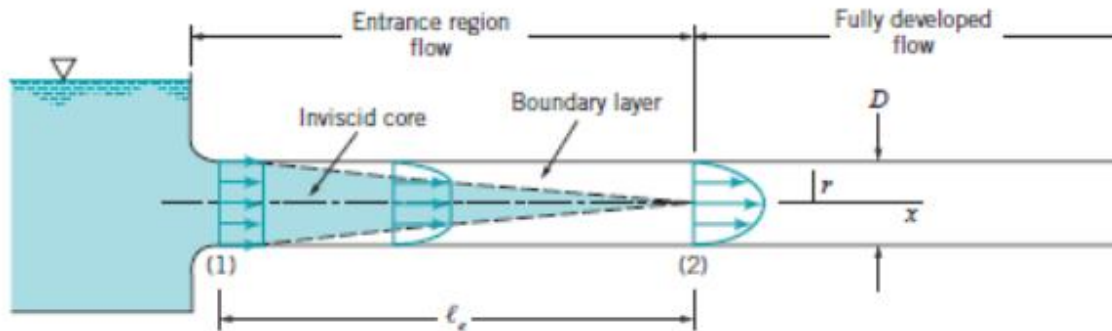


Flow in a Circular Pipe



1. Purpose

The Purpose of CFD Task 1 is to simulate steady **laminar** and **turbulent** pipe flow following a step-by-step instruction. Students should analyze axial velocity profiles in different sections, centerline velocity and pressure distributions, and friction factor. Students will compare **velocity profiles** with experimental and analytical data, analyze the differences between laminar and turbulent flows, and present results in CFD Lab report.

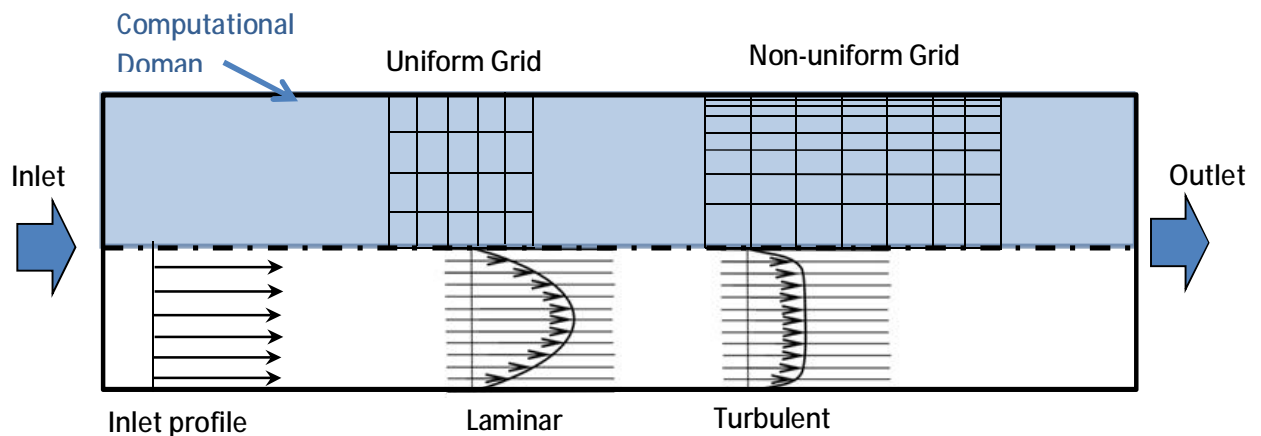
2. Simulation Design

The numerical simulation will be conducted for **laminar and turbulent** pipe flows.

For laminar mode, the Reynolds number is in the range of 250-1250 (for every student the value is defined by teacher).

For turbulent flow, the Reynolds number equal to $1 \cdot 10^5$ is considered. The schematic of the problem and the parameters for the simulation are shown below.

Since the flow is axisymmetric we only need to solve the flow in a single plane from the centerline to the pipe wall. The pipe diameter and length are equal to 0.02 m and 6 m correspondingly.



Boundary conditions need to be specified include **inlet**, **outlet**, **wall**, and **axis**. 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 outlet. Symmetric boundary condition will be applied on the pipe axis. Uniform grid will be used for the laminar flow whereas non-uniform grid will be used for the turbulent flow. Mesh characteristics is presented in the table below:

Table 1

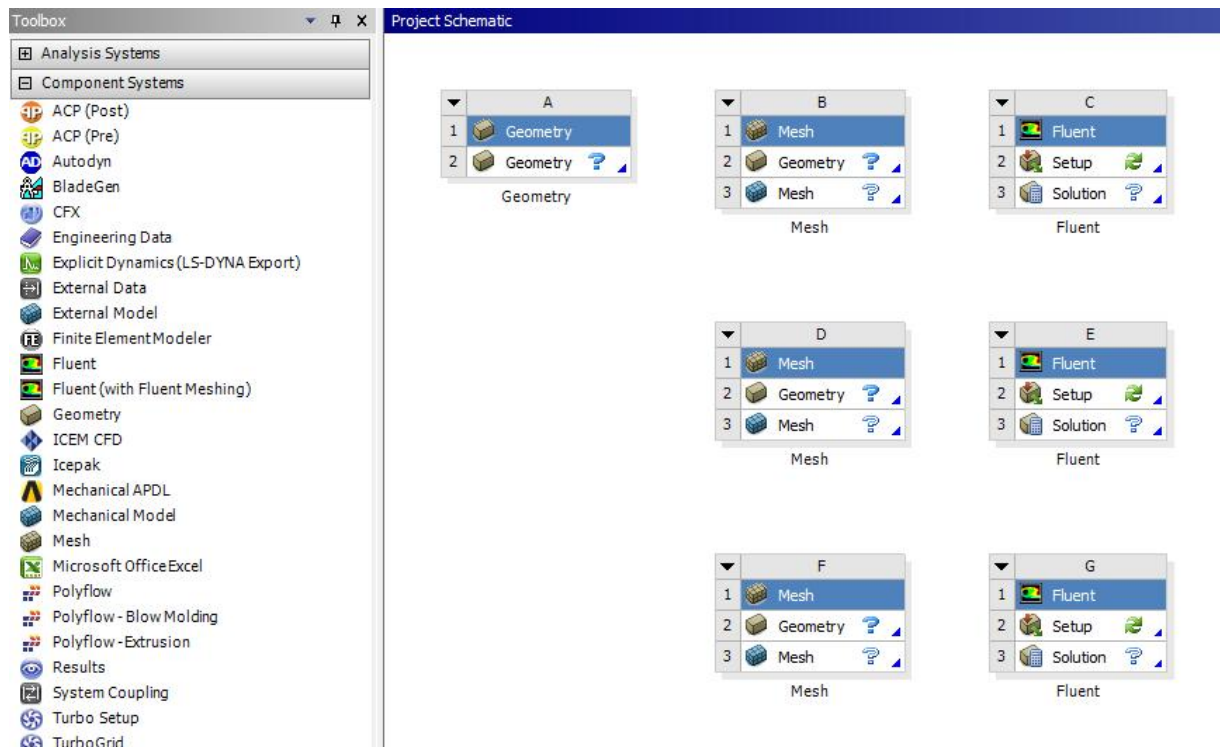
Mesh	Type	Number of Divisions	
		X	R
Laminar	Uniform	450	45
Laminar	Uniform	150	15
Turbulent	Non-uniform	600	15

Simulation results will be compared with analytical and empirical data. All analytical and empirical equations required for comparison are provided.

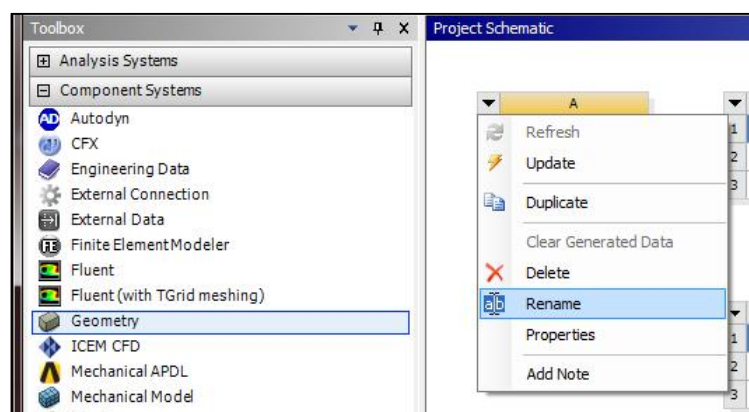
3. Project Schematic in Ansys Workbench

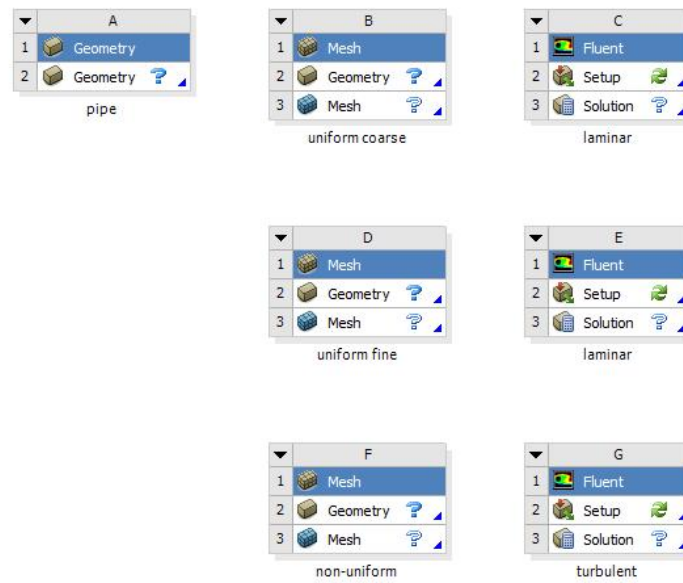
3.1. Start ANSYS Workbench

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

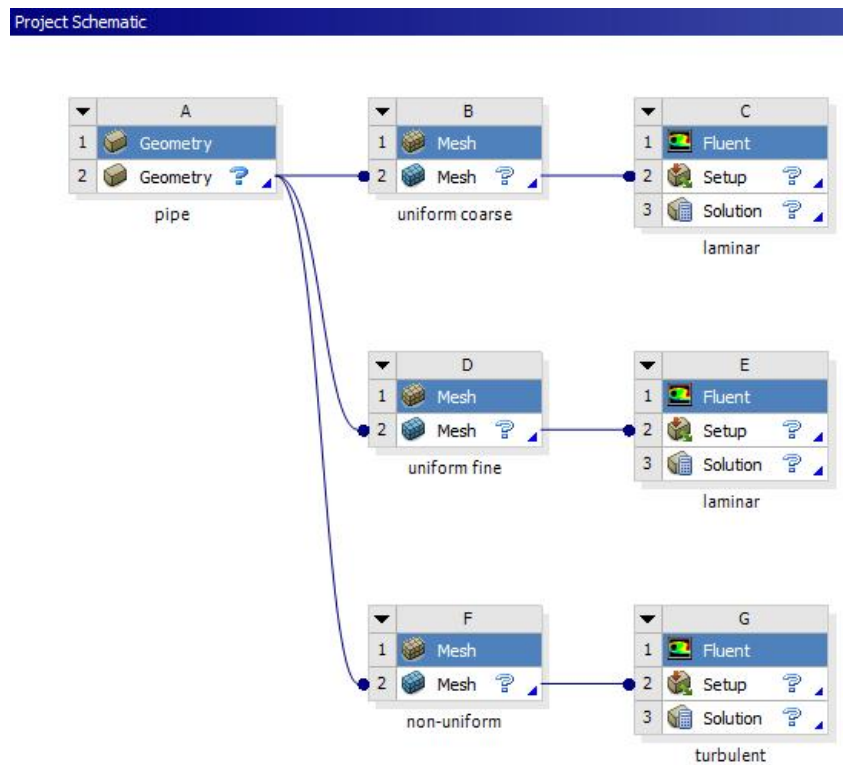


3.3. Click on the drop down arrow and select **Rename**. Change the names as per below.

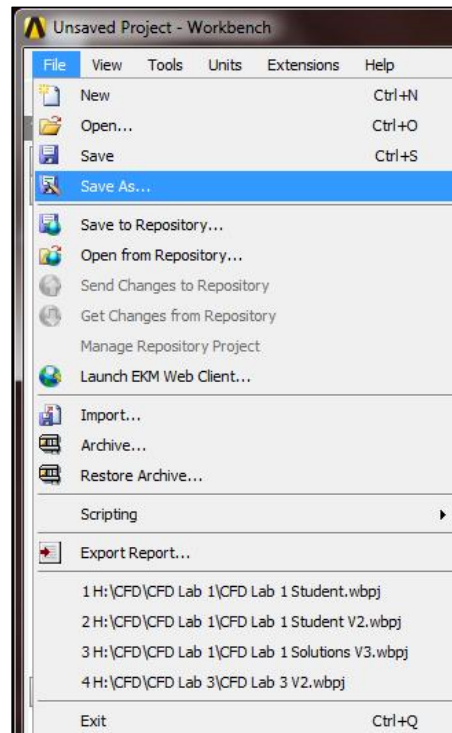




3.4. Create connections between component as per below. To make connections, click and drag the **Geometry** box to the **Mesh** box, and the **Mesh** box to the **Setup** box as per below.

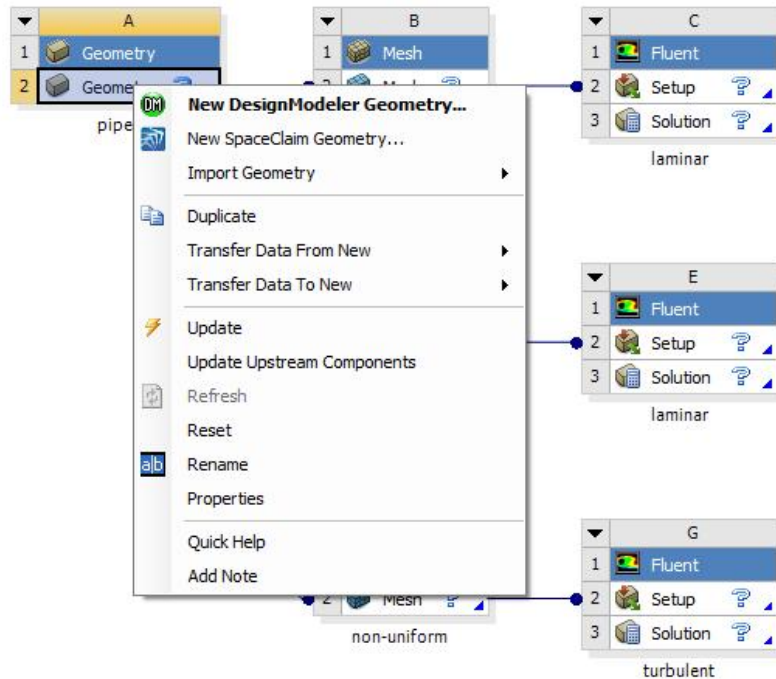


File > Save As. Save the workbench file.

