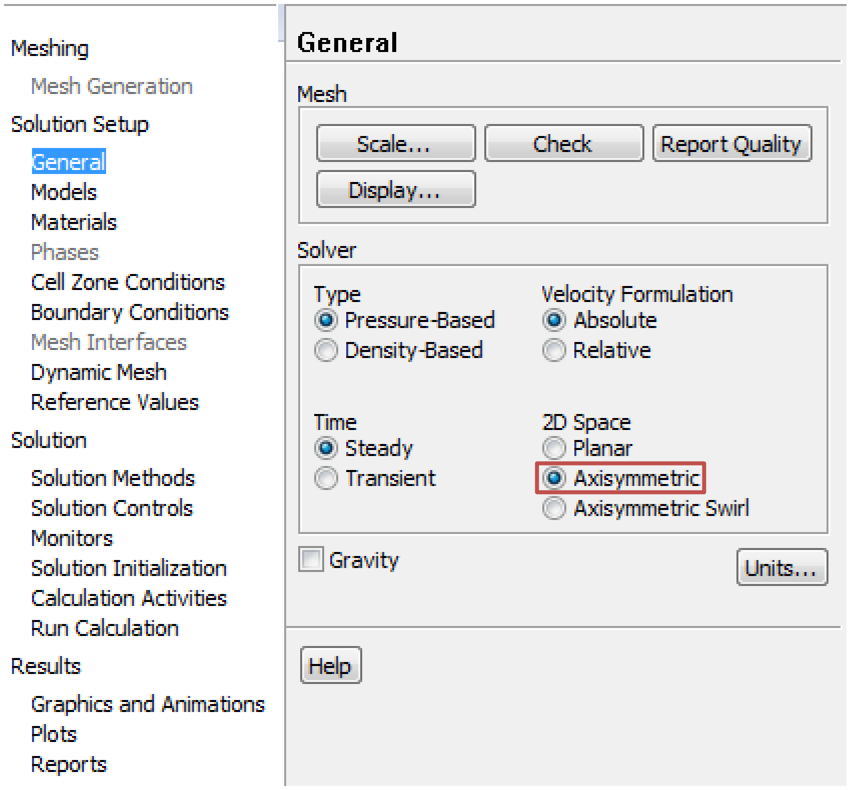
6.2. Under options check **Double Precision** and click **OK**.



6.3. **Solution Setup** > **General** > **Check**. (Note: If you get and error message you may have made a mistake while creating you mesh)

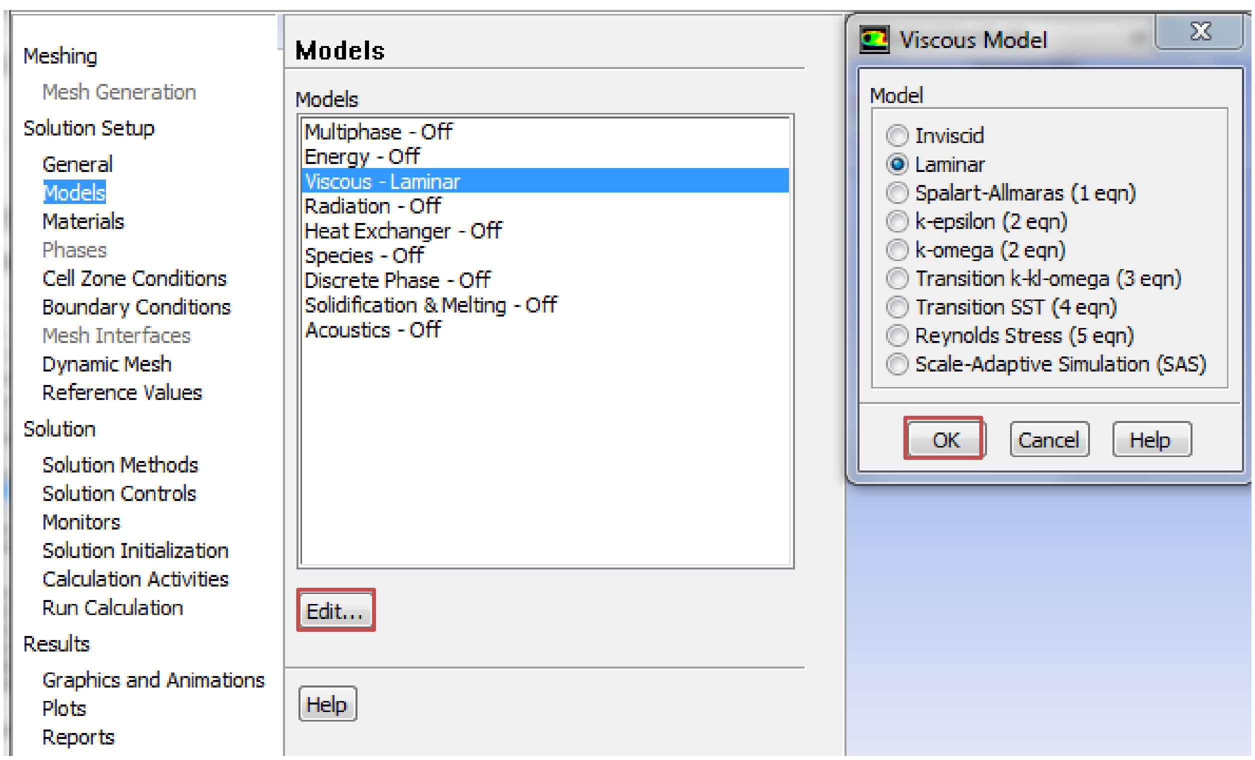


6.4. **Solution Setup** > **General** > **Solver**. Choose options shown below.

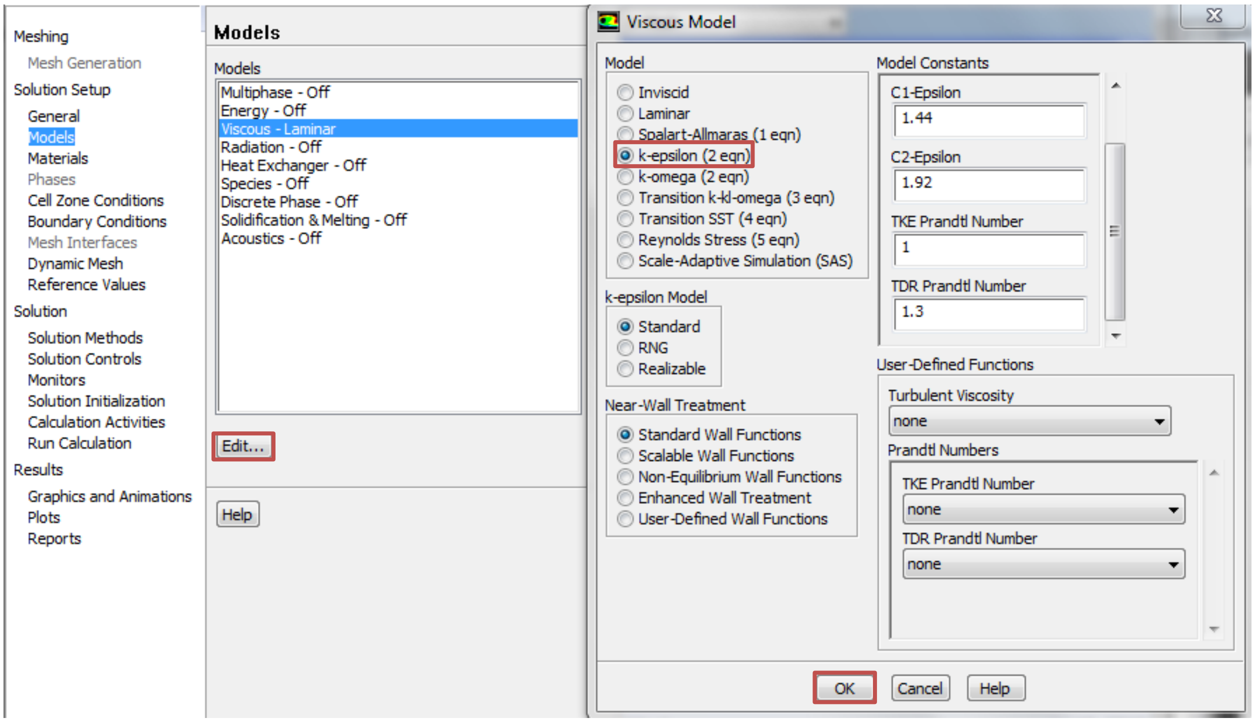


6.5. **Solution Setup** > **Models** > **Edit**. Select parameters as per below.

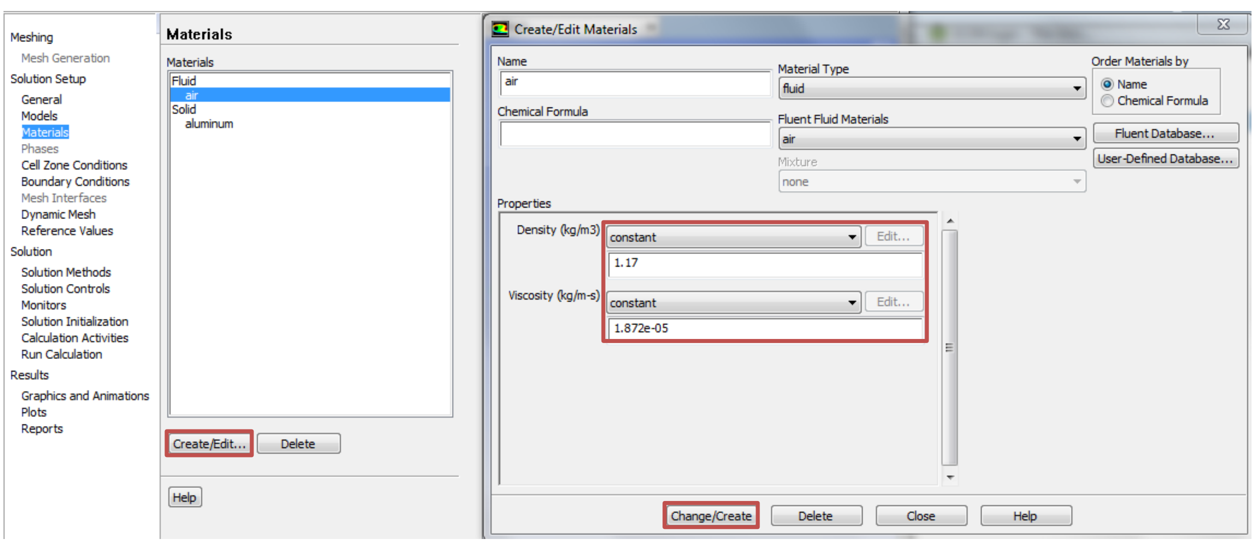
Laminar flow



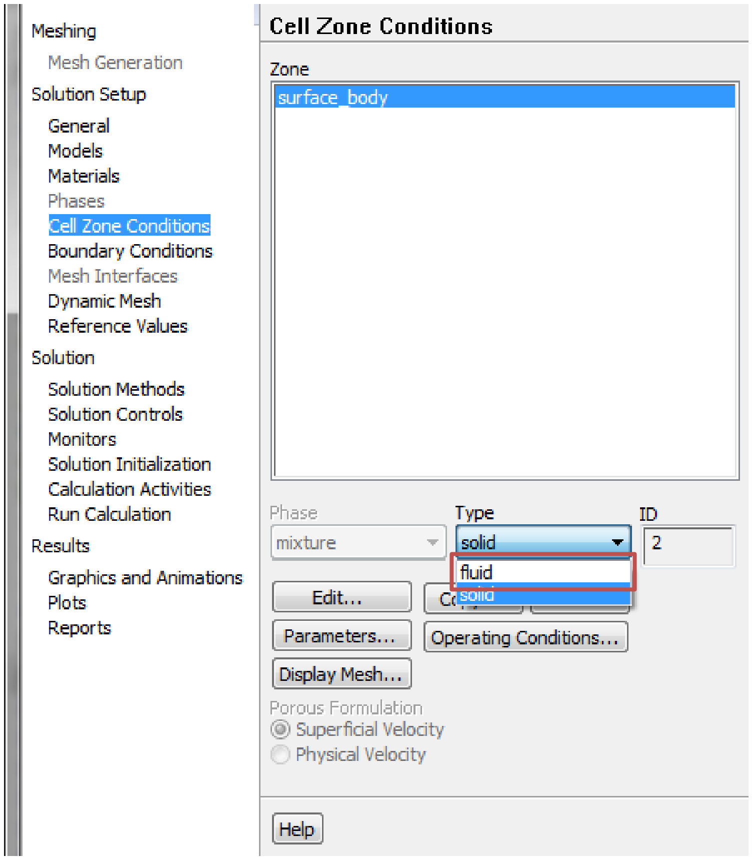
Turbulent flow

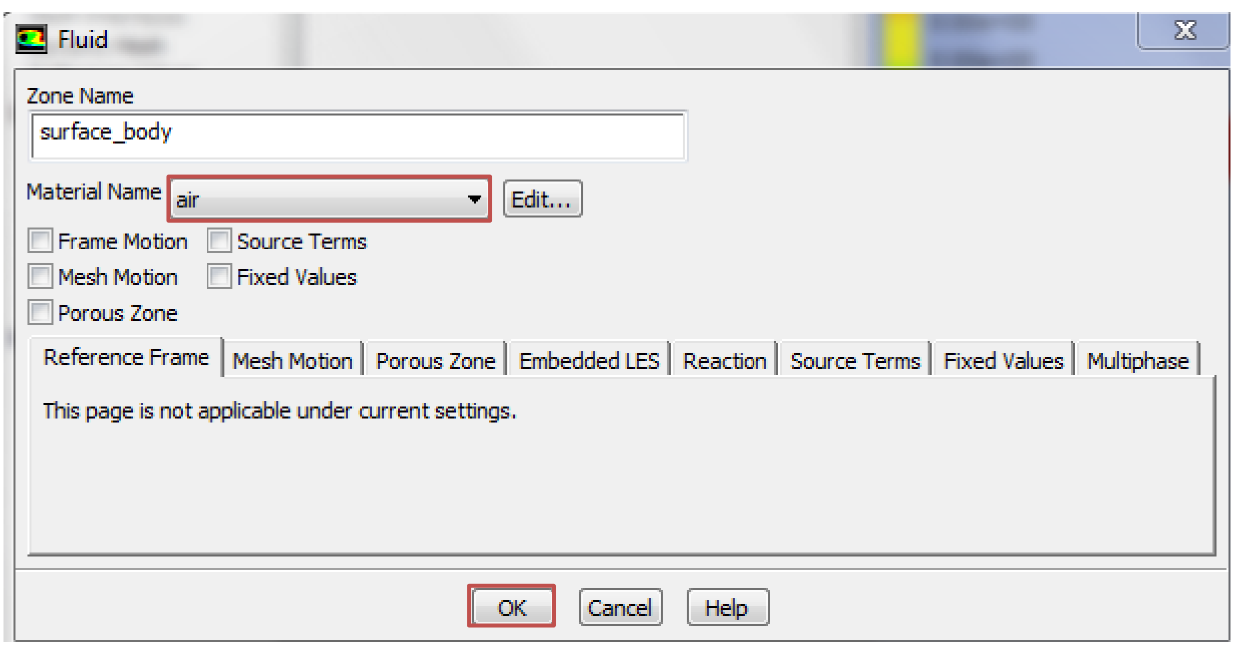


6.6. **Solution Setup** > **Materials** > **air** > **Create/Edit…** Change the **Density** and **Viscosity** as per below and click **Change/Create**. Close the dialog box when finished.



