Indian Institute of Technology Gandhinagar

ME 207

FLUID DYNAMICS

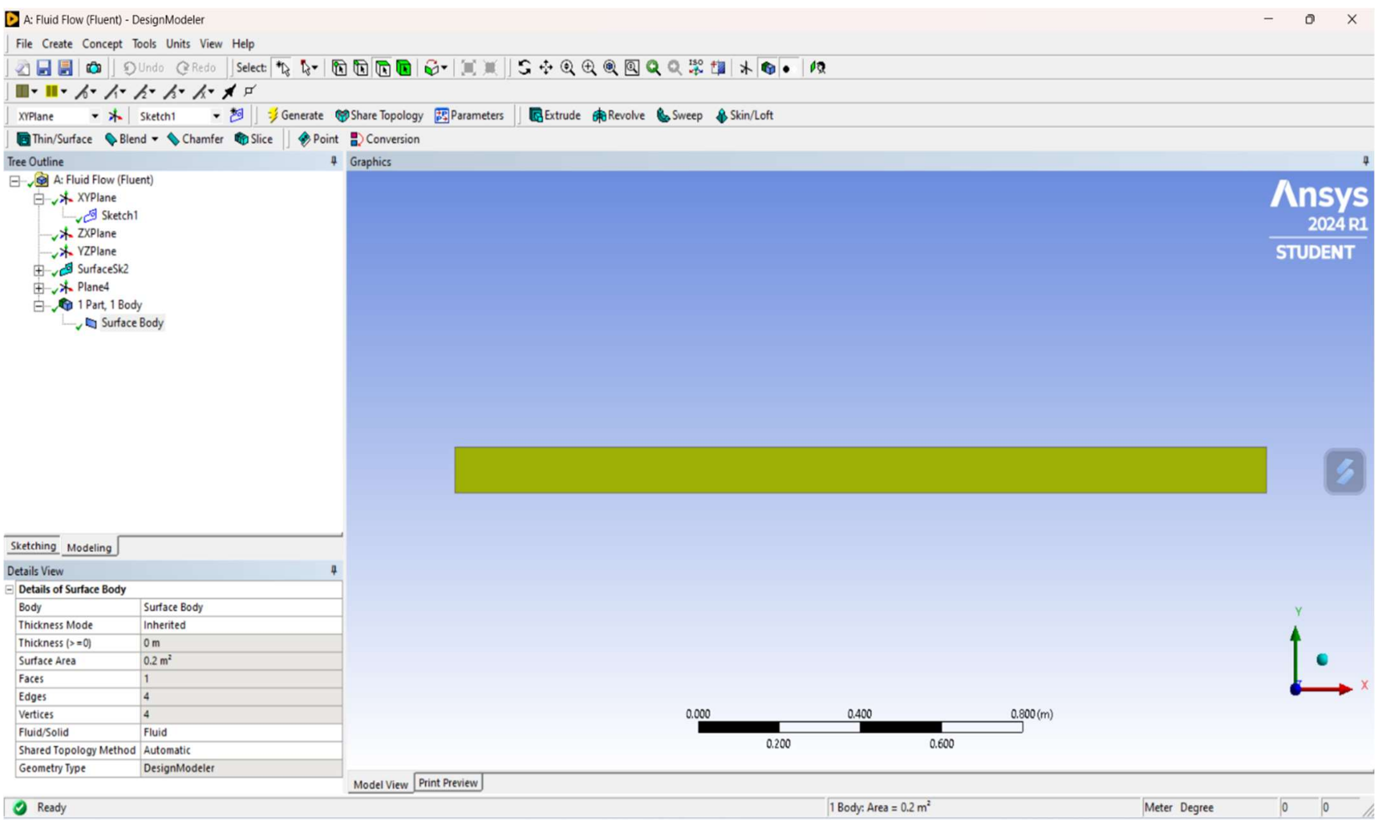
Computational Fluid Dynamics Lab-1 Report
Roll No. :-22110276

April 3, 2024

Mentored by-
Prof. Dilip Srinivas Sundaram

1) <u>Geometry and Boundary conditions</u>:

- The geometrical design includes the pipe with diameter of 20 cm and length of 2 m.
- Therefore, sketch a rectangle of 2m and 0.1m as its length and width respectively.
- Go to the New DesignModeller by a right click on Geometry section, and create a random rectangle from the sketching section.
- Now, give the above dimensions to the rectangle and create a surface from the concepts section.
- Change the type to fluid in the details of the surface.
- The density and viscosity of the crude oil can be taken as 860 kg/m3 and 17.2 cP.
- The below is the final surface produced:

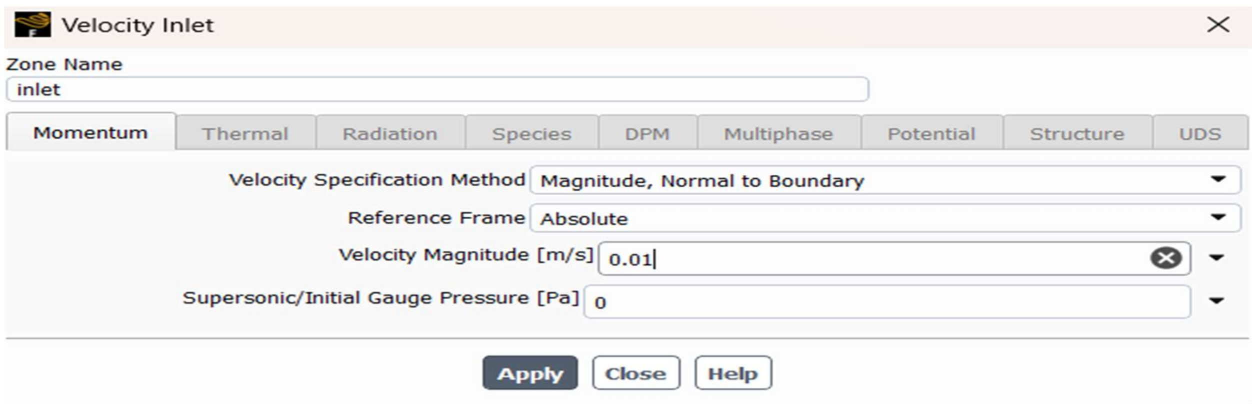


Boundary Conditions:-

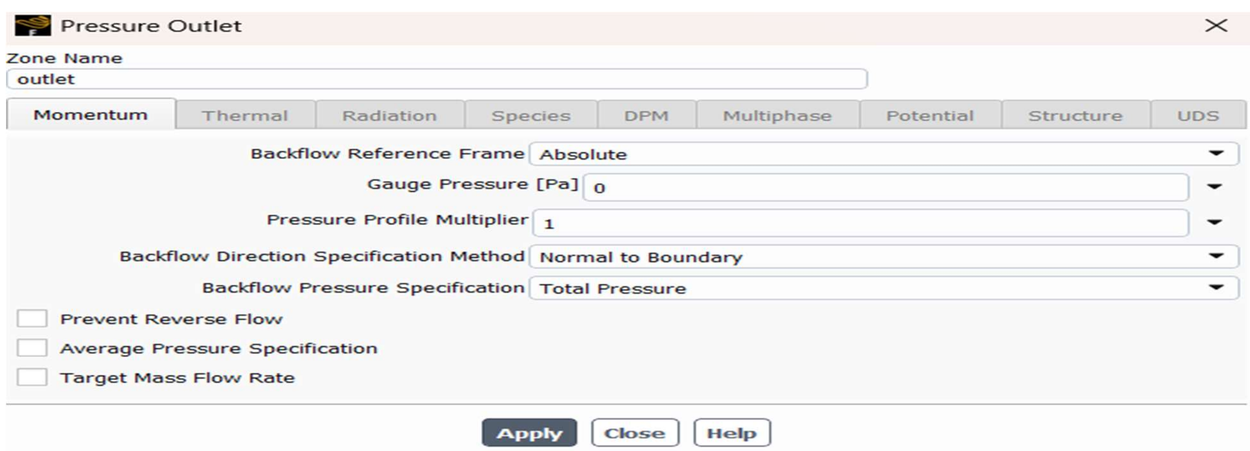
I) Name the four sides of the rectangle to be inlet, wall, axis, outlet.

II) We have to ensure that the type of inlet is Velocity-inlet and that of outlet is Pressure-outlet.

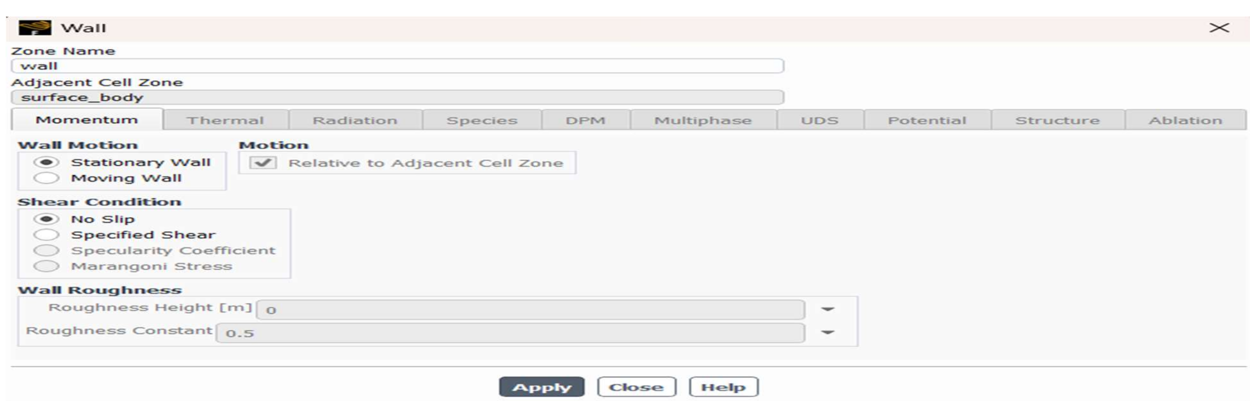
III) Edit the inlet properties in the setup section, and give the respective inlet velocity.



IV) Edit the outlet properties in the setup section, and give the outlet gage pressure to be zero.



V) Also, ensure that the shear condition for the wall is no-slip.



2) Mesh statistics and Mesh independence study:

- Ensure that the element type in the mesh is quadrilateral.
- We also have to analyze that the density distribution along the mesh is throughout.
- The skewness and aspect ratio in the mesh should not be very high, as high skewness or high aspect ratio in the mesh indicates high distortion in its elements.
- The below is a table representing the nodes and elements corresponding to the given element size:

Element size	Nodes	Elements
0.005	8421	8000
0.007	4305	4004
0.01	2111	2000

Mesh Independence:

- Mesh independence is the condition where the results stop changing with the further refinement in the mesh.
- We may create velocity profiles at different element sizes and compare them. (plots are there in Q-3 of results and discussion section)
- On comparing the plots, it can be seen that for different mesh sizes the plot is remaining comparatively same.

3) Discretisation schemes and solution methodology:

Some of the Discretisation schemes we used are:

- We used finite volume method to discretize the governing equations of fluid flow. The computational domain is divided into a finite number of control volumes, and the governing equations are integrated over these volumes to obtain discretized forms.

- As it is a laminar flow, implicit methods are commonly used due to their stability and ability to handle stiffness associated with viscous terms.

- Also, due to having Laminar flow, we typically use the central difference scheme as the Gradient scheme, as it provides second-order accuracy in the calculations.

Solution Methodology:

- We used pressure-based solver as it, iteratively solves for the pressure and velocity fields until convergence is achieved.

- It uses linear equation solvers, such as the Gauss-Seidel method to solve the discretized equations and efficiently handles large sparse linear systems resulting from the finite volume discretization.

- For laminar pipe flow, boundary conditions includes specifying the velocity profile or volumetric flow rate at the inlet and setting either a pressure or velocity boundary condition at the outlet.

By appropriate discretization schemes and solution methodologies, we can accurately simulate laminar pipe flow.

