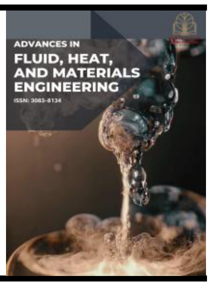
KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134




# Laminar Flow in a Circular Pipe: Verification of Hagen–Poiseuille Law

Ahmad Alif Ashraf Azwan[1*]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

**ARTICLE INFO**

***Article history:***
Received 15 October 2025
Received in revised form 12 December 2025
Accepted 16 December 2025
Available online 21 December 2025

***Keywords:***
Laminar flow; Hagen–Poiseuille law; CFD; circular pipe; pressure drop; velocity profile

**ABSTRACT**

This study presents a computational verification of the Hagen–Poiseuille law for steady, incompressible laminar flow through circular pipes using Computational Fluid Dynamics (CFD). The investigation aims to analyse the relationship between pressure drop, velocity distribution, and pipe diameter under laminar flow conditions. Three pipe models with diameters of 15 mm, 20 mm, and 25 mm, each with a length of 1.2 m, were simulated using ANSYS Fluent. The governing equations of continuity and momentum were solved using the Finite Volume Method (FVM), with water at 25°C as the working fluid. A uniform inlet velocity of 0.03 m/s was applied, while a pressure outlet boundary condition of 0 Pa was set at the exit. Mesh independence testing ensured numerical stability and accuracy. The geometry was discretized using a multizone mesh, and a grid independence test was performed to ensure numerical accuracy. Velocity contours revealed the development of the expected parabolic profile, while pressure results showed a linear decrease along the pipe length for all diameters. The theoretical pressure drops were compared with CFD predictions resulting in errors of 19.46%, 32.87%, and 82.70%. The deviation increased with diameter due to longer entrance length effects, yet the overall flow behaviour remained consistent with laminar theory. The study demonstrates that CFD can reliably reproduce laminar pipe flow characteristics and provides quantitative validation of the Hagen–Poiseuille relationship.

## 1. Introduction

The study of internal fluid flow in circular pipes is fundamental in fluid mechanics, particularly in applications involving microchannels, biomedical devices, heat exchangers, and pipeline transport systems [1-4]. Accurate prediction of pressure drops, velocity distribution, and wall shear stress is essential for validating theoretical laws such as the Hagen–Poiseuille equation. The Hagen–Poiseuille law represents one of the most fundamental analytical solutions in fluid mechanic which describes the behaviour of steady, incompressible, and fully developed laminar flow in a circular pipe [5-7].

In recent years, Computational Fluid Dynamics (CFD) has become a powerful tool for analysing internal pipe flows, providing detailed insight into flow behaviour, velocity gradients and pressure losses across different pipe diameters and flow conditions. Computational Fluid Dynamics (CFD) has

* *Corresponding author.*
*E-mail address: dd220034@student.uthm.edu.my*

https://doi.org/10.37934/afhme.7.1.3139a

become an indispensable tool for analysing internal flows due to its ability to resolve fluid velocity fields, wall shear stress, and pressure variations with high accuracy [8-11]. In CFD simulations, the computational domain is divided into a finite number of control volumes or elements where the governing equations are solved numerically. The accuracy of the results largely depends on how finely the domain is discretized.

Despite significant improvements in CFD technologies, achieving highly reliable results in internal flow simulation still presents challenges, particularly related to mesh quality, boundary condition selection, and appropriate modelling of the near-wall region. Previous studies have highlighted that mesh refinement is crucial for capturing the result accurately mainly for internal flow problems [12-15]. While finer meshes greatly enhance accuracy, they also increase computational cost. Therefore, balancing mesh density between the pipe centre and near-wall region is essential for cost-effective modelling.

To address this gap, the present study investigates the behaviour of steady, incompressible laminar flow in straight pipes of varying diameters using CFD. The simulations aim to validate the Hagen–Poiseuille law by examining the agreement between analytical and numerical predictions of pressure drop and velocity distribution. The simulations are conducted using ANSYS Fluent, where meshing, solver setup, and post-processing are performed.

