Computational Fluid Dynamics Lab 1

ME 207 | Fluid Dynamics

Diya Mehta 22110078

1. Problem Statement

Crude oil enters the pipe with diameter of 20 cm and length of 2 m. The density and viscosity of the crude oil can be taken as 860 kg/m3 and 17.2 cP, respectively. Perform steady-state axisymmetric fluid flow simulation in the Ansys software and complete the following exercises.

2. Geometry and Boundary Conditions

2.1. Geometry Conditions

The geometry of the simulation domain consists of a cylindrical pipe with the following dimensions:

- Diameter: 20 cm
- Length: 2 m

The pipe serves as the conduit for the flow of crude oil. The axisymmetric nature of the problem allows for simplification, wherein only a 2D cross-section of the pipe needs to be modeled.

2.2. Boundary Conditions

The boundary conditions applied to the simulation are crucial in defining the flow characteristics within the pipe. They are as follows:

- **Inlet Boundary Condition:** The velocity at the inlet is determined based on the desired Reynolds number. This boundary condition represents the flow entering the pipe.
- **Outlet Boundary Condition:** The pressure at the outlet is specified to maintain the desired pressure distribution within the system. This boundary condition ensures that the flow exits the domain without any reflections or disturbances. Here we have kept the outlet gauge pressure to be 0.
- **Wall Boundary Condition:** No-slip condition. At the walls of the pipe, the fluid velocity is set to zero, simulating the physical interaction between the fluid and the pipe wall. This condition ensures that the fluid adheres to the walls and prevents slip, which is typical for viscous flows.

Additionally, a constant wall temperature is maintained throughout the simulation to account for any temperature effects on the flow behavior.

3. Mesh Statistics and Mesh Independence Study

A mesh independence study is conducted to ensure that the results are not significantly affected by the mesh resolution. Several levels of mesh refinement are employed, with increasing mesh density. Mesh statistics including the number of nodes and elements are recorded for each refinement level.

It is observed that as the mesh density increases, the solution approaches convergence, i.e ΔP becomes constant, indicating independence from mesh resolution beyond a certain level of refinement.

Element Size	Nodes	Elements	Pressure Difference
0.03 m	252	186	0.28 Pa
0.01 m	2178	1970	0.31 Pa
0.009 m	2640	2409	0.31 Pa
0.007 m	4215	3920	0.31 Pa
0.005 m	8169	7760	0.31 Pa

Table 1. *Mesh Statistics*
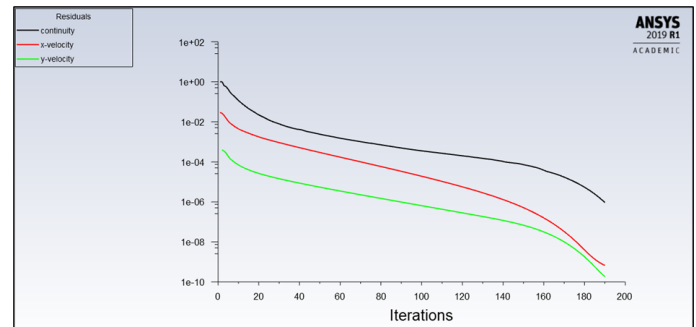

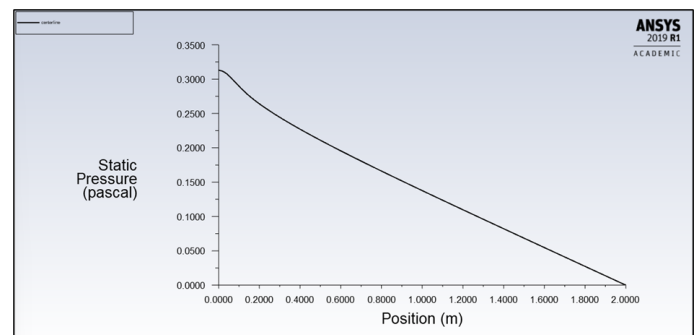**Figure 1.** *3.1 Iterations for element size 0.009 m*

Figure 2. *3.2 Pressure at the centreline for element size 0.009 m*
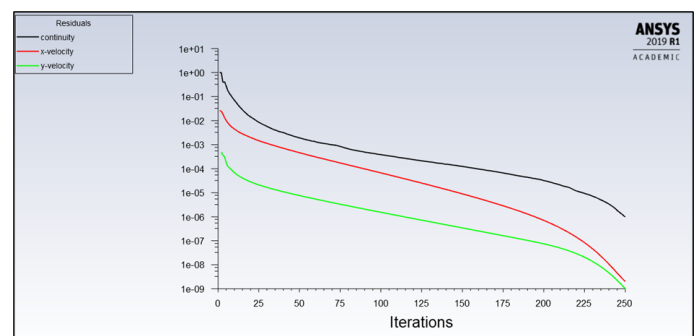