4) Results and Discussion:

Q-1

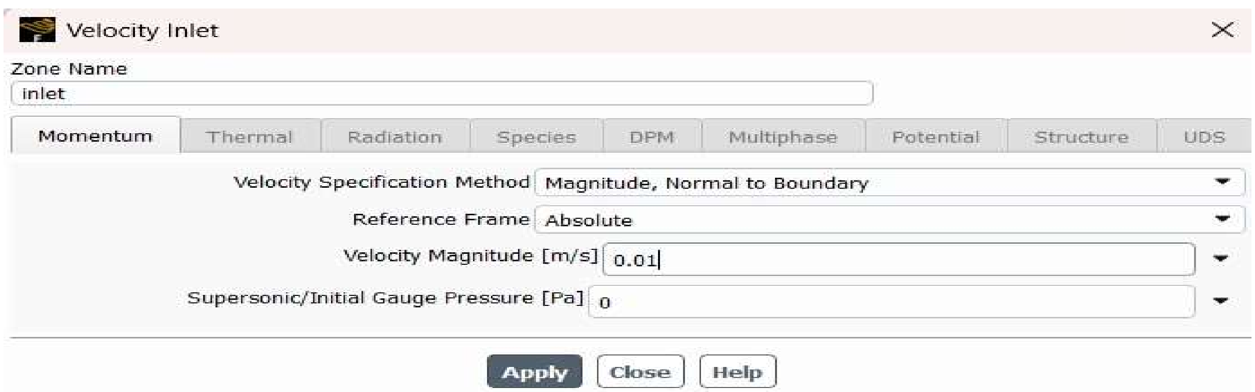
❖ For Re = 100, plot velocity and pressure at the centreline.

$$\mathrm{Re} = \frac{\rho \cdot V \cdot D}{\mu}$$

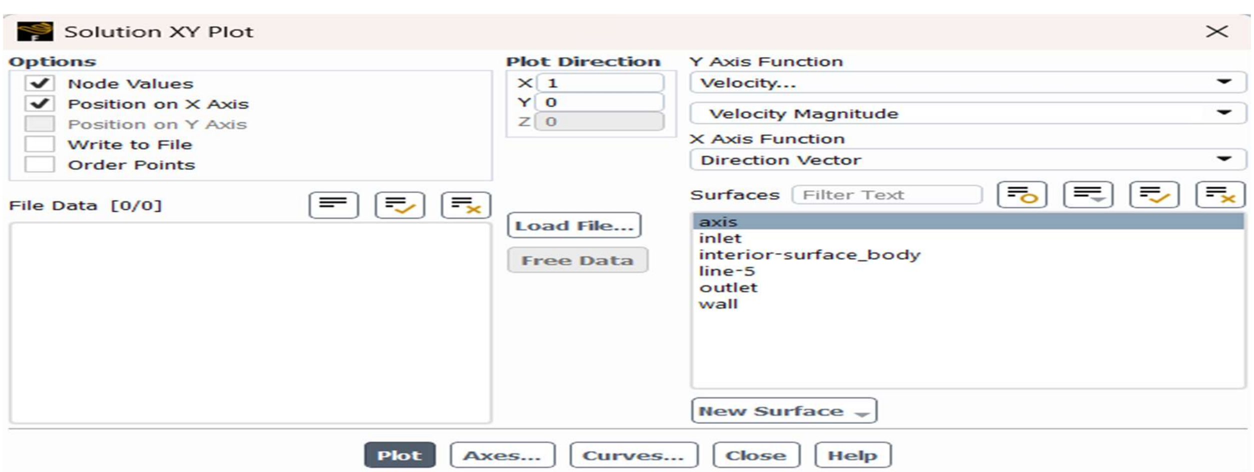
After substituting the values, we get V=0.01m/s

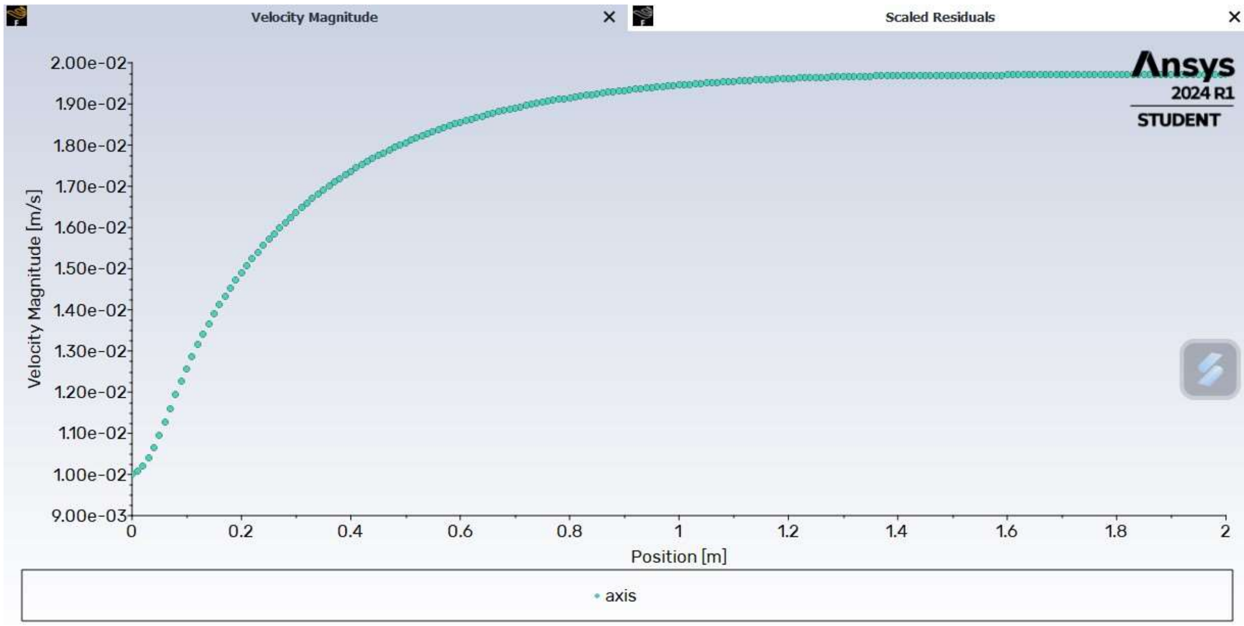
i) Getting the plot of velocity at the centerline (axis):-

Give the input velocity for the inlet in the “Boundary Conditions”



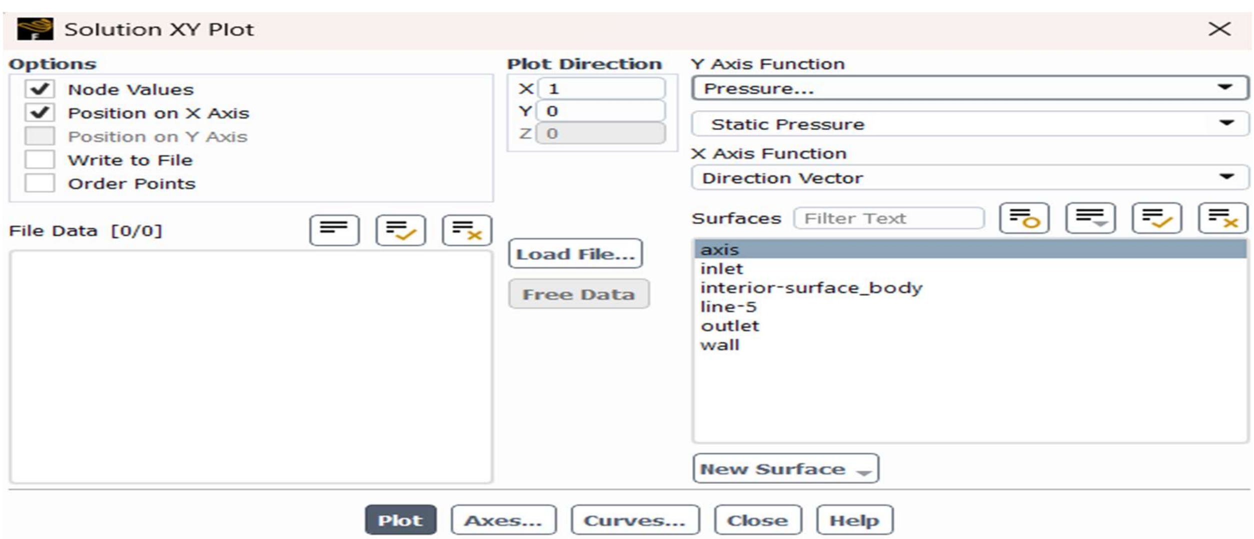
Now plot the velocity curve using XY plot feature as below:

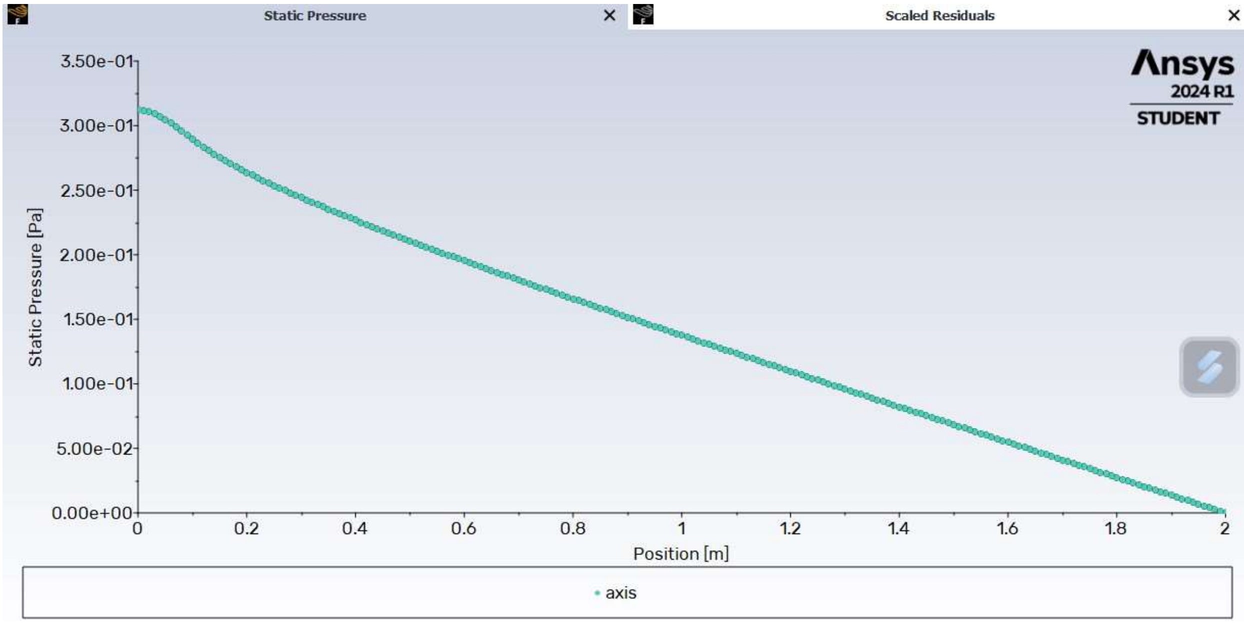




ii) Getting the plot of pressure at the centerline (axis):-

Keeping the input velocity same, plot the pressure curve as below:



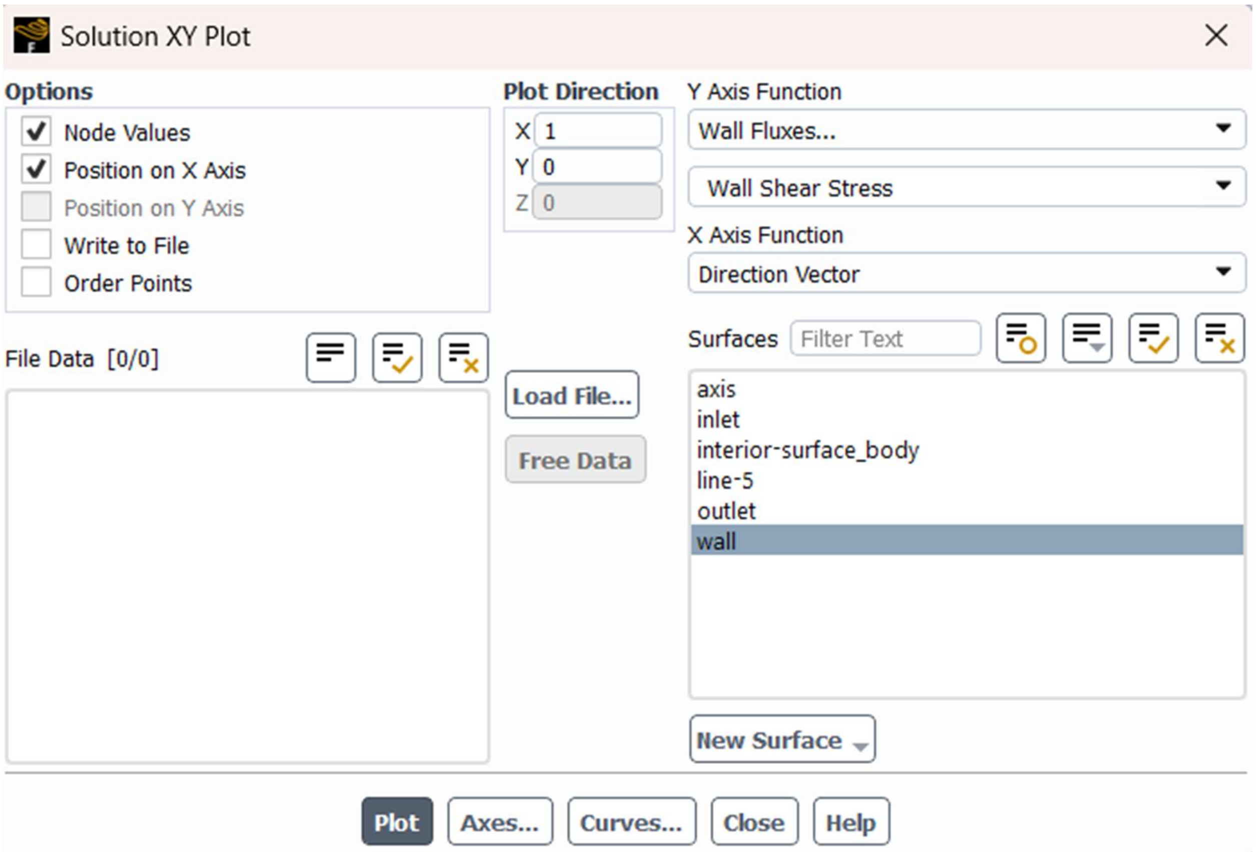


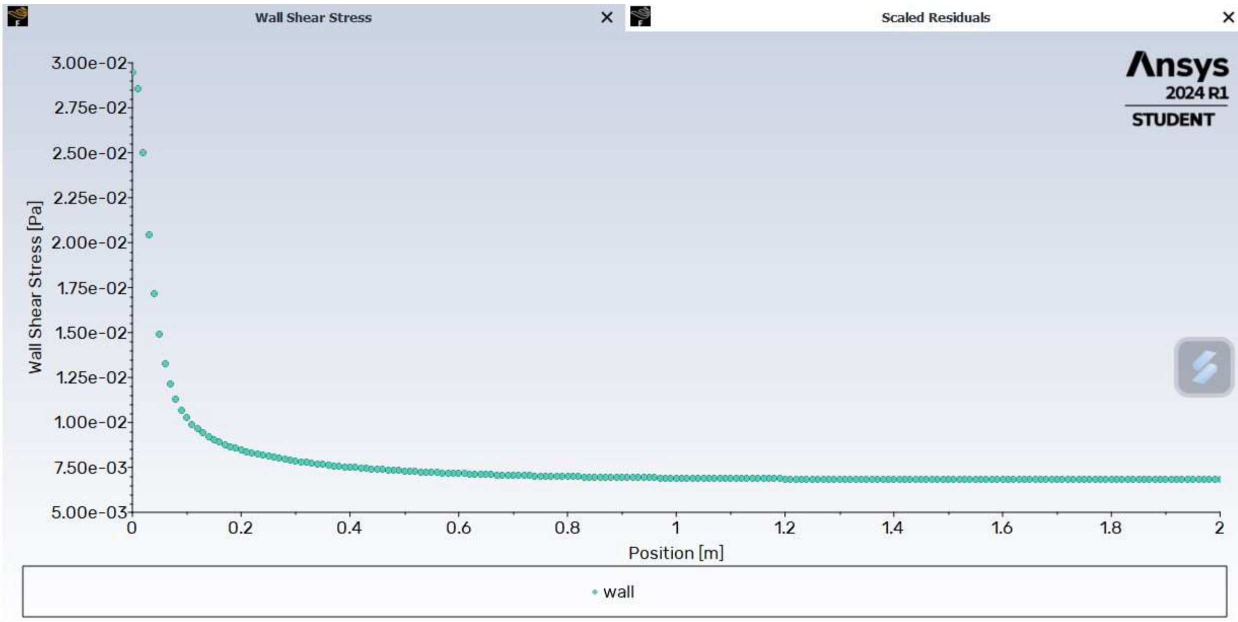
As the fluid flows along the pipe walls, a thin layer called the boundary layer. This boundary layer affects the pressure and velocity distribution within the pipe, contributing to the decreasing in pressure and increase in velocity at the centerline.

