IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 4.819
Volume 7, Issue 1, July 2019

Hydrodynamic Analysis of Pipe Flow: Coupling Hagen–Poiseuille Theory with CFD Simulation

Rajanish Kumar Shukla
Department of Mathematics
V. B. S. Govt. Degree College, Campierganj, Gorakhpur (U.P.)
rajanish_shukla@rediffmail.com

Abstract: *Fluid dynamics is a fundamental branch of physics concerned with the behaviour of fluids (liquids and gases) in motion and at rest. It plays a crucial role in understanding a wide range of natural phenomena and engineering applications, including atmospheric circulation, ocean currents, aerodynamics, and industrial fluid transport processes. This study presents an overview of the governing principles of fluid motion based on the conservation laws of mass, momentum, and energy, typically expressed through the Navier–Stokes equations. The distinction between laminar and turbulent flow regimes is discussed, along with key dimensionless parameters such as Reynolds number that characterize flow behavior. Additionally, the effects of viscosity, pressure gradients, and external forces on fluid motion are examined. Advances in computational fluid dynamics (CFD) have significantly enhanced the ability to simulate complex flow systems, providing deeper insights into fluid behavior under varying conditions. The findings highlight the importance of fluid dynamics in optimizing system performance, improving energy efficiency, and addressing challenges in environmental and technological contexts..*

Keywords: *Fluid dynamics*.

I. INTRODUCTION

Fluid flow through pipes is a fundamental problem in fluid mechanics with wide-ranging applications in engineering systems such as water supply networks, oil and gas transportation, chemical processing, and biomedical devices. Understanding the behavior of fluid flow in conduits is essential for the efficient design and operation of these systems. The nature of pipe flow is primarily governed by the balance of inertial, viscous, and pressure forces, which are described by the Navier–Stokes equations [1].

One of the key aspects of pipe flow is the distinction between laminar and turbulent regimes, typically characterized by the Reynolds number. Laminar flow, occurring at low Reynolds numbers, is well described by the classical Hagen–Poiseuille theory, which predicts a parabolic velocity profile and a linear relationship between pressure drop and flow rate [2]. In contrast, turbulent flow is inherently complex due to chaotic fluctuations and enhanced mixing, requiring empirical correlations such as the Darcy–Weisbach equation and friction factor relations for accurate prediction [3].

Despite the availability of analytical solutions for simplified cases, many practical flow problems involve complex geometries and boundary conditions that necessitate numerical approaches. Computational Fluid Dynamics (CFD) has emerged as a powerful tool for analyzing such flows by solving the governing equations numerically using discretization techniques like the finite volume method [4]. Turbulence models, such as the k–ε model, further enhance the capability of CFD to simulate realistic flow behaviour.

In this study, a combined analytical and computational approach is employed to investigate fluid flow through a circular pipe. The objective is to analyze velocity distribution, pressure drop, and frictional characteristics under different flow regimes and to validate the results against established theoretical and empirical models.

II. METHODOLOGY

The investigation of fluid flow through a circular pipe was conducted using both analytical and computational approaches based on the fundamental principles of fluid mechanics. The flow was assumed to be steady, incompressible, and Newtonian unless otherwise specified.

The governing equations for the system include the continuity equation and the Navier–Stokes equations:

Continuity equation (mass conservation):

$$\nabla \cdot \mathbf{V} = 0$$

Navier–Stokes equation (momentum conservation):

$$\rho \left(\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} \right) = -\nabla P + \mu \nabla^2 \mathbf{V}$$

For fully developed, steady flow in a horizontal circular pipe, the velocity depends only on the radial coordinate, and the axial component simplifies the Navier–Stokes equation to:

$$\frac{1}{r} \frac{d}{dr} \left(r \frac{du}{dr} \right) = \frac{1}{\mu} \frac{dP}{dx}$$

Solving this equation with no-slip boundary conditions u = 0 at r = R, the velocity profile is obtained as:

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

The volumetric flow rate Q is then calculated by integrating the velocity profile over the pipe cross-section:

$$Q = \int_0^R 2\pi r u(r) \, dr = \frac{\pi R^4}{8\mu} \left(-\frac{dP}{dx} \right)$$

This leads to the Hagen–Poiseuille equation:

$$Q = \frac{\pi R^4 \Delta P}{8 \mu L}$$

To characterize the flow regime, the Reynolds number was evaluated:

$$Re = \frac{\rho u_{avg} D}{\mu}$$

where uavg = Q/A. Flow was considered laminar for Re < 2300 and turbulent for higher values.

For turbulent flow conditions, empirical relations such as the Darcy–Weisbach equation were used to estimate pressure drop:

$$\Delta P = f \frac{L}{D} \frac{\rho u_{avg}^2}{2}$$

where f is the friction factor, determined using correlations like the Blasius relation or Moody chart.

