Computational Fluid Dynamics Laboratory 1

Hari Balaji

Department of Mechanical Engineering

IIT Gandhinagar

Under Prof. Dilip Srinivas Sundaram and Prof. Uddipta Ghosh

Abstract—*This report includes the answers to the questions presented as part of the CFD Lab 1 for the ME 207 course. It attempts to use the ANSYS simulation software to simulate the laminar pipe flow problem and interpret the results of the simulation effectively.*

Keywords—*CFD, ANSYS, Simulation, Fluid Dynamics*

1. Introduction

Fluid flow analysis plays a pivotal role in engineering design and optimization, particularly in industries reliant on efficient fluid transport, such as oil and gas. Understanding the behavior of crude oil within pipelines is critical for ensuring operational efficiency and reliability. Computational Fluid Dynamics (CFD) simulations offer a powerful means to study fluid flow phenomena, enabling engineers to predict flow patterns, pressure distributions, and frictional losses under various conditions.

In this study, we focus on the steady-state axisymmetric flow of crude oil through a pipe, employing ANSYS software to investigate flow characteristics across different Reynolds numbers. By examining parameters such as velocity profiles, pressure gradients, and wall shear stress, we aim to gain insights into the flow regime transitions, boundary layer development, and overall flow behavior, providing valuable guidance for pipeline design and optimization in the oil and gas industry.

2. Geometry and Boundary Conditions

Given Data:
$D = 20cm$, $L = 2m$, $\rho = 860 kg/m^3$ and $\mu = 17.2cP$

- **Geometry:** For the analysis of the given cylindrical pipe, an infinitesimal rectangular cross section along the axis was considered since the flow is axisymmetric.

- **Boundary Conditions:**
  Inlet: The inlet boundary condition is the inlet velocity which is determined using the Reynolds Number.
  Outlet: The outlet boundary condition is the specification of the pressure at the outlet.
  Wall: The no-slip boundary condition is applied at the wall to account for viscous effects.

3. Mesh Statistics and Mesh Independence Study

3.1. Mesh Statistics

Below are the relevant statistics of the primary mesh used to perform the simulation:

Physics Preference	CFD
Solver Preference	Fluent
Element Size	1e-2 m
Nodes	2211
Elements	2000

Table 1. *Mesh Statistics*

3.2. Mesh Independence Study

To demonstrate mesh independence, there should be a very small or negligible difference in the values on convergence. The values obtained are as follows:

Refinement Level	Convergence Iteration	Pressure Difference
No Refinement	171	0.33228369 Pa
Level 1	418	0.341331 Pa
Level 2	728	0.34152987 Pa
Level 3	995	0.34749202 Pa

Table 2. *Values for Different Levels of Mesh Refinement*

The parameter chosen to demonstrate independence of mesh refinement was the pressure difference between the inlet and the outlet. This is because pressure difference is one of the primary characteristics of the flow field of a fluid and it is directly related to the physical behaviour of the system. Also, pressure is sensitive to mesh refinements, particularly in areas of flow acceleration and deceleration and sudden geometry changes.

From table 2, we can see that the variation in pressure difference is very small between multiple iterations of mesh refinement, and it goes on decreasing as we refine the mesh further. We can hence say that the results are mesh independent. Below are the graphs for all the above mentioned cases:
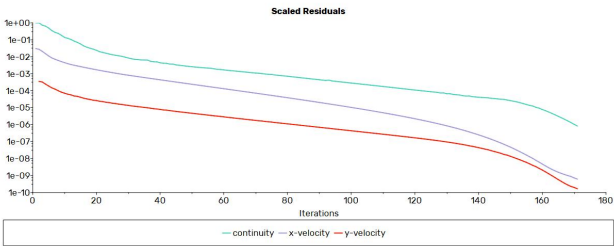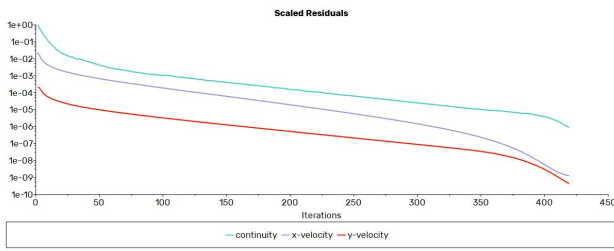