Q-2

- We have to calculate the Wall shear stress and Darcy's friction factor along the wall.

i) Plotting the Wall shear stress:





The decrease in wall shear stress along the length of the pipe is due to the decrease in velocity gradient near the wall as the flow progresses downstream.

ii) Calculating the Darcy's Friction factor:

$$Darcy's\ friction\ factor = \frac{\tau \cdot \text{viscosity}}{\text{density} \cdot \text{diameter}}$$

From the plot of wall shear stress, the saturated value of shear stress comes to be 0.0075

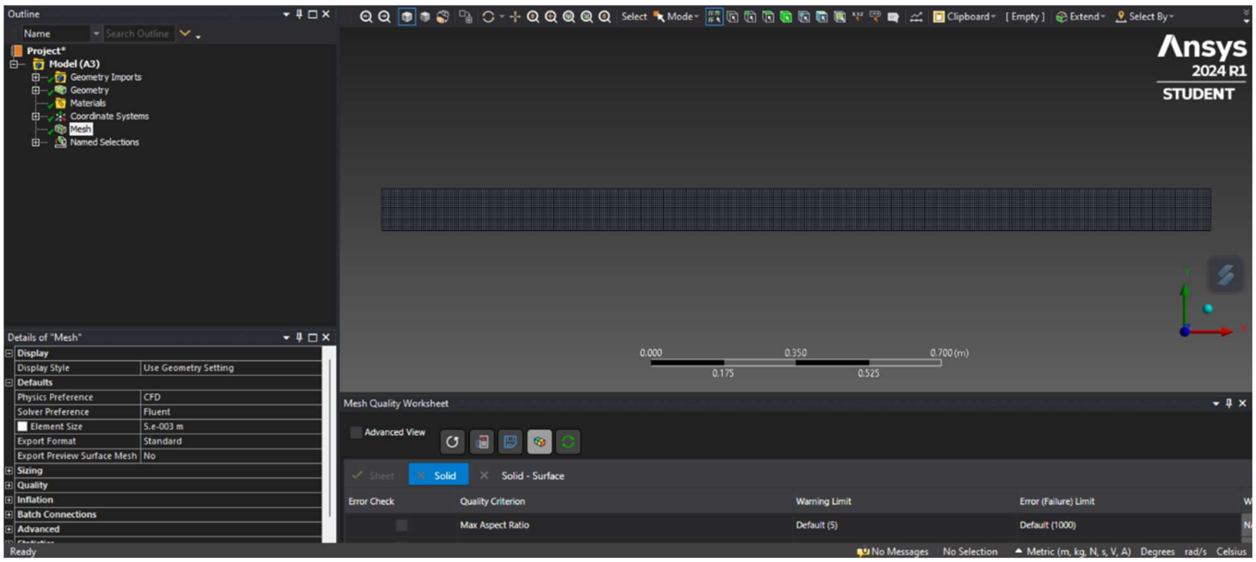
Substituting the values, we get:

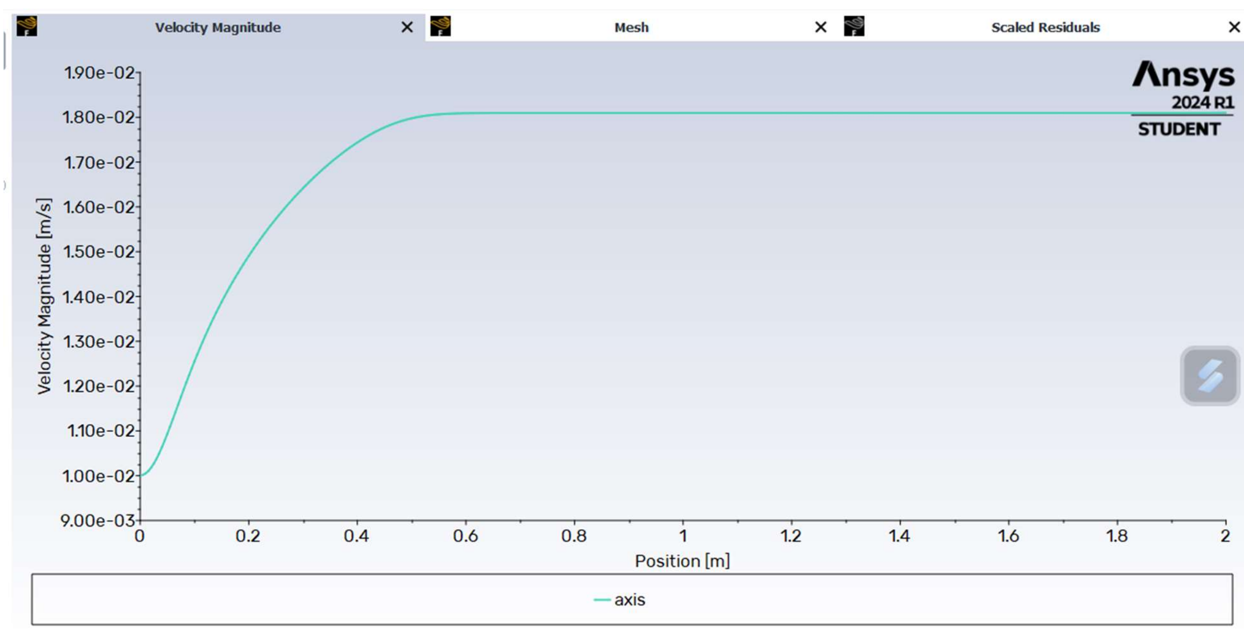
$$\text{Darcy's friction factor} = 0.697$$

Q-3

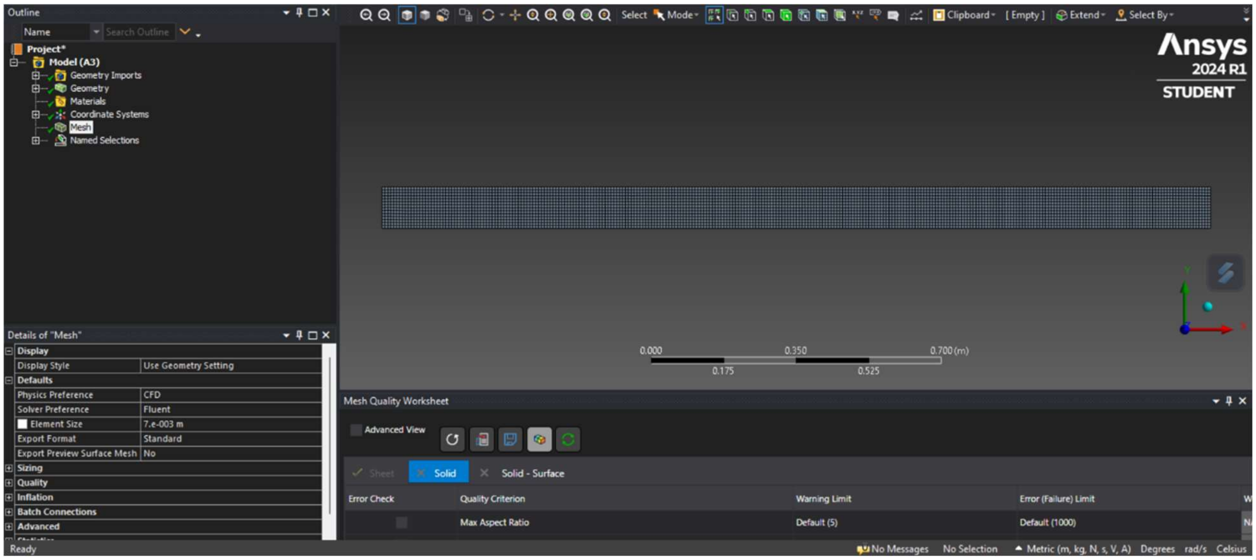
❖ We have to perform mesh independence study at least three levels of mesh refinement to demonstrate the mesh independence