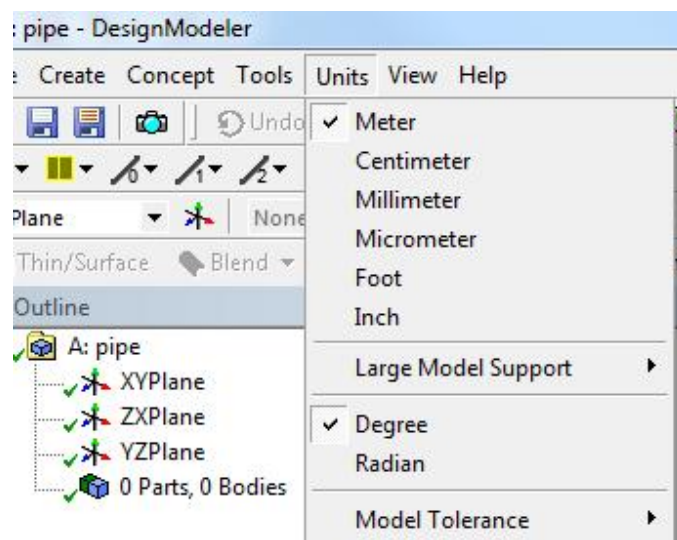


4. Geometry

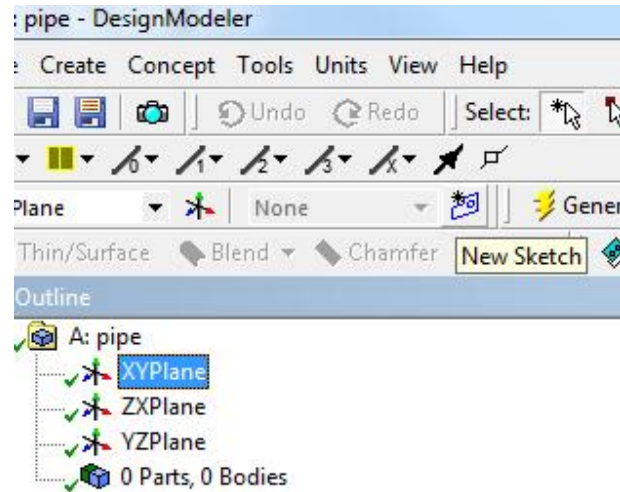
4.1. Right click **Geometry** and select **New DesignModelerGeometry....**



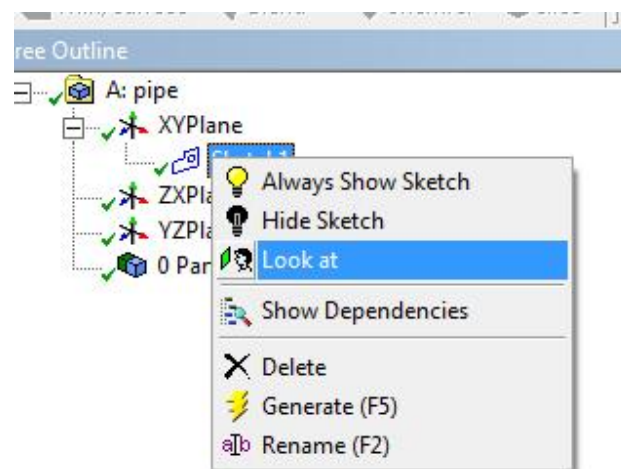
4.2. Make sure that Unit is set to **Meter** (default value).



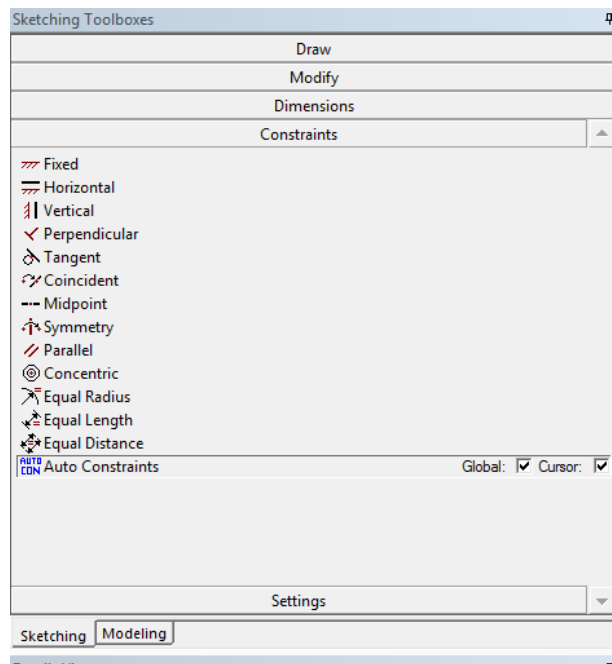
4.3. Select the **XYPlane** under the **Tree Outline** and click **New Sketch** button.



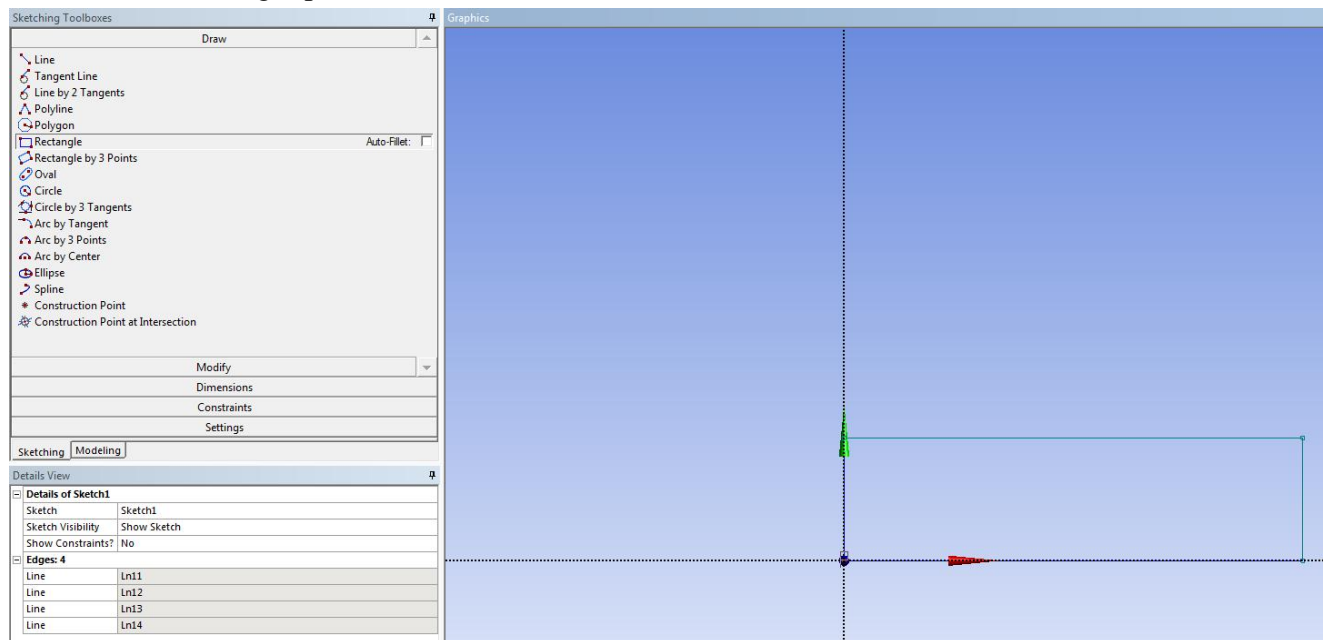
4.4. Right click **XYPlane** and select **Look at**.



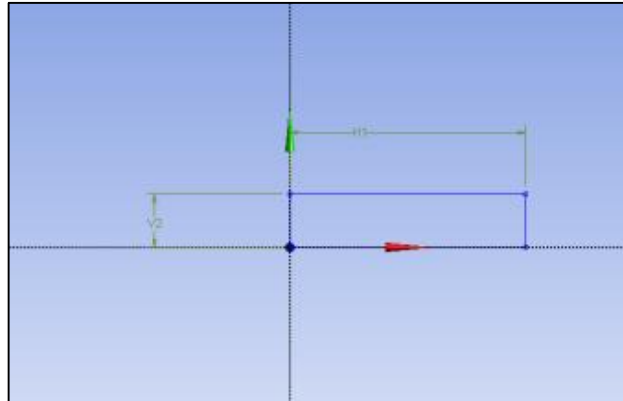
4.5. Enable the auto constraints option to pick the exact point as below. Select **Sketching > Constraints > Auto Constraints** > make sure Cursor is selected.



4.6. Select **Sketching > Draw > Rectangle**. Create a rectangle geometry. The cursor will show “P” when it is on the origin point.



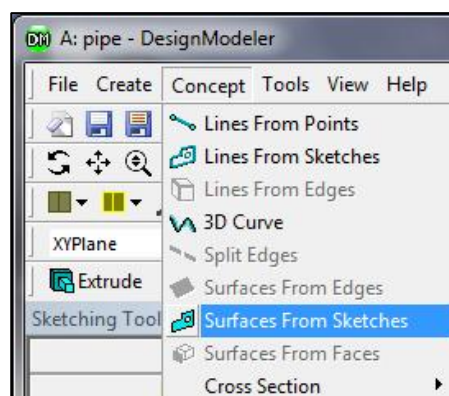
- 4.7. Select **Sketching > Dimensions > General**. Click on top edge then click anywhere. Repeat the same thing for one of the vertical edges. You should have a similar figure as per below.



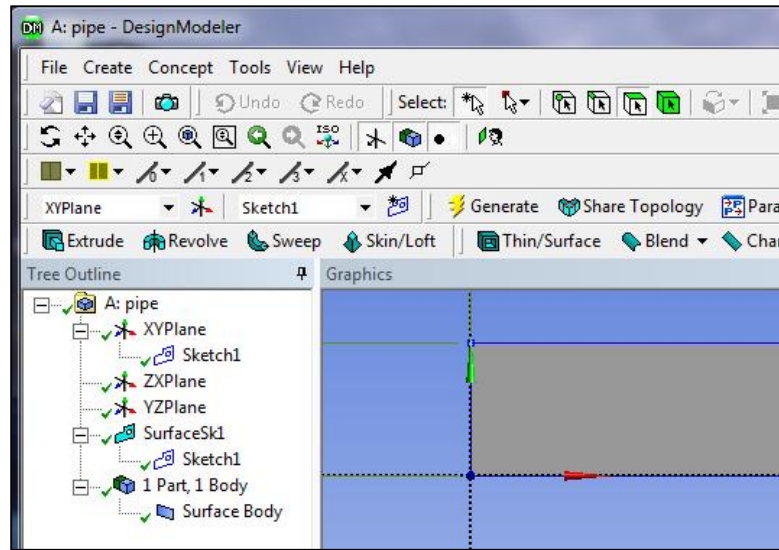
- 4.8. Click on **H1** under **Details View** and change it to **6 m**. Click on **V2** and change it to **0.02 m**.

Details View	
[-] Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
[-] Dimensions: 2	
<input type="checkbox"/> H1	6 m
<input type="checkbox"/> V2	0.02 m
[-] Edges: 4	
Line	Ln11
Line	Ln12
Line	Ln13
Line	Ln14

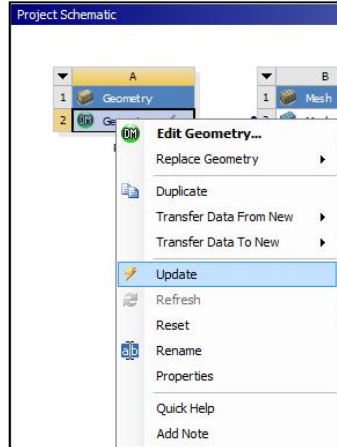
- 4.9. **Concept > Surfaces From Sketches** and select **Sketch1** from the **Tree Outline** and hit **Apply** on **Base Objects** under **Details view**.



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

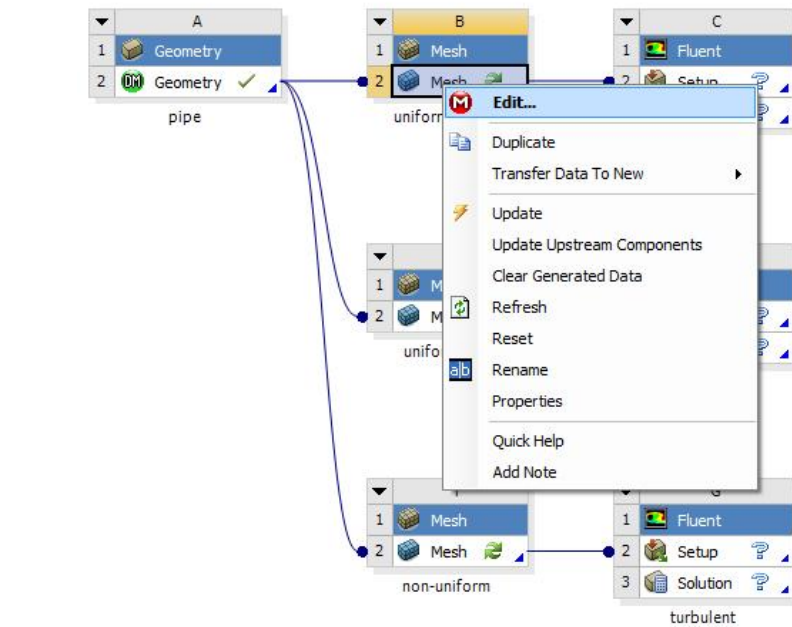


- 4.11. **File > Save Project**. Save project and close window.
- 4.12. If you see the lightning sign next to **Geometry** in the workbench then right click on **Geometry** and click **Update** as shown below.