Figure 3. *3.3 Iterations for element size 0.007 m*

4. Discretisation Schemes and Solution Methodology

4.1. Discretisation Schemes

In the computational fluid dynamics (CFD) simulation, the discretisation schemes play a critical role in approximating the continuous governing equations of fluid flow on a discrete grid. The discretisation schemes determine how the spatial and tem-
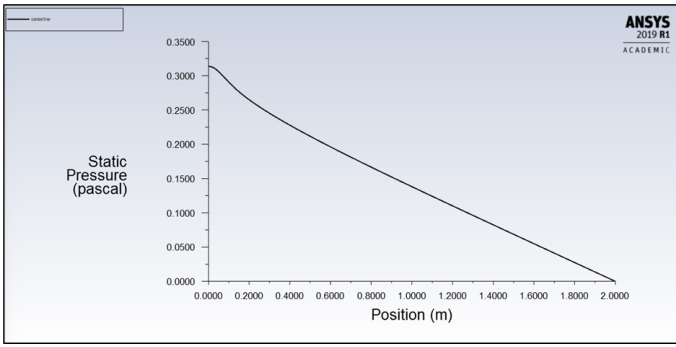
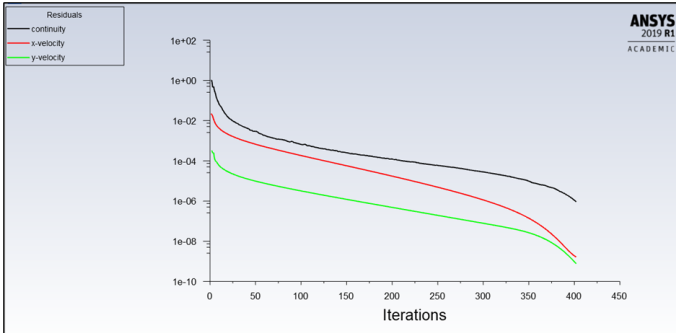**Figure 4.** *3.4 Pressure at the centreline for element size 0.007 m*

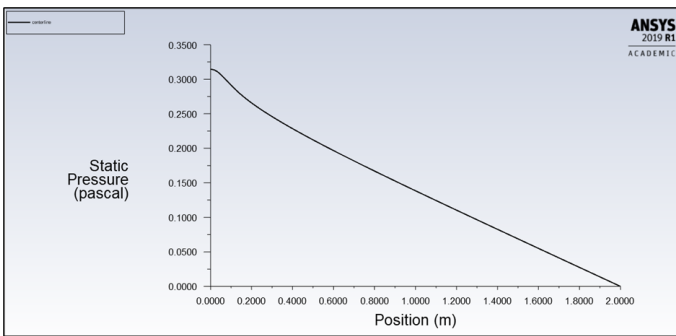**Figure 5.** *3.5 Iterations for element size 0.005 m*

Figure 6. *3.6 Pressure at the centreline for element size 0.005 m*

poral derivatives in the governing equations are approximated and computed numerically. In this study, the following discretisation schemes are employed:

- Spatial Discretisation:
  - Finite Volume Method (FVM): The governing equations, namely the Navier-Stokes equations, are discretised using the finite volume method. In FVM, the computational domain is divided into control volumes, and the integral form of the governing equations is applied to each control volume. This approach conserves mass, momentum, and energy within each control volume, making it well-suited for fluid flow simulations.

- Temporal Discretisation:
  - Steady-State Simulation: The simulation is conducted in a steady-state regime, where the flow variables remain constant with time. Therefore, no temporal discretisation is required for time-dependent terms in the governing equations.

4.2. Solution Methodology

The solution methodology refers to the numerical techniques and algorithms used to solve the discretised equations and obtain the numerical solution for the flow field. The following solution methodology is employed:

Solver: Ansys Software: The simulation is performed using the Ansys software suite, which provides powerful tools for conducting CFD simulations. Ansys offers a range of solvers and numerical methods tailored for various fluid flow problems. In this study, Ansys is utilized to solve the discretised continuity, x-momentum and y-momentum equations (Navier-Stokes Equations) and obtain numerical solutions for the flow field within the cylindrical pipe.

5. Results and Discussion

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

The velocity and pressure distributions along the centerline of the pipe are analyzed for a Reynolds number of 100.