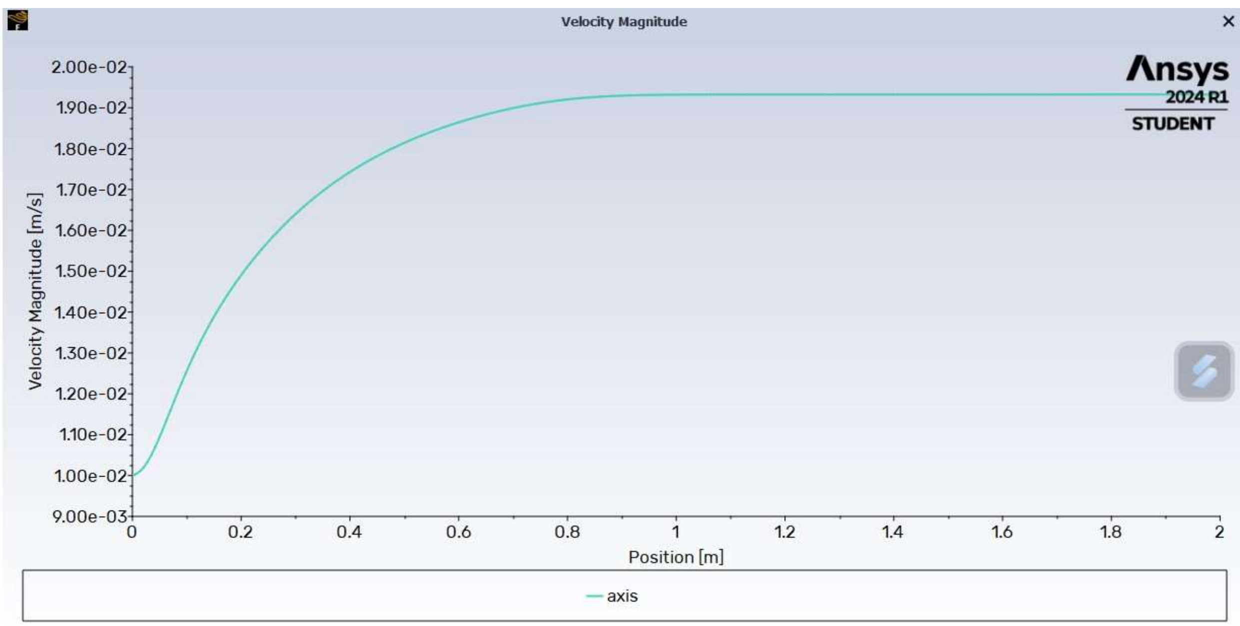
I) Element size= 0.005



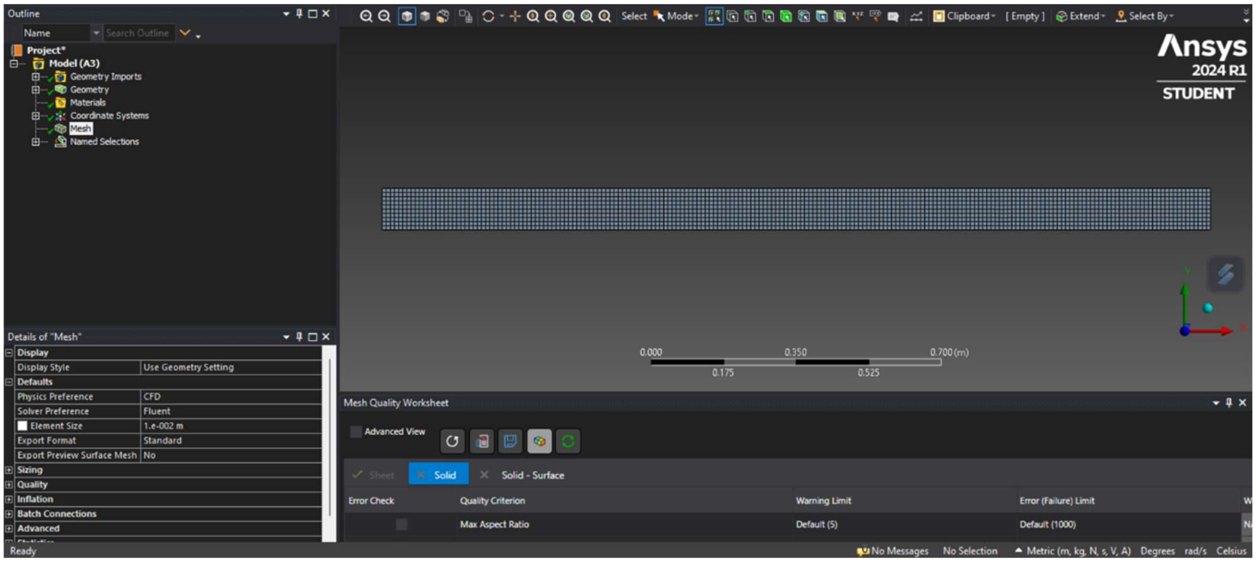


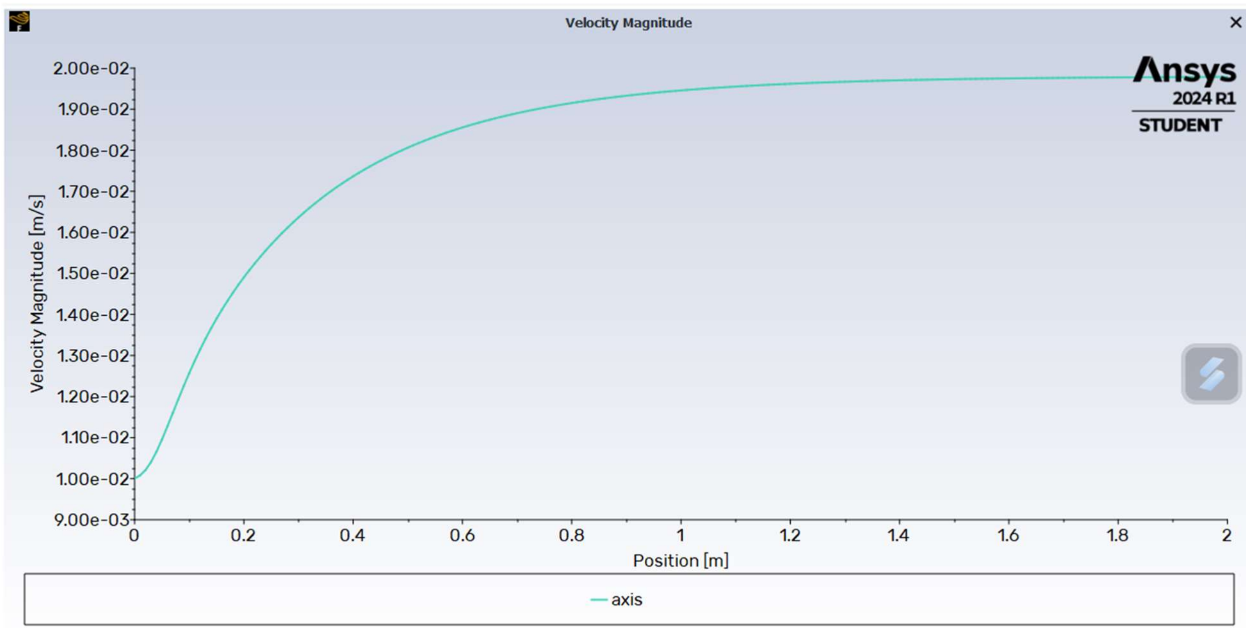
II) Element size= 0.007





III) Element size= 0.01





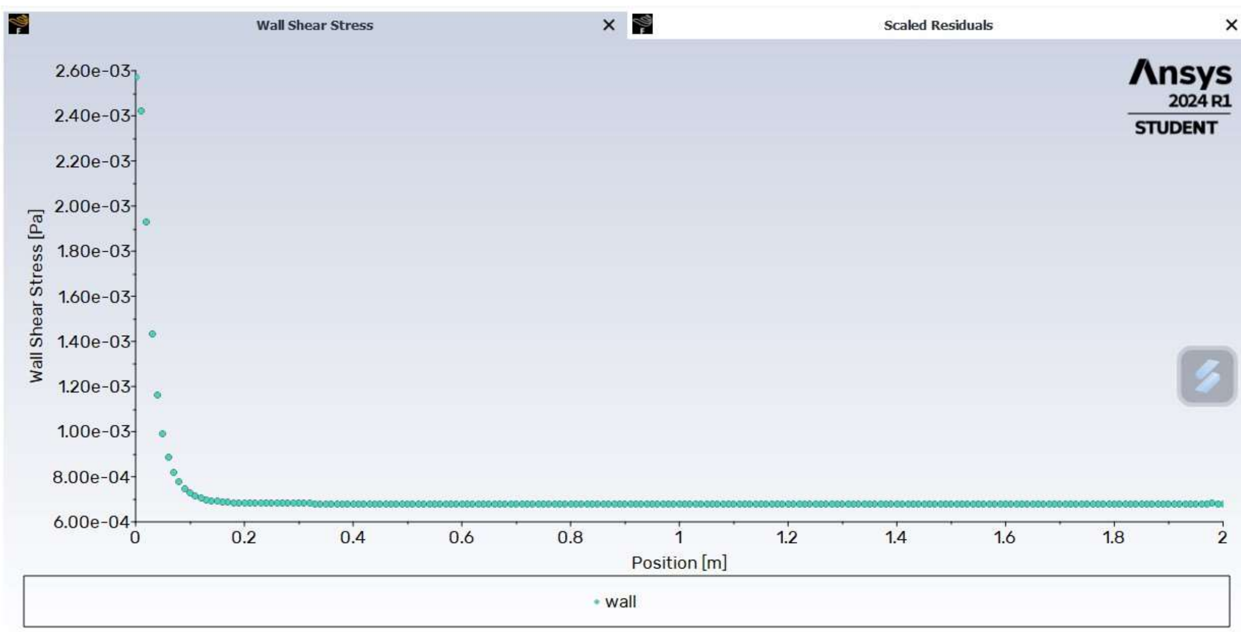
The Mesh Independence happens due to accurate conservation of mass and momentum in the numerical solution.

Q-4

❖ We have to select 5 different Reynolds numbers in the range of 10 - 1000 and plot Darcy's friction factor Vs Reynolds number. The plot should be on a log-log scale.

$$Darcy's\ friction\ factor\ = \frac{\tau\ \cdot\ \text{viscosity}}{\text{density}\ \cdot\ \text{diameter}}$$