In addition to analytical treatment, CFD simulations were carried out. The pipe geometry was discretized into a structured mesh, and the governing equations were solved using the finite volume method. Appropriate boundary conditions were applied: uniform velocity at the inlet, constant pressure at the outlet, and no-slip condition at the pipe wall. For turbulent cases, suitable turbulence models (e.g., k–ε model) were employed. The results were analysed in terms of velocity profiles, pressure distribution, and frictional losses, and were validated against theoretical predictions such as the parabolic velocity profile for laminar flow and empirical correlations for turbulent flow [5]. Figure 1 shows Laminar and turbulent flow through the pipe of circular cross section of radius R.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 4.819
Volume 7, Issue 1, July 2019

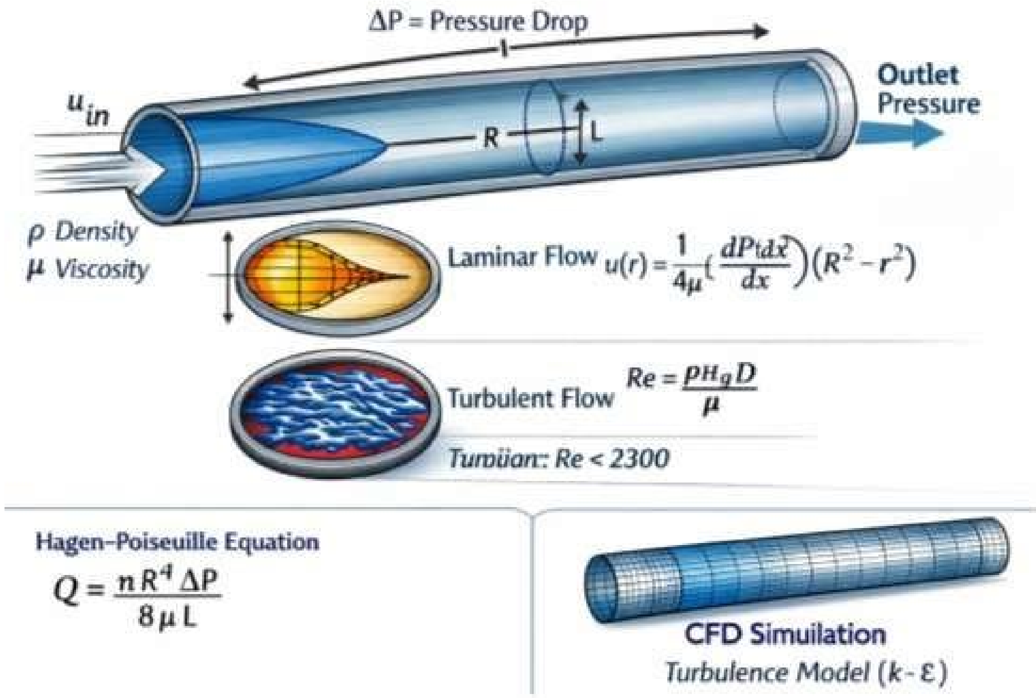

Figure 1: Laminar and turbulent flow through the pipe of circular cross section of radius R.

II. RESULTS AND DISCUSSION

The flow characteristics inside the circular pipe were analyzed for both laminar and turbulent regimes. For laminar flow ($Re < 2300$), the velocity profile obtained from analytical and numerical solutions exhibited a parabolic distribution, with maximum velocity at the centerline and zero velocity at the pipe wall due to the no-slip condition. The computed results were in excellent agreement with the Hagen–Poiseuille model, confirming the validity of the governing assumptions. The pressure drop along the pipe length was found to vary linearly with axial distance and directly proportional to fluid viscosity, consistent with theoretical predictions.

As the Reynolds number increased beyond the critical limit, the flow transitioned to a turbulent regime. In this case, the velocity profile became flatter in the core region with a steep gradient near the wall, indicating enhanced momentum transfer due to eddy formation. The pressure drop increased significantly compared to laminar flow and showed a nonlinear dependence on flow velocity. The friction factor estimated from the simulations closely followed empirical correlations, validating the applicability of the Darcy–Weisbach equation.

The CFD results further provided detailed insight into flow behavior. Contour plots of velocity and pressure revealed uniform flow development after a certain entrance length, beyond which fully developed conditions were achieved. The use of the k–ε turbulence model effectively captured the turbulent characteristics, although minor deviations from empirical values were observed at very high Reynolds numbers, likely due to model limitations.

Overall, the study demonstrates that analytical models accurately describe laminar flow behavior, while numerical methods are essential for capturing complex turbulent features. The combined approach ensures reliable prediction of velocity distribution, pressure drop, and frictional losses, which are critical for the design and optimization of pipe flow systems in engineering applications.