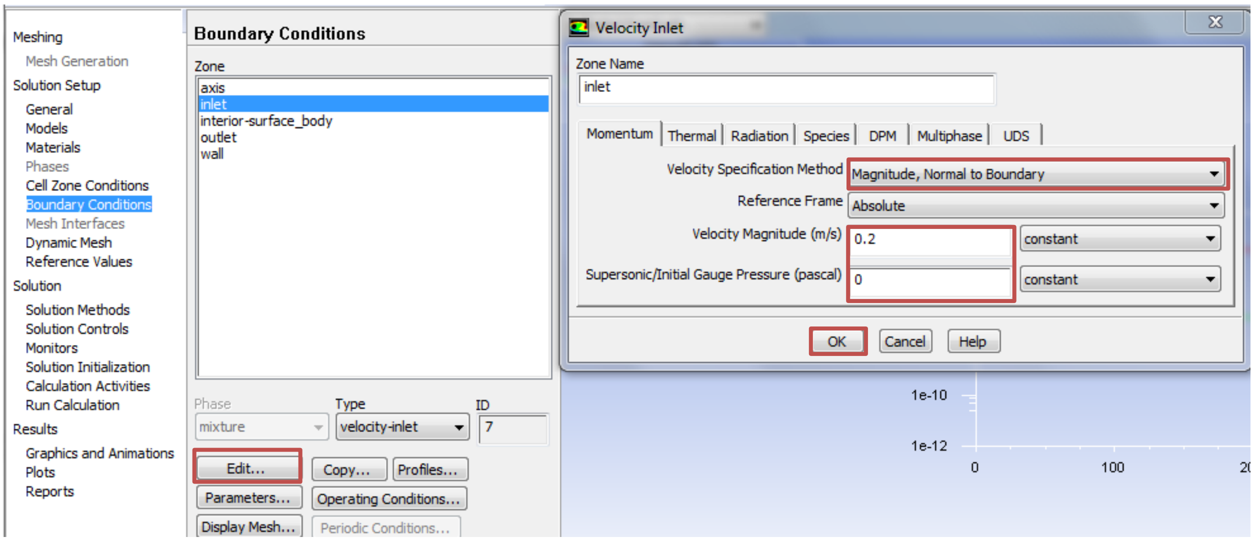
6.7. **Cell Zone Conditions** > **Zone** > **surface_body**. Change type to **fluid** and click **OK**. Select **Material Name** as **air** and click **OK**.



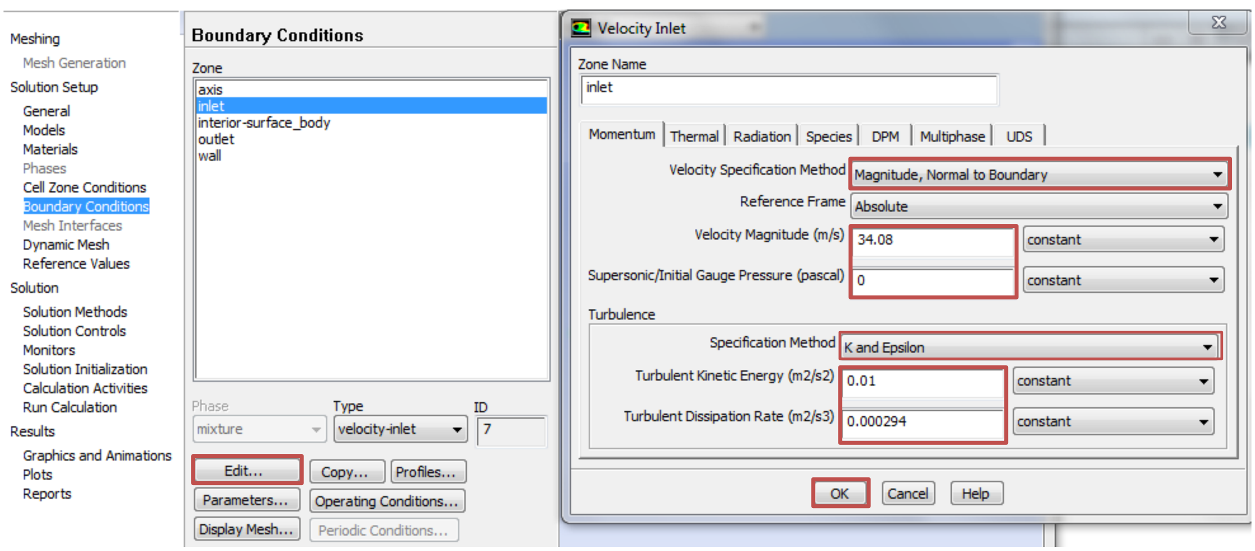


6.8. **Solution Setup** > **Boundary Conditions** > **inlet** > **Edit…** Change parameters as per below and click **OK**. (Note: Change inlet velocity to 0.2 m/s for laminar flow)

Laminar flow

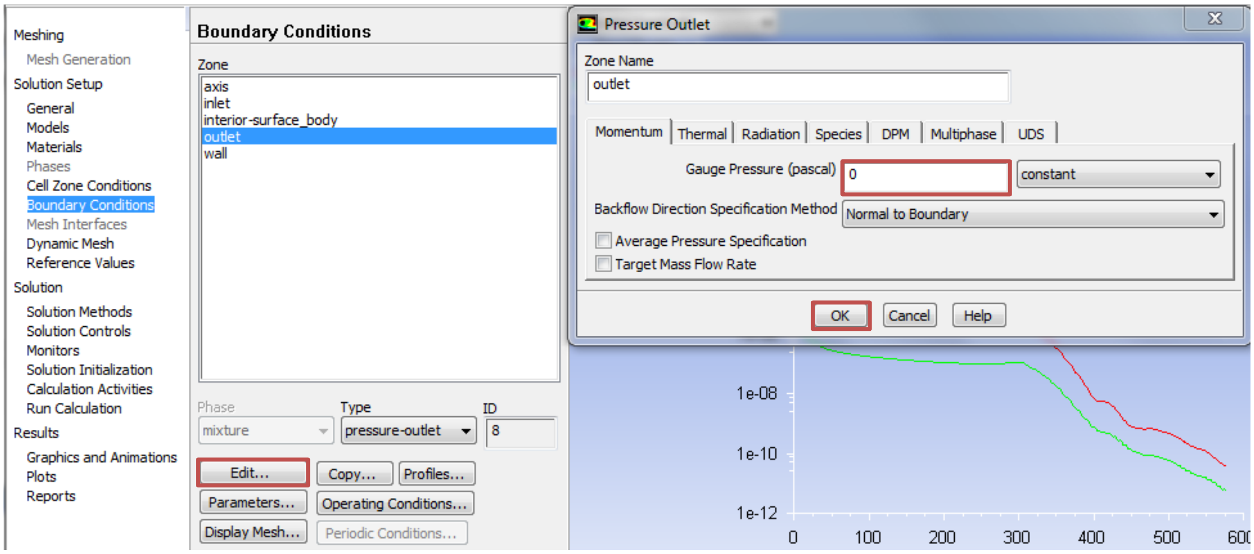


Turbulent flow

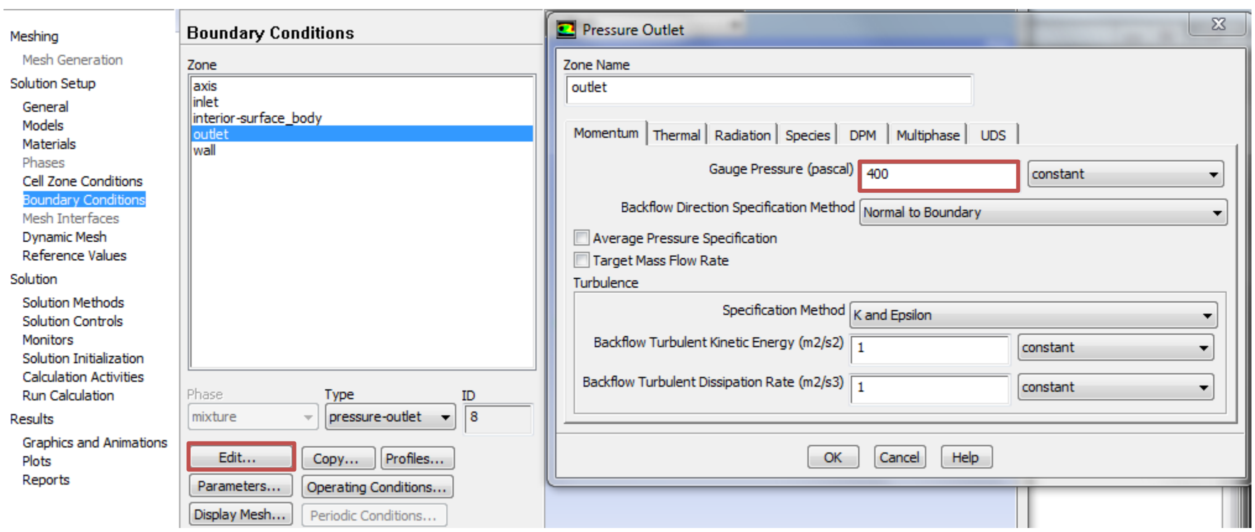


6.9. **Solution Setup** > **Boundary Conditions** > **outlet** > **Edit**. Change parameters as per below and click **OK**. (Note: Outlet pressure is 0 Pa for laminar flow)