Using $Re = \dfrac{\rho v D}{\mu}$, $v = 0.01 m/s$ as the inlet velocity. Towards the pipe outlet, the velocity becomes constant as it achieves a steady-state value, indicating fully developed flow conditions.
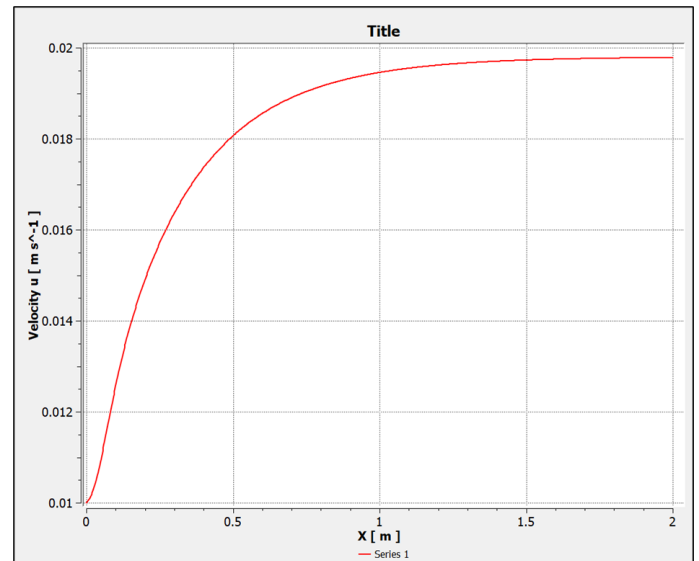

Figure 7. *5.1.1 Velocity at the centreline for Re=100*

The pressure distribution shows a gradual decrease along the length of the pipe, reflecting the pressure drop caused by viscous losses.

The simulation is showing that as the pressure decreases along the length of the pipe, the velocity increases.

5.2. Calculate wall shear stress and Darcy's friction factor along the wall.

The wall shear stress and Darcy's friction factor are computed along the pipe wall using the simulated data.

For wall shear stress,

$$\Delta P = \frac{4\tau L}{D}$$
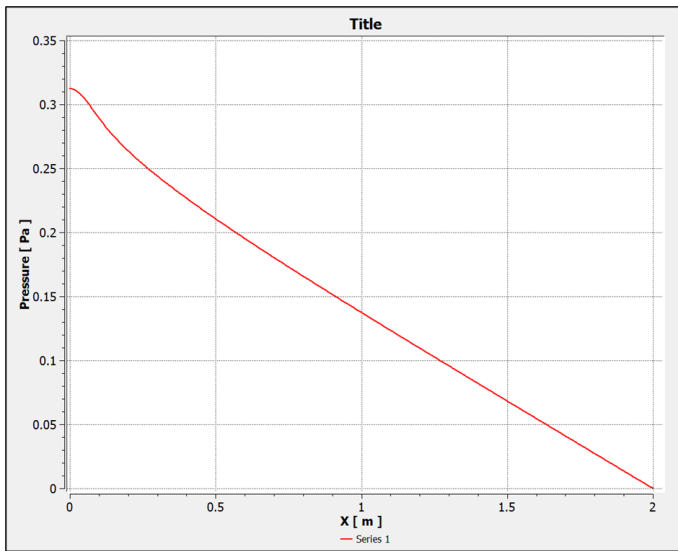
Where $\Delta P = 0.31 Pa$, $L = 2m$, $D = 20cm$.

Figure 8. *5.1.2 Pressure at the centreline for Re=100*

Thus,

$$\tau = 0.00775 Pa$$

For Darcy's friction factor,

$$\Delta P = \frac{1}{2}\rho u^2 f \frac{L}{D}$$

where $\rho = 860 kg/m^3$, $u = 0.01 m/s$, $L = 2m$, $D = 20cm$, $\Delta P = 0.31 Pa$

Then,

$$f = 0.72$$

The wall shear stress varies along the pipe length, with higher values near the inlet and lower values towards the outlet. This variation is attributed to changes in the velocity gradient and flow conditions.
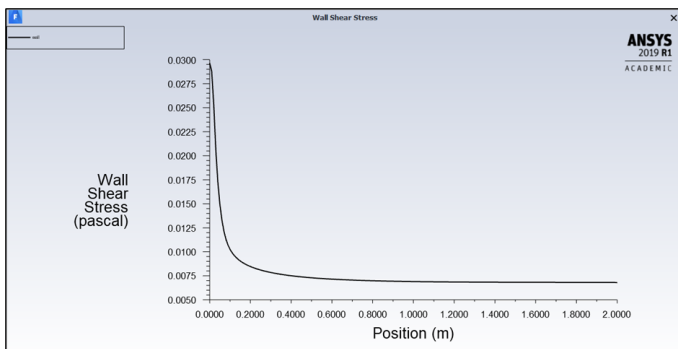

Figure 9. *5.2.1 Wall shear stress*

From the plot, we can see that the wall shear stress becomes constant and is approximately equal to the calculated value.

5.3. Perform mesh independence study

A mesh independence study is conducted to ensure the reliability and accuracy of the simulation results.

Results obtained from different levels of mesh refinement are compared to assess mesh independence.