## 2. Methodology

### *2.1 Geometry of Straight Pipe Design*

The design of this study is based on the development of laminar flow by considering the length of the pipe before the flow achieves a fully developed laminar flow [16]. Hence, the pipe length of 1.2m has been selected to analysed laminar flow behaviour and verifying the Hagen–Poiseuille law as shown in Figure 1. Three pipe diameters of the pipe which are 15 mm, 20 mm, and 25 mm in Table 1 are chosen to examine how the size of the pipe influences velocity distribution, pressure drop, and laminar flow characteristics. These diameters fall within a practical range commonly found in laboratory experiments, small-scale industrial piping, and fluid transport components, allowing the results to reflect realistic engineering conditions [17]. The computational fluid domain corresponds directly to the internal volume of the pipe. Since internal pipe flow is fully constrained by solid walls, no external domain is required. The pipe length of 1.2 m is sufficiently long to allow the development of the velocity profile and to analyse the pressure drop along the pipe for the range of inlet velocities considered.

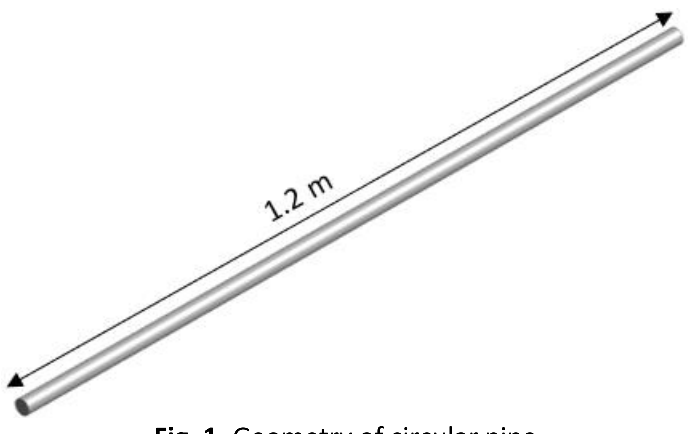



**Fig. 1.** Geometry of circular pipe

**Table 1**
Straight pipe specification

| Model | Length, m | Diameter, m |
|---|---|---|
| A | 1.2 | 0.015 |
| B | 1.2 | 0.020 |
| C | 1.2 | 0.025 |

### *2.2 Discretization*

In this study, the circular pipe was discretized to enable numerical solution of the governing fluid flow equations. The continuous domain was divided into finite control volumes using a structured mesh, ensuring adequate resolution in both the radial and axial directions. This discretization approach allows the simulation to approximate the parabolic velocity profile predicted by Hagen–Poiseuille law while balancing computational efficiency and solution accuracy.

#### *2.2.1 Mesh generation of the circular pipe*

The computational domain of the circular pipe was discretized using the path-conforming meshing method to ensure that the mesh accurately follows the curved geometry of the pipe walls. A multizone mesh was employed to provide flexibility in capturing complex geometrical features, while body sizing controls were applied to maintain consistent element sizes throughout the domain as shown in Figure 2. This approach allows for a finer mesh near the pipe walls to optimize computational efficiency. The combination of path-conforming method and body sizing ensures a balance between solution accuracy and computational cost. Mesh quality metrics such as skewness and orthogonality were monitored to ensure numerical stability and reliability of the simulation results.