Laminar flow

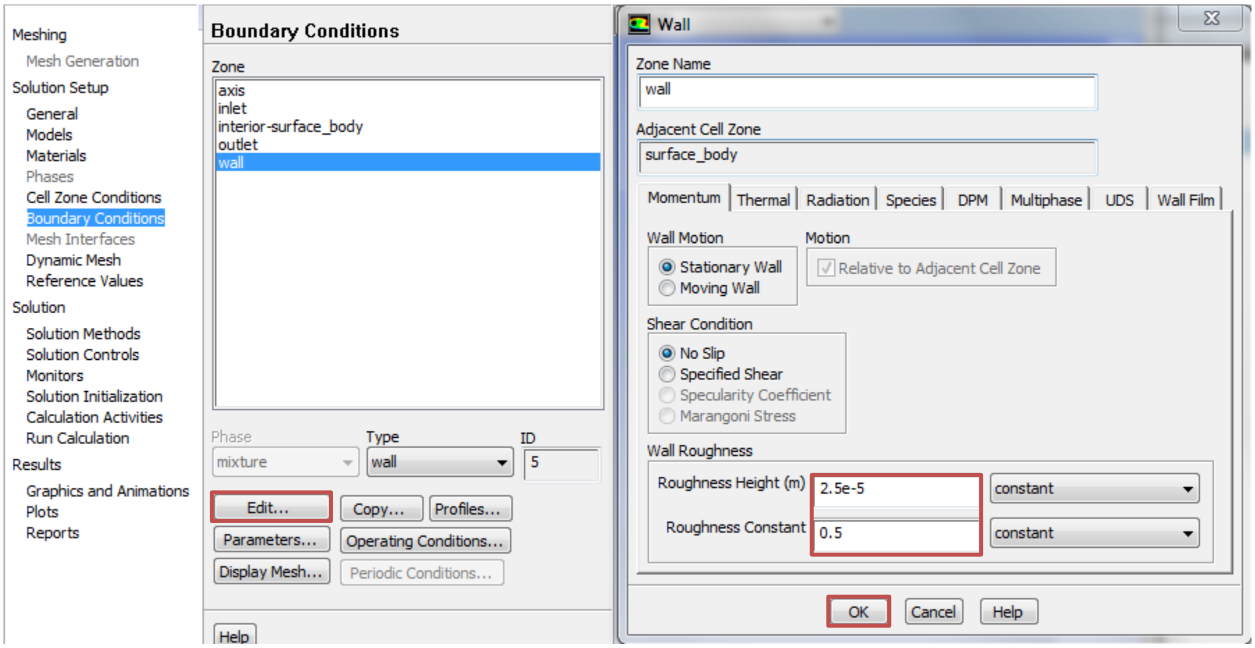


Turbulent flow

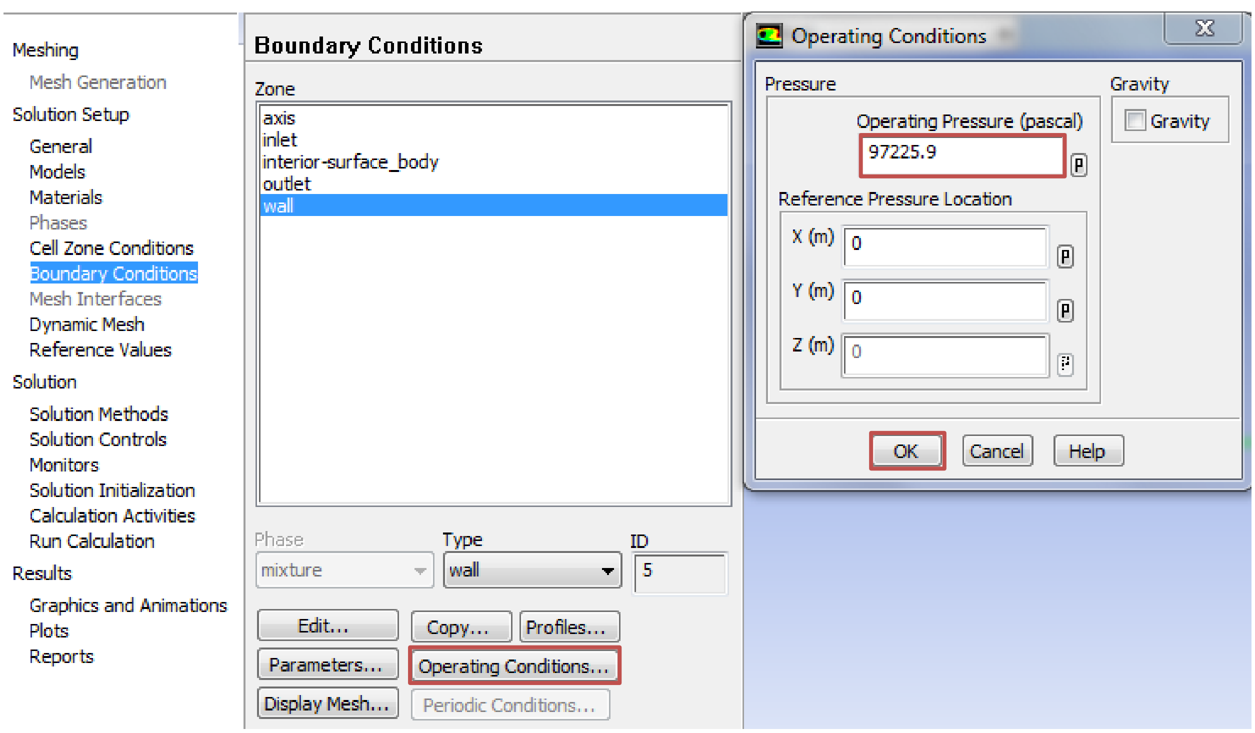


6.10. **Solution Setup** > **Boundary Conditions** > **wall** > **Edit…** Change parameters as per below and click **OK**.

Turbulent flow

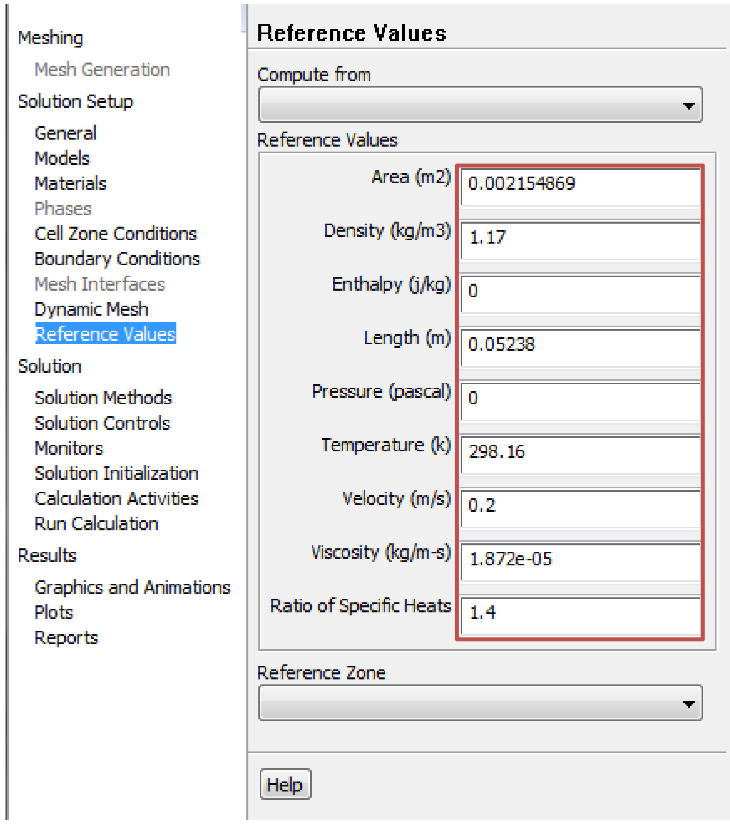


6.11. **Solution Setup** > **Boundary Conditions** > **Operating Condition**. Change parameters as per below and click **OK**.

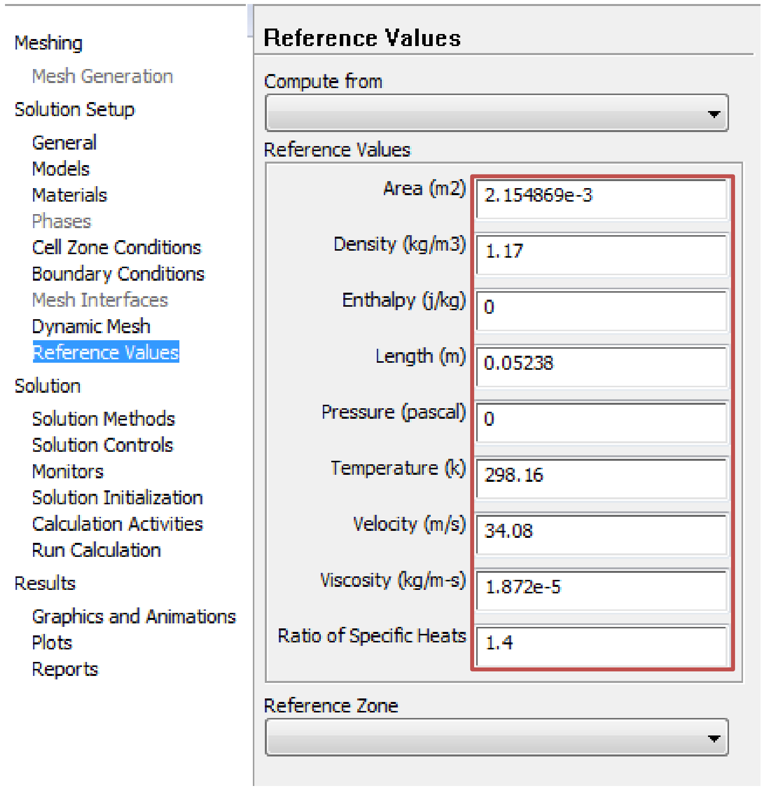


6.12. **Solution Setup** > **Reference Values**. Change parameters as per below.

Laminar flow

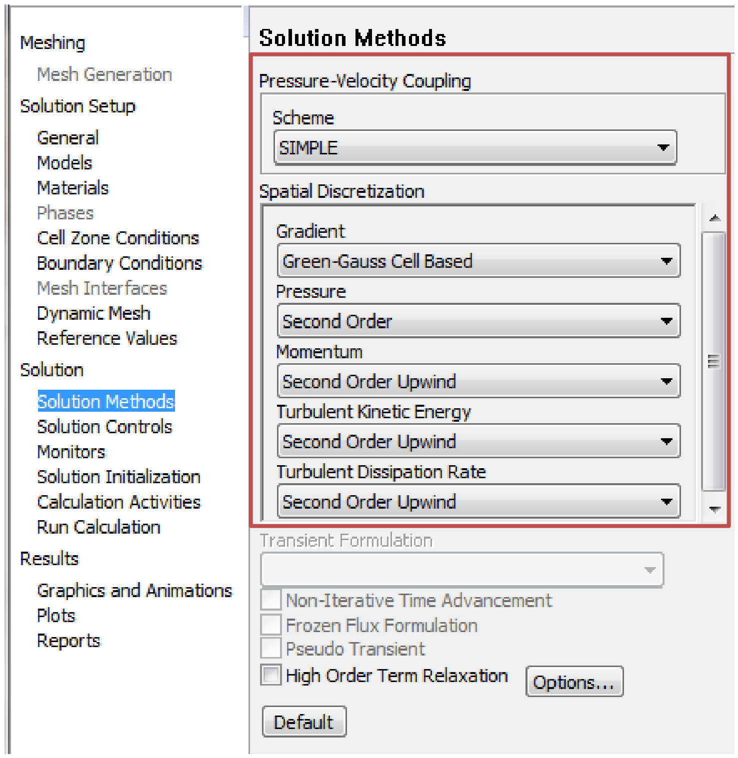


Turbulent flow

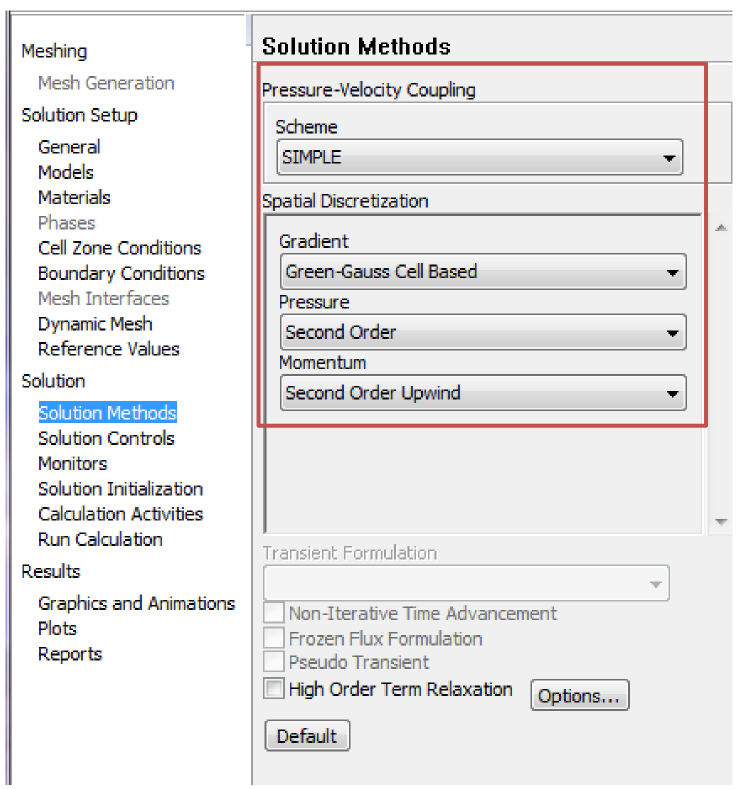


6.13. **Solution** > **Solution Methods**. Change parameters as per below.

Turbulent flow

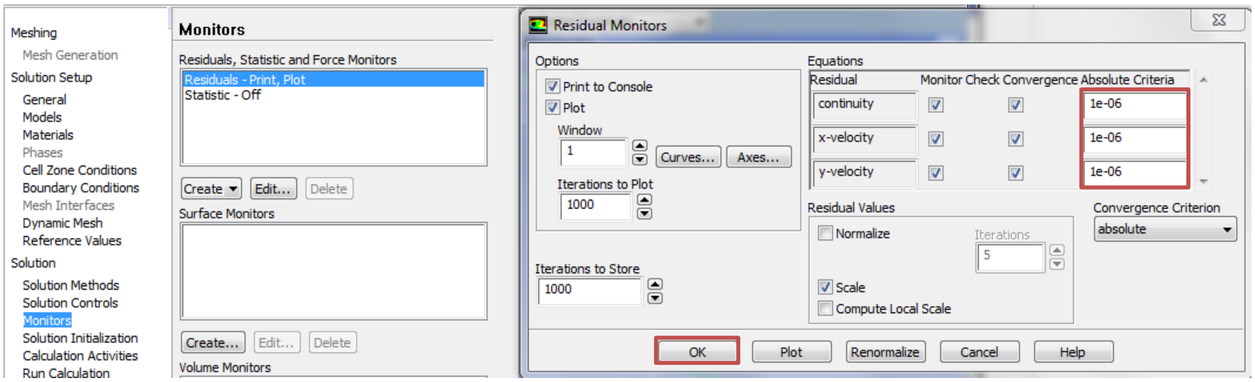


Laminar flow

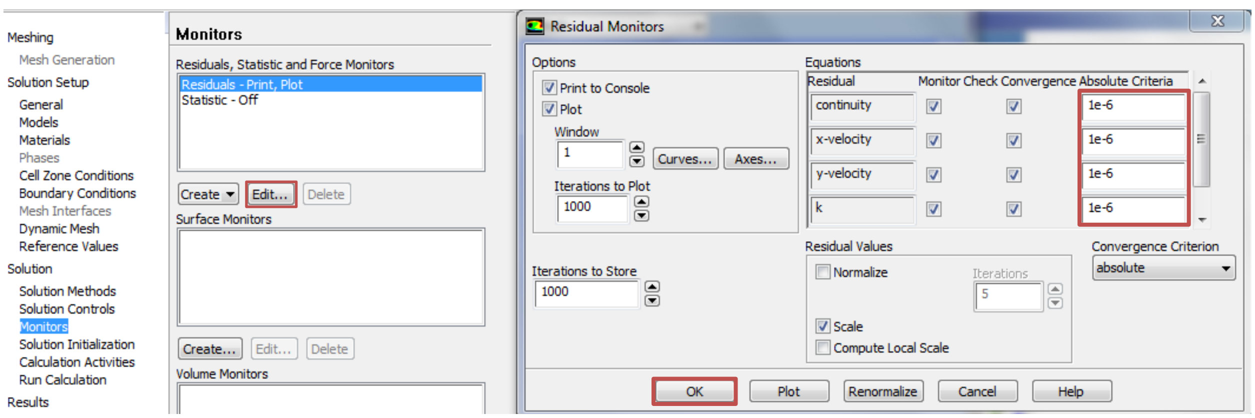


6.14. **Solution** > **Monitors** > **Residuals** > **Edit**. Change convergence criterion to 1e-6 for all five equations as per below and click **OK**. (Note: for iterative error study you will need to use 1e-5)