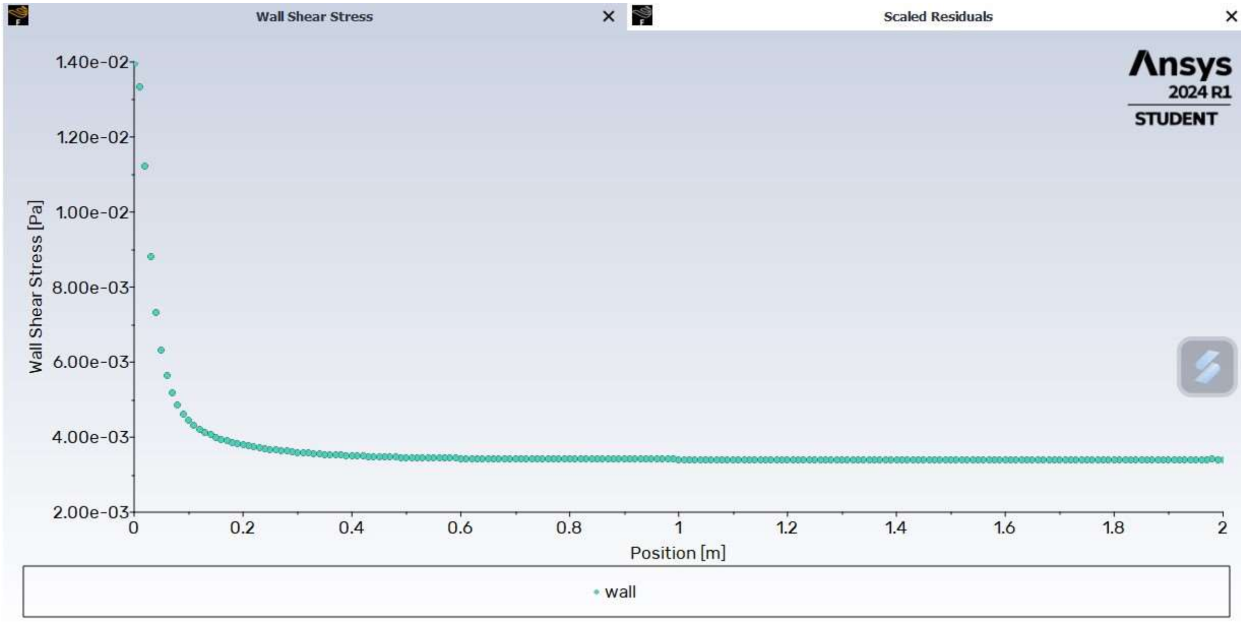
I) Reynold's number $= 10$



From the above plot, the saturated wall shear stress comes out to be 0.0007 Pa

After substituting the values, Darcy's friction factor comes to be 6.511

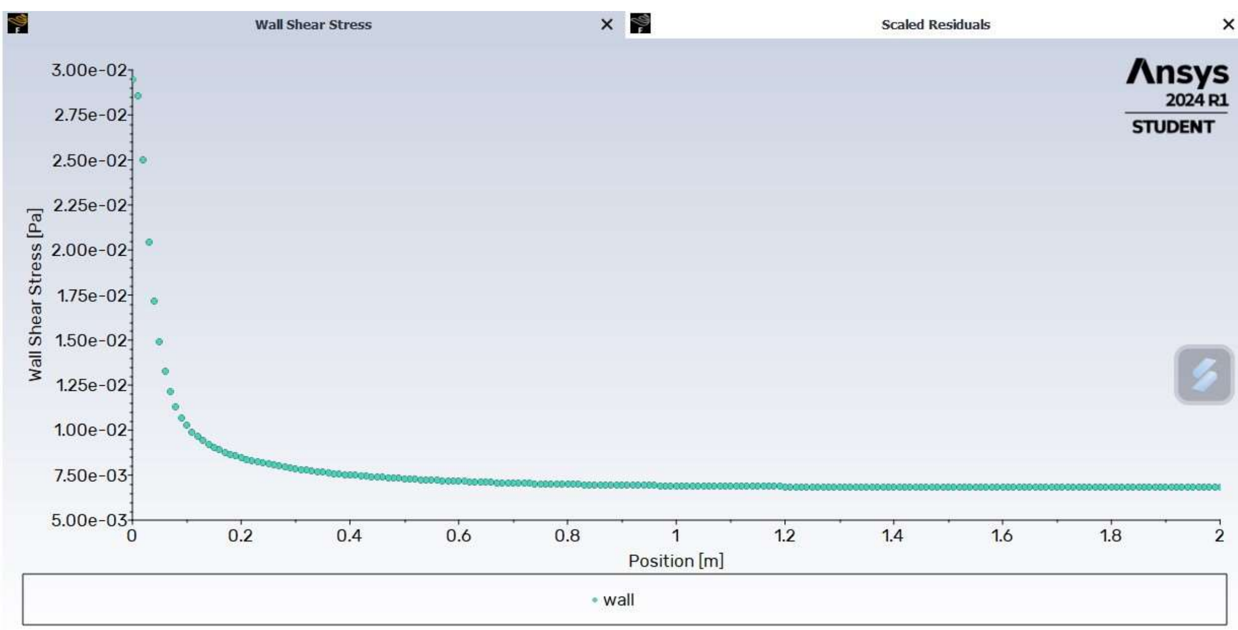
II) Reynold's number $= 50$



From the above plot, the saturated wall shear stress comes out to be 0.004 Pa

After substituting the values, Darcy's friction factor comes to be 1.489

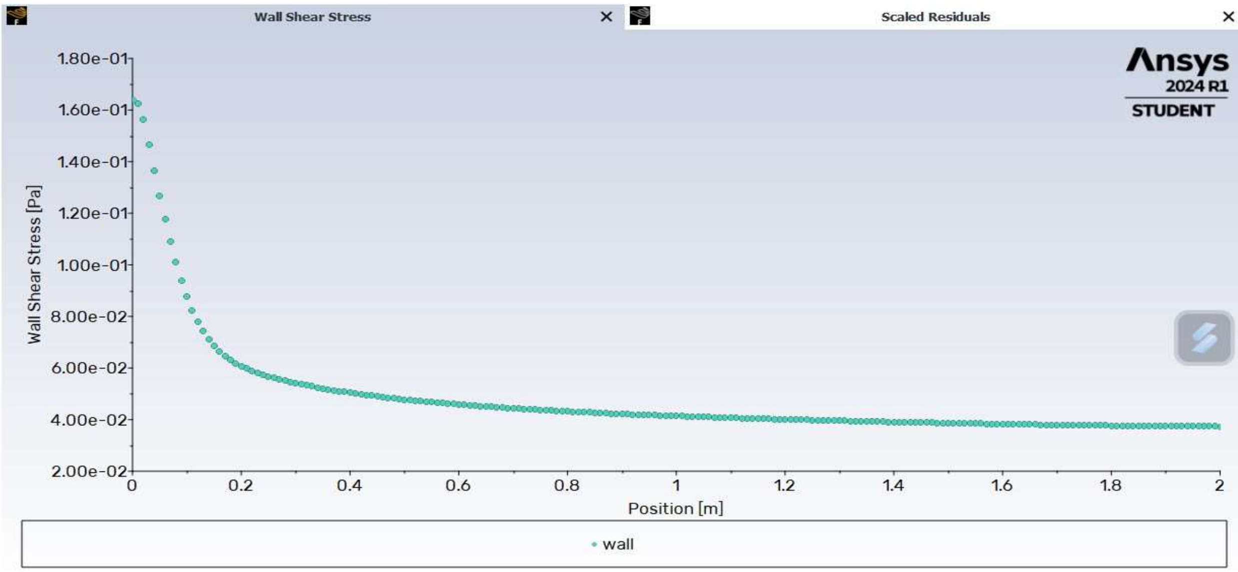
III) Reynold's number $= 100$



From the above plot, the saturated wall shear stress comes out to be 0.0075 Pa

After substituting the values, Darcy's friction factor comes to be 0.697

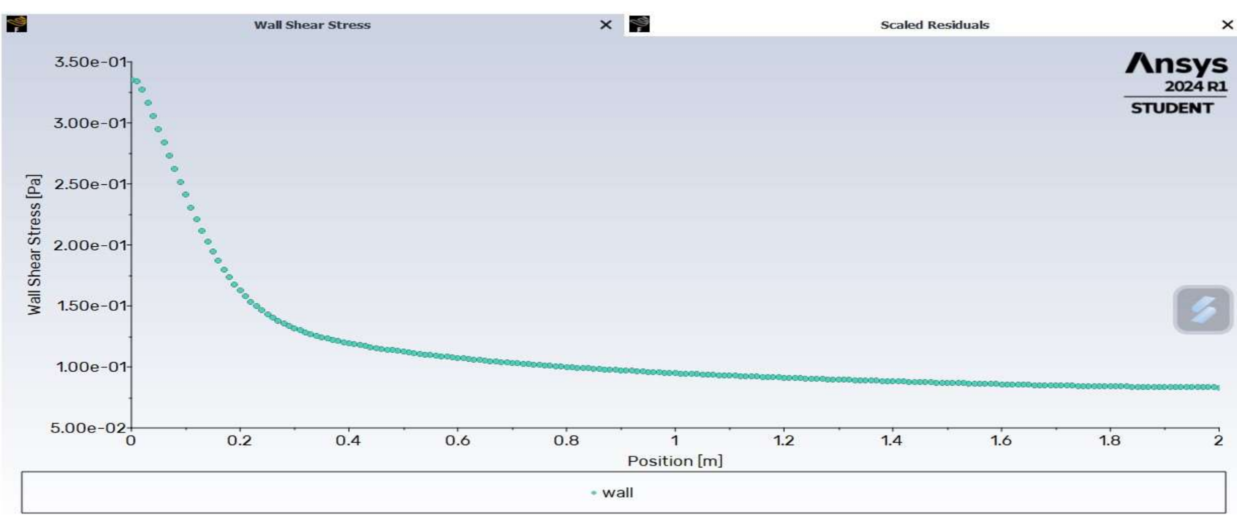
IV) Reynold's number $= 500$



From the above plot, the saturated wall shear stress comes out to be 0.04 Pa

After substituting the values, Darcy's friction factor comes to be 0.149

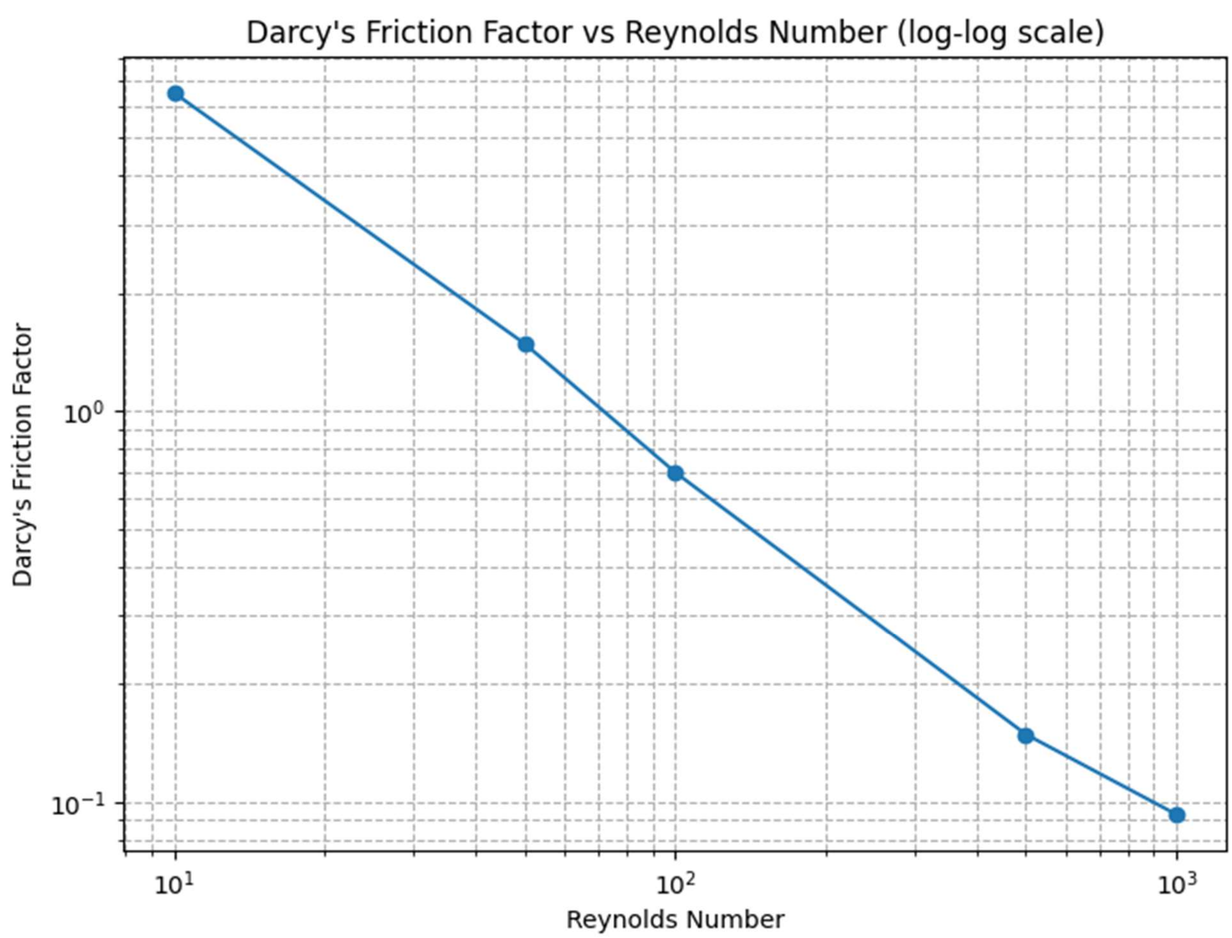
V) Reynold's number $= 1000$



From the above plot, the saturated wall shear stress comes out to be 0.1 Pa

After substituting the values, Darcy's friction factor comes to be 0.093

Reynolds number	Friction factor
10	6.511
50	1.489
100	0.697
500	0.149
1000	0.093

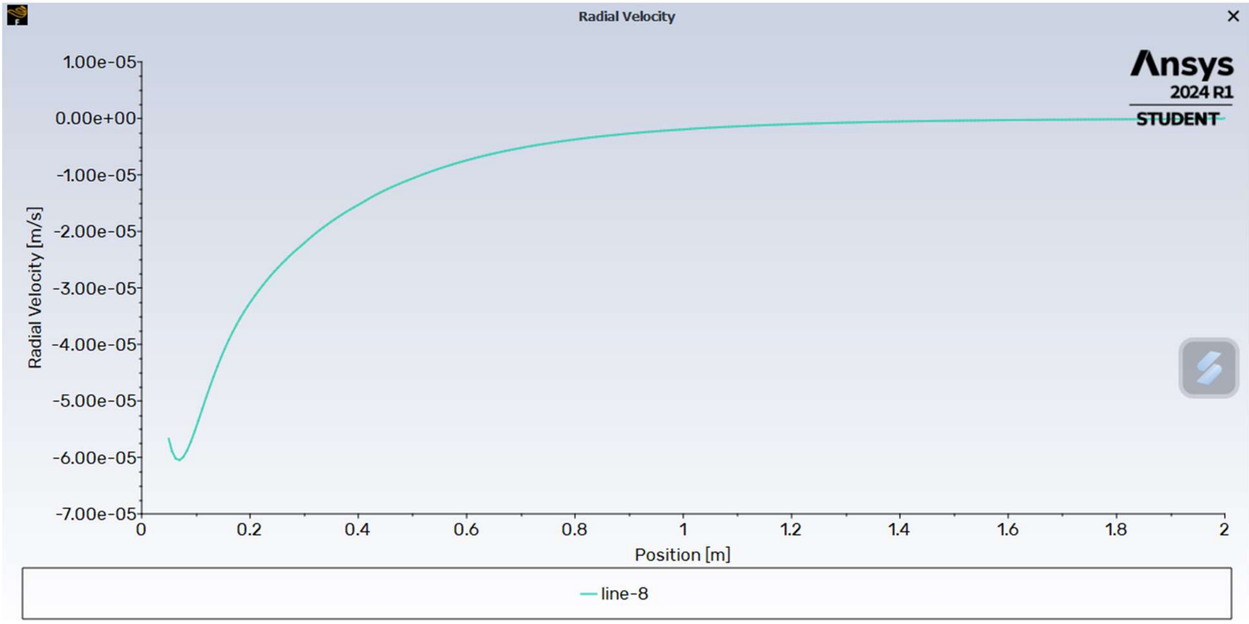


As the Reynolds number increases, the flow becomes more turbulent, as a result reduces the resistance to flow, which leads to decrease in friction factor

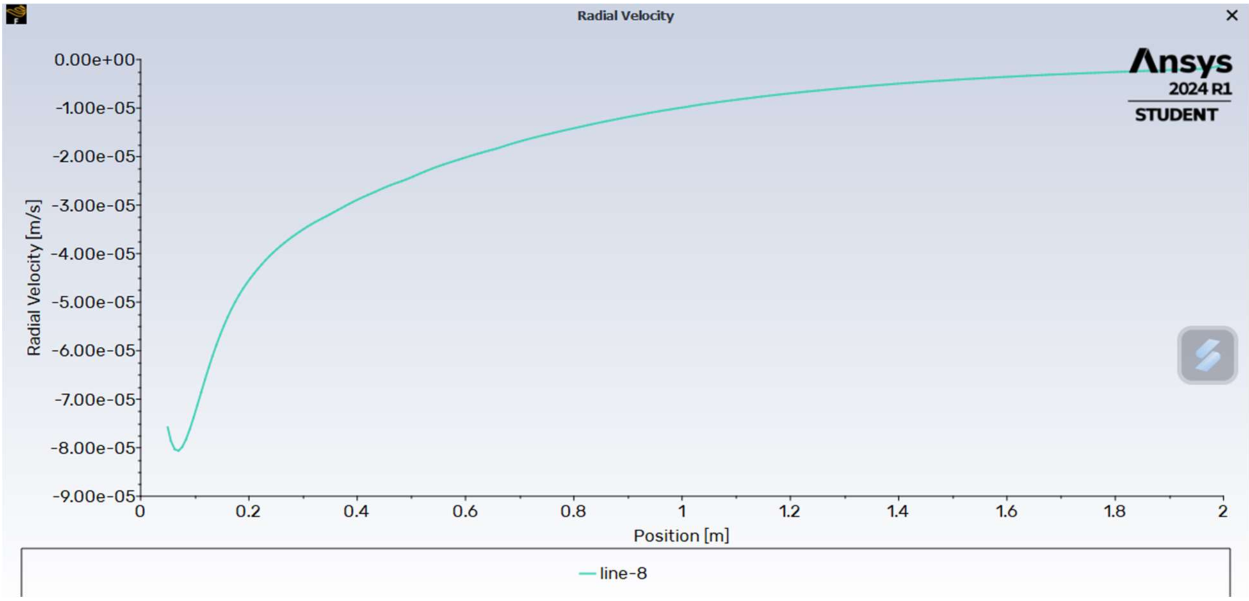
Q-5

❖ Plot the radial velocity along the length (z-direction) at 5 cm from the centreline for 3 different Reynolds numbers.