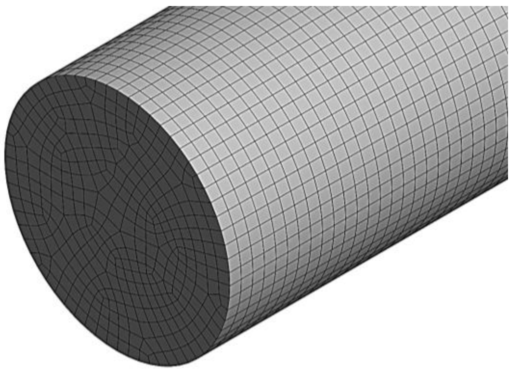


**Fig. 2.** Computational mesh for fluid domain

#### *2.2.2 Grid independence test*

The Grid Independence Test (GIT) is a crucial step in Computational Fluid Dynamics (CFD) analysis used to ensure that the numerical results are not dependent on the mesh size or grid resolution. In the grid independence process, several mesh sizes are tested by running simulations under the same boundary conditions and physical parameters. Key output parameters, such as velocity, pressure drop, or mass flow rate are then compared across the different mesh levels. When further refinement of the mesh produces negligible changes in these parameters, the solution is considered grid independent. This procedure helps balance accuracy and computational efficiency by selecting a mesh that provides reliable results without unnecessary computational cost [18-20].

### *2.3 Governing Equation*

The simulation of fluid flow inside the straight pipe is modelled as steady-state, laminar, incompressible, and fully developed internal flow. The fluid properties such as density and viscosity are assumed uniform throughout the pipe. The numerical solution is based on the Navier–Stokes equations and solved using the Finite Volume Method (FVM). The governing equations used in this study are the continuity equation and the momentum equation for incompressible flow. For incompressible flow, the continuity equation ensures that mass is conserved:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{1}$$

In vector form, $\nabla \cdot \vec{V} = 0$ (2)

where $\vec{V} = (u, v, w)$ is the velocity vector and $\nabla$ represents the spatial derivative operator. The incompressible Navier–Stokes equation for steady internal flow is expressed as:

$$\rho(\vec{V} \cdot \nabla\vec{V}) = -\nabla P + \mu\nabla^2\vec{V} \tag{3}$$

where $\rho$= fluid density (kg/m³), $\mu$= dynamic viscosity (Pa·s), $P$= pressure (Pa). For fully developed laminar pipe flow (Hagen–Poiseuille condition), axial velocity develops in the $x$-direction only, and the simplified form becomes:

$$\frac{dP}{dx} = \mu\frac{d^2u}{dr^2} \tag{4}$$

This equation forms the basis for the analytical Hagen–Poiseuille solution.

The pressure drop in a circular pipe under laminar flow can be predicted using:

$$\Delta P = \frac{128\mu LQ}{\pi D^4} \tag{5}$$

where $L$= pipe length (m), $D$= pipe diameter (m), $Q$= volumetric flow rate (m³/s), $V_{\text{avg}}$= average flow velocity (m/s). This analytical equation is used as the reference solution for validating CFD results. The predicted pressure drops and velocity profile from CFD are compared against these theoretical equations. These comparisons are crucial to evaluate simulation accuracy and to verify whether the pipe flow behaviour follows the Hagen–Poiseuille law under laminar conditions.

### *2.4 Boundary Condition*

The boundary conditions were defined to ensure a steady-state and fully developed laminar regime consistent with the Hagen–Poiseuille assumptions. At the inlet, a uniform velocity boundary condition was imposed with velocity of 0.03 m/s for all pipe diameters. At the outlet, a pressure outlet boundary condition was specified, with the gauge pressure set to 0 Pa. This setup allows the solver to naturally determine the static pressure distribution along the pipe length. The use of a zero-pressure outlet ensures that pressure variations within the domain result solely from viscous effects, consistent with laminar flow assumptions.

The pipe wall was defined as a no-slip boundary condition, where the fluid velocity at the wall surface was set to zero (u = 0). The wall roughness was neglected, assuming an ideal smooth internal surface, which aligns with theoretical Hagen–Poiseuille flow conditions. The operating pressure was maintained at 1 atm. (101325 Pa) to simulate standard atmospheric conditions, and the flow was treated as steady state. The laminar viscous model in ANSYS Fluent was used, with double-precision calculation enabled to minimize numerical errors.