Laminar flow

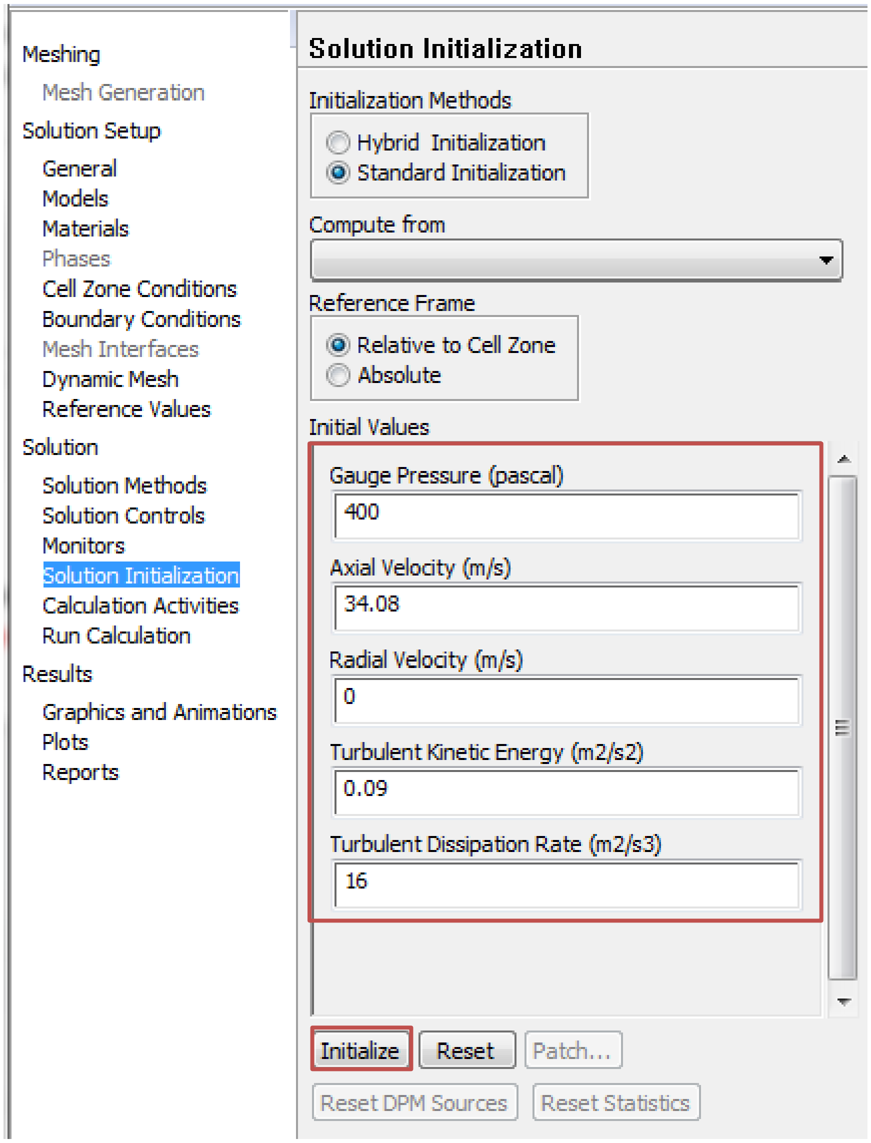


Tubulent flow

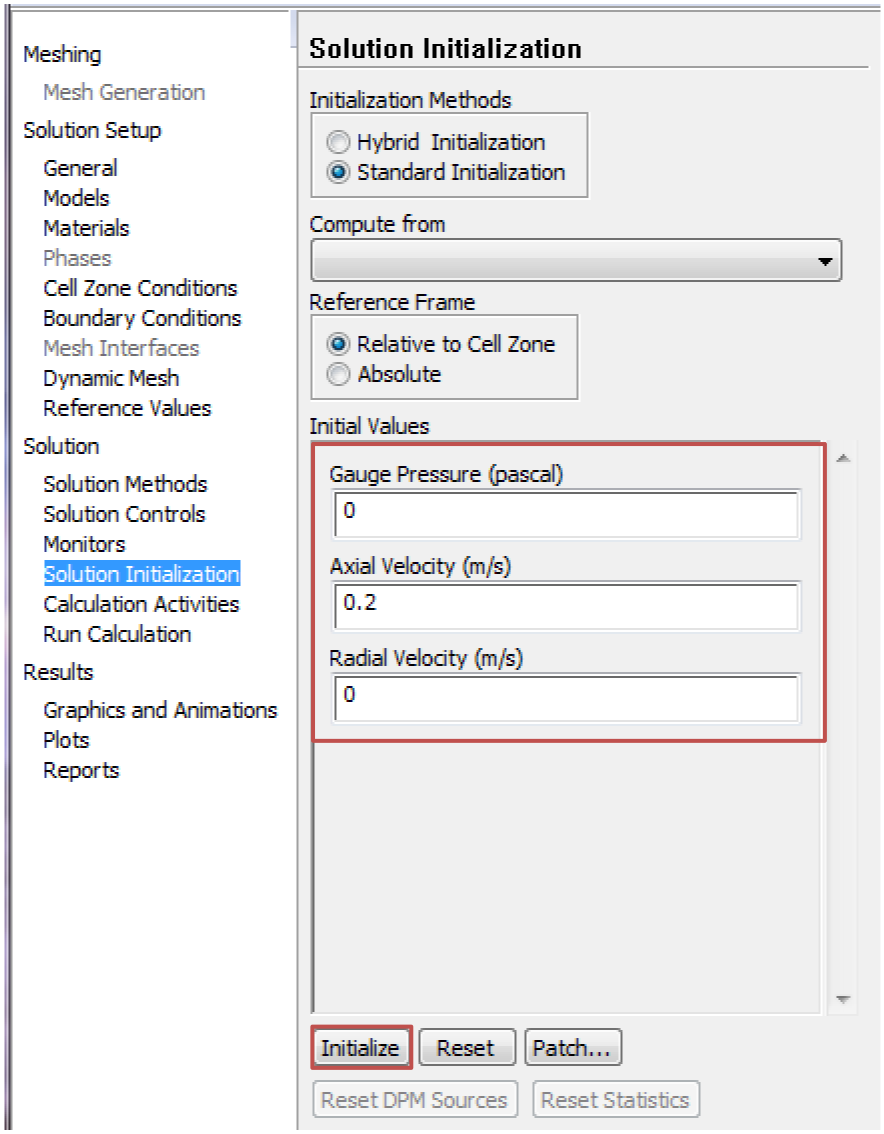


6.15. **Solution** > **Solution Initialization**. Change parameters as per below and click **Initialize**. (Note: use 0 Pa and 0.2 m/s for laminar flow for pressure and velocity respectively)

Turbulent flow



Laminar flow

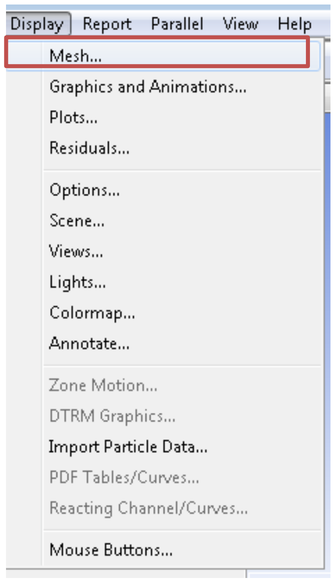


6.16. **Solution** > **Run calculation**. Change number of iterations to 1000 and click **Calculate**.

7. Post Processing

Displaying Mesh

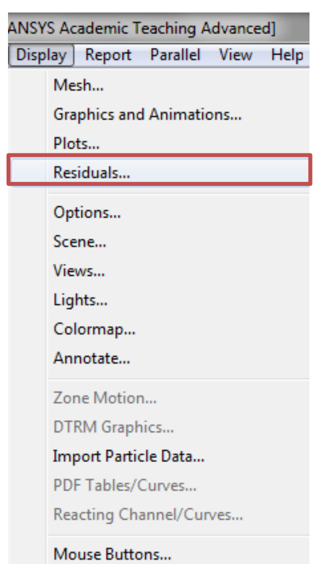
Display > **Mesh**

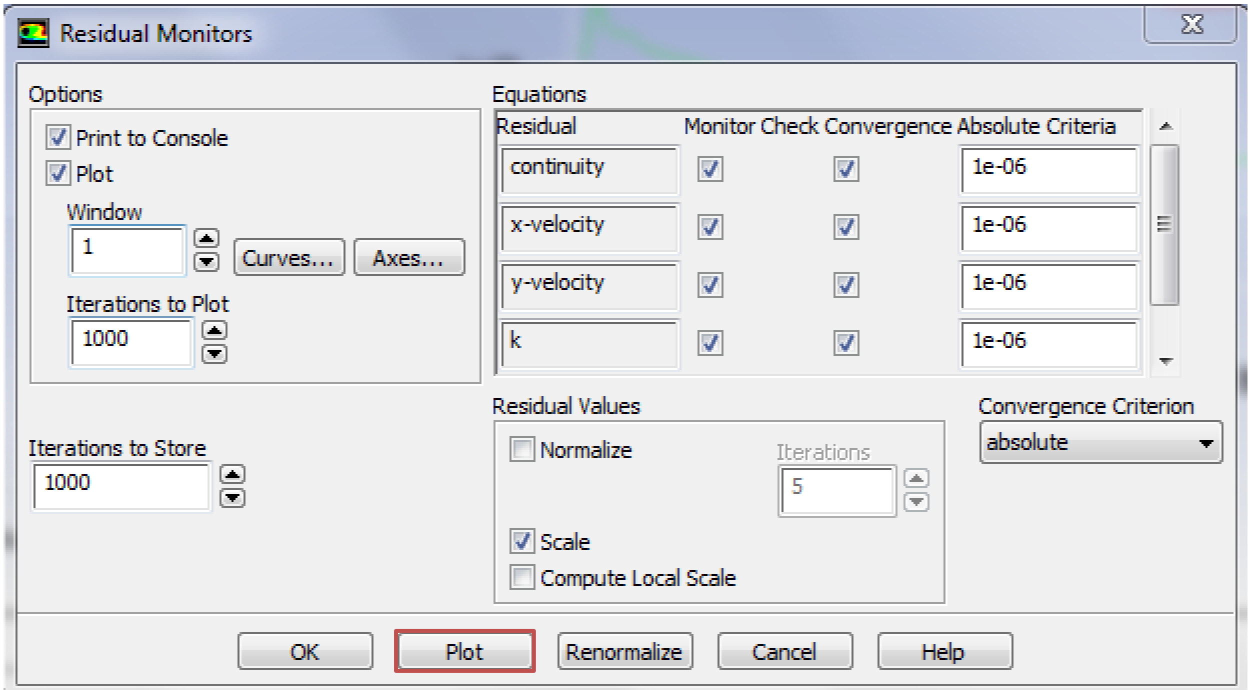


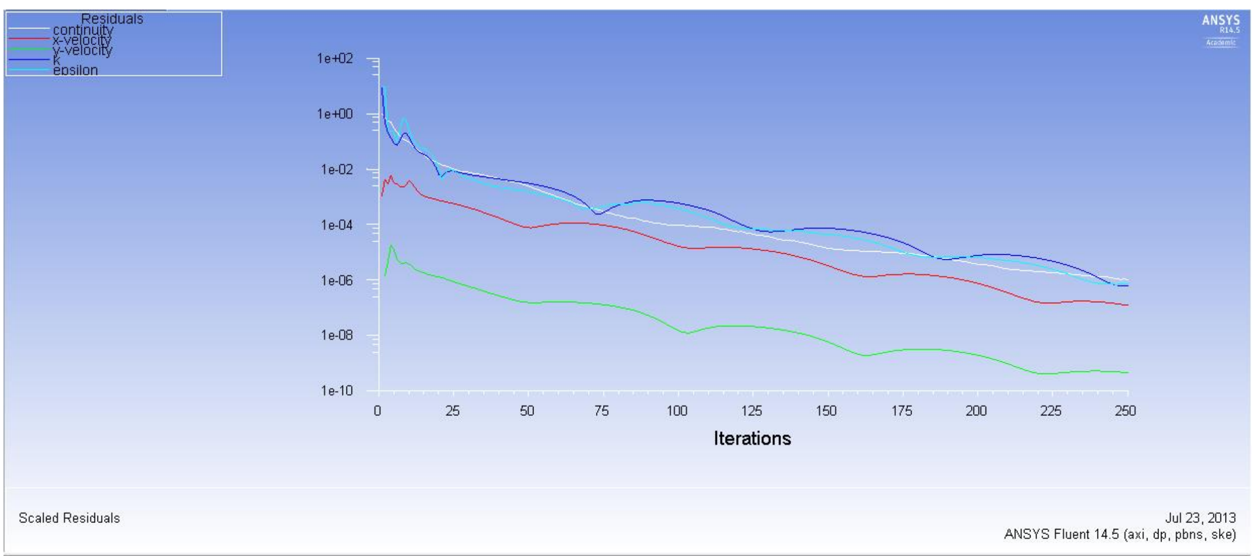
Select all the surface you want to display, lines and points you create can be displayed here as well.

Plotting and Printing Residuals

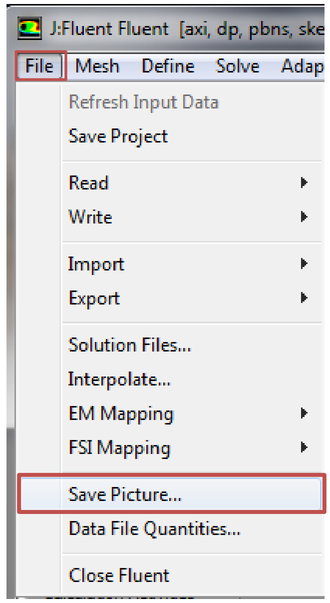
Display > **Residuals**. Click on **Plot** button then click on **OK**.