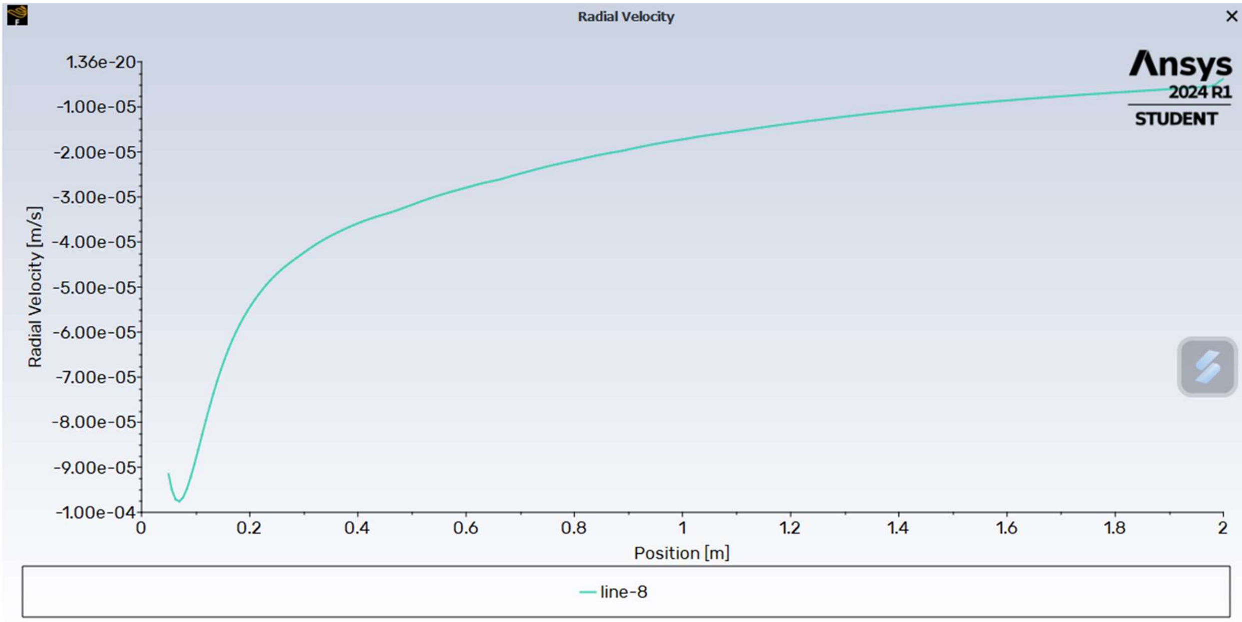
I) Reynold's number $= 100$



II) Reynold's number $= 200$



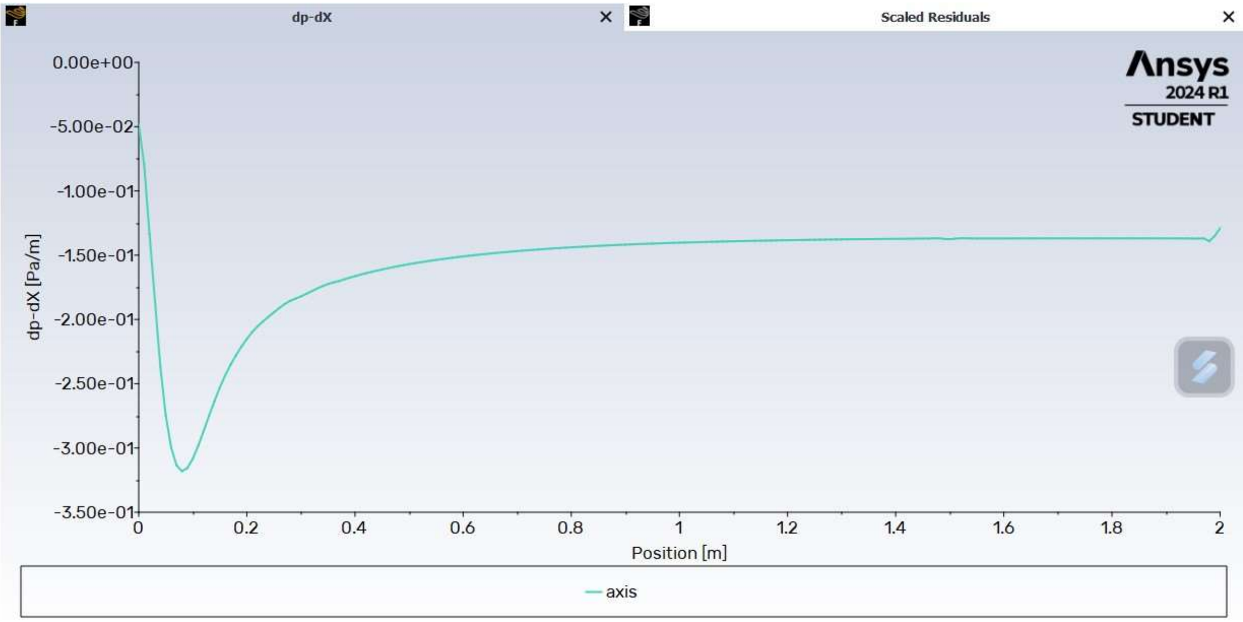
III) Reynold's number $= 300$



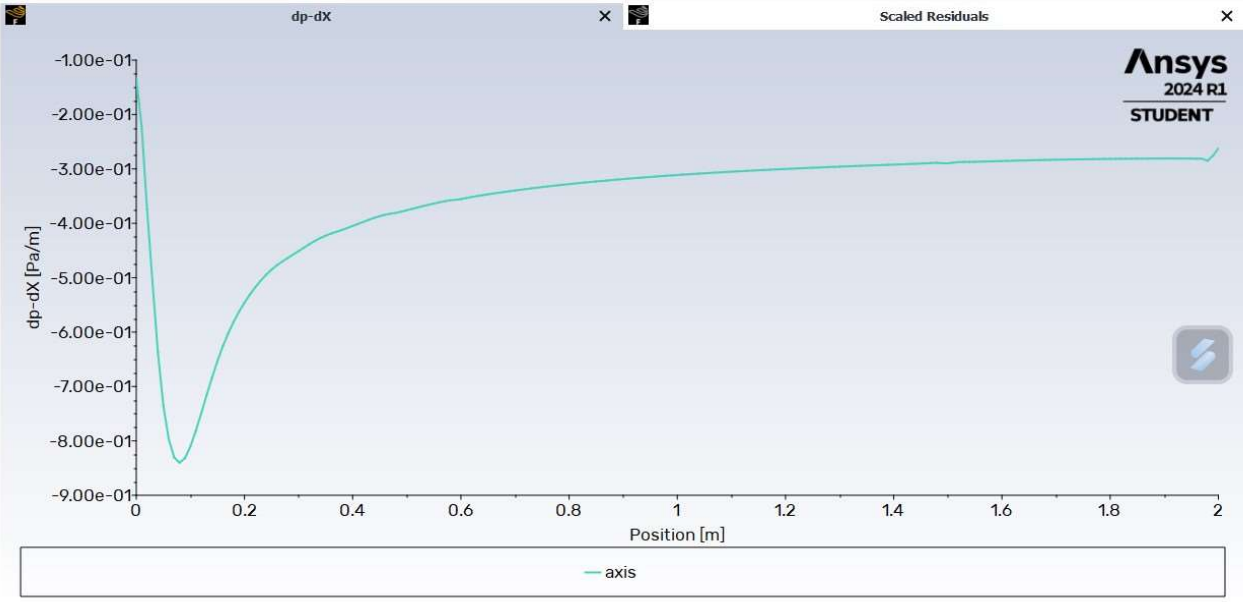
Q-6

❖ Plot the pressure gradient along the centreline for 3 different Reynolds numbers.

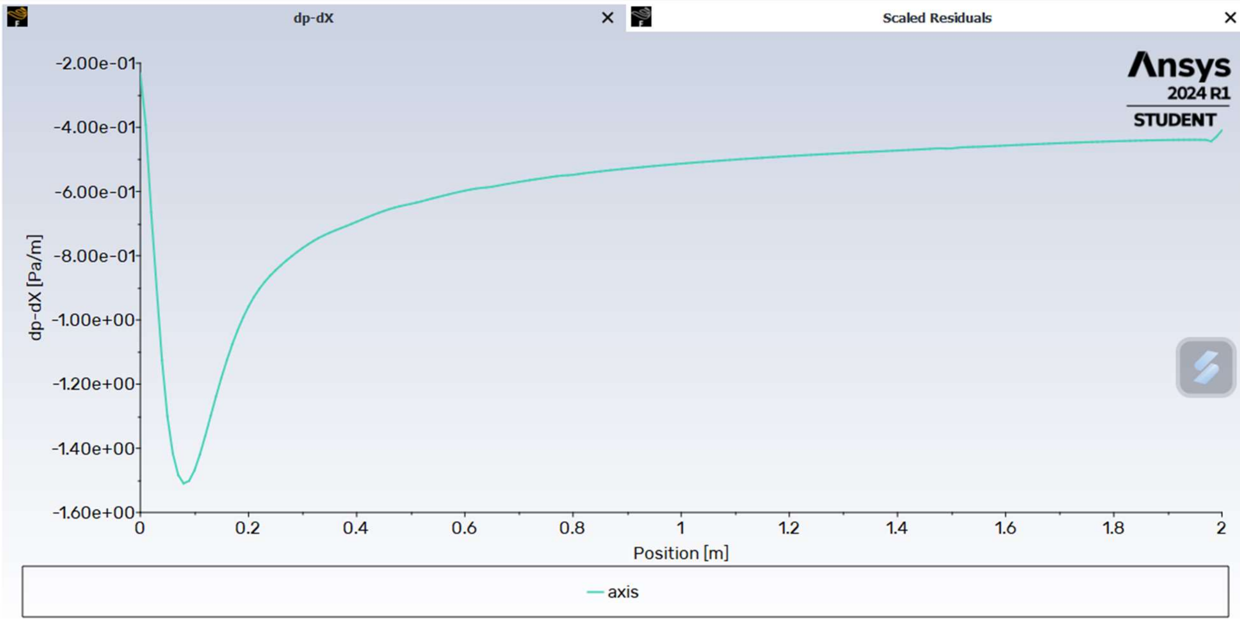
I) Reynold's number $= 100$



II) Reynold's number $= 200$



III) Reynold's number $= 300$

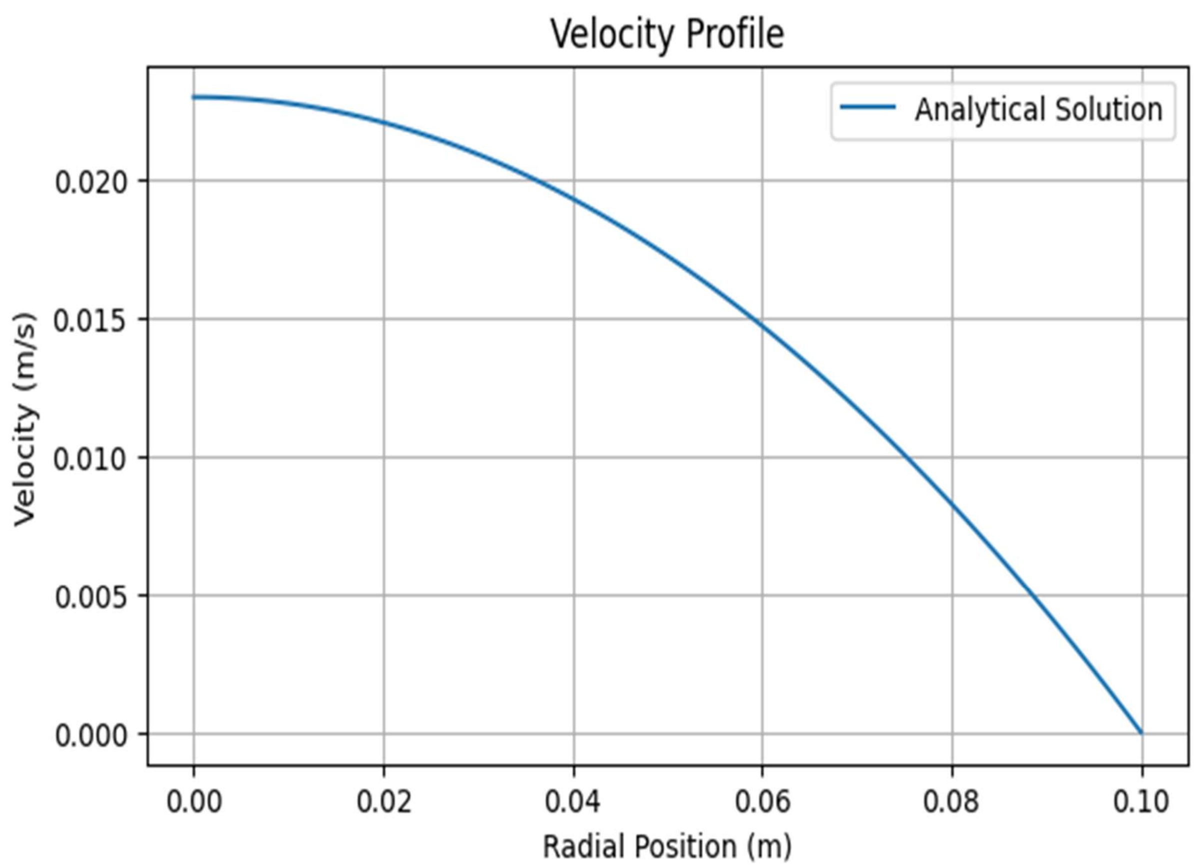


The initial decrease followed by an increase in the pressure gradient can be explained by viscous effects and boundary layer growth.

Q-7

- Calculate the fully developed velocity profile u(y) from analytical results using the pressure gradient from the CFD results. Show the analytically calculated velocity profile comparison with CFD results on same graph.

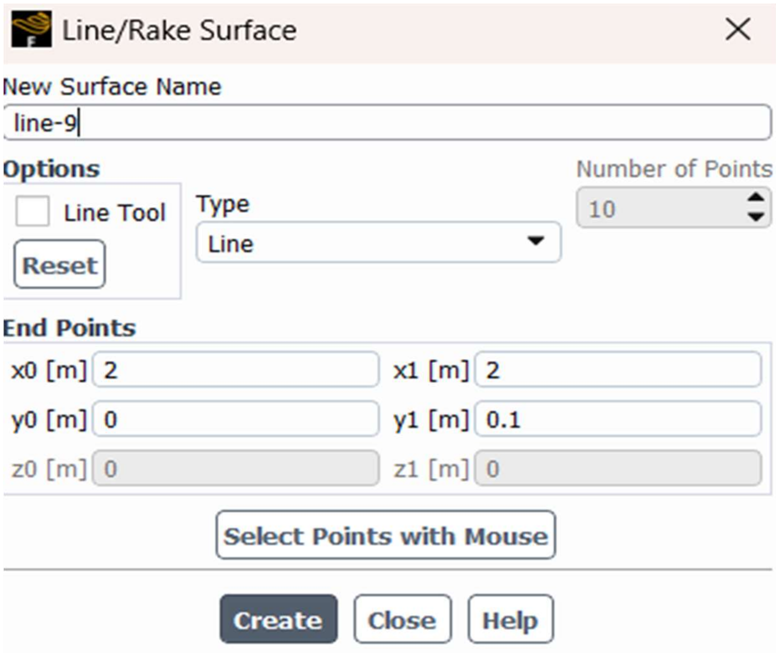
i) Analytically calculating the velocity field using Hagen-Poiseuille equation:

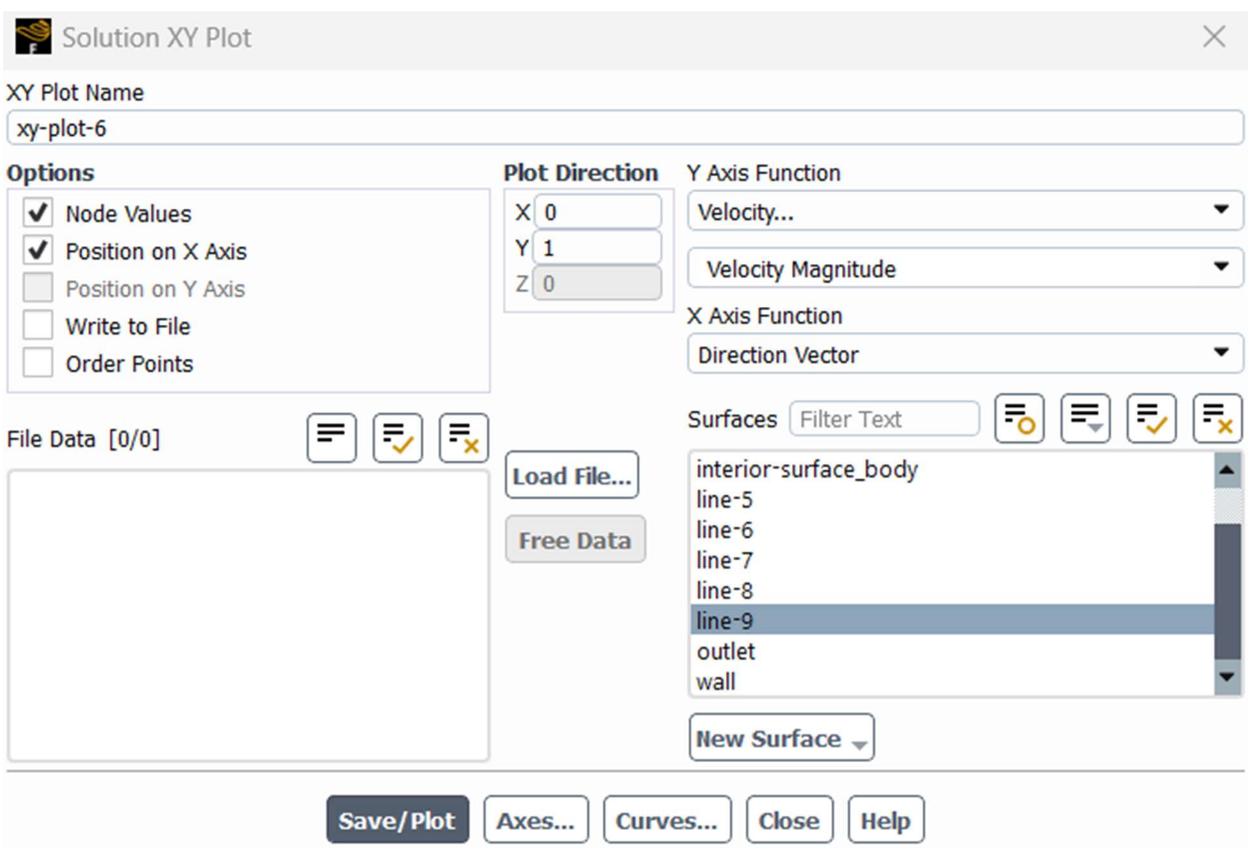


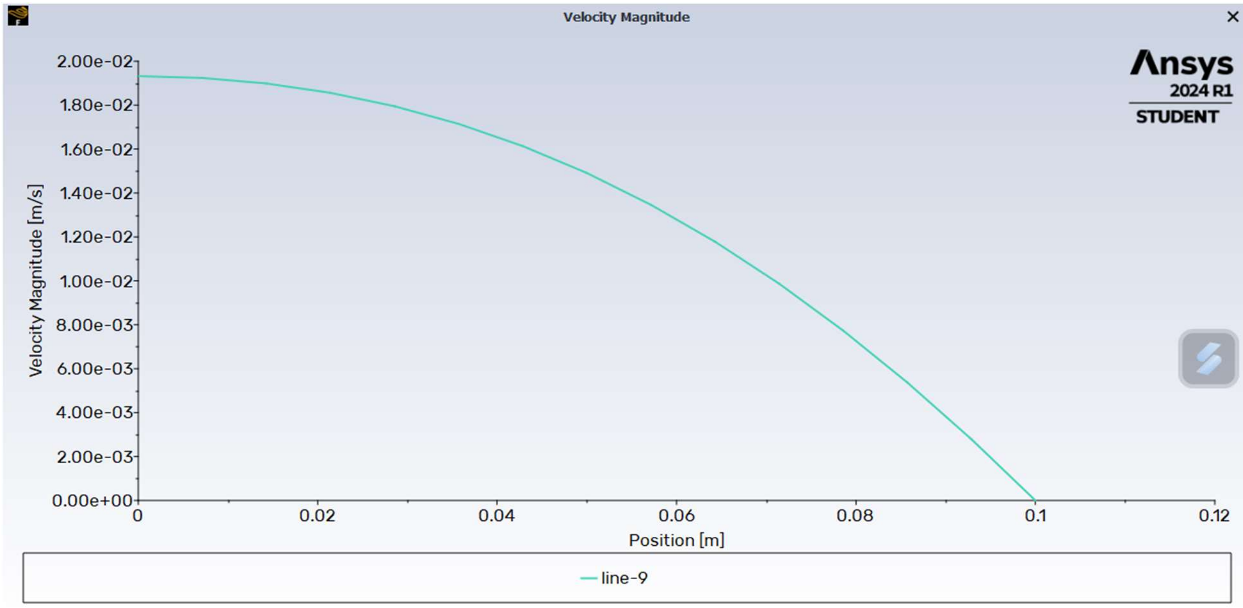
After loading the pressure gradient data from a CSV file, it performs Numerical integration is performed using Simpson's rule (**scipy.integrate.simps**) to calculate the pressure drop (**delta_P**) along the length of the pipe based on the pressure gradient from the CFD results.

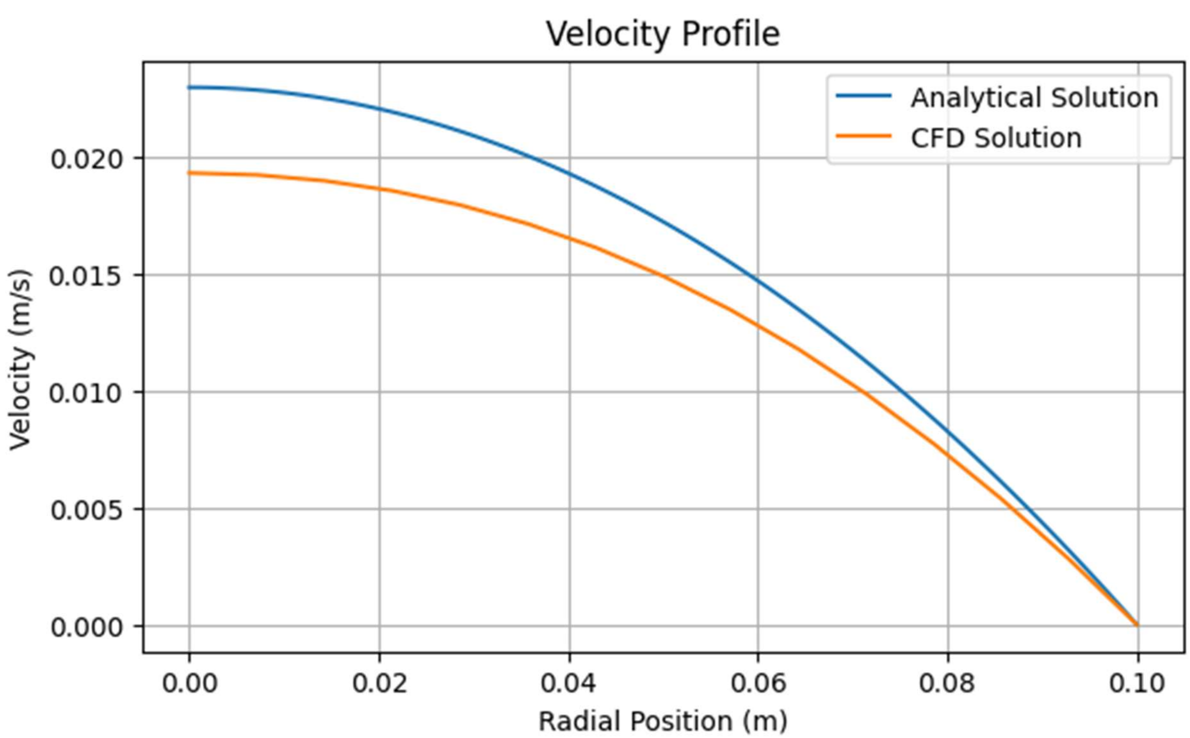
It uses Hagen-Poiseuille equation, which describes laminar flow in a pipe.

ii) Calculating the velocity field using CFD:









Both, analytical plot generated using python, and the CFD plot of velocity profile are comparatively same

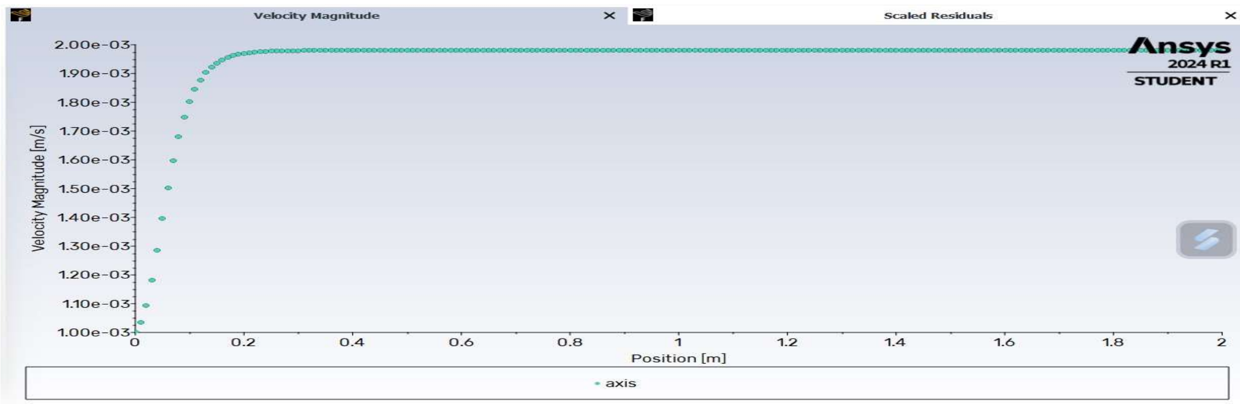
Q-8

- Find the entrance length of the flow for 5 different Reynolds numbers and plot your results.

For Laminar Flow:

$$L_h = 0.05 \times \mathrm{Re} \times D$$

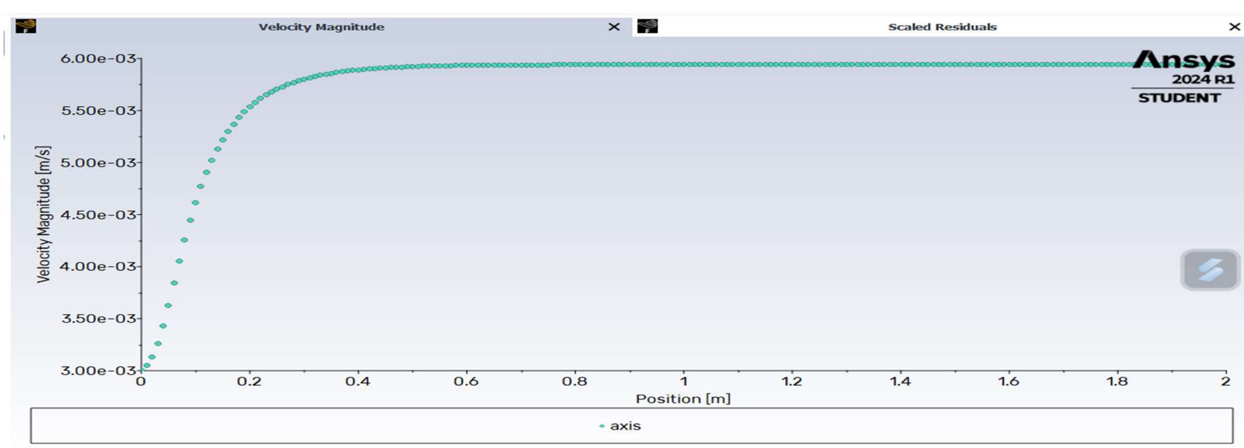
I) Reynold's number $= 10$



From the above plot, the Entrance length comes out to be 0.15

Using the formula, the Entrance length comes out to be 0.1

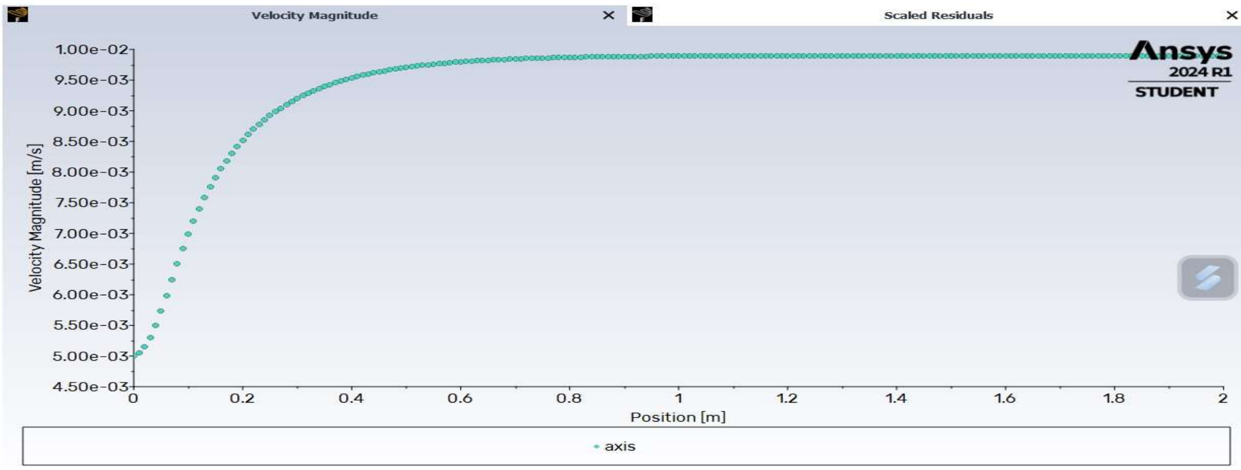
II) Reynold's number $= 30$



From the above plot, the Entrance length comes out to be 0.35

Using the formula, the Entrance length comes out to be 0.3

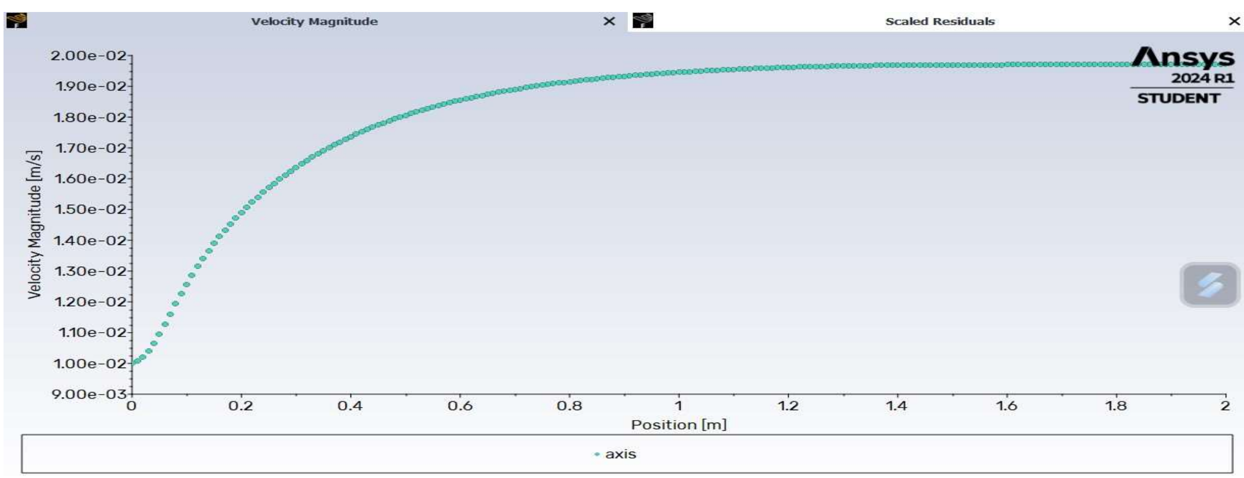
III) Reynold's number $= 50$



From the above plot, the Entrance length comes out to be 0.6

Using the formula, the Entrance length comes out to be 0.5

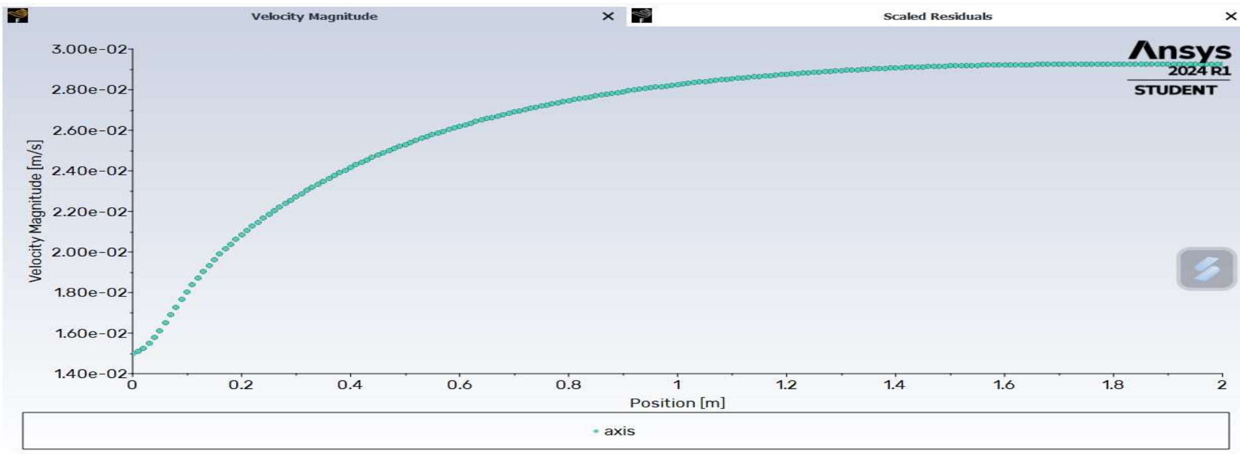
IV) Reynold's number $= 100$



From the above plot, the Entrance length comes out to be 1.1

Using the formula, the Entrance length comes out to be 1.0

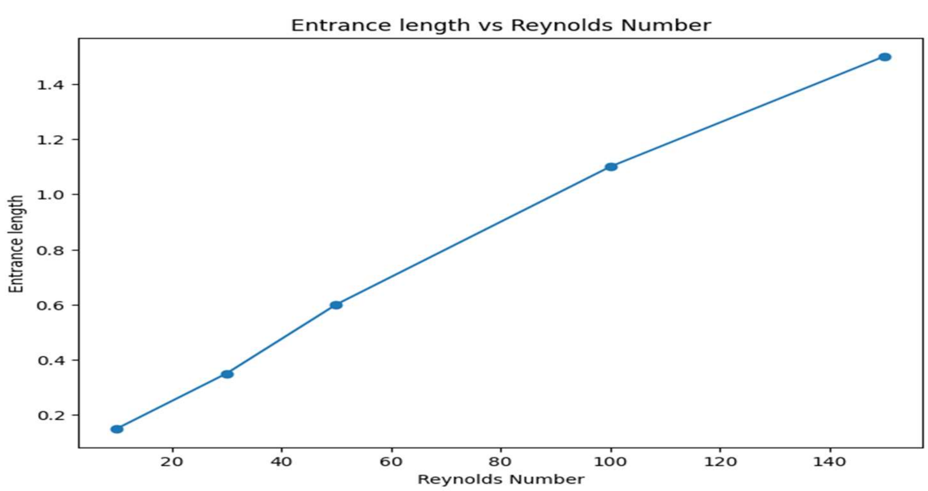
V) Reynold's number $= 150$



From the above plot, the Entrance length comes out to be 1.5

Using the formula, the Entrance length comes out to be 1.5

Reynolds number	Entrance length
10	0.15
30	0.35
50	0.6
100	1.1
150	1.5

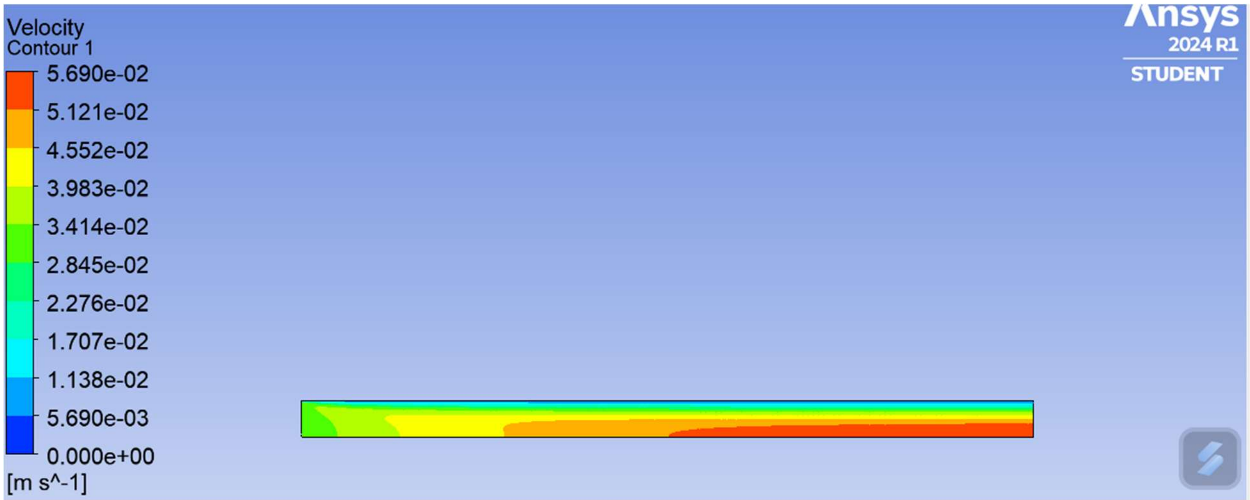


The result holds true as at higher Reynolds numbers, the entrance length increases because the flow requires more distance to transition from the initial unsteady state to the fully developed state.

Q-9

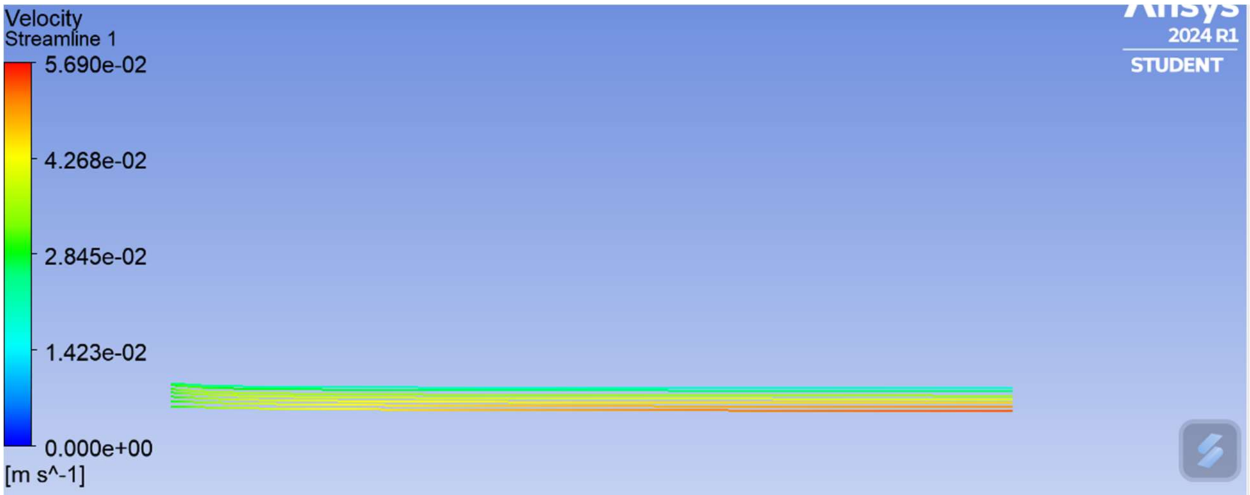
❖ Plot the r and z components of velocity contours and streamline contour separately.

i) z-component velocity contour:

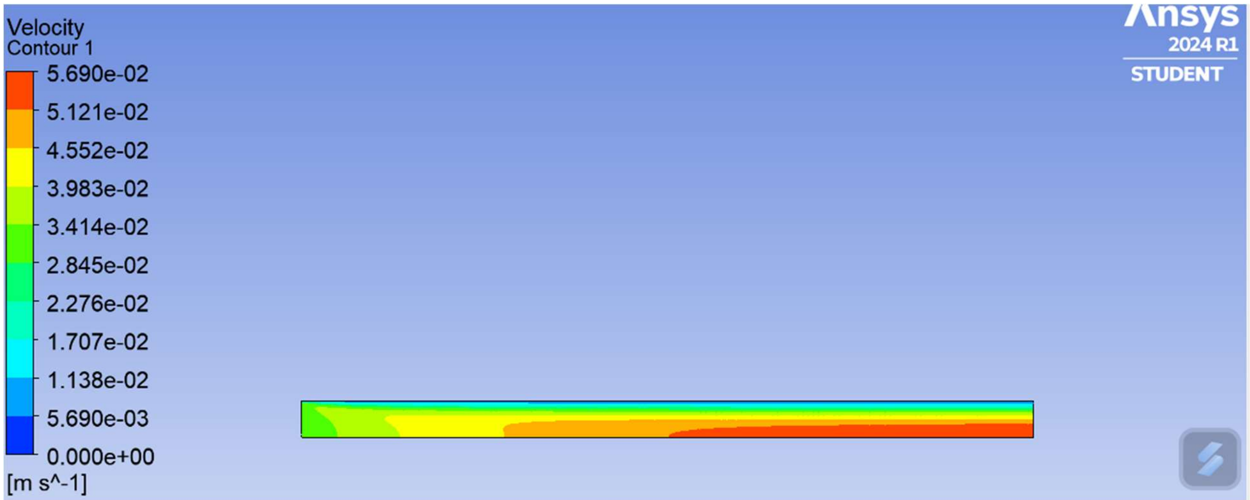


ii) r- component velocity contour:

iii) Streamline contour:



Q-10



a

Velocity contours shows different shades or colors to represent varying magnitudes of velocity. Regions of higher velocity are depicted with warmer colors (red or yellow), while regions of lower velocity are represented with cooler colors (blue or green).

In the first plot, as the boundary layer decreases with the forward flow, the velocity near the centerline increases, and decreases near the wall, finally obtaining a parabolic shape.

Velocity streamlines indicate the direction of fluid flow at every point in the domain. Streamlines follow the direction of the velocity vector at each location, allowing observers to easily visualize the overall flow pattern and directionality.

5) Conclusion:

This ANSYS simulation for the laminar flow provided detailed information of fluid flow. Through the simulation, comprehensive insights of pressure and velocity distribution along the pipe length were obtained. Pressure and velocity varied along the pipe according to the flow nature and other influencing factors such as geometry and boundary conditions. Also, the shear stress distribution within the pipe contributed to the understanding of frictional losses.

The new thing which I learnt is the mesh independence, analyzing the boundary layers, making the contours and streamlines and analyzing them.

The insights gained from this assignment will help me understanding fluid flow behavior in a better way and will be important for optimizing system performance, and ensuring operational efficiency.