It is observed that as the mesh density increases, the solution approaches convergence, indicating independence from mesh resolution beyond a certain level of refinement.

5.4. 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 was plotted against Reynolds number for five different Reynolds numbers ranging from 10 to 1000 on a log-log scale. This plot revealed the characteristic trend of Darcy's friction factor with Reynolds number.

The log-log scale allows for a clear visualization of the power-law relationship between Darcy's friction factor and Reynolds number.
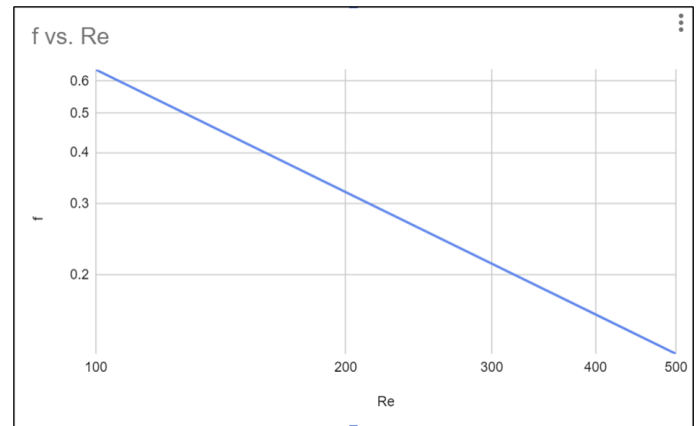
$$f = 64/Re$$

Figure 10. *5.4.1 Darcy's friction factor Vs Reynolds number on a log-log scale*

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

Radial velocity profiles at a distance of 5 cm from the centerline and pressure gradient along the centerline are analyzed for three different Reynolds numbers.

Near the pipe outlet, the radial velocity approachs zero as the flow transitions to a fully developed state and the velocity profile becomes more uniform.

A dip in the radial velocity is seen as this is the transition phase, so velocity has not reached its fully developed state.
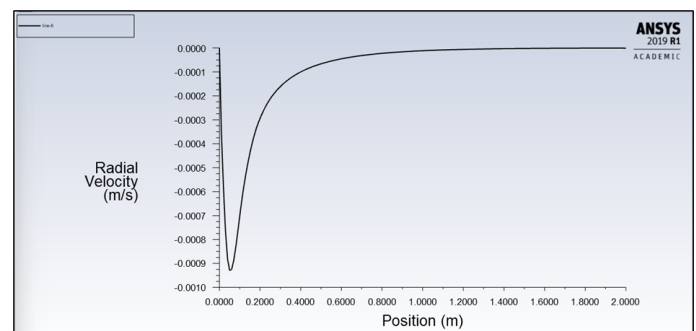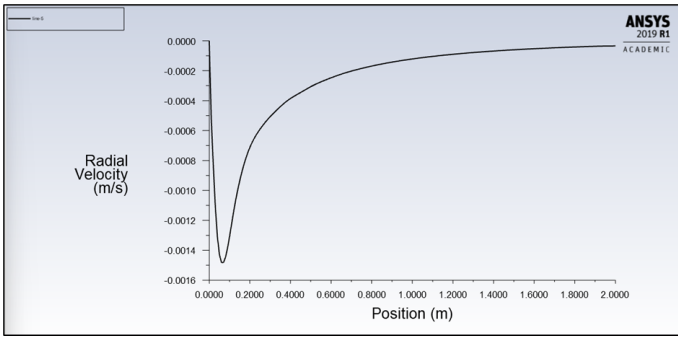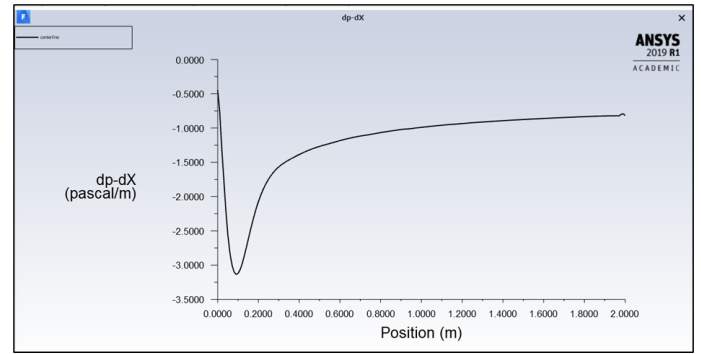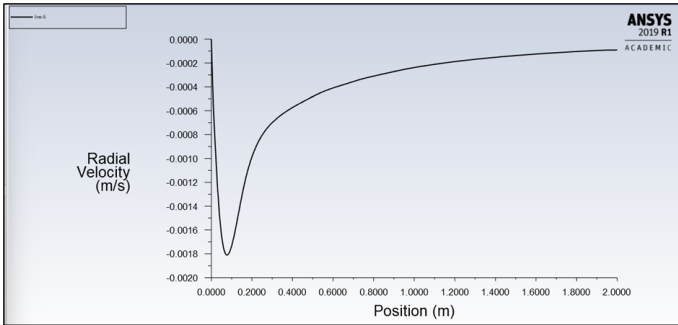

**Figure 11.** *5.5.1 Radial velocity for Re=100*

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

Pressure gradient along the centerline was analyzed for three different Reynolds numbers (Re = 100, 300, 500).