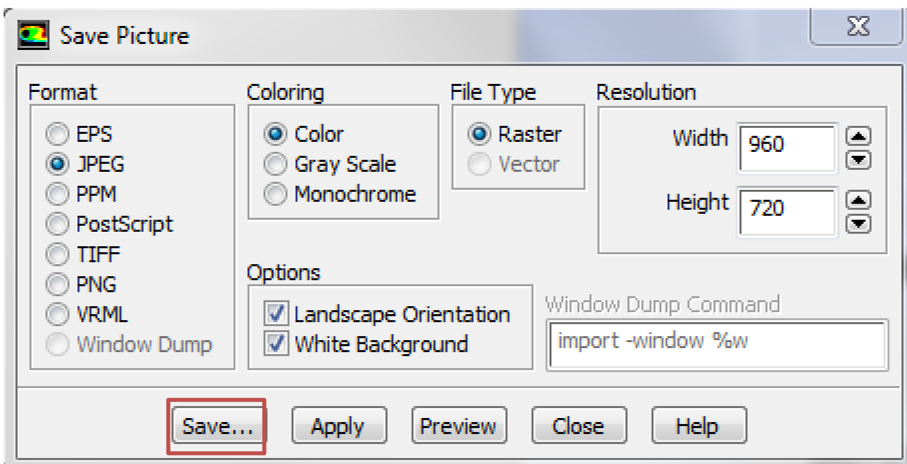






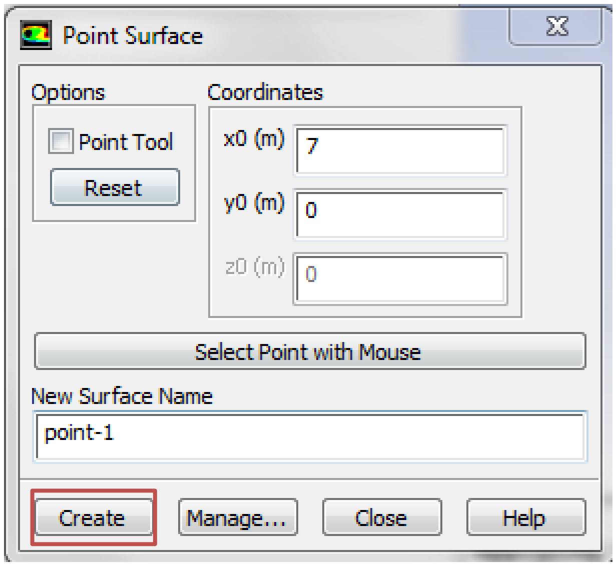
File > **Save Picture**. Using option as per below save the residuals.





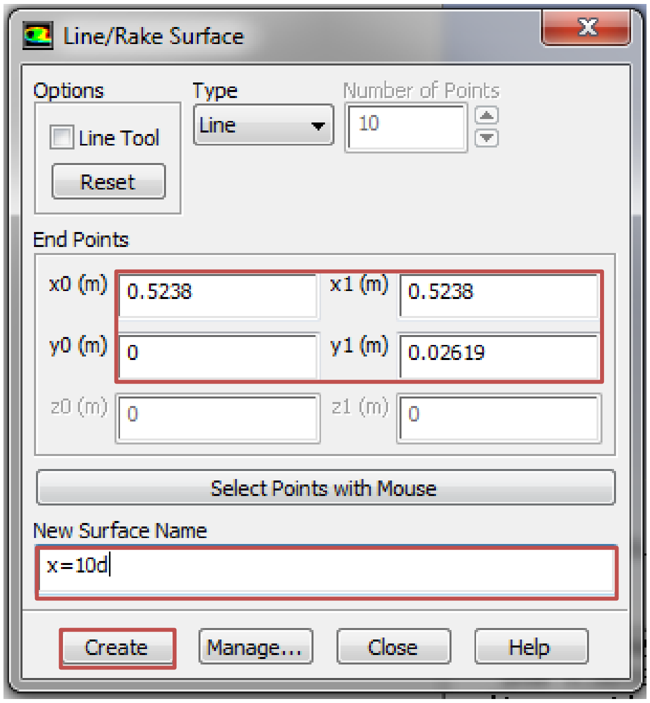
Creating Points and Lines

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



Point Name	x0	y0
point-1	7	0
point-2	7	0.005
point-3	7	0.010
point-4	7	0.015
point-5	7	0.020
point-6	7	0.021
point-7	7	0.022
point-8	7	0.023
point-9	7	0.024
point-10	7	0.025

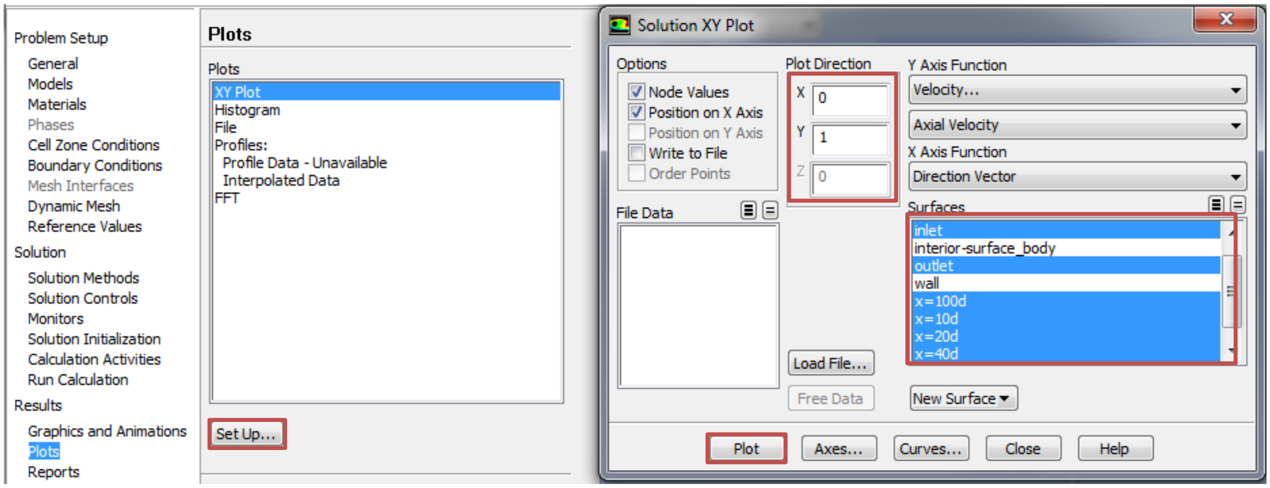
Surface > **Line/Rake**. Change x and y values as per below click **Create**. Repeat this for other lines shown in the table below.



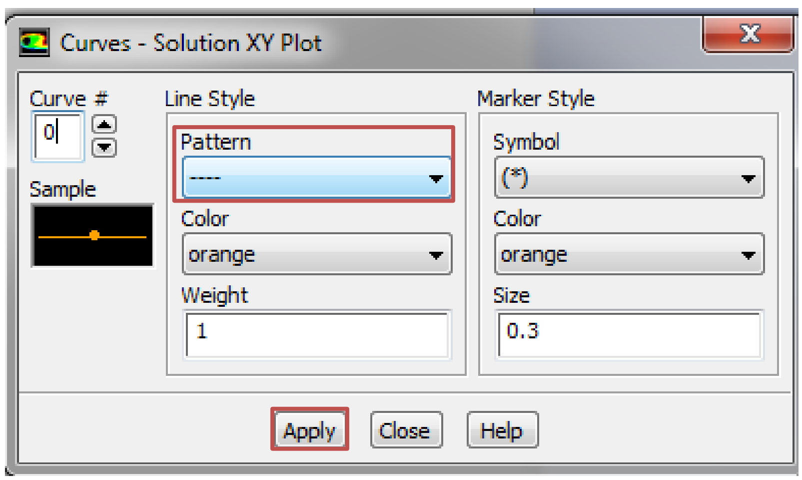
Surface Name	X0	Y0	X1	Y1
x=10d	0.5238	0	0.5238	0.02619
x=20d	1.0476	0	1.0476	0.02619
x=40d	2.0952	0	2.0952	0.02619
x=60d	3.1428	0	3.1428	0.02619
x=100d	5.238	0	5.238	0.02619

Plotting Results

Results > **Plots** > **XY Plot** > **Setup**. Select **inlet**, **outlet**, and the lines you created and change setting as per below then click **Plot**.

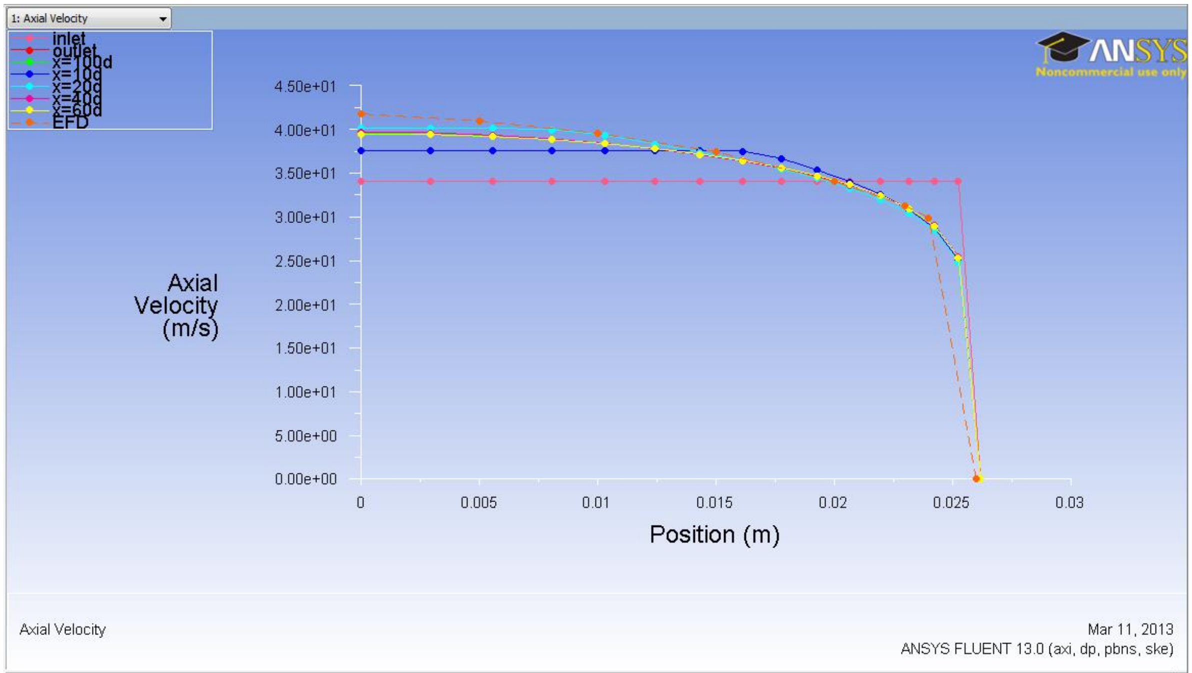


XY Plot > **Setup** > **Curves**. For Curve # 0 select the **Pattern** as per below and click **Apply**. Repeat this for all the curves 0 through 7.

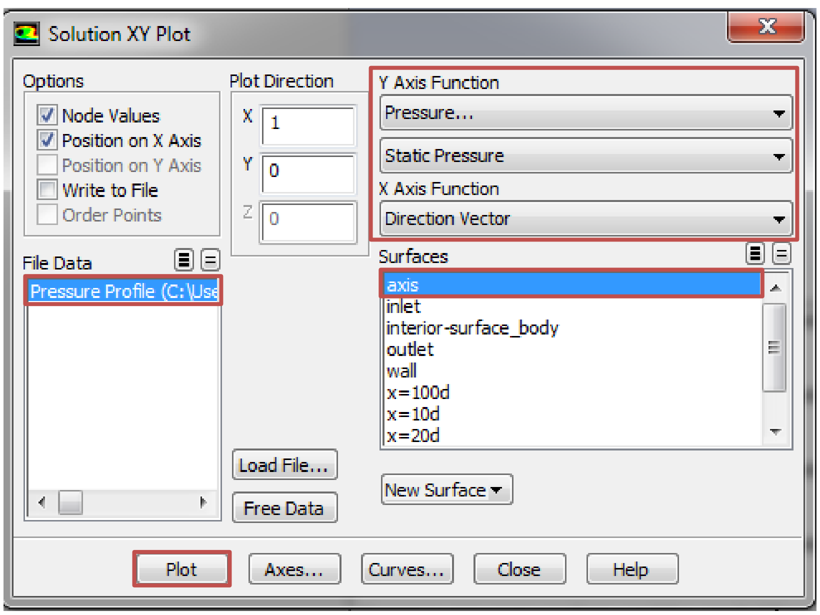


Download the data for the Simulation from the class website (http://www.engineering.uiowa.edu/~me_160/).

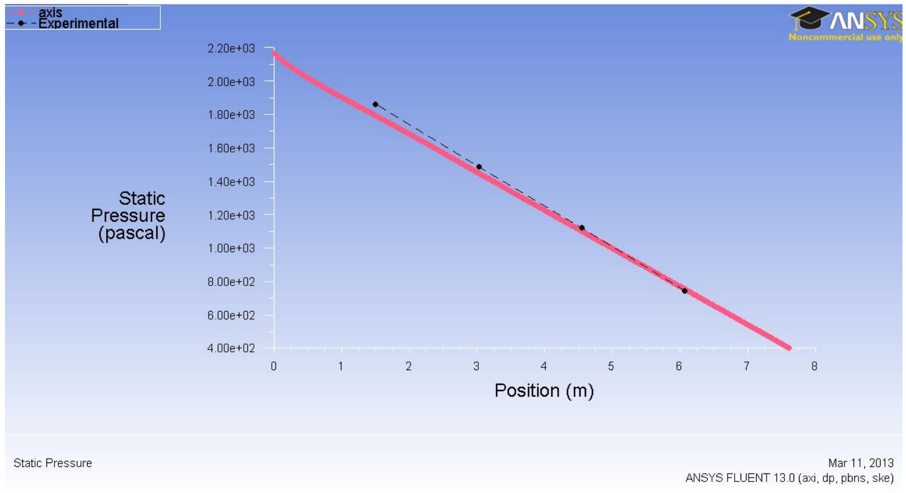
XY Plot > **Setup** > **Load File**. Select axialvelocityEFD-turbulent-pipe.xy and click **OK**.