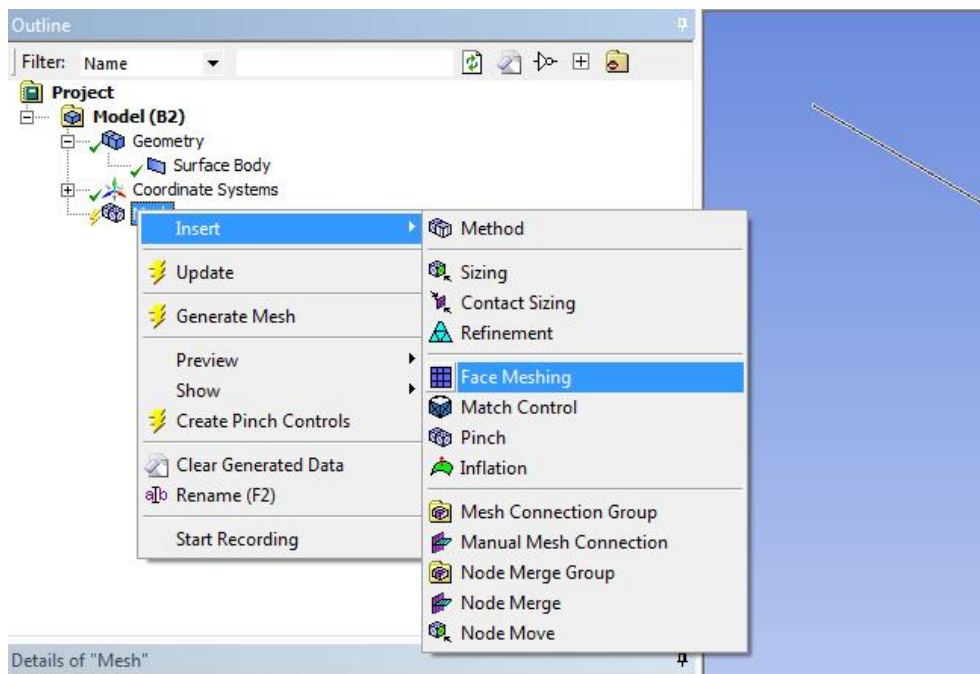


5. Meshing

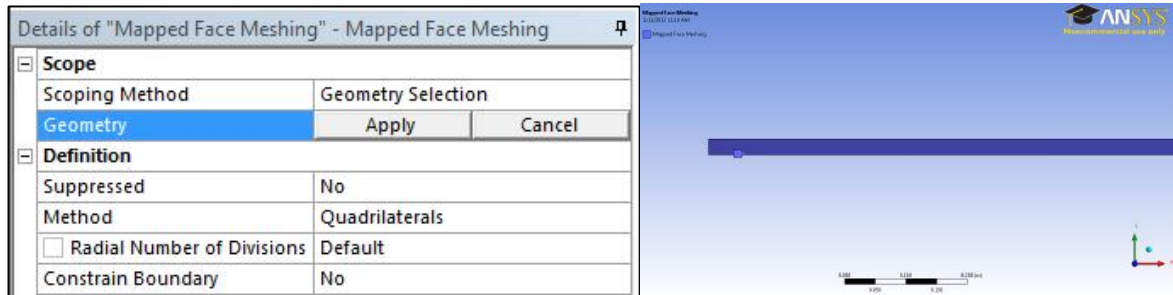
5.1. Right click on **Mesh** and select **Edit. Start Meshing**.



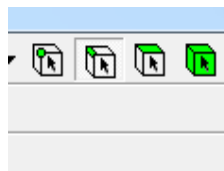
5.2. Right click on **Mesh** then select **Insert > Face Meshing**.



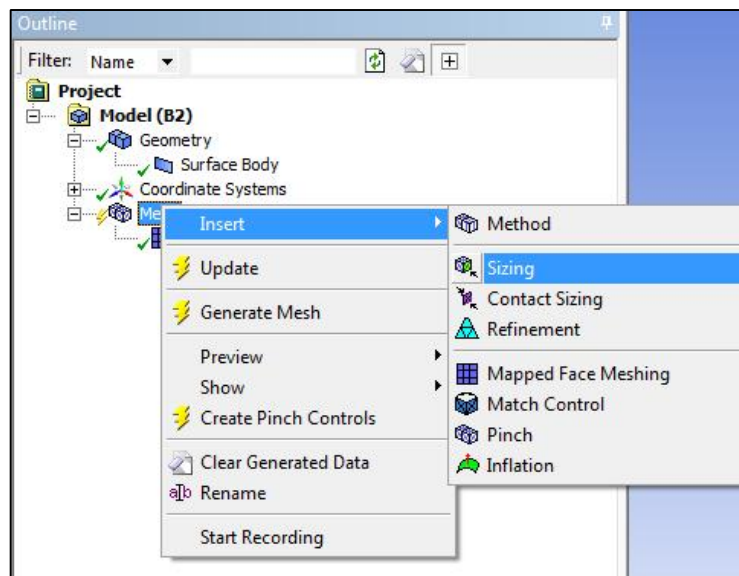
- 5.3. Select the pipe geometry by clicking anywhere on the pipe surface, then click the yellow box that says “No Selection” and click **Apply**. (From this stage: Rotate the view to xy-plane by clicking z-axis of 3D axis at right bottom of the screen. Drag and drop with right mouse button to zoom in. Press F7 to restore the view.)



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



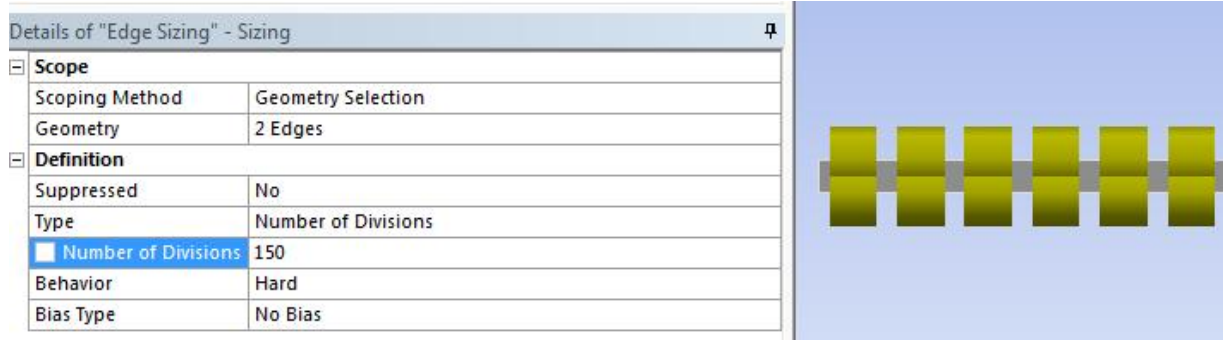
- 5.5. Right click on **Mesh** then select **Insert > Sizing**.



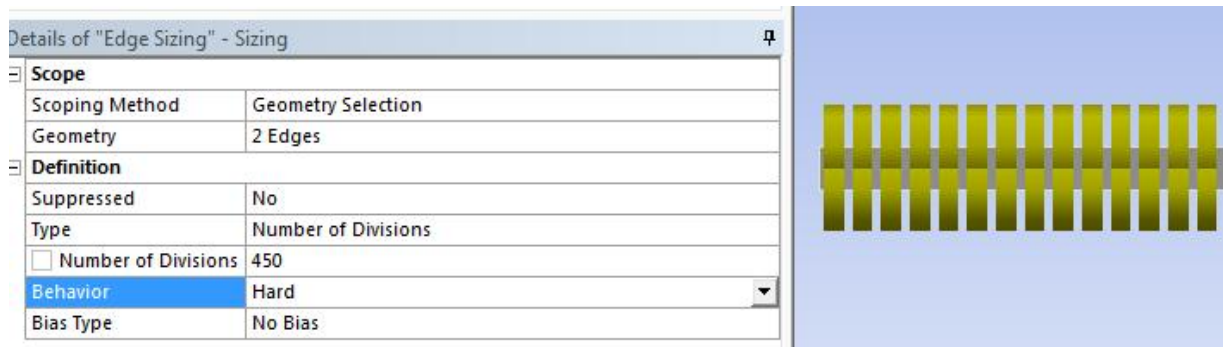
5.6. Hold Ctrl and select the top and bottom edge then click **Apply** in the **Details** box for **Geometry** on the right. Specify details of sizing as per below depending on case.

Note that the number of divisions should be different for fine and coarse mesh created for laminar flow and for turbulent flow.

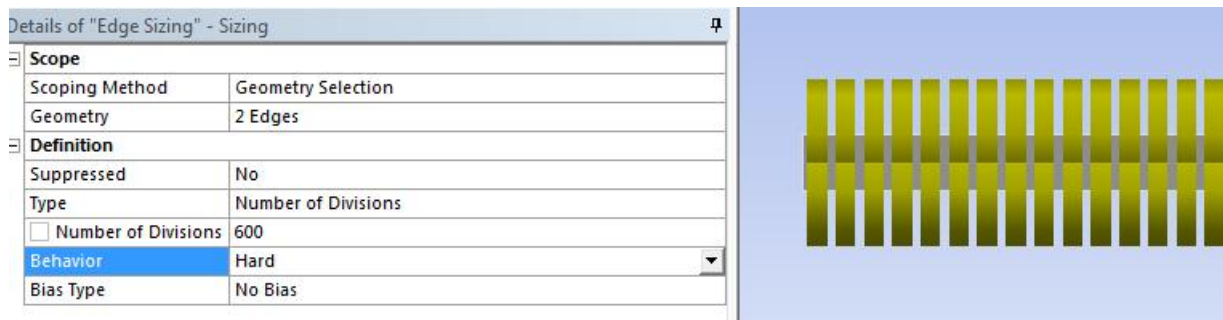
For laminar flow. Coarse mesh.



For laminar flow. Fine mesh.

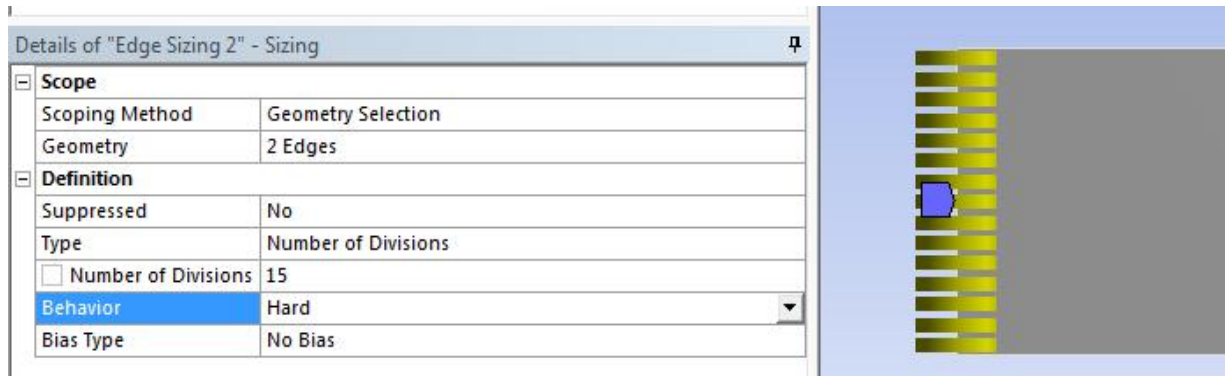


For turbulent flow

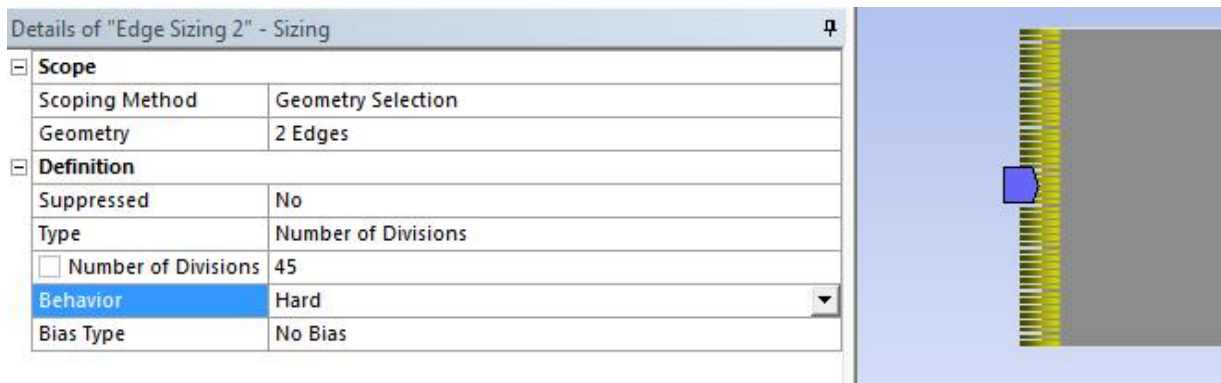


- 5.7. Repeat step 5.5. Select the left and right edge and click **Apply** for uniform grid flow and change sizing parameters as per below. Change the sizing parameters separately for non-uniform grid as per below. Make sure to select edges individually when changing sizing parameters for non-uniform grid.

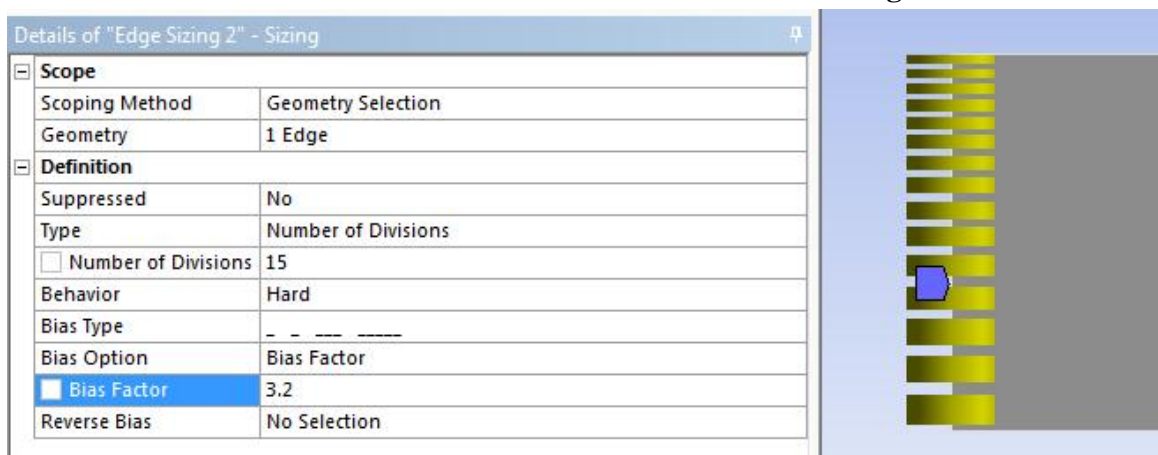
For laminar flow. Uniform coarse mesh.