### *2.5 Analysis*
#### *2.5.1 Velocity analysis*

Velocity is a fundamental parameter in understanding fluid behaviour within pipes, as it directly reflects how the flow develops and responds to viscous forces. Extracting the velocity distribution allows for the identification of the entrance length, the development of the parabolic profile characteristic of fully developed laminar flow, and the influence of pipe diameter on flow gradients. By analysing the velocity profiles along the pipe, it is possible to verify whether the CFD model accurately captures the balance between viscous resistance and the pressure driving force, as described by the Hagen–Poiseuille law. This ensures that the simulation provides reliable insight into the flow dynamics, particularly the maximum centreline velocity and shear effects near the pipe walls which are critical for validating laminar flow behaviour.

#### *2.5.2 Pressure analysis*

Pressure is another critical variable in pipe flow studies, as it quantifies the hydraulic resistance and energy loss caused by viscous effects. Extracting the pressure distribution along the pipe enables evaluation of pressure drops for different pipe diameters and allows direct comparison with theoretical predictions from the Hagen–Poiseuille equation. Monitoring the pressure also confirms the flow regime and validates that the solver maintains steady-state conditions. By analysing pressure along with velocity, the study ensures a comprehensive assessment of the CFD model's ability to reproduce classical laminar pipe flow, providing a basis for evaluating the influence of pipe geometry on hydraulic performance.

## **3. Results**
### *3.1 Grid Independence Test Result*

The grid independence test was conducted to ensure that the simulation results are not significantly affected by the mesh density. Figure 3 shows the velocity distribution along the pipe length for three different mesh configurations, Mesh 1 (98228 nodes), Mesh 2 (332836 nodes), and Mesh 3 (102926 nodes). It can be observed that the velocity profiles for the finer meshes closely overlap, indicating minimal variation in the predicted velocity once the mesh was sufficiently refined. Mesh 1 shows a slight deviation near the entrance region, suggesting a lower resolution in capturing the developing flow. Since the difference between Mesh 2 and Mesh 3 is negligible, Mesh 2 was selected for the subsequent simulations to achieve a balance between computational accuracy and efficiency.

Pressure was chosen as the key parameter for the grid independence test because it gives a clear indication of how mesh refinement affects the accuracy of the simulation. In pipe flow, even small changes in mesh size can influence how the pressure gradient develops along the length of the pipe. Since this study focuses on verifying the Hagen–Poiseuille law it is important to make sure the pressure results are not affected by the mesh density. By checking that the pressure drop remains nearly the same for different mesh sizes, the simulation can be confirmed to produce a reliable result for further comparison with theoretical predictions.