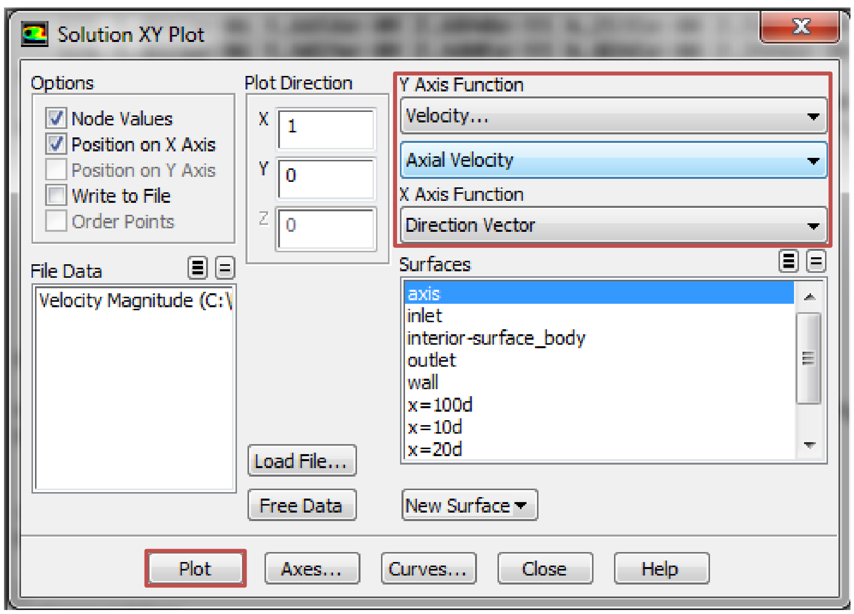
Results > **Plots** > **XY Plot** > **Setup**. Change Y function to **Pressure…** and select **axis** then click **Plot**.

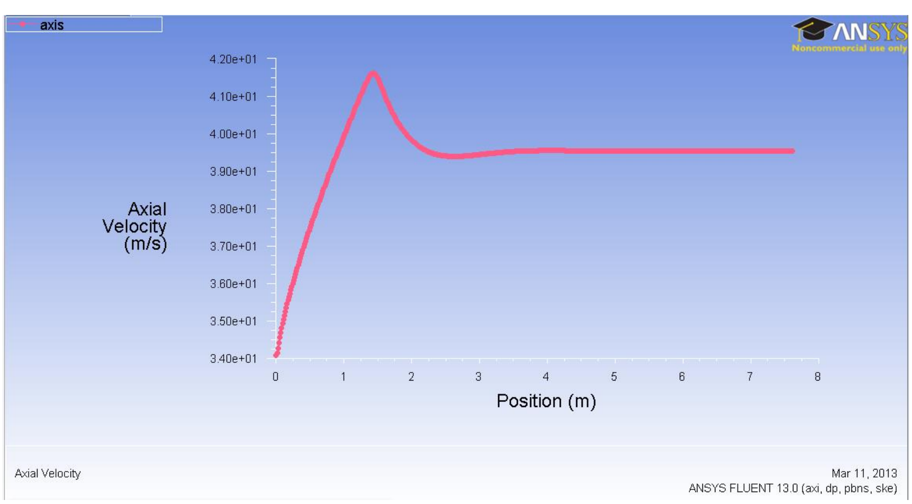


Load experimental data for the centerline pressure.



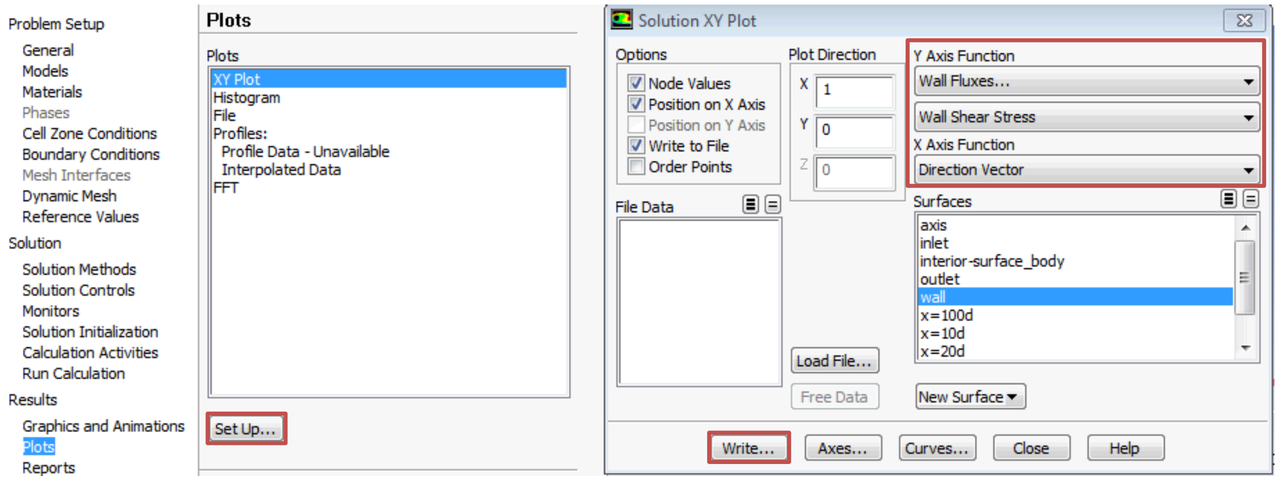
Select **axis** and change **Plot Direction** as per below. Then plot the figure.





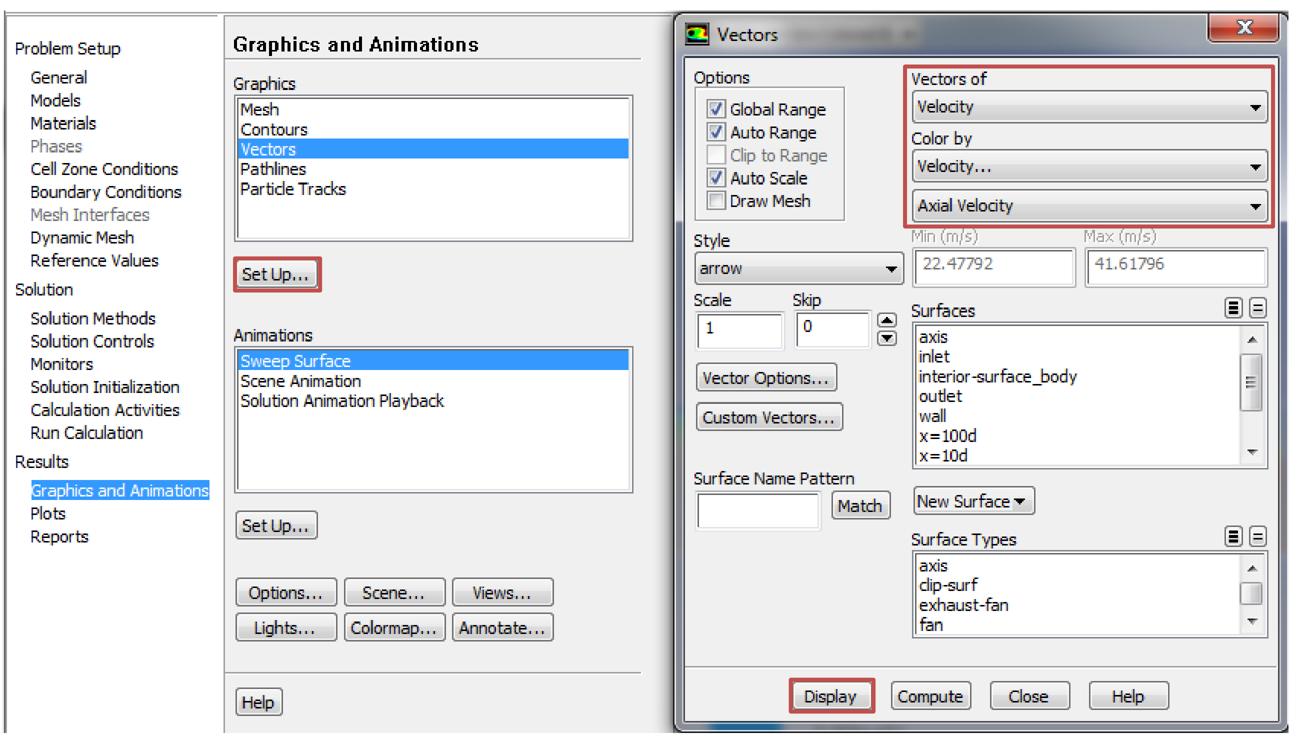
Exporting Data

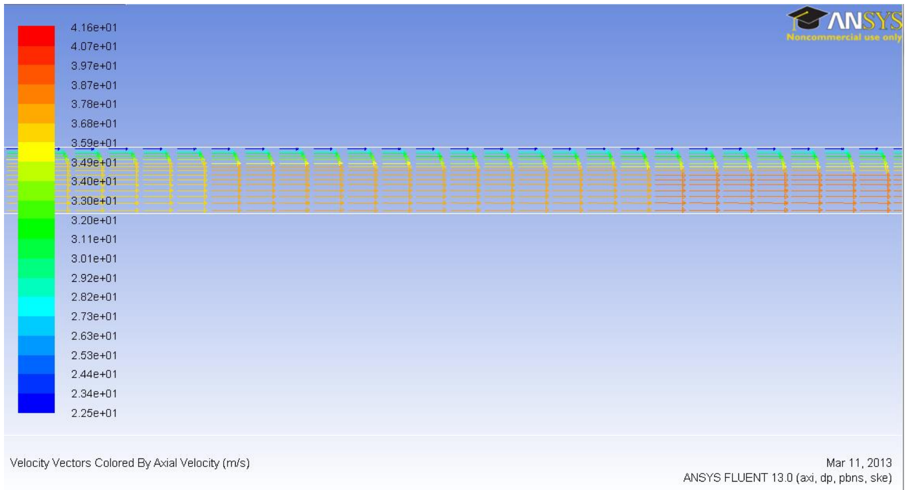
Select **Plots** > **XY Plot**. Then change parameter as per below and click **Write**. This will export the shear stress along the wall of the pipe. You will need this data to compute the shear stress coefficient at the developed region.



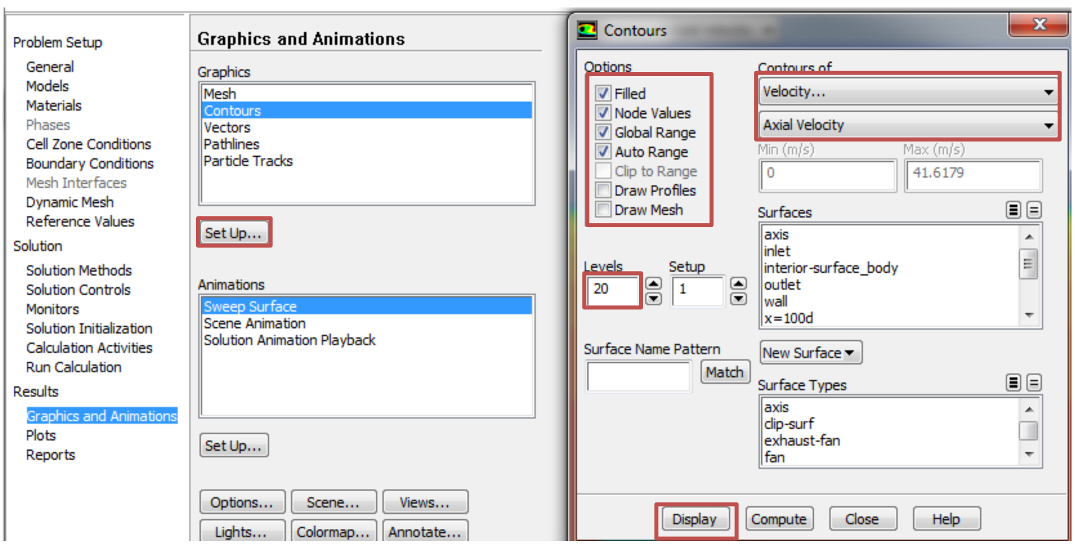
Plotting Vectors and Contours

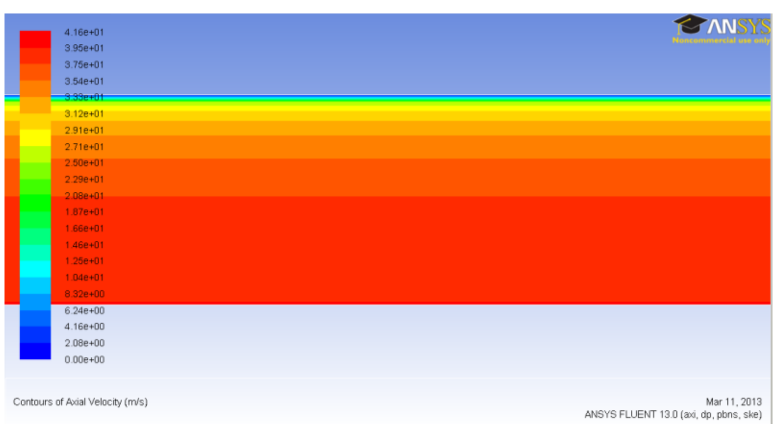
Results > **Graphics and Animations** > **Vectors** > **Set Up…** Change the vector parameters as per below and click **Display**.





Results > **Graphics and Animations** > **Contours** > **Set Up…** Change the vector parameters as per below and click **Display**.