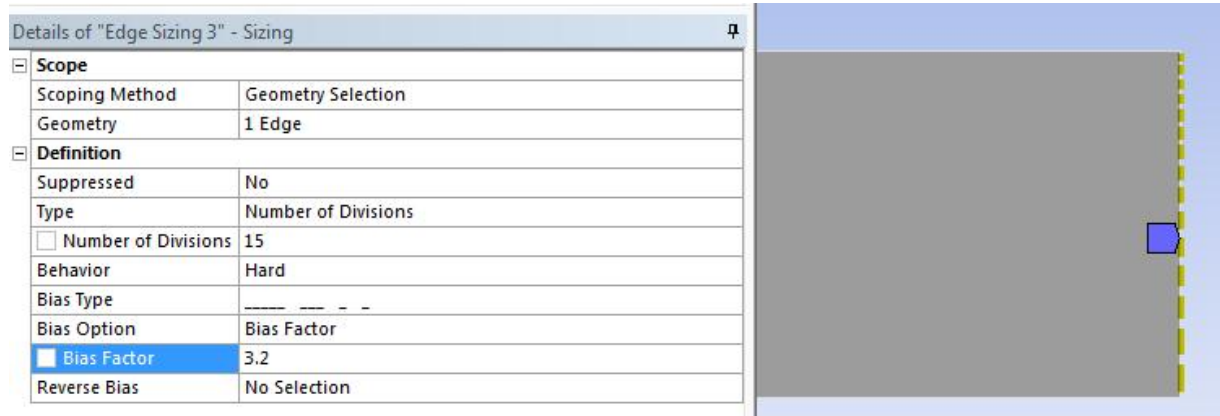
For laminar flow. Uniform mesh (fine).



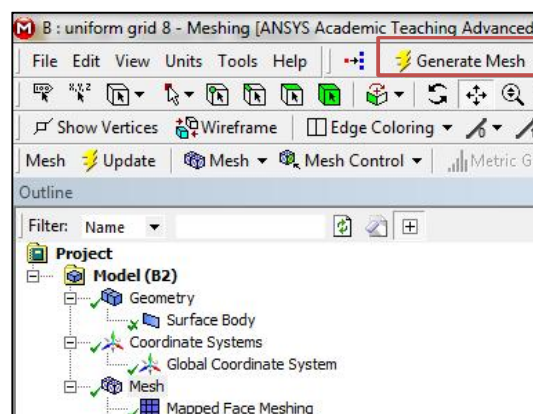
For turbulent flow. Non-uniform mesh. Left Edge



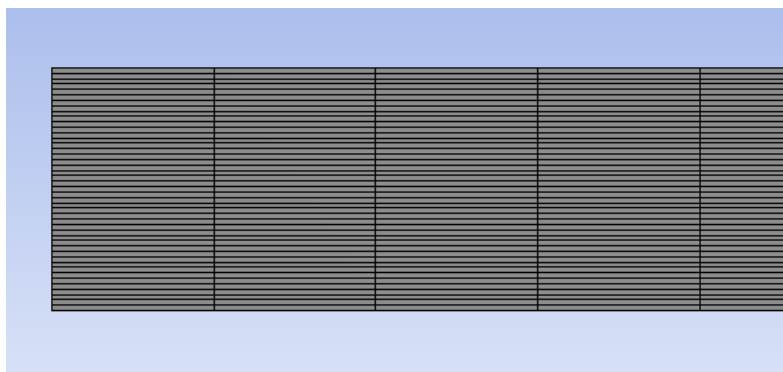
For turbulent flow. Non-uniform mesh. Right Edge



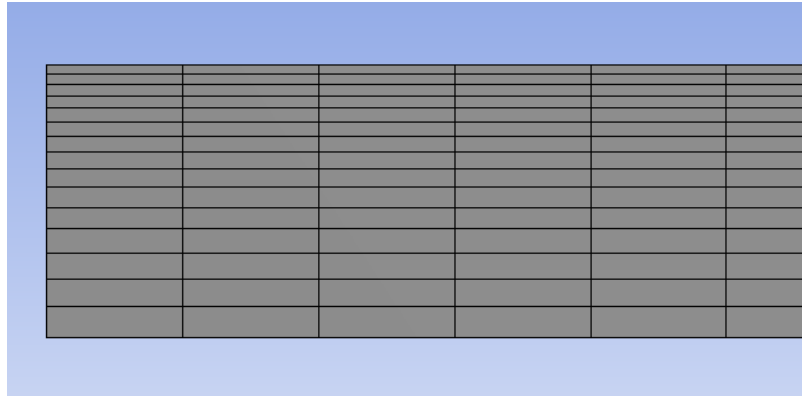
5.8. Click on **Generate Mesh** button and click **Mesh** under **Outline** to show mesh.



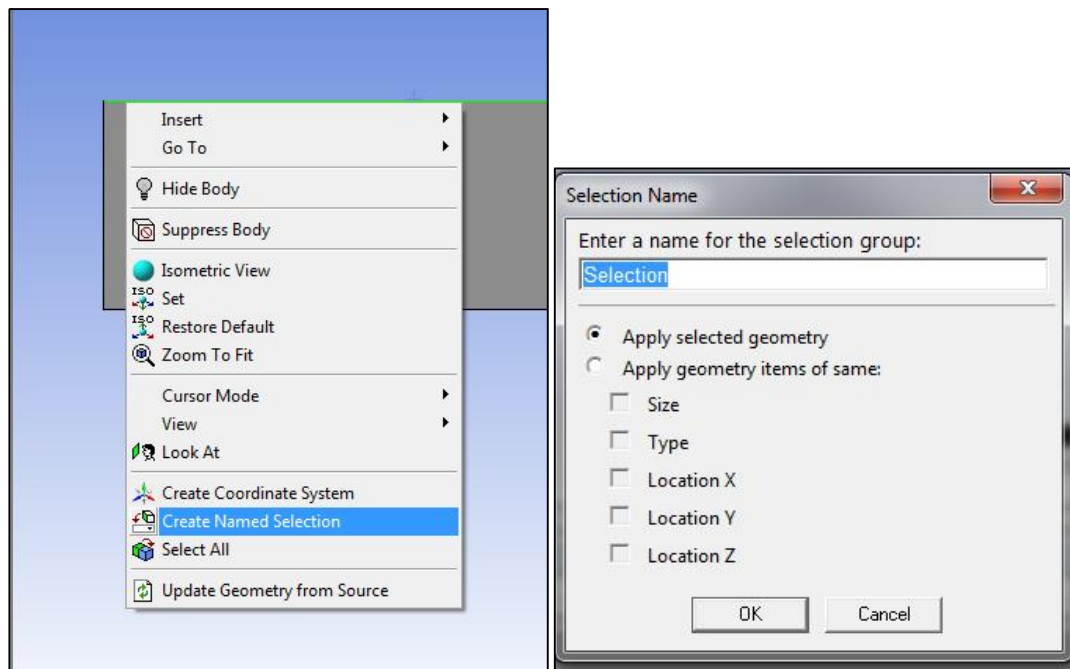
Uniform fine mesh

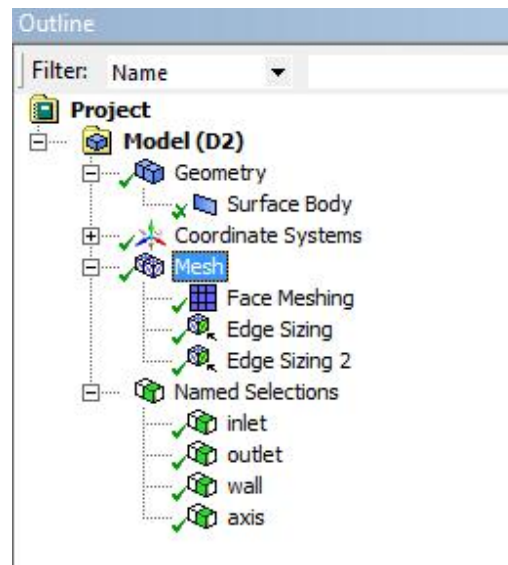


Non-uniform mesh.

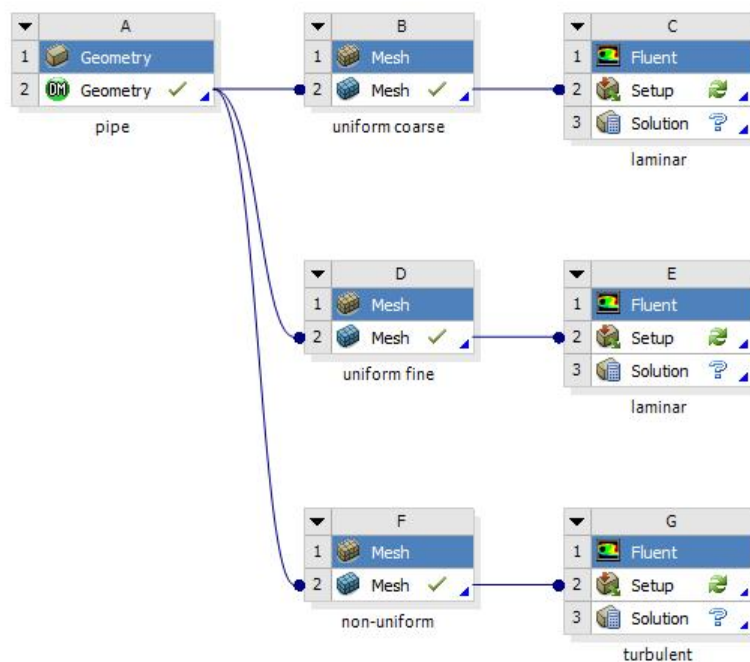


- 5.9. Change the edge names by right clicking on the edge and selecting **Create Named Selection**. Name left, right, bottom and top edges as inlet, outlet, axis and wall respectively. Your outline should look same as the figure below.



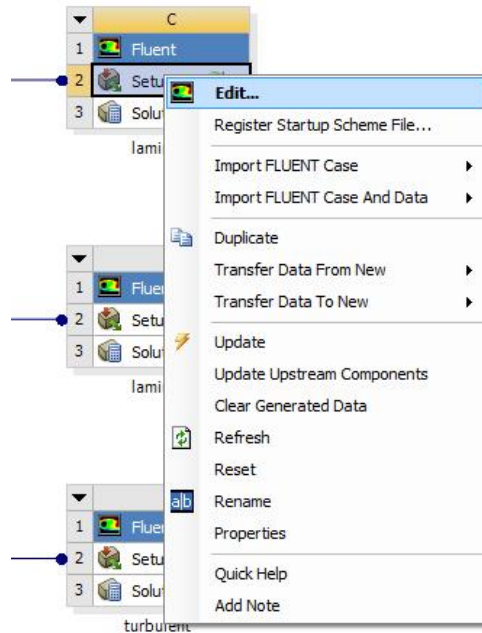


5.10. **File > Save Project.** Save the project and close the window. Update **Mesh** on Workbench if necessary. Your Project Schematic will be similar to the figure below.

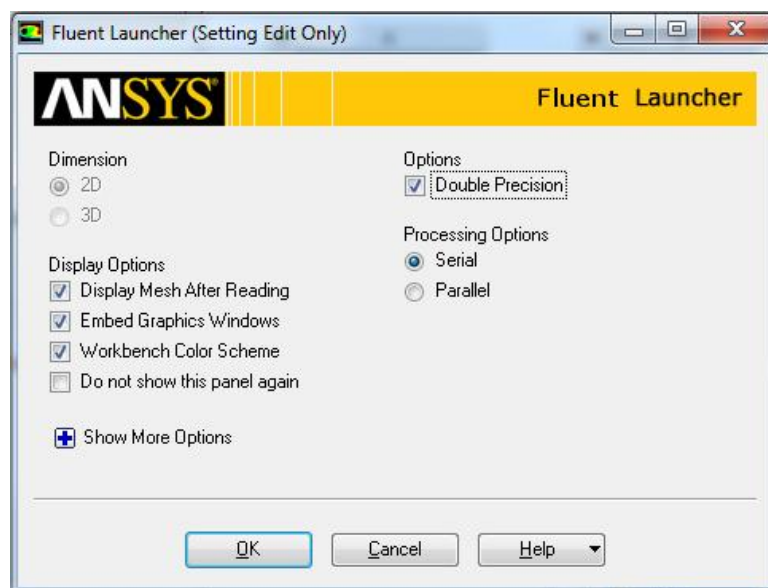


6. Solving in Fluent

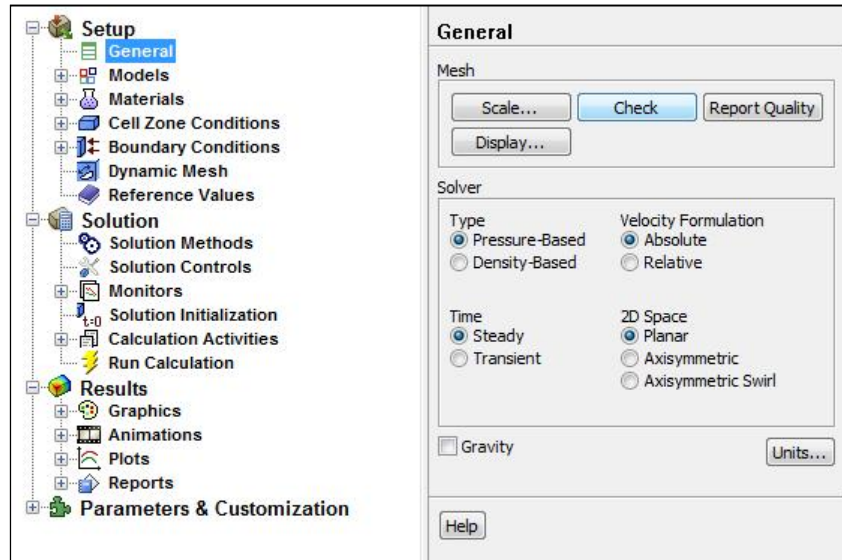
6.1. Right click **Setup** and select **Edit**.



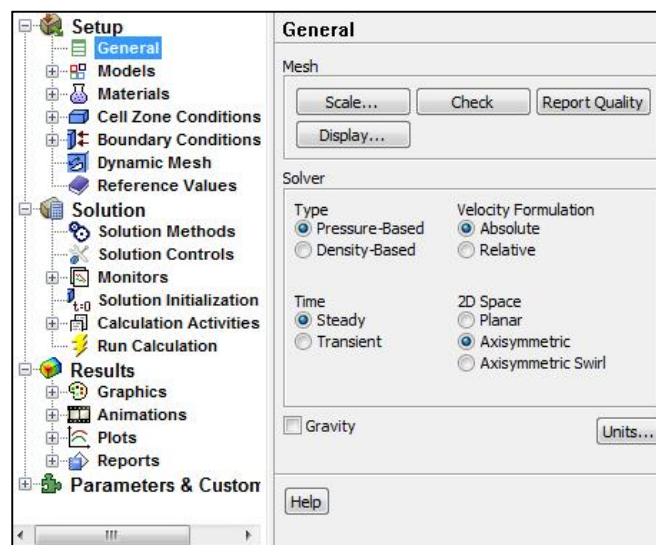
6.2. Under options check **Double Precision** and click **OK**.