3.1 Graph Descriptions:

3.1.1 Velocity Profile Across Pipe Radius

The velocity distribution across the pipe radius as shown in Figure 2 illustrates a clear distinction between laminar and turbulent flow regimes. For laminar flow, the profile is parabolic, with maximum velocity at the centre line and zero velocity at the pipe wall due to the no-slip boundary condition. In contrast, turbulent flow exhibits a flatter velocity profile in the central region with a sharp gradient near the wall, indicating enhanced mixing and momentum transfer.

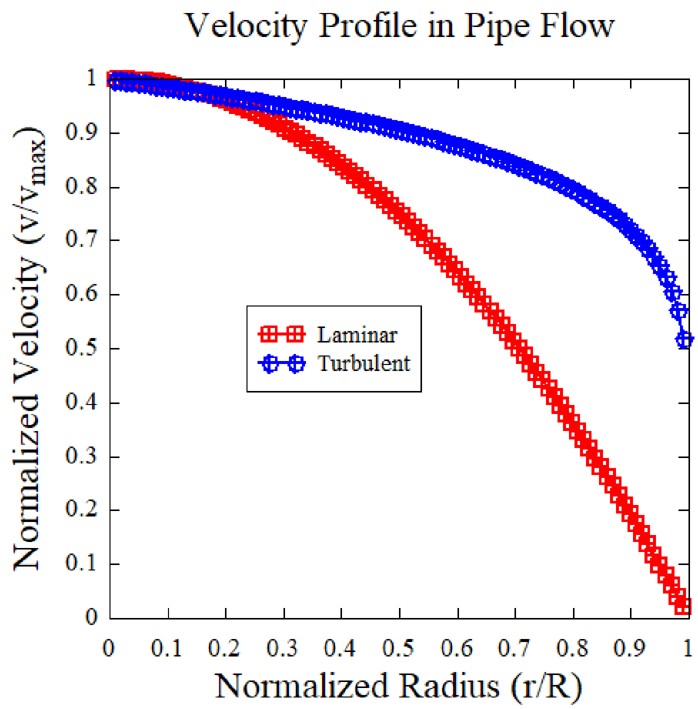

Figure 2: Velocity profile in a pipe of radius R.

3.1.2 Pressure Drop vs Pipe Length

The variation of pressure drop along the pipe length shows a linear decrease in pressure for laminar flow, consistent with the Hagen–Poiseuille relation. For turbulent flow, the pressure drop is significantly higher and deviates from linearity at higher velocities due to increased frictional losses and eddy formation.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 4.819
Volume 7, Issue 1, July 2019

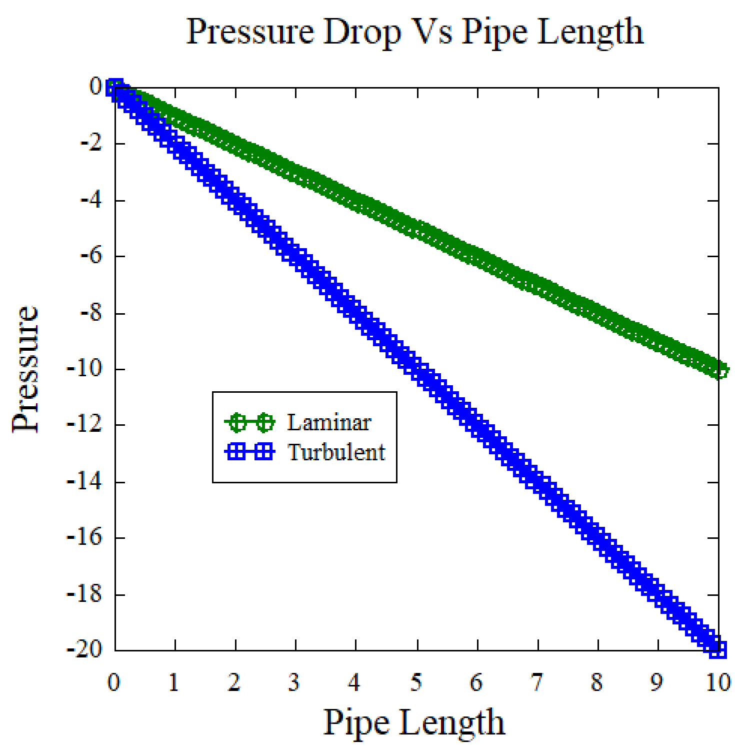

Figure 3: Drop in pressure in a pipe of radius R.