Figure 1. *No Mesh Refinement*
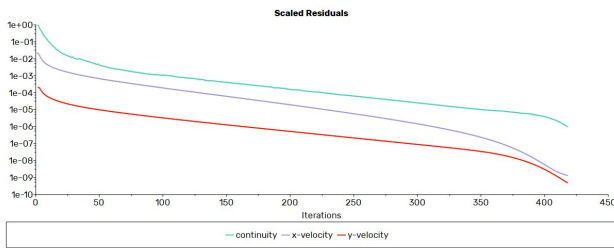
Figure 2. *Mesh Refinement Level 1*

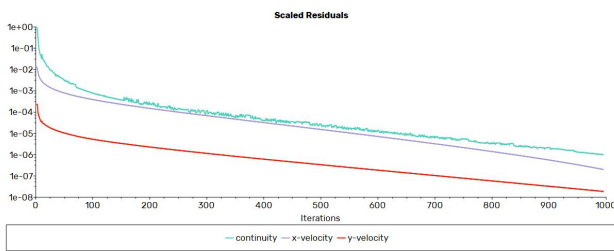**Figure 3.** *Mesh Refinement Level 2*

Figure 4. *Mesh Refinement Level 3*

4. Discretisation Schemes and Solution Methodology

4.1. Discretisation Schemes

The software used to perform this laboratory was ANSYS. The discretisation schemes used were:

1. **Finite Volume Method (FVM):** This is the core discretisation scheme used ANSYS's Fluent solver for CFD. Fluent discretises the computational domain into a mesh of control volumes, calculates the fluxes across the faces of these control volumes, and integrates the governing equations over each control volume to obtain numerical solutions.

2. **Spatial Discretisation Schemes:** Some spatial discretisation schemes like first and second order upwind schemes, quick scheme, etc. may have been used by the solver. Other than these, the Fluent solver also uses temporal discretisation schemes, turbulence modeling schemes and scalar transport schemes.

4.2. Solution Methodology

ANSYS's Fluent Solver uses the above discretisation schemes to obtain the solutions for the continuity, x-velocity and y-velocity.

The process followed to obtain the solution was streamlined by ANSYS as follows:

1. **Geometry:** Used to define the geometry of the domain to be analysed.

2. **Mesh:** Used to define the mesh which enables the discretisation of the geometrical domain.

3. **Setup:** Used to define boundary conditions and other parameters before computing the solution.

4. **Solution:** Computing the solution after defining all the required parameters.

5. **Results:** Post-processing the CFD simulation results to obtain useful analytical data.

5. Results and Discussion

5.1. Velocity and Pressure plots for *Re*=100

For Re=100, we need to calculate the inlet velocity:

$$Re = \frac{\rho v D}{\mu} \tag{1}$$
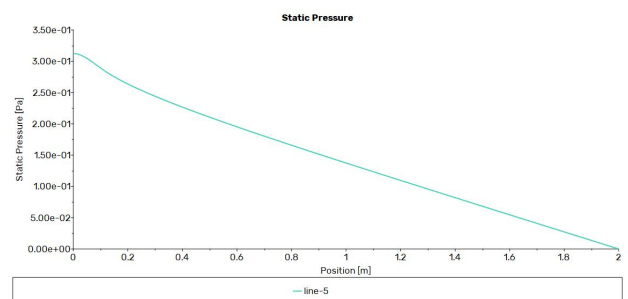
Rearranging the above equation, we get:

$$v = \frac{\mu Re}{\rho D} \tag{2}$$

Substituting the given values of μ, ρ, D and *Re*:

$$v = \frac{0.0172 \times 100}{860 \times 0.2}$$

$$\therefore v = 0.01 \text{ m/s}$$

Obtained Plots:

Figure 5. *Variation of Static Pressure along the centreline for Re=100*

There are some irregularities initially, but as we travel further down the pipe the static pressure varies linearly with a negative slope. The pressure decrease is actually the head loss which occurs due to viscous effects.
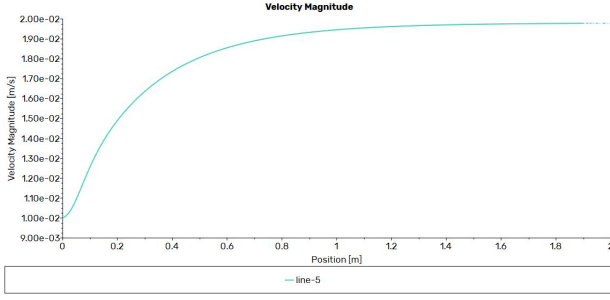
Figure 6. *Variation of Velocity along the centreline for Re=100*

The velocity increases until the flow is fully developed and the boundary layers have formed, after which it becomes almost constant.

5.2. Wall Shear Stress and Darcy's Friction Factor along the Wall (for Re=100)

We know that

$$\Delta P_L = f \frac{L}{D} \frac{\rho V_{avg}^2}{2} \tag{3}$$

where f is Darcy's Friction Factor[1]. If we rearrange the above equation, we get:

$$f = \frac{2\Delta P_L D}{\rho L V_{avg}^2} \tag{4}$$

$$\Delta P_L = P_{inlet} - P_{outlet}$$

We obtain the values of P_{inlet} and P_{outlet} to be 0.3323 Pa and 0 Pa respectively from the ANSYS Fluent Solver as shown:

Figure 7. *P_{inlet} and P_{outlet} for Re=100*

$$\therefore \Delta P_L = 0.3323 - 0 = 0.3323 \text{ Pa}$$

Now using equation 4, we get:

$$f = \frac{2 \times 0.3323 \times 0.2}{860 \times 2 \times (0.01)^2}$$

$$\therefore f = 0.7728$$

We also know that

$$f = \frac{8\tau_w}{\rho V_{avg}^2} \tag{5}$$

where τ_w is the wall shear stress. Rearranging the above equation, we get:

$$\tau_w = \frac{\rho f V_{avg}^2}{8} \tag{6}$$

Now using the above equation 6 and the calculated value of f, we get:

$$\tau_w = \frac{860 \times 0.7728 \times (0.01)^2}{8}$$

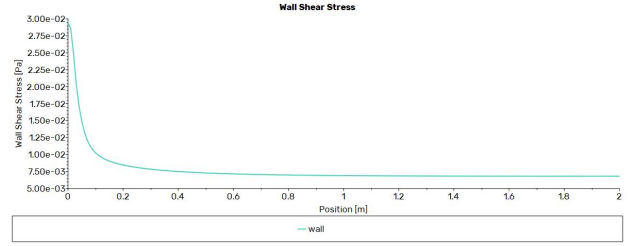$$\therefore \tau_w = 8.3076 \times 10^{-3} \text{ N/m}^2$$

Figure 8. *Variation of τ_w along the z-direction*

The wall shear stress is initially high, but it decreases as the boundary layer forms and finally reaches an almost consant value.

5.3. Darcy's Friction Factor vs. Reynolds Number

Re	v (in m/s)	ΔP_L (in Pa)	f
200	0.02	0.7646	0.4445
390	0.039	1.8031	0.2756
580	0.058	3.0612	0.2116
770	0.077	4.4981	0.1764
960	0.096	6.0898	0.1537

Table 3. *Values of v, ΔP_L and f for different values of Re*

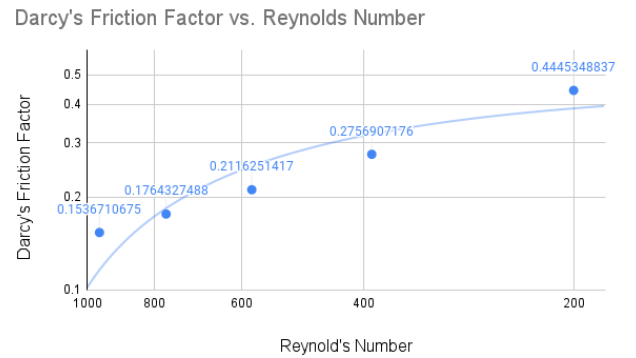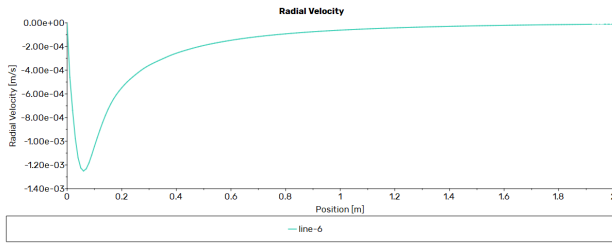The plot of Darcy's Friction Factor (f) vs. Reynolds Number (Re) on a logarithmic scale is as shown:

Figure 9. *f vs. Re on a logarithmic scale*

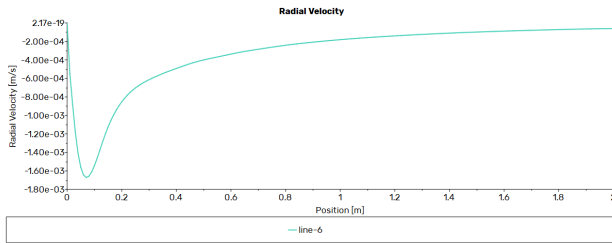## 5.4. Radial Velocity at along the z-direction at r=5cm

$$\boxed{Re\text{=}200}$$

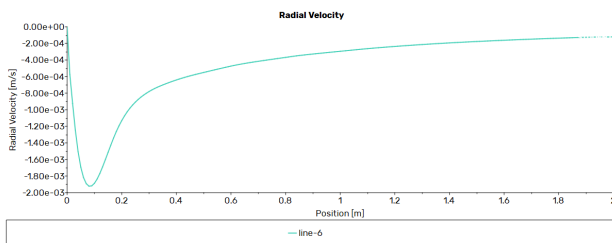Using equation 2, we get the axial velocity v as:

$$v = \frac{0.0172 \times 200}{860 \times 0.2} = 0.02 \text{ m/s}$$

Figure 10. *Radial Velocity at r=5cm for Re=200*

$$\boxed{Re\mathbf{=400}}$$

Using equation 2, we get the axial velocity v as:

$$v = \frac{0.0172 \times 400}{860 \times 0.2} = 0.04 \text{ m/s}$$

Figure 11. *Radial Velocity at r=5cm for Re=400*

$$\boxed{Re\mathbf{=600}}$$

Using equation 2, we get the axial velocity v as:

$$v = \frac{0.0172 \times 600}{860 \times 0.2} = 0.06 \text{ m/s}$$

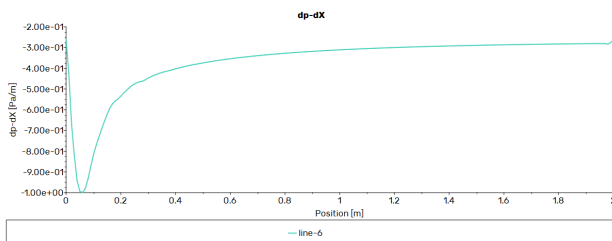**Figure 12.** *Radial Velocity at r=5cm for Re=600*
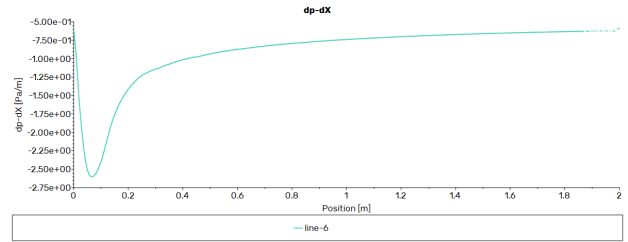
5.5. Pressure Gradient along the Centreline

We need to plot the pressure gradient along the centreline for three different values of Re.