Figure 12. *5.5.2 Radial velocity for Re=300*

Figure 13. *5.5.3 Radial velocity for Re=500*

The pressure gradient becomes constant after reaching the fully developed flow since the pressure varies linearly with the length, so the slope is constant.

The pressure gradient profiles indicate changes in pressure along the pipe length, which influence the flow characteristics and energy dissipation.
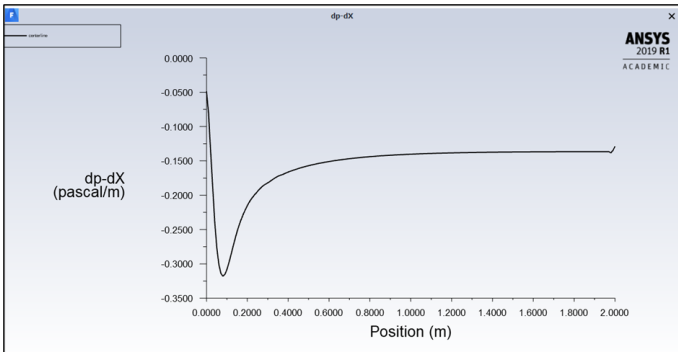
Figure 14. *5.6.1 dP/dx along the centreline for Re=100*
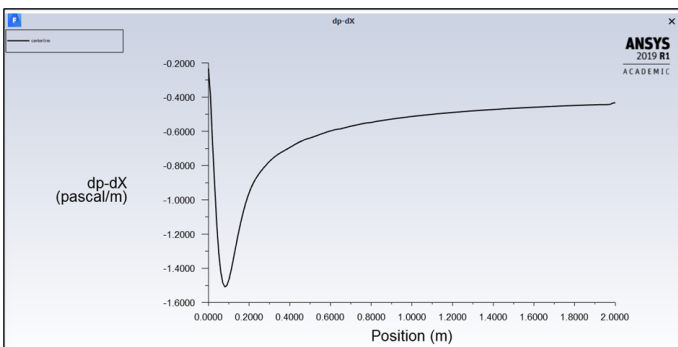
Figure 15. *5.6.2 dP/dx along the centreline for Re=300*

Figure 16. *5.6.3 dP/dx along the centreline for Re=500*

5.7. 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 the same graph.

Fully developed velocity profile (u(y)) was calculated analytically using the pressure gradient from the CFD results and compared with the CFD velocity profile.

The velocity profile exhibits a parabolic shape, characteristic of laminar flow, with maximum velocity at the pipe center. The velocity gradually decreases towards the pipe wall due to viscous effects.
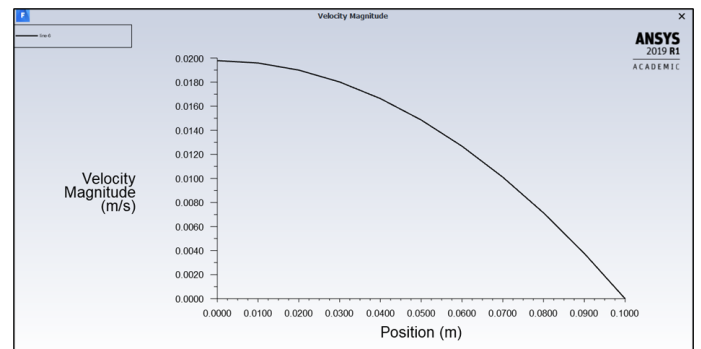
Figure 17. *5.7.1 CFD result for velocity profile u(y)*

Formula used for analytical calculations

$$u = \frac{dP}{dx}\frac{R^2}{4\mu}[1 - \frac{r^2}{R^2}]$$

From the above plot, we take dP/dx = 0.14 Pa/m. $R = 0.1m$, $\mu = 17.2cP$.

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

Entrance length of the flow was determined for five different Reynolds numbers to understand flow development from the pipe entrance.

Calculated entrance lengths provided insights into the influence of flow velocity and viscosity on the development of flow profile within the pipe.