6.3. **Setup > General > Check.** You may ignore the warning messages. (Note: If you get an error message you may have made a mistake while creating your mesh)

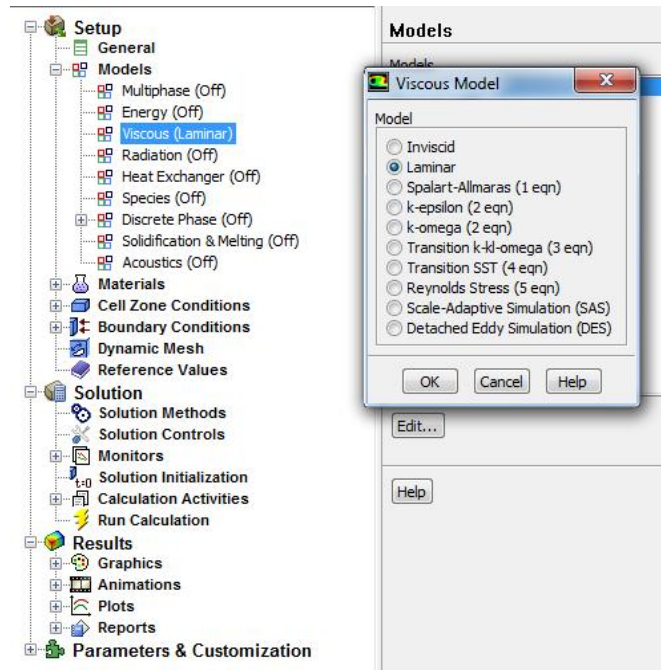


6.4. **Setup > General > Solver.** Choose options shown below.

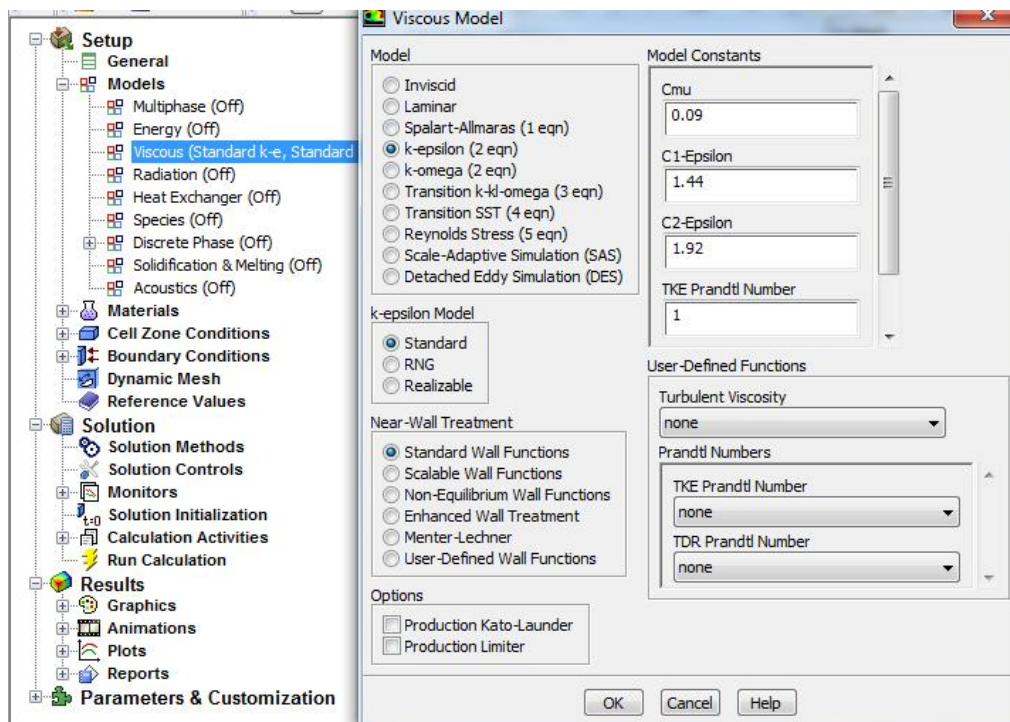


6.5. **Setup > Models > Viscous (Laminar)** (double click). Select parameters as per below and click OK.

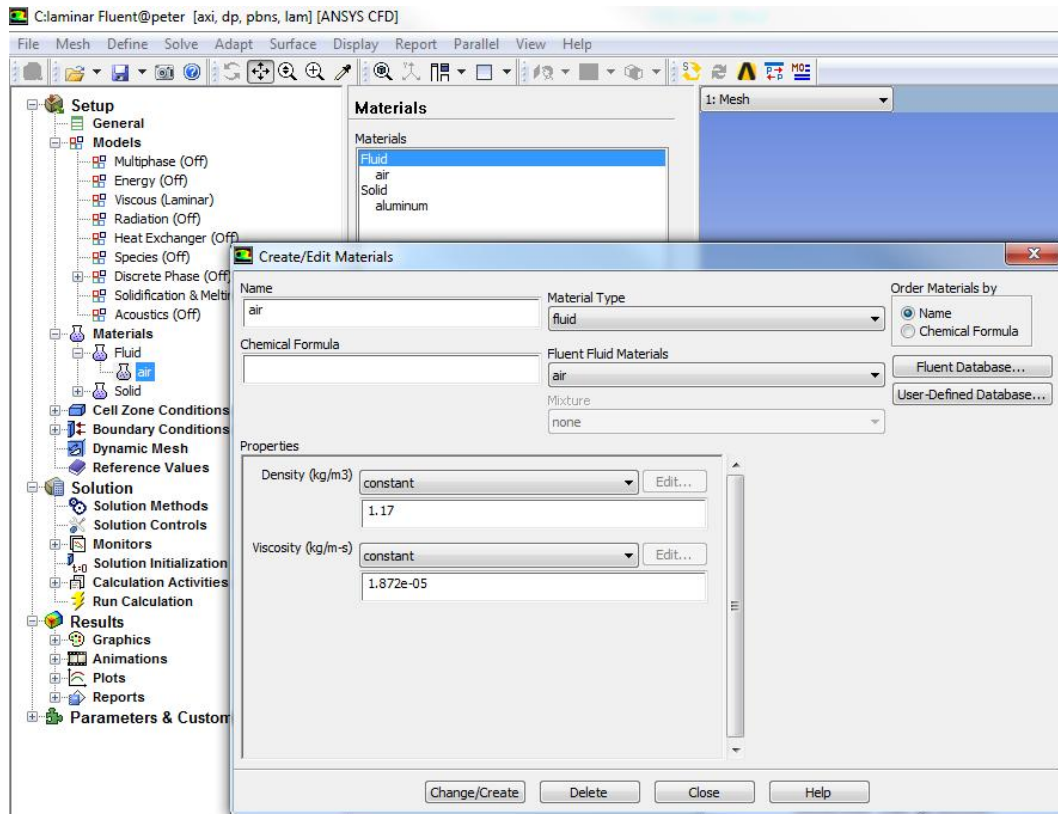
Laminar flow



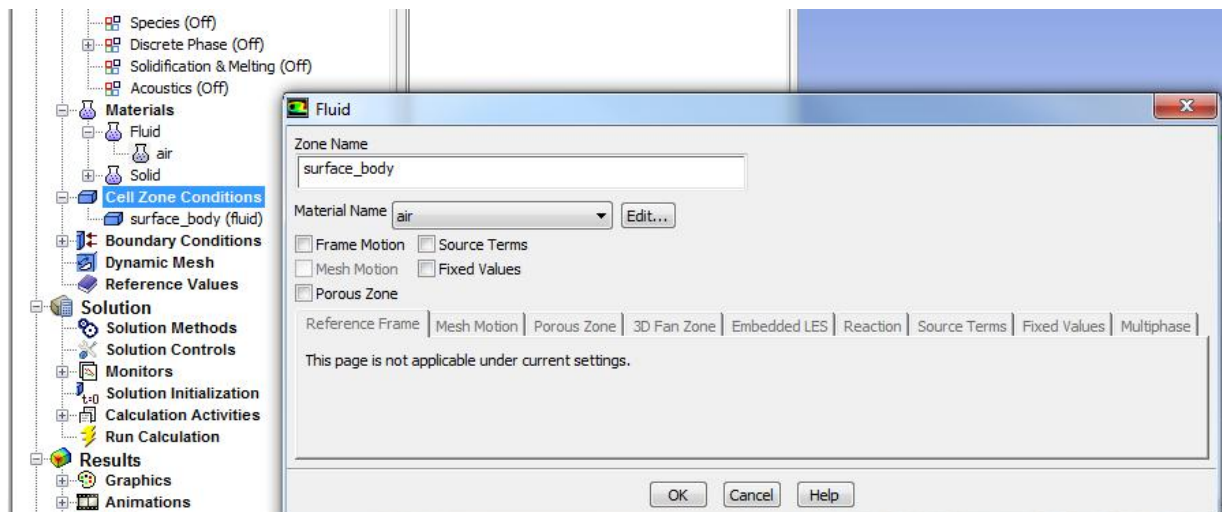
Turbulent flow



6.6. **Setup > Materials > Fluid > air** (double click). Change the **Density** and **Viscosity** as per below and click **Change/Create**. Close the dialog box when finished.

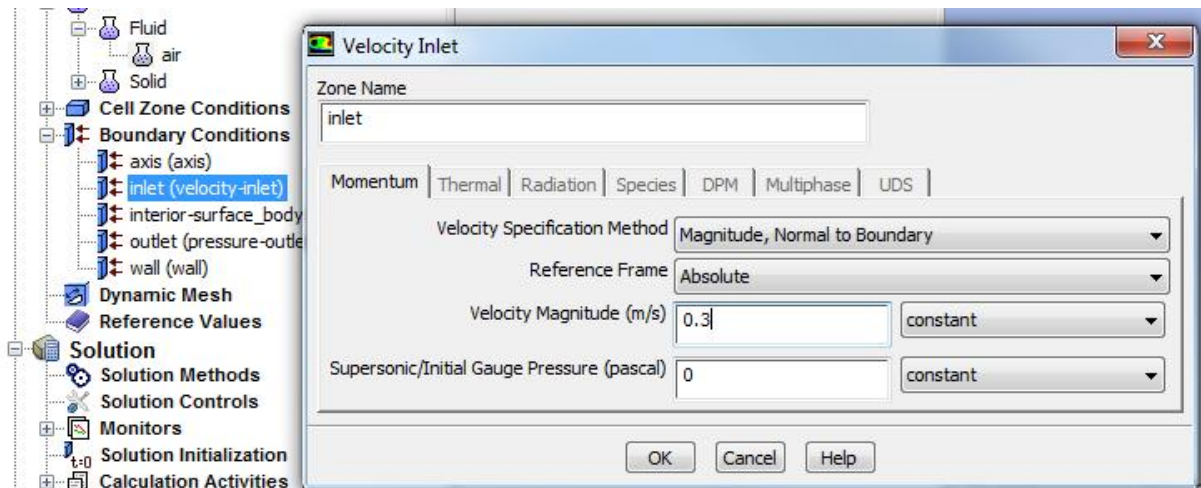


6.7. **Setup > Cell Zone Conditions > Zone > surface_body**. Make sure **air** is selected and click **OK**.



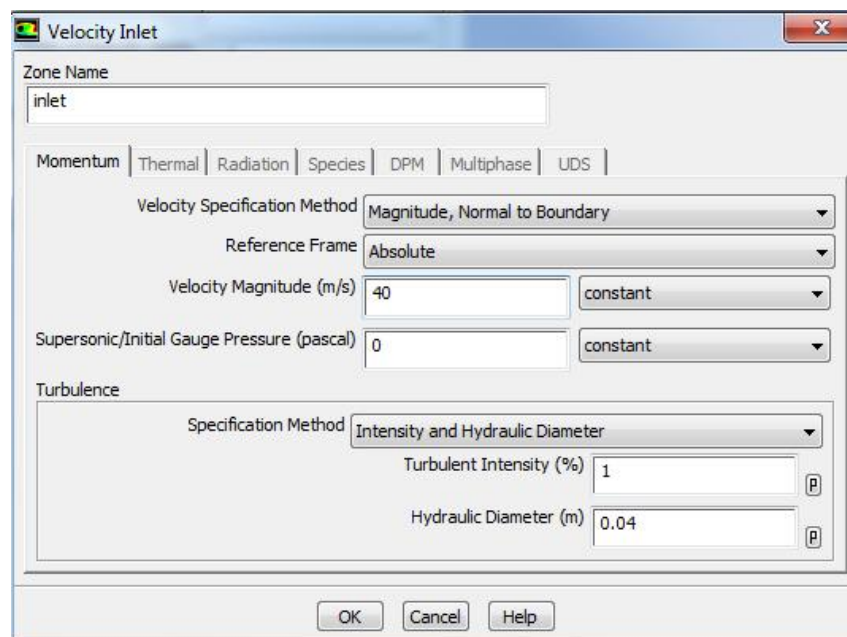
6.8. **Setup > Boundary Conditions > inlet** (double click). Change parameters as per below and click **OK**.

Laminar flow

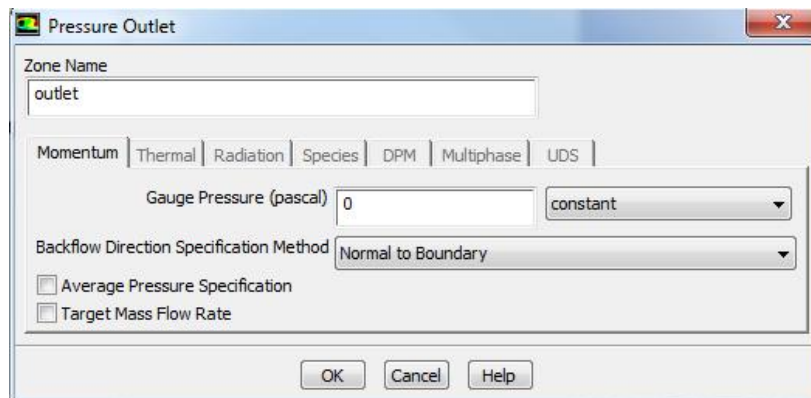


!!! The velocity value may be different. It should be calculated by student for a given Reynolds number.