$$\boxed{Re\mathbf{=200}}$$

Using equation 2, we get the axial velocity v as:

$$v = 0.02 \text{ m/s}$$

Figure 13. *Variation of Pressure Gradient along the centreline for Re=200*

$$\boxed{Re\mathbf{=400}}$$

Using equation 2, we get the axial velocity v as:

$$v = 0.04 \text{ m/s}$$

Figure 14. *Variation of Pressure Gradient along the centreline for Re=400*

$$\boxed{Re\mathbf{=600}}$$

Using equation 2, we get the axial velocity v as:

$$v = 0.06 \text{ m/s}$$
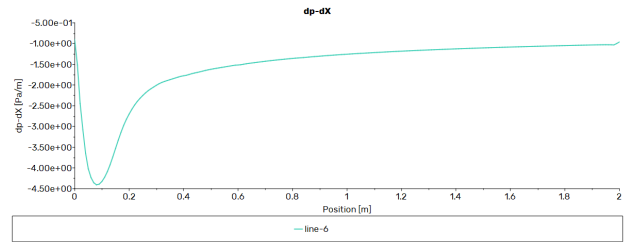
Figure 15. *Variation of Pressure Gradient along the centreline for Re=600*

In Figs. 13, 14 and 15, there is an initial dip before the pressure gradient becomes constant because initially the flow is not laminar and the flow conditions are unstable. After the flow conditions stabilise, the pressure gradient also stabilises.

5.6. Velocity Profile

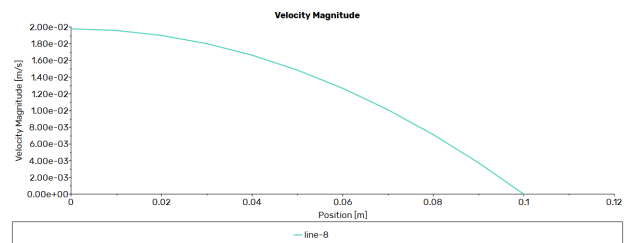Here is the velocity profile calculated using the CFD results:

Figure 16. *Velocity Profile of the Flow*

The following formula was used to calculate the velocity profile using analytical results:

$$u = \frac{dP}{dx}\frac{R^2}{4\mu}\left(1 - \frac{y^2}{R^2}\right) \tag{7}$$

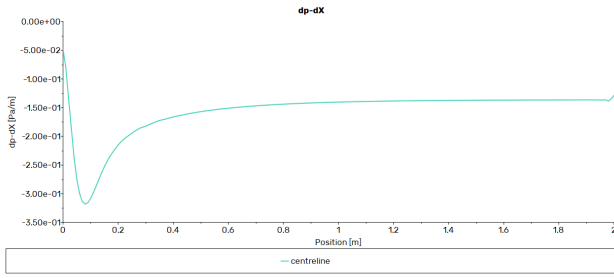To calculate the value of dP/dx, the following graph was used:

Figure 17. *dP/dx for Re=100*

Figure 19. *Entrance Length visualised for different Reynolds Numbers*

From the above graph, the value of dP/dx was obtained to to be -0.137 after the flow develops fully (i.e. when the graph almost becomes constant), and the following analytical velocity profile was obtained:
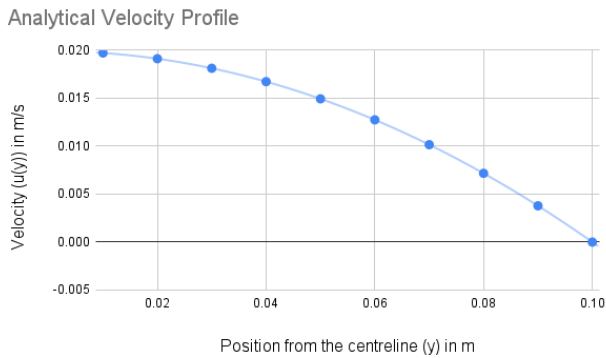
5.8. r and z Components of the Velocity and Streamline Contours
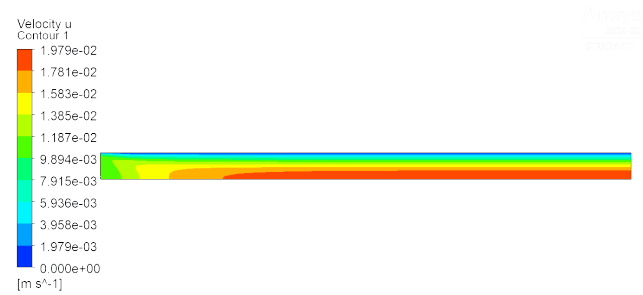
Figure 18. *Analytical Velocity Profile for Re=100*