From the plots, we find the entrance length from where the velocity becomes constant.
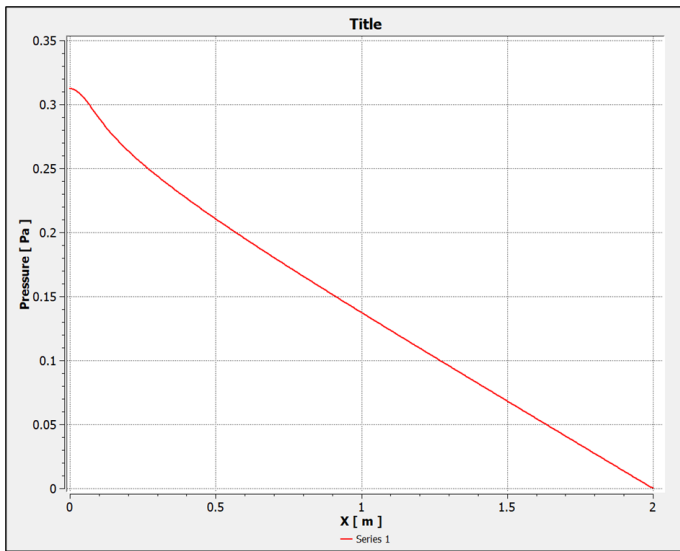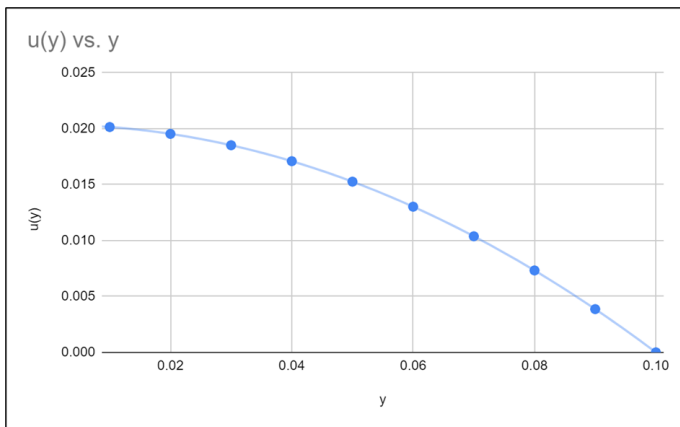
Figure 18. *5.7.2 Pressure along the length*

Figure 19. *5.7.3 Analytical result for velocity profile u(y)*

Re	Entrance Length (in m)
50	0.60
70	0.80
90	1.00
100	1.20
120	1.40

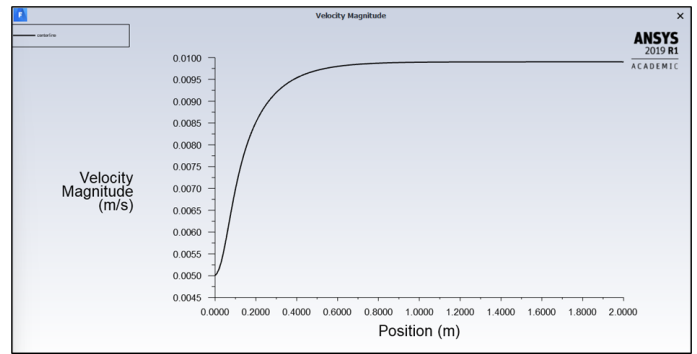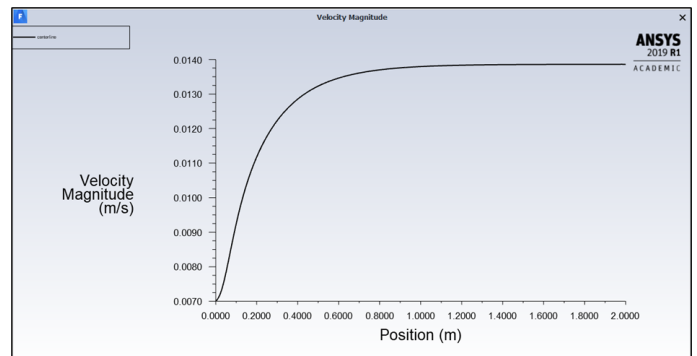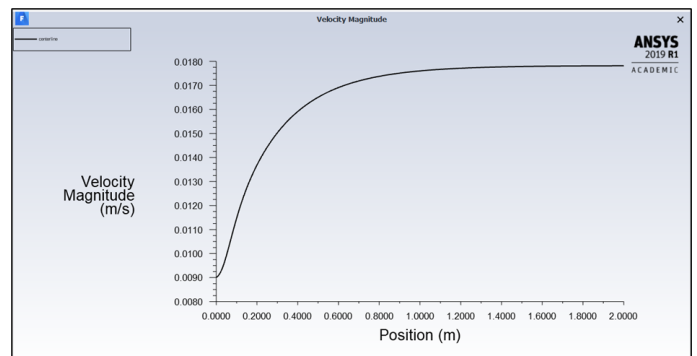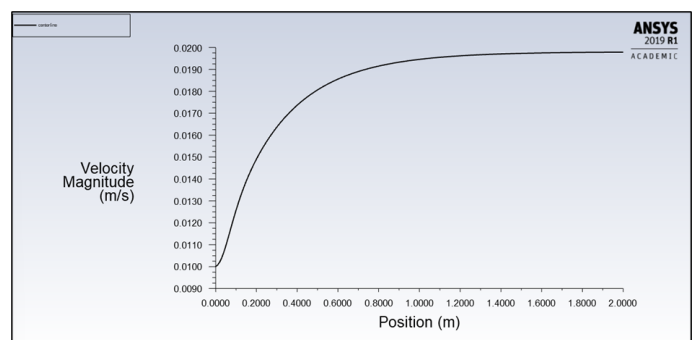**Table 2.** *Entrance length for different Reynolds numbers*
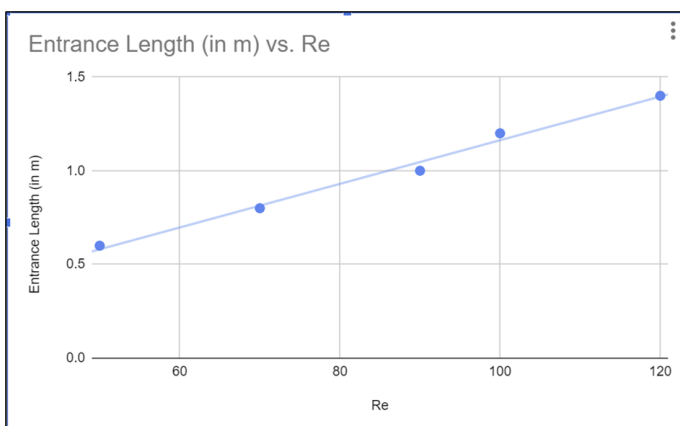
Figure 20. *5.8.1 Entrance length vs Reynolds number*