3.1.3 Friction Factor vs Reynolds Number

The plot of friction factor against Reynolds number demonstrates the transition between flow regimes. In the laminar region, the friction factor follows the analytical relation f = 64/Re, showing an inverse dependence on Reynolds number. In the turbulent region, the friction factor decreases more gradually and aligns with empirical correlations such as the Blasius equation, highlighting the complex dependence on flow conditions.

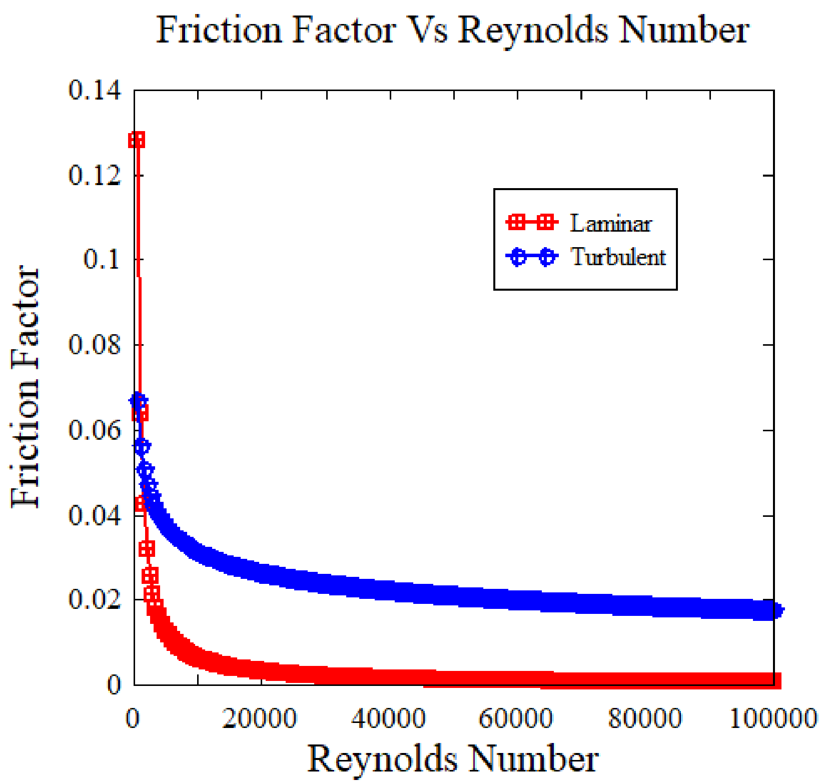

Figure 4: Friction factor vs Reynolds number in a pipe of radius R.

3.1.4 Velocity Contours (CFD Results)

The contour plots of velocity magnitude show the development of flow from the pipe inlet to the fully developed region. Near the entrance, the boundary layer grows along the pipe wall, eventually merging at the centerline. Beyond this entrance length, a stable and fully developed velocity profile is observed.

3.1.5 Pressure Contours (CFD Results)

The pressure contour plots indicate a continuous decrease in pressure along the axial direction of the pipe. The gradient is steeper for turbulent flow, confirming higher energy losses. The uniformity of pressure contours across cross-sections supports the assumption of fully developed flow.

IV. CONCLUSIONS

The present study systematically investigated fluid flow behaviour in a circular pipe under laminar and turbulent regimes using analytical formulations and computational techniques. The results confirm that laminar flow is accurately described by the Hagen–Poiseuille model, exhibiting a parabolic velocity profile and a linear pressure drop along the pipe length. In contrast, turbulent flow demonstrates a flattened velocity distribution and significantly higher pressure losses due to enhanced momentum transfer and eddy formation. The transition between flow regimes, characterized by the Reynolds number, plays a critical role in determining flow characteristics and energy losses. The friction factor analysis showed strong agreement with established theoretical and empirical correlations, validating the reliability of the adopted approach. Furthermore, CFD simulations effectively captured detailed flow features, including velocity distribution and pressure variation, particularly under turbulent conditions where analytical solutions are limited. Overall, the combined analytical and numerical methodology provides a robust framework for predicting pipe flow behavior. The findings are highly relevant for the design and optimization of fluid transport systems, contributing to improved efficiency and reduced energy consumption in engineering applications.

REFERENCES

[1]. White, F. M. Fluid Mechanics, 8th ed., McGraw-Hill, 2016.
[2]. Fox, R. W., McDonald, A. T., and Pritchard, P. J. Introduction to Fluid Mechanics, 9th ed., Wiley, 2015.
[3]. Çengel, Y. A., and Cimbala, J. M. Fluid Mechanics: Fundamentals and Applications, 3rd ed., McGraw-Hill, 2014.
[4]. Versteeg, H. K., and Malalasekera, W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed., Pearson, 2007.
[5]. Munson, B. R., Young, D. F., and Okiishi, T. H. Fundamentals of Fluid Mechanics, 7th ed., Wiley, 2013