Figure 20. *Axial Velocity (z-component) of the Velocity Contours*

5.7. Entrance Length of the Flow

To calculate the entrance length of the described laminar flow, we use the following formula:

$$L_{\text{entrance}} = 0.0575 Re_{\text{D}} D \tag{8}$$

Figure 21. *Radial Velocity (r-component) of the Velocity Contours*

[2] Here are the values of the entrance lengths of the flow for five different values of the Reynolds Number:

Re	L_{entrance} (in m)
200	2.3
390	4.485
580	6.67
770	8.855
960	11.04

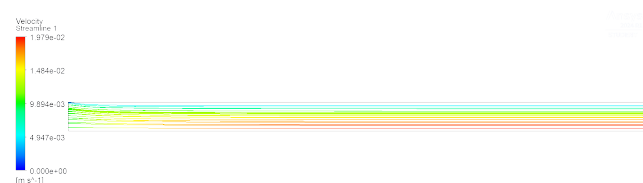**Table 4.** *Values of L_{entrance} for different values of Re*

Figure 22. *Streamline Contours*

5.9. Flow Features in the Contours
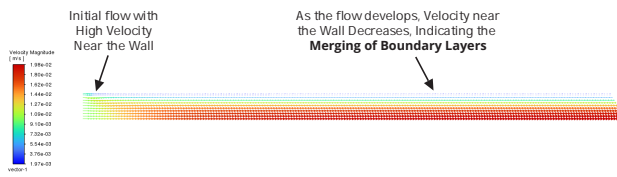
Figure 23. *Vector Field of the Flow Velocity*

Figure 24. *Strain Rate*

1. **Boundary Layer Development:**

   - *Velocity Vector Field (Fig. 22)* : Initially, near the entrance of the pipe, the velocity profile is relatively uniform with maximum velocity at the center and decreasing towards the walls due to the no-slip condition. As the fluid moves downstream, the velocity profile develops a parabolic shape. Near the pipe walls, the velocity gradients are higher due to the no-slip condition and the velocity decreases rapidly. This variation in velocity creates shear stress at the wall, initiating the boundary layer development.

   - *Strain Rate Contour (Fig. 23)*: The strain rate, which represents the rate of deformation within the fluid flow, is highest near the pipe wall where velocity gradients are the steepest. As fluid particles move along the boundary layer, they experience shear deformation due to the velocity gradient, causing the strain rate to be higher near the wall.

2. **Boundary Layer Merging:**

   - *Velocity Vector Field (Fig. 22)*: In laminar flow, the boundary layer thickness grows gradually along the pipe length due to the continuous accumulation of viscous effects. As the boundary layers from opposite walls grow, they eventually meet at the centerline of the pipe. At this point, the fluid flow in the center of the pipe has a much higher velocity compared to the fluid near the walls, which leads to a reduced velocity gradient at the centerline.

   - *Strain Rate Contour (Fig. 23)*: The strain rate near the centerline of the pipe is lower compared to the near-wall region due to the reduced velocity gradient. This indicates that the deformation rate of the fluid near the centerline is lower.
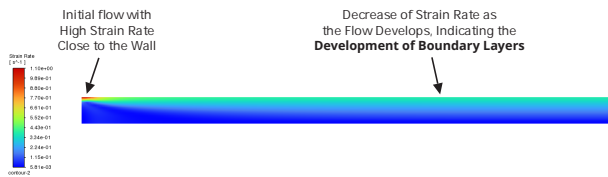
- *Merging*: When the boundary layers from opposite walls meet at the centerline, the decreasing velocity gradients near the centerline and the lower strain rates indicate a merging of the boundary layers. At this stage, the flow becomes fully developed, with a consistent velocity profile across the pipe diameter.

6. Conclusion

References

[1] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*, 3rd. New York, NY: McGraw-Hill Education, 2013, SI Units edition.

[2] Wikipedia, *Entrance length (fluid dynamics)*, [Online; accessed 31-March-2024]. [Online]. Available: https://en.wikipedia.org/wiki/Entrance_length_(fluid_dynamics)#:~:text=In%20fluid%20dynamics%2C%20the%20entrance,the%20flow%20becomes%20fully%20developed..