Figure 21. *5.8.2 Velocity at the centreline for Re=50*

Figure 22. *5.8.3 Velocity at the centreline for Re=70*

Figure 23. *5.8.4 Velocity at the centreline for Re=90*

Figure 24. *5.8.5 Velocity at the centreline for Re=100*

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

Radial and axial velocity contours, along with streamline contours, were plotted separately to visualize flow patterns within the pipe.
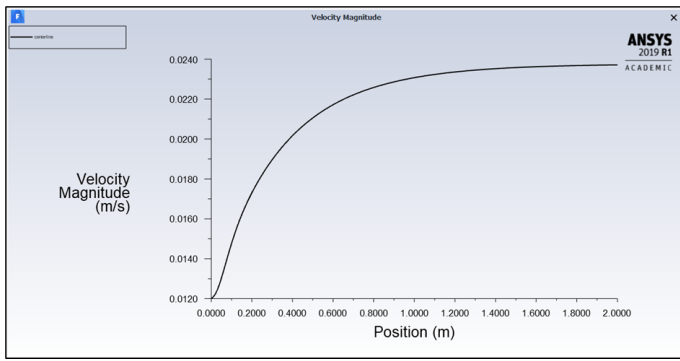
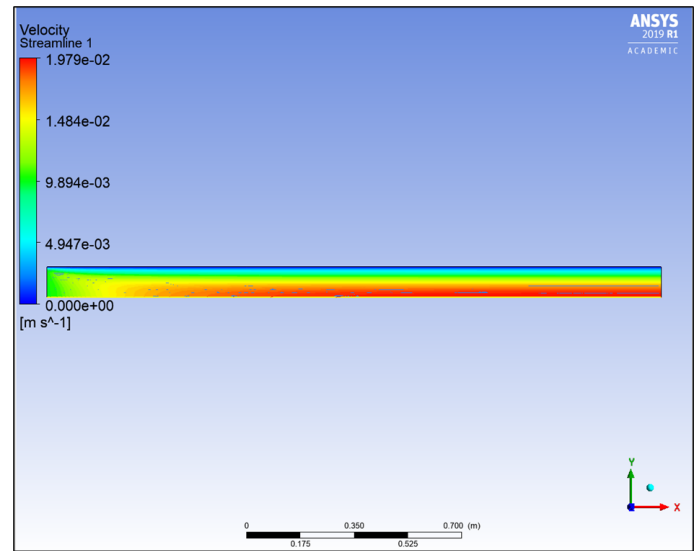**Figure 25.** *5.8.6 Velocity at the centreline for Re=120*
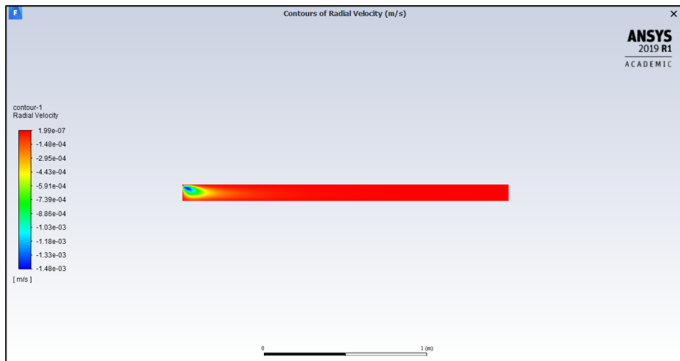