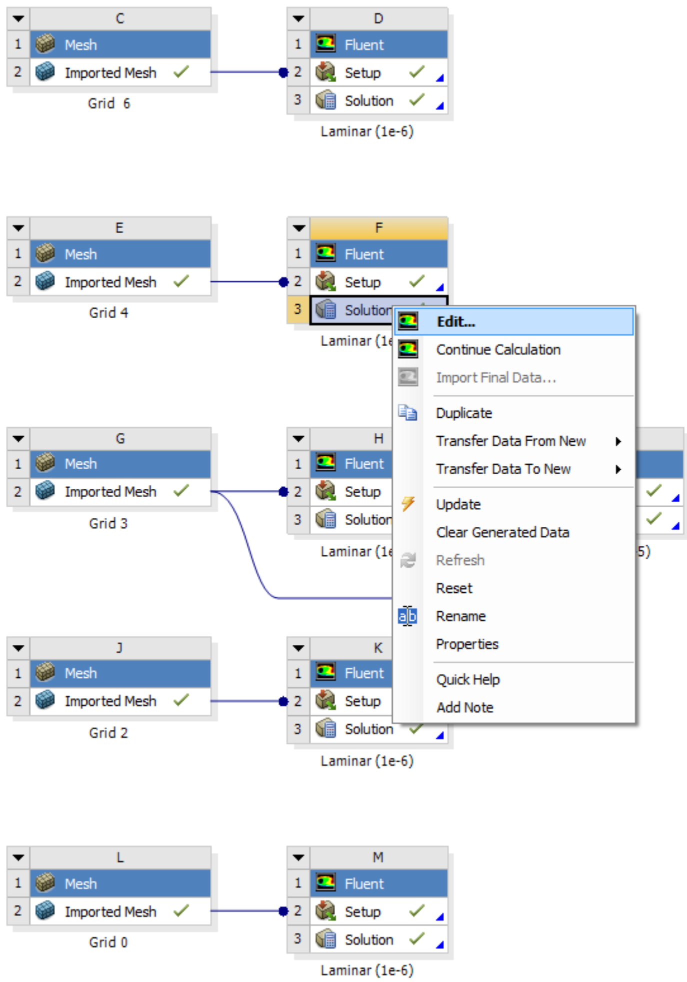


Close window and save workbench file.

V&V Instructions

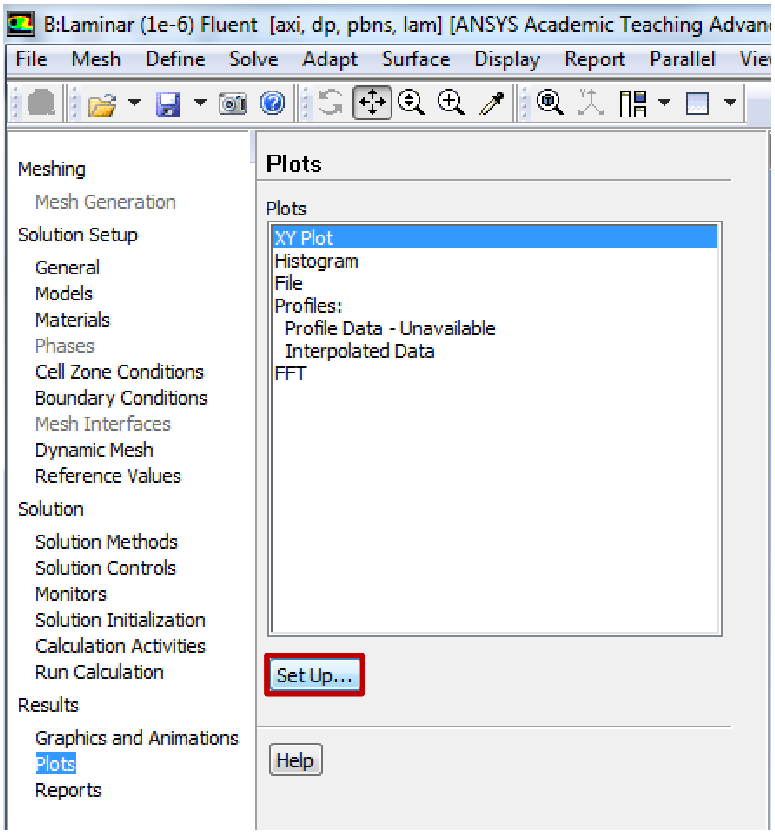
V&V Instructions for Velocity Profile

Right click **Solution** > Select **Edit…**

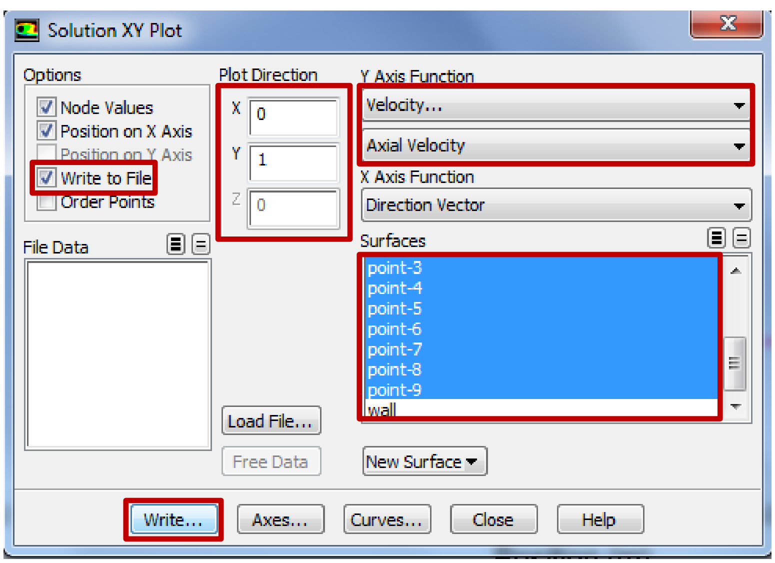


Create reference points

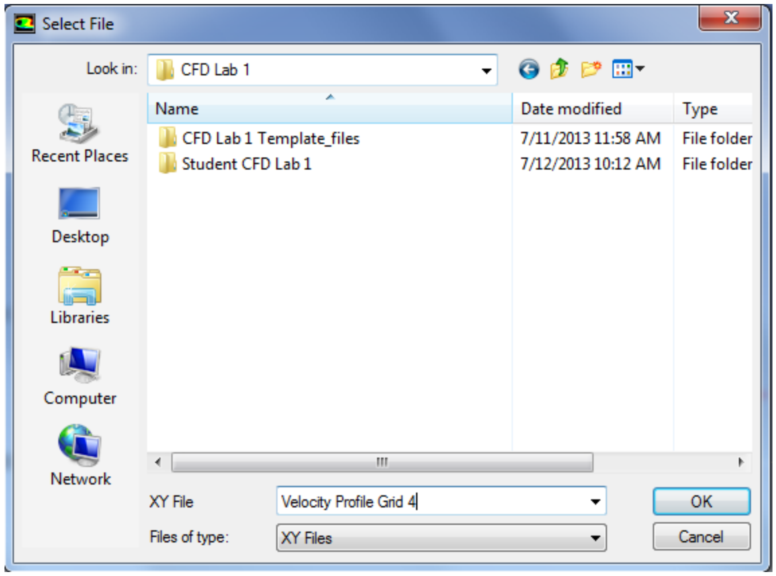
Results > **Plots** > **XY Plot** > **Set Up…**



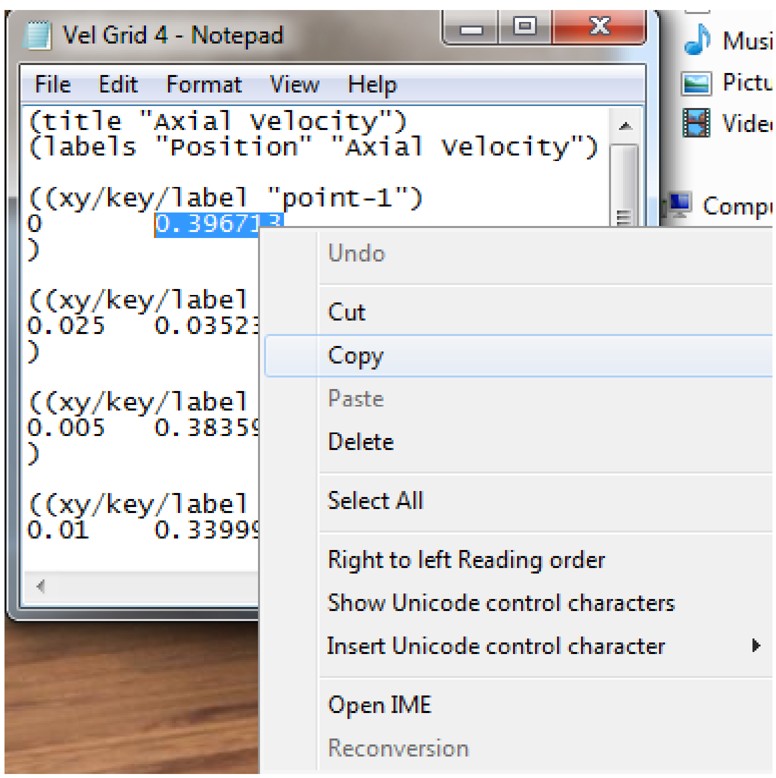
Change parameters as per below and click **Write…** Make sure to select points 1 through 10.



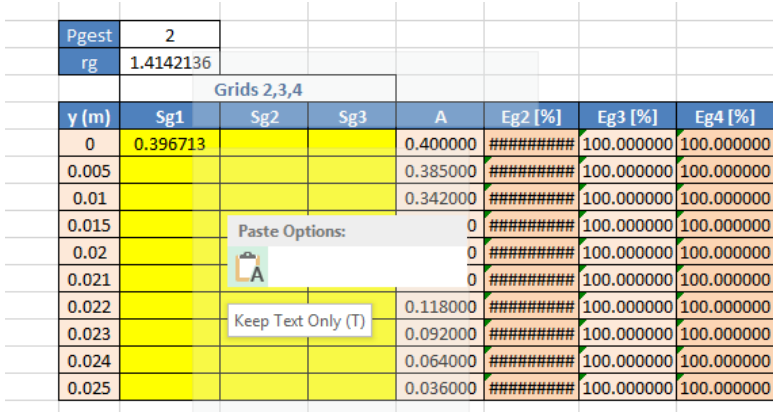
Name file according to which grid solution you are using.



Open file using Wordpad, copy points to input into V&V Excel file.



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

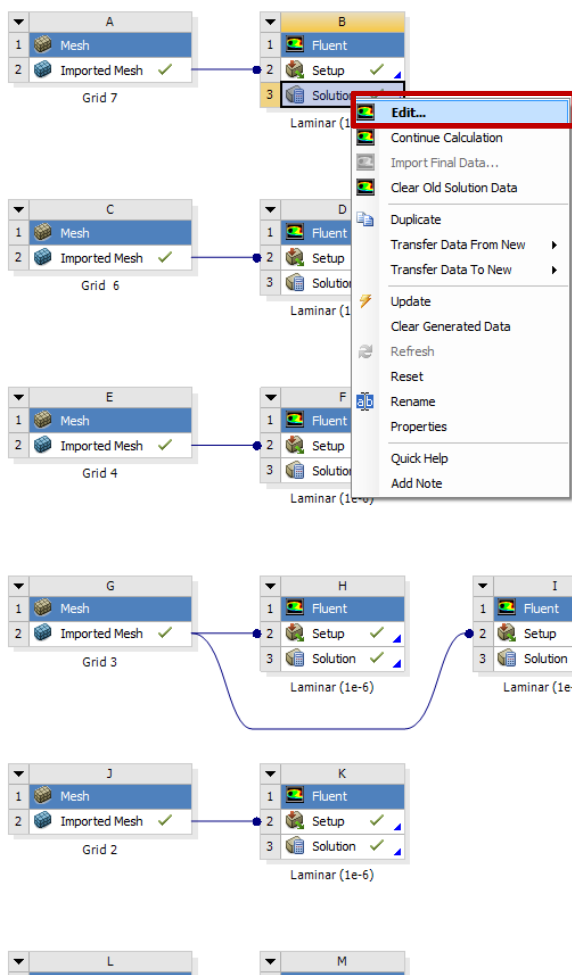


Pgest	2						
rg	1.4142136						
	Grids 2,3,4						
y (m)	Sg1	Sg2	Sg3	A	Eg2 [%]	Eg3 [%]	Eg4 [%]
0	0.396713			0.400000	##########	100.000000	100.000000
0.005				0.385000	##########	100.000000	100.000000
0.01				0.342000	##########	100.000000	100.000000
0.015				0	##########	100.000000	100.000000
0.02				0	##########	100.000000	100.000000
0.021				0	##########	100.000000	100.000000
0.022				0.118000	##########	100.000000	100.000000
0.023				0.092000	##########	100.000000	100.000000
0.024				0.064000	##########	100.000000	100.000000
0.025				0.036000	##########	100.000000	100.000000

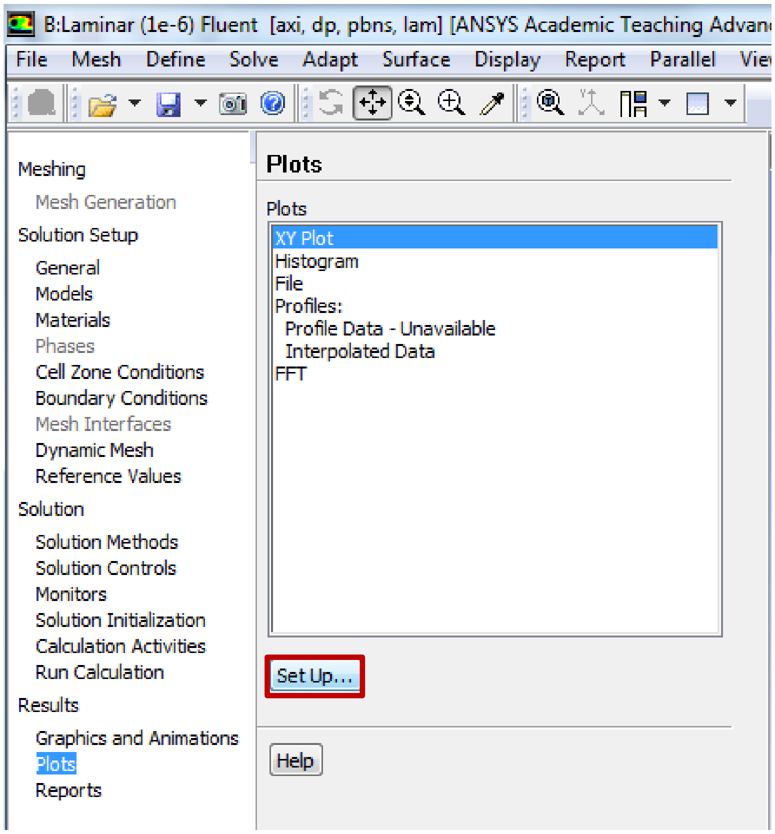
Repeat this process for the remaining y location points and then the two remaining grid solutions. All yellow cells should be filled.