Turbulent flow

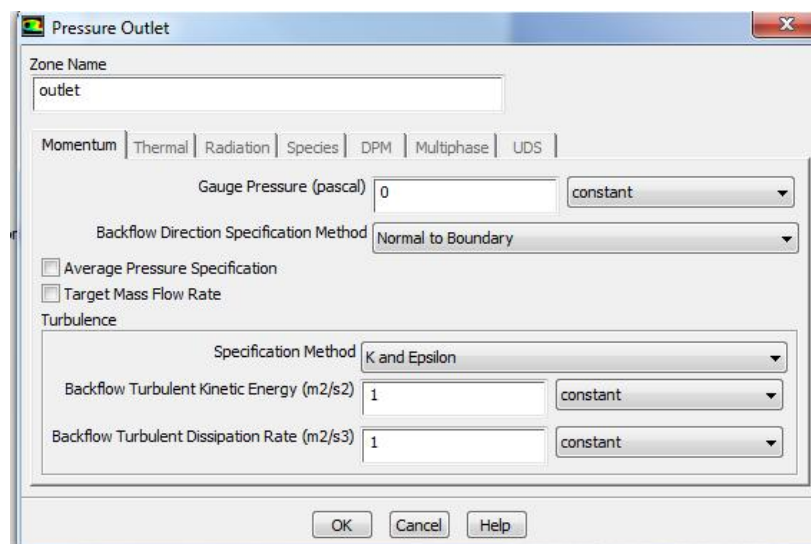


6.9. **Setup > Boundary Conditions > outlet** (double click) or click **Edit....** Change parameters as per below and click **OK. Laminar flow**



The 'Pressure Outlet' dialog box is shown with the 'Momentum' tab selected. The 'Zone Name' field contains 'outlet'. The 'Gauge Pressure (pascal)' is set to 0 with a 'constant' dropdown. The 'Backflow Direction Specification Method' is set to 'Normal to Boundary'. The 'Average Pressure Specification' and 'Target Mass Flow Rate' checkboxes are unchecked. The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

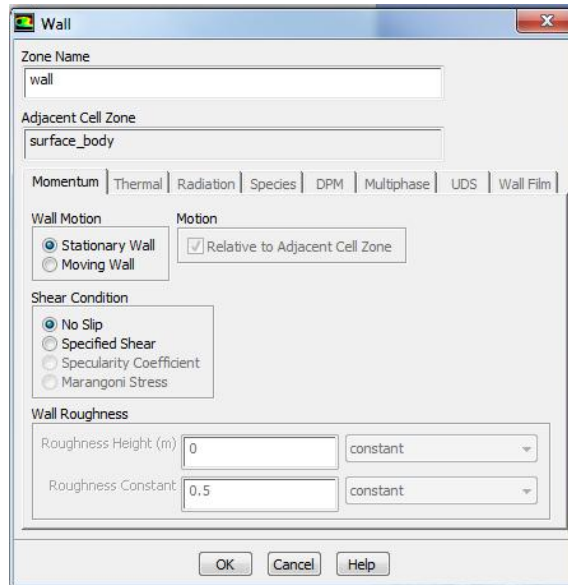
Turbulent flow



The 'Pressure Outlet' dialog box is shown with the 'Momentum' tab selected. The 'Zone Name' field contains 'outlet'. The 'Gauge Pressure (pascal)' is set to 0 with a 'constant' dropdown. The 'Backflow Direction Specification Method' is set to 'Normal to Boundary'. The 'Average Pressure Specification' and 'Target Mass Flow Rate' checkboxes are unchecked. The 'Turbulence' section is expanded, showing 'Specification Method' set to 'K and Epsilon'. The 'Backflow Turbulent Kinetic Energy (m2/s2)' is set to 1 with a 'constant' dropdown. The 'Backflow Turbulent Dissipation Rate (m2/s3)' is set to 1 with a 'constant' dropdown. The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

- 6.10. **Setup > Boundary Conditions > wall** (double Click) Change parameters as per below and click **OK**. No need to change for laminar cases.

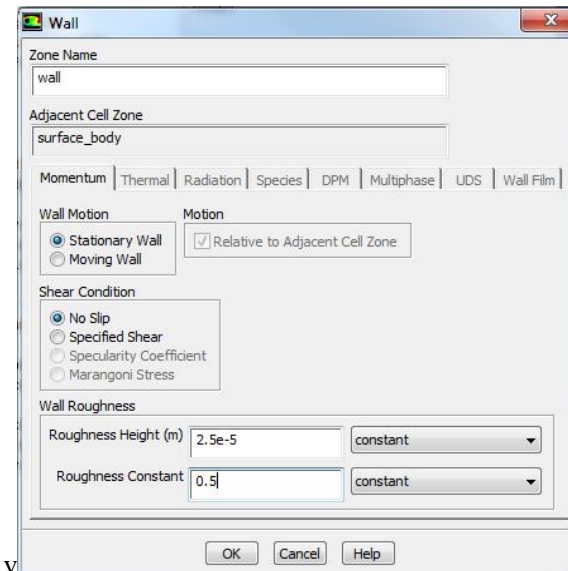
Laminar flow



The screenshot shows the 'Wall' dialog box with the following settings:

- Zone Name:** wall
- Adjacent Cell Zone:** surface_body
- Momentum:** Thermal | Radiation | Species | DPM | Multiphase | UDS | Wall Film
- Wall Motion:** Stationary Wall (selected), Moving Wall
- Motion:** Relative to Adjacent Cell Zone (checked)
- Shear Condition:** No Slip (selected), Specified Shear, Specularity Coefficient, Marangoni Stress
- Wall Roughness:**
 - Roughness Height (m): 0, constant
 - Roughness Constant: 0.5, constant
- Buttons:** OK, Cancel, Help

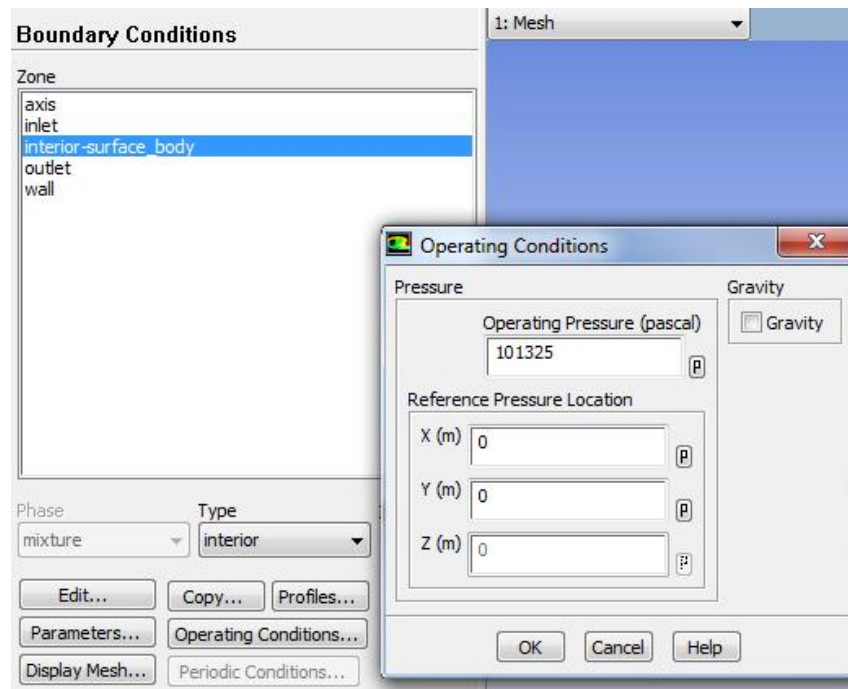
Turbulent flow



The screenshot shows the 'Wall' dialog box with the following settings:

- Zone Name:** wall
- Adjacent Cell Zone:** surface_body
- Momentum:** Thermal | Radiation | Species | DPM | Multiphase | UDS | Wall Film
- Wall Motion:** Stationary Wall (selected), Moving Wall
- Motion:** Relative to Adjacent Cell Zone (checked)
- Shear Condition:** No Slip (selected), Specified Shear, Specularity Coefficient, Marangoni Stress
- Wall Roughness:**
 - Roughness Height (m): 2.5e-5, constant
 - Roughness Constant: 0.5, constant
- Buttons:** OK, Cancel, Help

- 6.11. **Setup > Boundary Conditions > Operating Condition....** Check the operating pressure (it should be equal to 1 atmosphere) and click **OK**.



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

Laminar flow

Reference Values	
Compute from	<input type="text"/>
Reference Values	
Area (m ²)	<input type="text" value="1"/>
Density (kg/m ³)	<input type="text" value="1.17"/>
Enthalpy (j/kg)	<input type="text" value="0"/>
Length (m)	<input type="text" value="0.04"/>
Pressure (pascal)	<input type="text" value="0"/>
Temperature (k)	<input type="text" value="300"/>
Velocity (m/s)	<input type="text" value="0.3"/>
Viscosity (kg/m-s)	<input type="text" value="1.872e-05"/>
Ratio of Specific Heats	<input type="text" value="1.4"/>

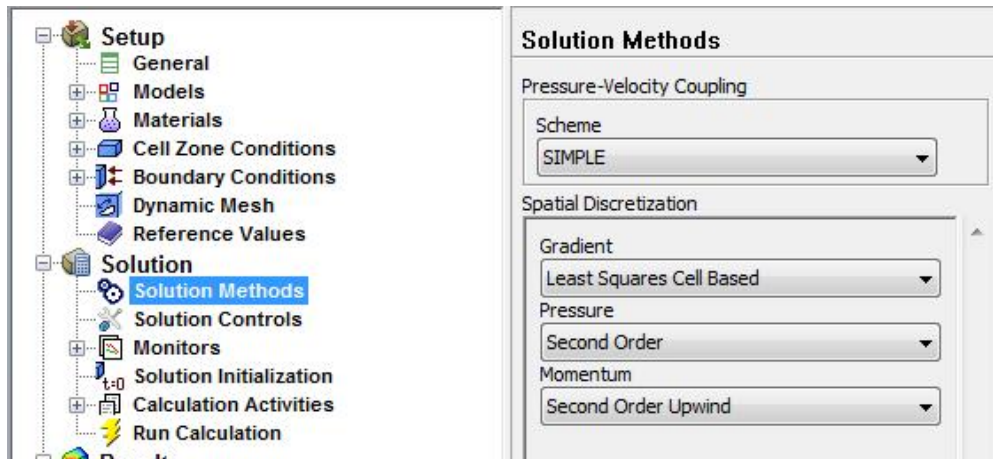
!!! The velocity is different for different Reynolds numbers

Turbulent flow

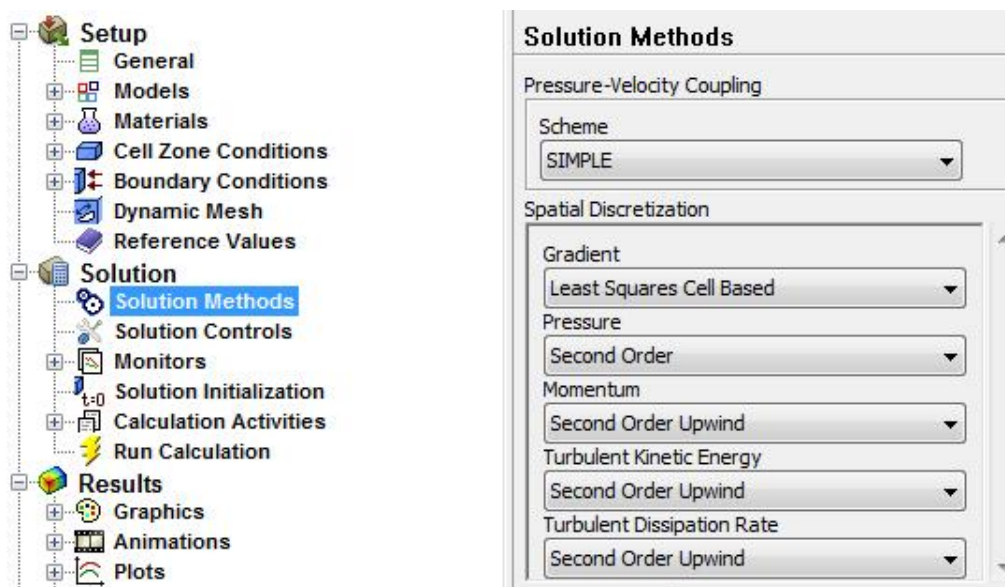
Reference Values	
Area (m ²)	<input type="text" value="1"/>
Density (kg/m ³)	<input type="text" value="1.17"/>
Enthalpy (j/kg)	<input type="text" value="0"/>
Length (m)	<input type="text" value="0.04"/>
Pressure (pascal)	<input type="text" value="0"/>
Temperature (k)	<input type="text" value="300"/>
Velocity (m/s)	<input type="text" value="40"/>
Viscosity (kg/m-s)	<input type="text" value="1.872e-05"/>
Ratio of Specific Heats	<input type="text" value="1.4"/>

- 6.13. **Solution > Solution Methods.** Check the parameters are set as on the figure below.

Laminar flow

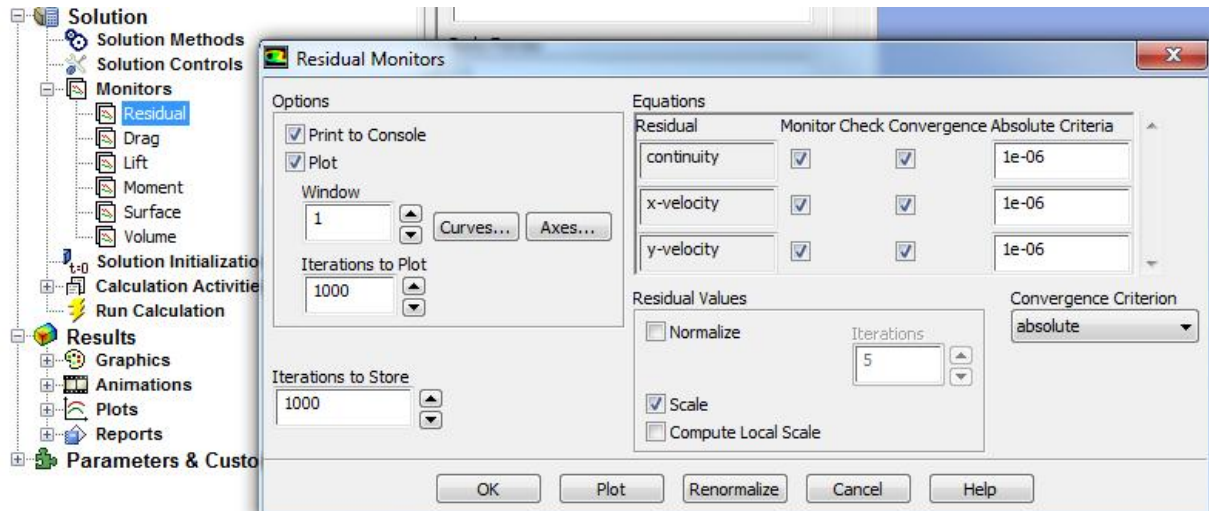


Turbulent flow

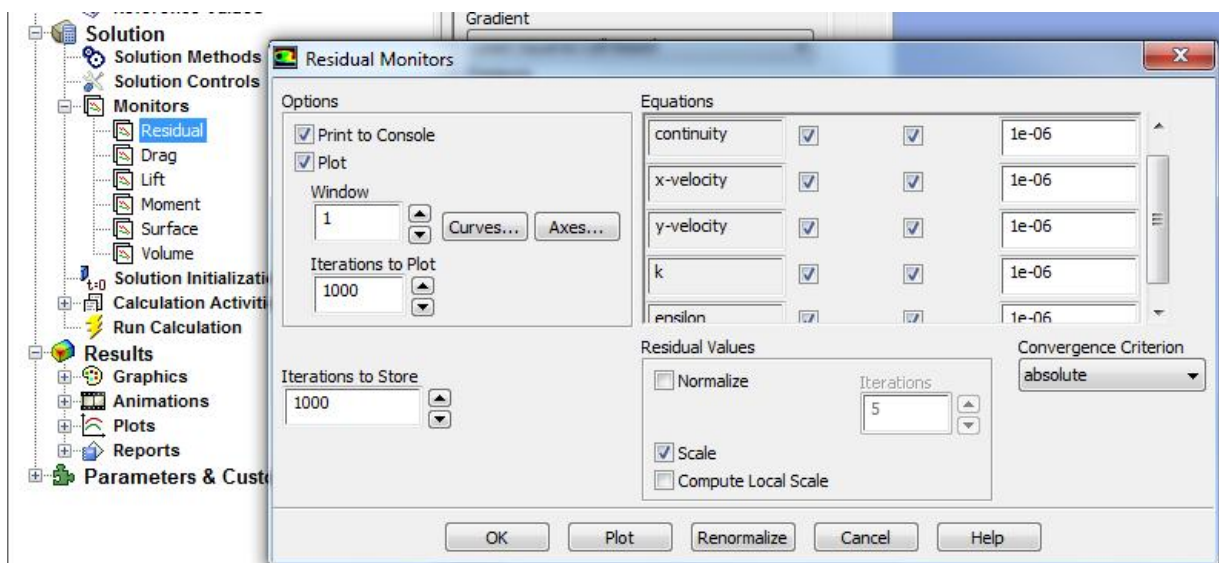


- 6.14. **Solution > Monitors > Residual.** Change convergence criterion to **1e-6** for all three and five equations as per below for laminar and turbulent cases respectively and click **OK**.