Figure 26. *5.9.1 Radial r-velocity contour*

Figure 27. *5.9.2 Axial z-velocity contour*
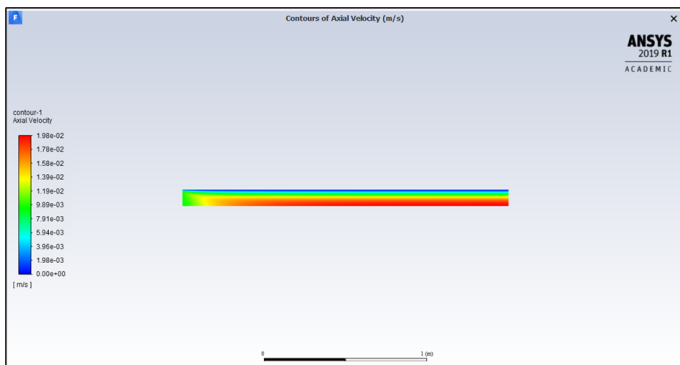

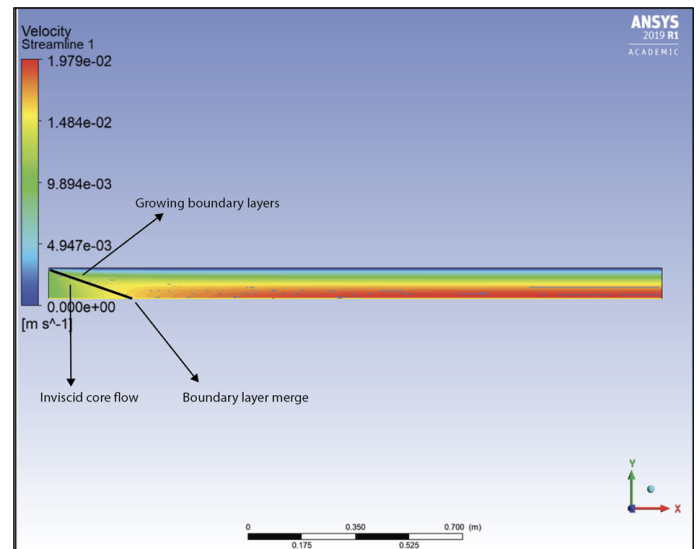**Figure 28.** *5.9.3 Streamline contour*

Figure 29. *5.10.1 Flow features in the contour*

5.10. Mark and describe the flow features in the contours (should include boundary layer development and merging).

Flow features observed in the contours included boundary layer development near the pipe wall, separation zones at sharp bends, and merging of boundary layers along the length of the pipe.

Entrance Region: At the entrance of the flow, the contours indicate the transition from an undeveloped flow state to a developed flow state. In this region, the boundary layers are just beginning to form near the walls of the conduit or channel. The contours typically show steep velocity gradients near the walls as the flow accelerates from the entrance.

Boundary Layer Development: The contours reveal the gradual thickening of the boundary layers as the flow adjusts to the geometry of the conduit. Velocity gradients within the boundary layers become less steep as the flow reaches a more uniform velocity profile across the cross-section of the conduit.

Merging of Boundary Layers: In the developing flow region, the contours show the merging of boundary layers from different regions of the conduit.

Fully Developed Flow: In the fully developed flow region, the contours indicate a stable and uniform flow profile across the entire cross-section of the conduit. The boundary layers have fully developed, with a consistent thickness along the walls. Velocity profiles become fully developed and remain constant along the flow direction, indicating a steady-state flow condition.

6. Conclusion

The comprehensive analysis of fluid flow within the cylindrical pipe through computational fluid dynamics (CFD) simulation has provided valuable insights into the behavior and performance of the pipe system under varying Reynolds numbers.

The velocity and pressure distributions along the centerline of the pipe exhibited typical behaviors associated with laminar flow, with velocity profiles showing a parabolic shape and pressure decreasing gradually along the pipe length. These

characteristics are consistent with the expected behavior of fluid flow in a cylindrical pipe under laminar flow conditions.

Visualization of velocity contours, streamline contours, and radial velocity profiles provided insights into flow patterns within the pipe, including boundary layer development, separation zones, and merging of boundary layers.