V&V Instructions for Friction

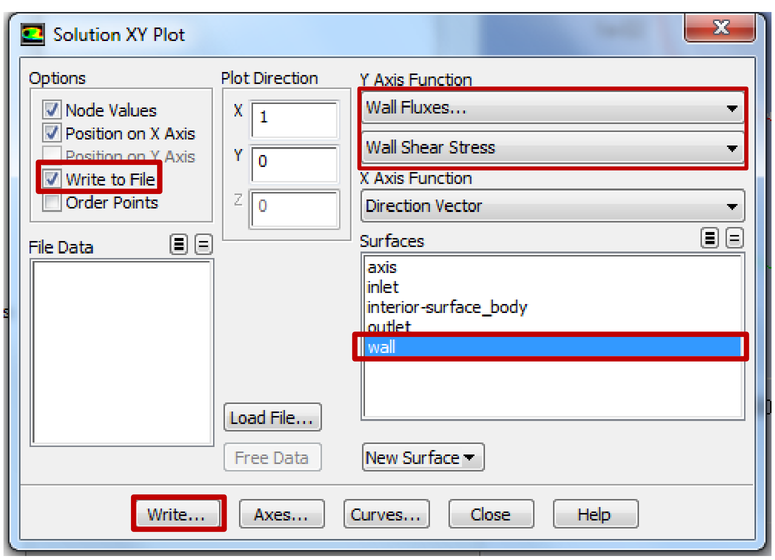
Right click **Solution** > Select **Edit…**



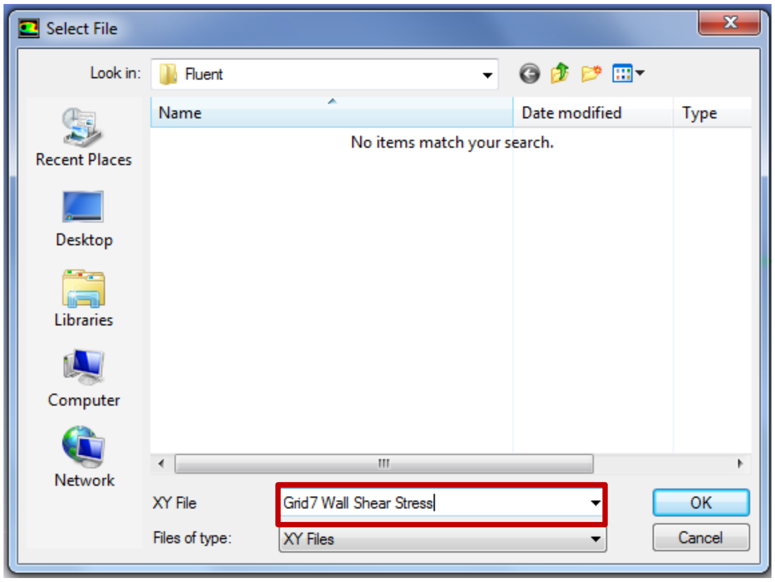
Results > **Plots** > **XY Plot** > **Set Up…**



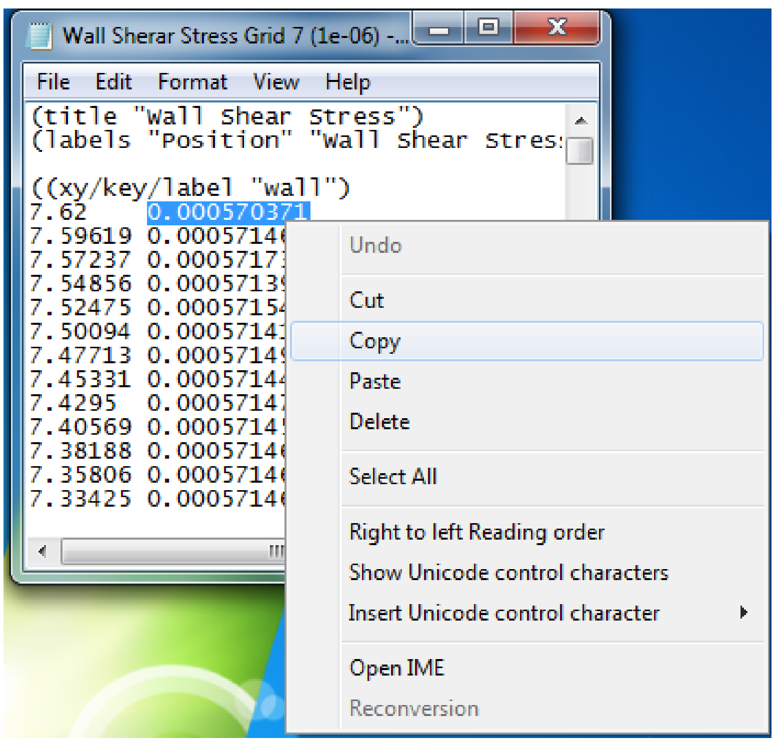
Change parameters as per below and click **Write…**



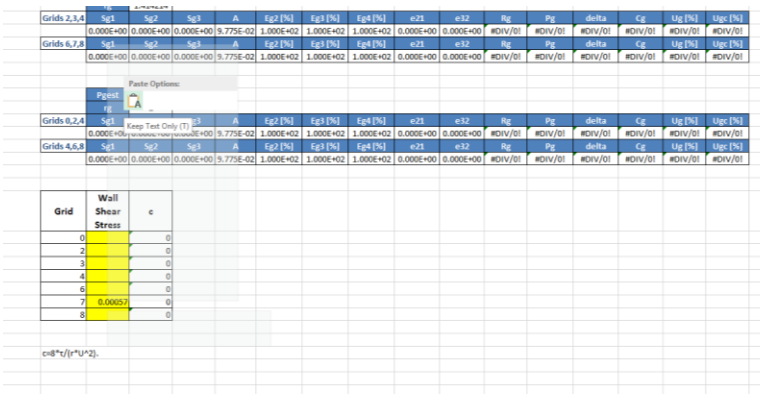
Name the file according to grid number and save to project folder.



Open file with Notepad and copy wall shear stress at the x location of 7.62m.



Paste the value into corresponding cell in the V&V template.



Grids 2,3,4	Sg1	Sg2	Sg3	A	Eg2 [%]	Eg3 [%]	Eg4 [%]	e21	e32	Rg	Pg	delta	Cg	Ug [%]	Ugc [%]
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	1.000E+02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
Grids 6,7,8	Sg1	Sg2	Sg3	A	Eg2 [%]	Eg3 [%]	Eg4 [%]	e21	e32	Rg	Pg	delta	Cg	Ug [%]	Ugc [%]
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	1.000E+02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!

Paste Options:

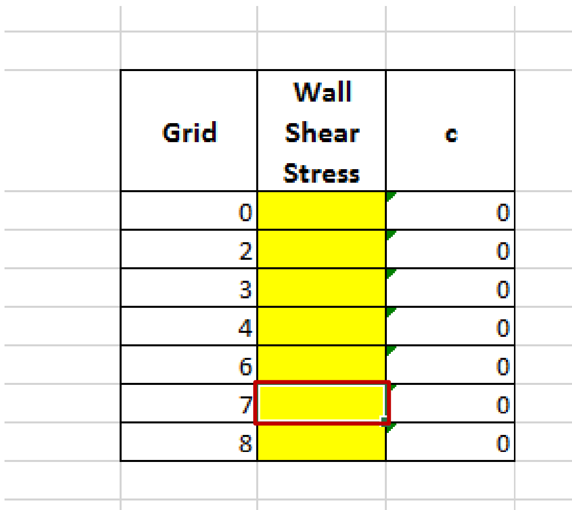
Keep Text Only (T)

	Pgest														
	rg														
Grids 0,2,4	Sg1	[illegible]	[illegible]	A	Eg2 [%]	Eg3 [%]	Eg4 [%]	e21	e32	Rg	Pg	delta	Cg	Ug [%]	Ugc [%]
	0.000E+00	[illegible]	[illegible]	9.775E-02	1.000E+02	1.000E+02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!
Grids 4,6,8	Sg1	Sg2	Sg3	A	Eg2 [%]	Eg3 [%]	Eg4 [%]	e21	e32	Rg	Pg	delta	Cg	Ug [%]	Ugc [%]
	0.000E+00	0.000E+00	0.000E+00	9.775E-02	1.000E+02	1.000E+02	1.000E+02	0.000E+00	0.000E+00	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!	#DIV/0!

Grid	Wall Shear Stress	c
0		0
2		0
3		0
4		0
6		0
7	0.00057	0
8		0

c=8*τ/(r*U^2).

Make sure when pasting you select **Keep Text Only** and you select the proper cell corresponding to the grid number.



Grid	Wall Shear Stress	c
0		0
2		0
3		0
4		0
6		0
7		0
8		0

Repeat this process for the remaining six grids. Each yellow cell should be filled.

8. Exercises

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

*** 1-4 and 6 are for laminar flows, 5 is for turbulent flows**

8.1. Iterative error studies: Use grid #4 and #8 with laminar flow conditions. Use two different convergent limits 10^{-5} and 10^{-6} and fill in the following table for the values on friction factors. Find the relative error between AFD friction factor (0.097747231) and friction factor computed by CFD, which is computed by:

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

To get the value of $Factor_{CFD}$, you need export wall shear stress data. Then use the wall shear stress at the developed region to calculate the friction factor. The equation for the friction factor is C=8*τ/(r*U^2). Where C is the friction factor, t is wall shear stress, r is density and U is the inlet velocity. Discuss the effect of convergent limit on results for these two meshes

Mesh No.	f (10^{-5})	F(10^{-6})
4	(%)	(%)
8	(%)	(%)

NOTE: (1). X and R should be NX+1 and NR+1. So, when you can create mesh manually, you need use NX, NR (112×10) for mesh 4 and (452×44) for mesh 8.

- Figure need to be saved: residuals history for mesh 8 for two convergent limits.
- Data need to be saved: the above table with values.
- ANSYS case need to be saved: mesh 8 with convergent limit 10^{-6}

8.2. Verification study for friction factor of laminar pipe flow: Run the simulations with the meshes shown in the table. Using mesh 4 as the "fine" mesh, and run verification with grid refinement ratio 1.414 and convergence limit 10^{-6}. Compute the parameters in the table (Refer to class website for V&V instructions). Using Mesh 8 as the "fine" mesh and repeat the above procedure using the same grid refinement ratio 1.414.

Meshes	Pg	Cg	Ug(%)	Ugc (%)
2,3,4				
6,7,8				

Which set of meshes is closer to the asymptotic range (i.e. Cg close to 1.0)? Which set has a lower grid uncertainty (Ug)? Which set is closer to the theoretical value of order of accuracy (2nd order). For the fine mesh 8, also compare its relative error of the friction factor (the one using convergent limit 10^{-6} in the table in exercise 1) with the grid uncertainty for 6,7,8, which is higher and what does that mean?

- Figure need to be saved: Figures and tables from V&V spread sheet.
- Data need to be saved: the above table with values

8.3. **Effect of grid refinement ratio on verification results (friction factor)**: Still use mesh 4 and 8 as the "fine mesh", but run verification with grid refinement ratio 2 for laminar pipe flow and convergence limit 10^{-6}.

Meshes	Pg	Cg	Ug(%)	Ugc (%)
0,2,4				
4,6,8				

Compared to results in 2, which set of meshes is sensitive to grid refinement ratio? Why?

- Figures need to be saved: Figures and tables from V&V spread sheet.
- Data need to be saved: the above table with values

8.4. **Verification study of axial velocity profile**: Use mesh 4 as the "fine mesh", use grid refinement ratio 1.414 and convergence limit 10^{-6}. Follow the V&V for velocity how to in the post processing section. Save the figures and discuss if the simulation has been verified.

- Figures need to be saved: Figures showing Ug, Ugc with |E|. Discuss which mesh solution is closest to the AFD data, why?
- Data need to be saved: None.

8.5. **Simulation of turbulent pipe flow**

Run simulation with convergence limit 10^{-6} and compare with EFD data on axial velocity profile and pressure distribution along the pipe. Export the axial velocity profile data at x=100D, use EXCEL to open the file you exported and normalize the profile using the centerline velocity magnitude at x=100D. Plot the normalized velocity profile in EXCEL and paste the figure into WORD.

- Figures need to be saved: Axial velocity profile with EFD data, normalized axial velocity profile at x=100D, centerline pressure distribution with EFD data, "centerline velocity distribution", contour of axial velocity, velocity vectors showing the developing region and developed regions.
- Data need to be saved: Developing length and compared it with that using formula 6.6 in textbook.

8.6. **Comparison between laminar and turbulent pipe flow**

Compare the results of laminar pipe flow using mesh 8 in exercise 1 (convergent limit 10^{-6}) with results of turbulent pipe flow in exercise 5. Analyze the difference in normalized axial velocity profile and developing length for laminar and turbulent pipe flows.

NOTE: (1). Since you have finished laminar simulation using mesh 8 in exercise 1, you can just open the case file you saved and output the figures and data you need.

- Figures need to be saved: Axial velocity profile with AFD data, normalized axial velocity profile at x=100D, “centerline velocity distribution” for laminar flows.
- Data need to be saved: Developing length for laminar pipe flow and compared it with that using formula 6.5 in textbook.

8.7. Questions need to be answered in CFD Lab report

8.7.1. Answer all the questions in exercises 1 to 6

8.7.2. Analyze the difference between CFD/AFD and CFD/EFD and possible error sources.

8.7.3. Analyze the difference between ANSYS predictions and your own calculations (using formula in CFD lecture) for order of accuracy and grid uncertainties.