Laminar flow

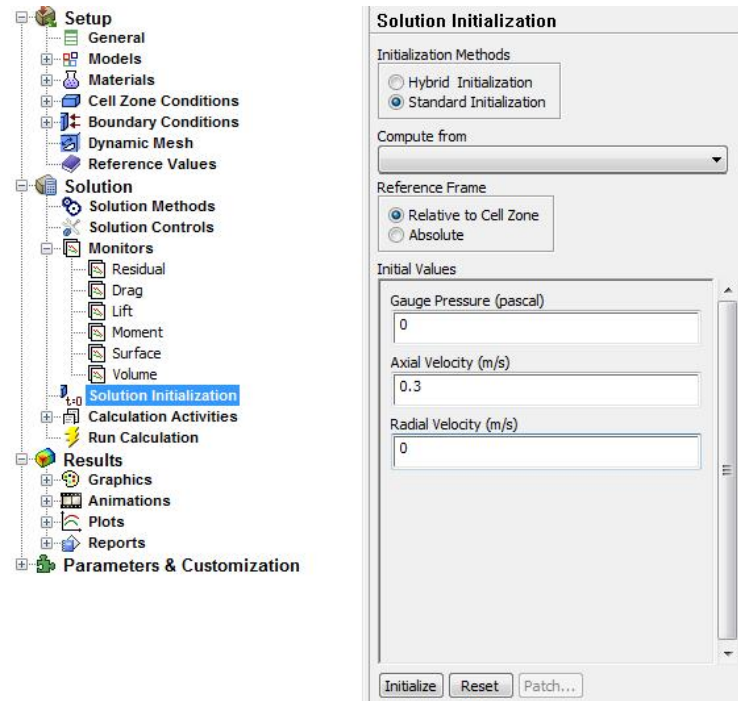


Turbulent flow

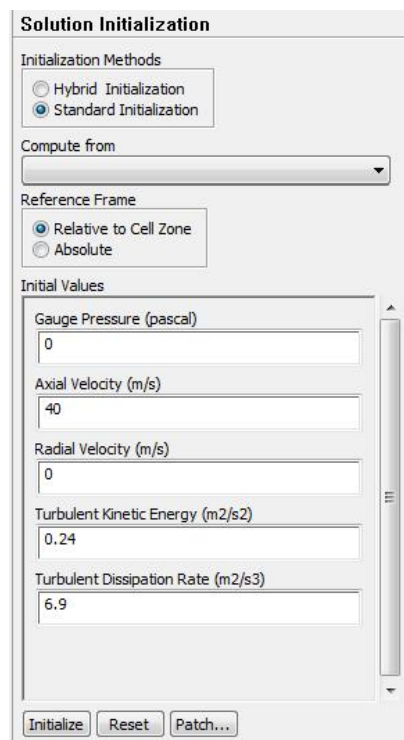


- 6.15. **Solution > Solution Initialization.** Change parameters as per below and click **Initialize.**

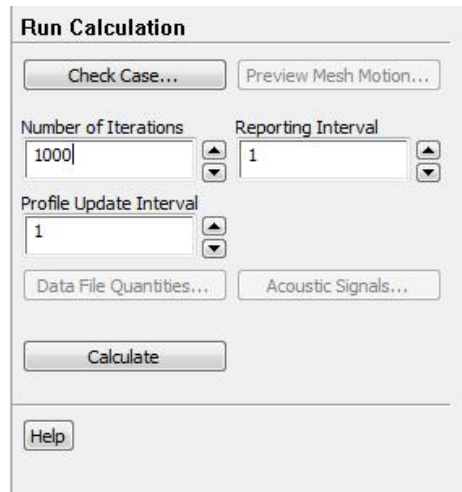
Laminar flow



Turbulent flow



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



The image shows a 'Run Calculation' dialog box with the following elements:

- Check Case...** and **Preview Mesh Motion...** buttons at the top.
- Number of Iterations** set to 1000 with up/down arrows.
- Reporting Interval** set to 1 with up/down arrows.
- Profile Update Interval** set to 1 with up/down arrows.
- Data File Quantities...** and **Acoustic Signals...** buttons.
- A large **Calculate** button.
- A **Help** button at the bottom.

- 6.17. **File > save project.**

7. Results post-processing

After solution is converged (with convergence limit 10^{-6}) you need to do postprocessing. The following figures and data should be included into the report. In addition, the answers on the following questions should be given.

Pictures to be presented for laminar and turbulent flows:

- Convergence history (Residuals)
- Axial velocity profile in different sections. + Comparison with analytical/empirical data.
- Centerline pressure distribution
- Centerline velocity distribution
- Velocity vectors showing the developing region and developed regions

Data to be reported:

- Developing length. + Comparison with empirical data.
- Pressure drop along the pipe. + Comparison with empirical data.

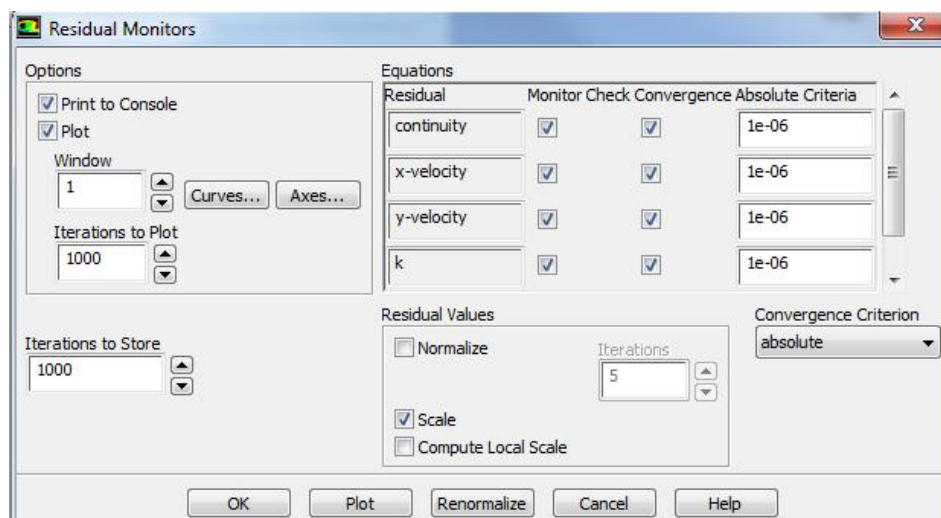
Questions to be answered:

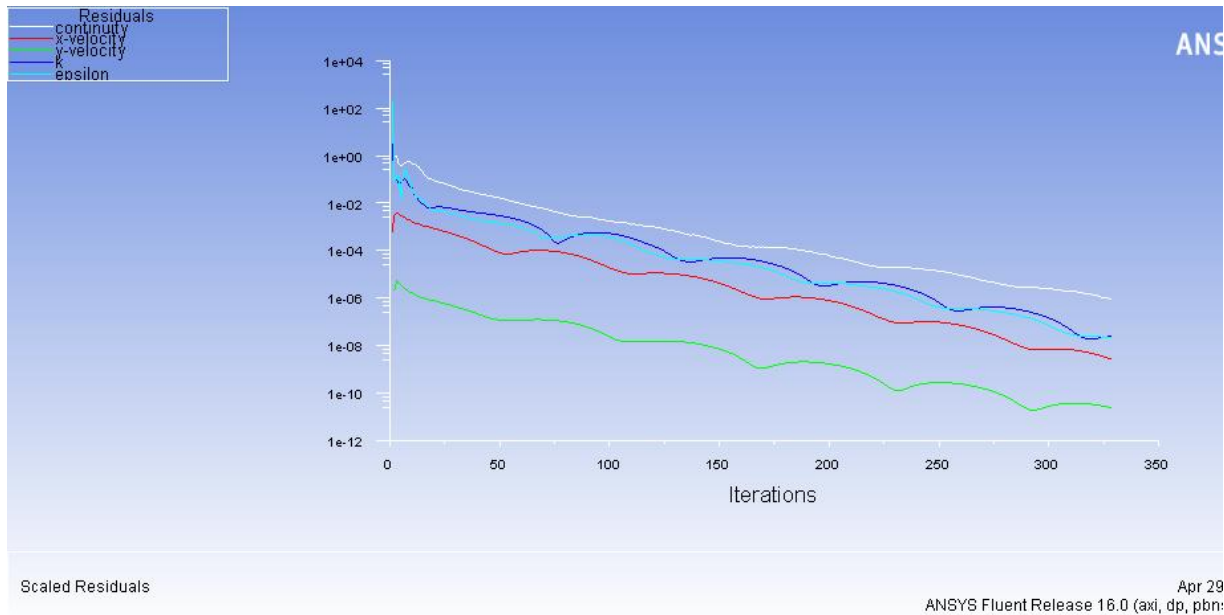
- Compare the results of laminar pipe flow obtained for fine and coarse mesh.
- Analyze the difference between laminar flow (obtained by using fine mesh) and turbulent flow.
- Analyze the difference between simulation and empirical data for laminar and turbulent flow and possible sources of errors.

This section shows how to prepare the figures and perform calculations.

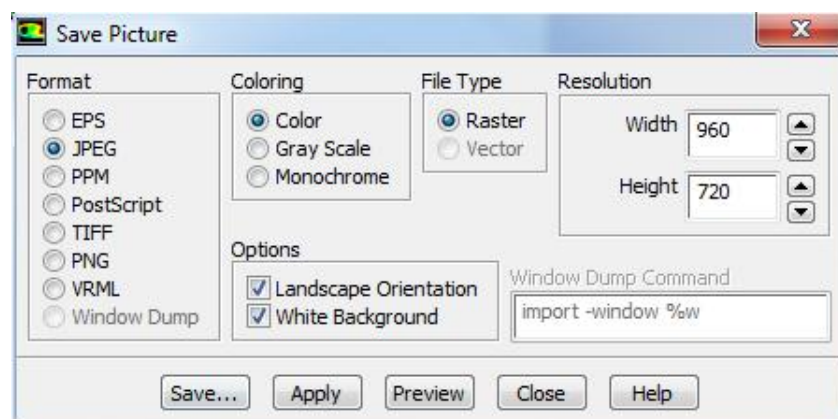
7.1. Plotting and Printing Residuals

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





File > Save Picture. Using option as per below save pictures.



7.2. Creating a File with Analytical/Empirical data

Create a text file using **Notepad**. The text file should have the following structure

```
title "Velocity Magnitude")
(labels "Position" "Velocity Magnitude")
```

```
((xy/key/label "analytical/empirical")
0      velocity value
0.0025 velocity value
0.005  velocity value
0.0075 velocity value
0.01   velocity value
0.015  velocity value
0.016  velocity value
0.017  velocity value
0.018  velocity value
0.019  velocity value
0.02   velocity value
)
```

The following formulas can be used to calculate the velocity profile.

For laminar flow:

$$v = v_{max} \cdot \left(1 - \frac{r^2}{r_0^2}\right), \quad v_{max} = 2 \cdot v_{inlet}$$

For turbulent flow:

$$v = v_{max} \cdot \left(1 - \frac{r}{r_0}\right)^{1/7}, \quad v_{max} = 1.23 \cdot v_{inlet}$$

Save the file. Give a name “axial_vel_empirical.xy”

This file will be loaded later and used for verification of numerical results.

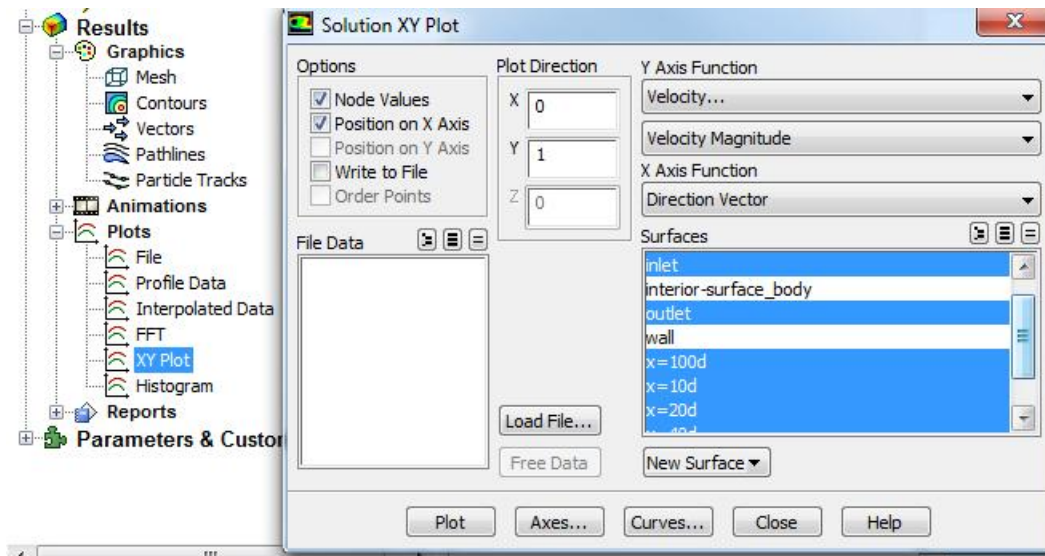
Surface > Line/Rake. Change x and y values as per below click **Create**. Do this for all the lines shown in the table below.

Surface Name	X0	Y0	X1	Y1
x=10d	0.4	0	0.4	0.02
x=20d	0.8	0	0.8	0.02
x=40d	1.6	0	1.6	0.02
x=60d	2.4	0	2.4	0.02
x=100d	4	0	4	0.02

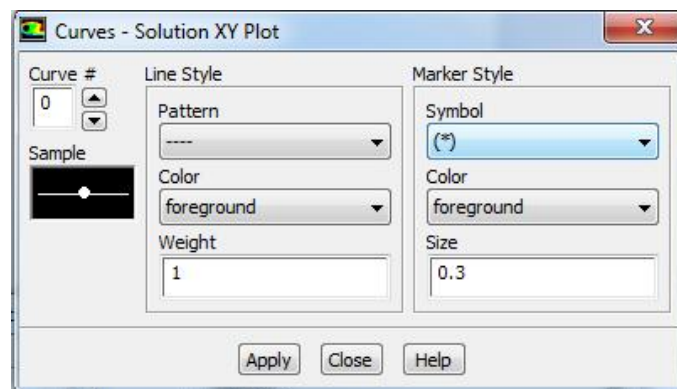
7.3. Plotting Results

Velocity Profile

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

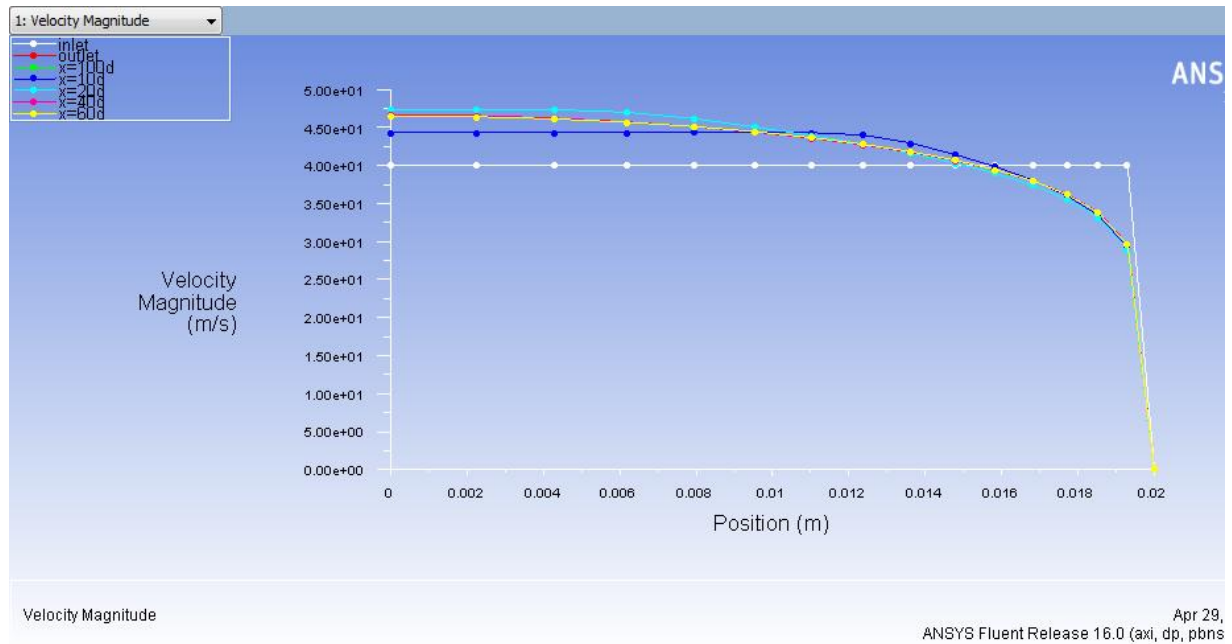


Results > XY Plot > Setup > Curves.... For Curve # 0 select the **Pattern** as per below and click **Apply**. Repeat this for all the curves.



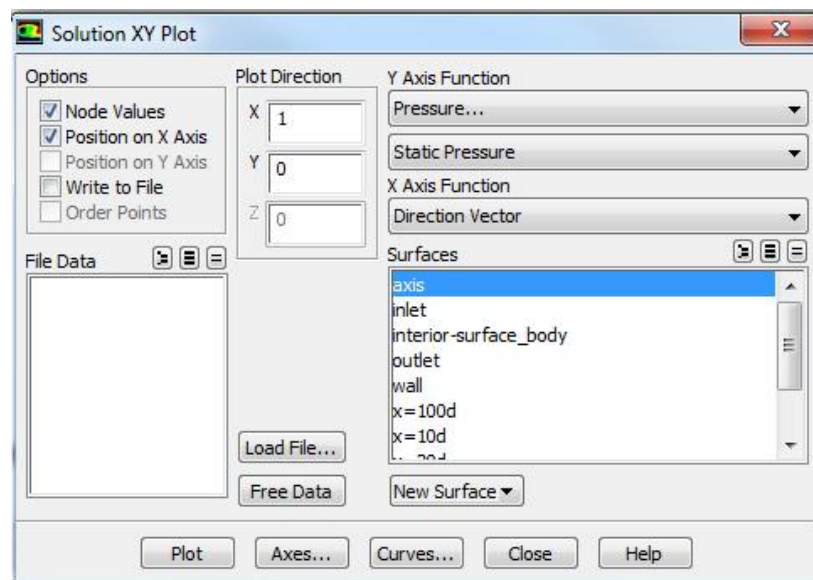
Download the file with empirical data.

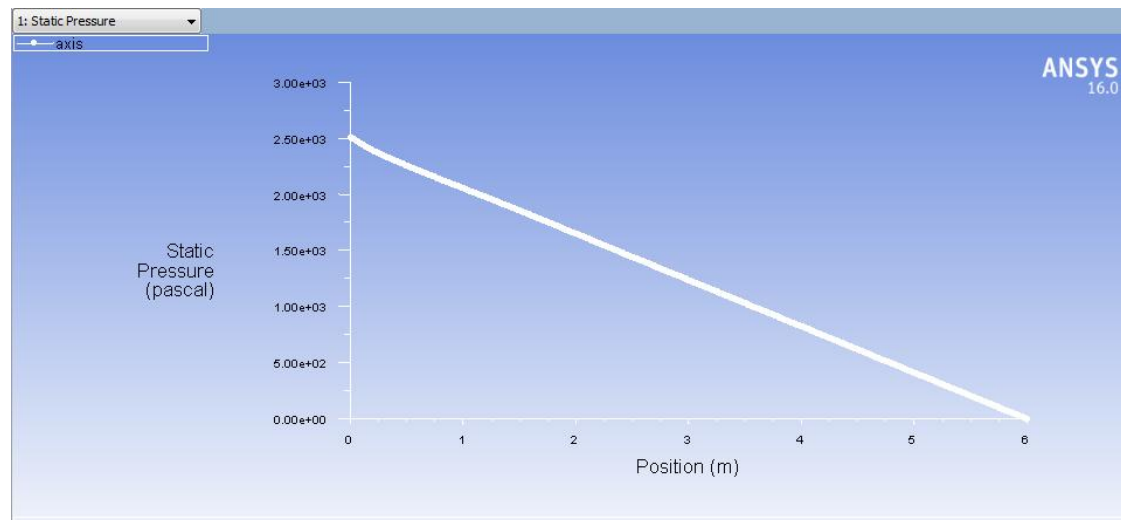
Results > XY Plot > Setup > Load File. Select axial_vel_empirical.xy and click **OK**.



Static Pressure Profile at Centerline

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





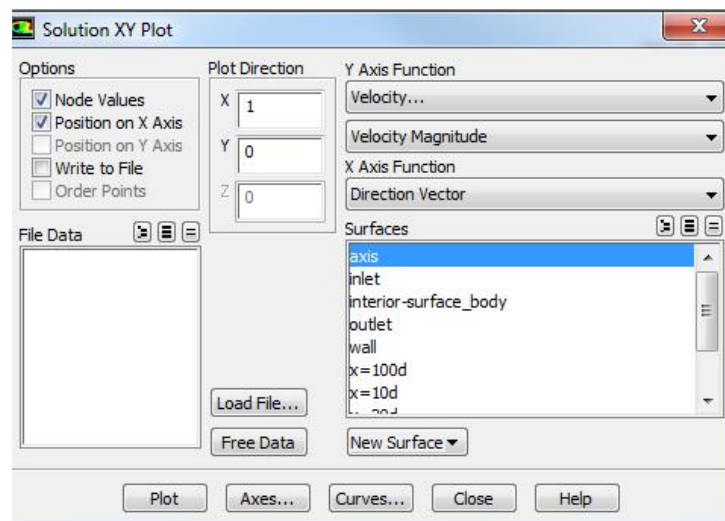
! Compare the pressure drop with Darcy-Weisbach Equation

$$\Delta p = \lambda \frac{l}{d} \frac{\rho v^2}{2}$$

Here λ – friction factor

Velocity at Centerline

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



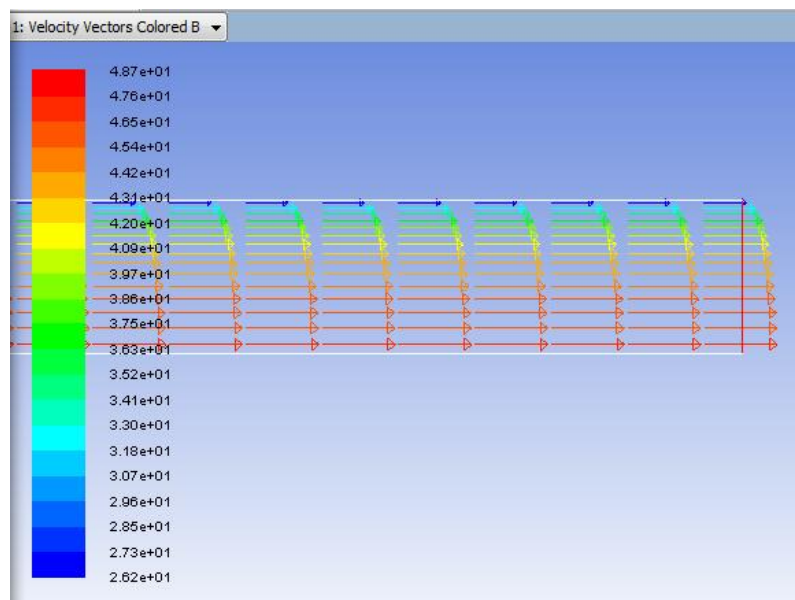
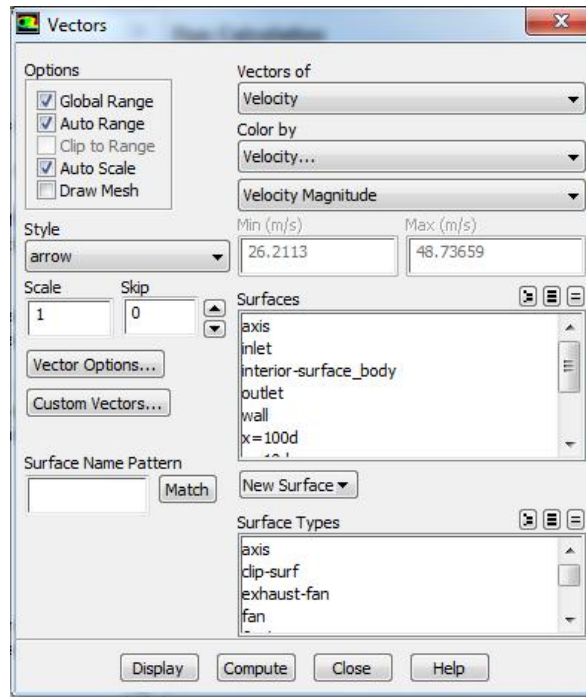
Examine the graph and try to evaluate the value of **entrance length**. Compare the results with empirical calculations.

For laminar flow $L_{entrance} \cong 0.06 \cdot Re \cdot d$

For turbulent flow $L_{entrance} \cong 4.4 \cdot Re^{1/6} \cdot d$

7.4. Plotting Vectors and Contours

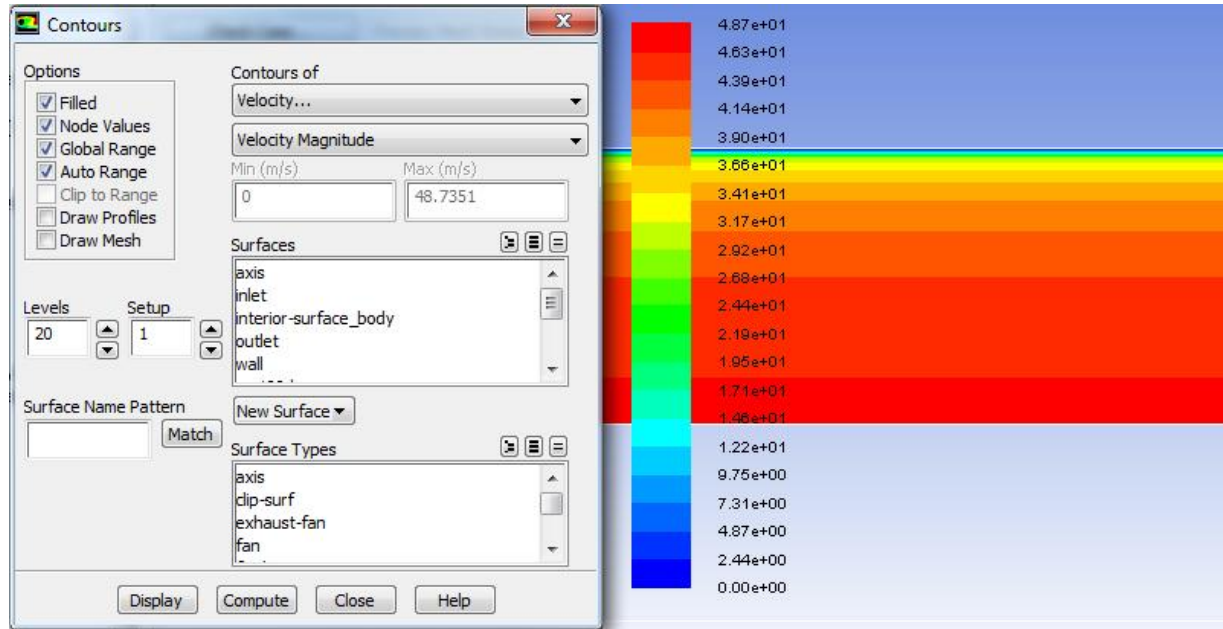
Results > Graphics > Vectors (double click). Change the vector parameters as per below and click **Display**.



! To show vectors in the entrance region choose the area near the inlet of the pipe

! To show vectors in the developed region choose the area near the end of the pipe

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



Close window and save workbench file.