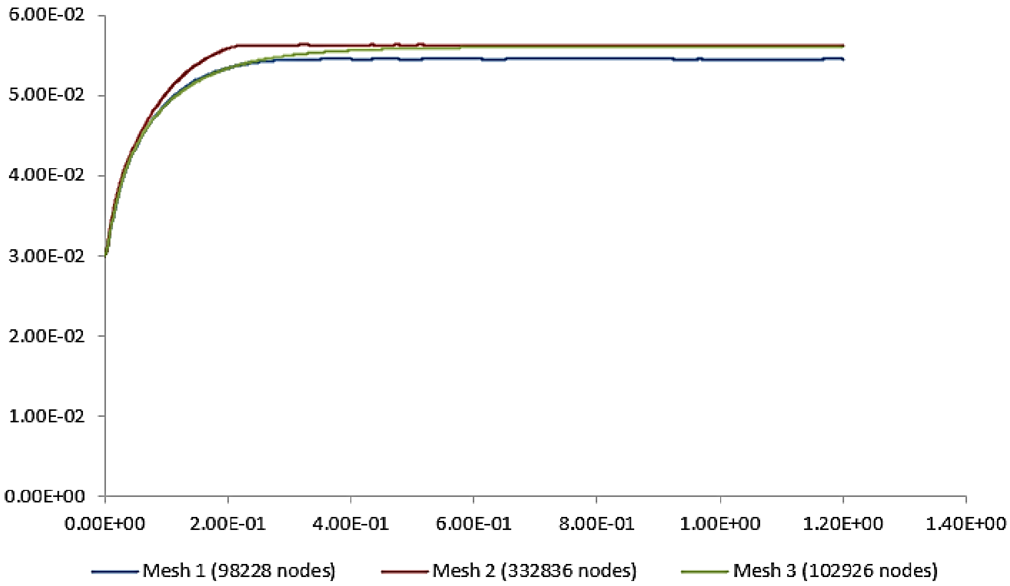
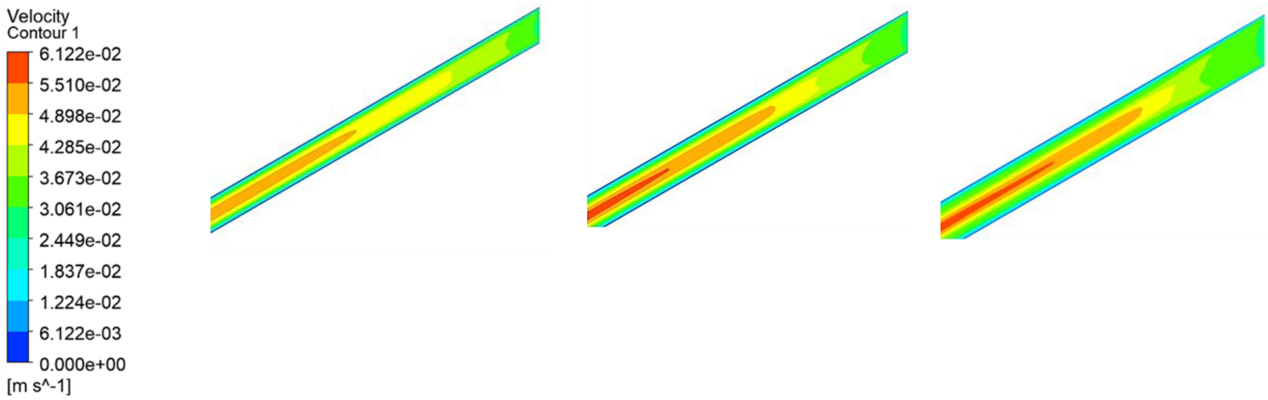



**Fig. 3.** Pressure chart for grid independence test

### *3.2 Velocity Contour of Three Different Pipe Diameter*

The velocity contours obtained from the CFD simulation show the characteristic parabolic velocity distribution expected for fully developed laminar flow in circular pipes as shown in Figure 4. Near the pipe inlet, the flow initially exhibits a relatively uniform velocity profile. However, as the fluid progresses downstream, viscous effects along the wall gradually decelerate the near-wall region, allowing the centreline velocity to increase and form the classical parabolic shape. This developing region corresponds to the entrance length, which depends on the pipe diameter and Reynolds number.




(a) (b) (c)

**Fig. 4.** Velocity contour (a) 15 mm (b) 20 mm (c) 25 mm circular pipe

The velocity profile remains unchanged, confirming that the flow becomes fully developed before reaching the outlet. Among the three pipe diameters analysed, the 15 mm pipe displays the highest velocity gradient near the wall and the largest maximum velocity at the centreline due to greater viscous influence, while the 20 mm and 25 mm pipes exhibit more uniform distributions with smaller gradients. The smooth and symmetric contours confirm that the flow remains stable and laminar throughout the domain, demonstrating that the CFD simulation effectively reproduces the theoretical velocity behaviour predicted by the Hagen–Poiseuille law.

*3.3 Pressure Contour of Three Different Pipe Diameter*

The results obtained from the CFD simulation as shown in Figure 5 demonstrate the expected pressure behaviour for laminar flow in circular pipes of varying diameters. For all three pipe sizes, the pressure decreases steadily along the length of the pipe, confirming fully developed behaviour. The 15 mm pipe shows the highest pressure drop due to greater frictional resistance caused by its smaller hydraulic diameter. The 20 mm and 25 mm pipes experience progressively lower pressure gradients, which is consistent with the theoretical inverse fourth-power relation between diameter and pressure drop.

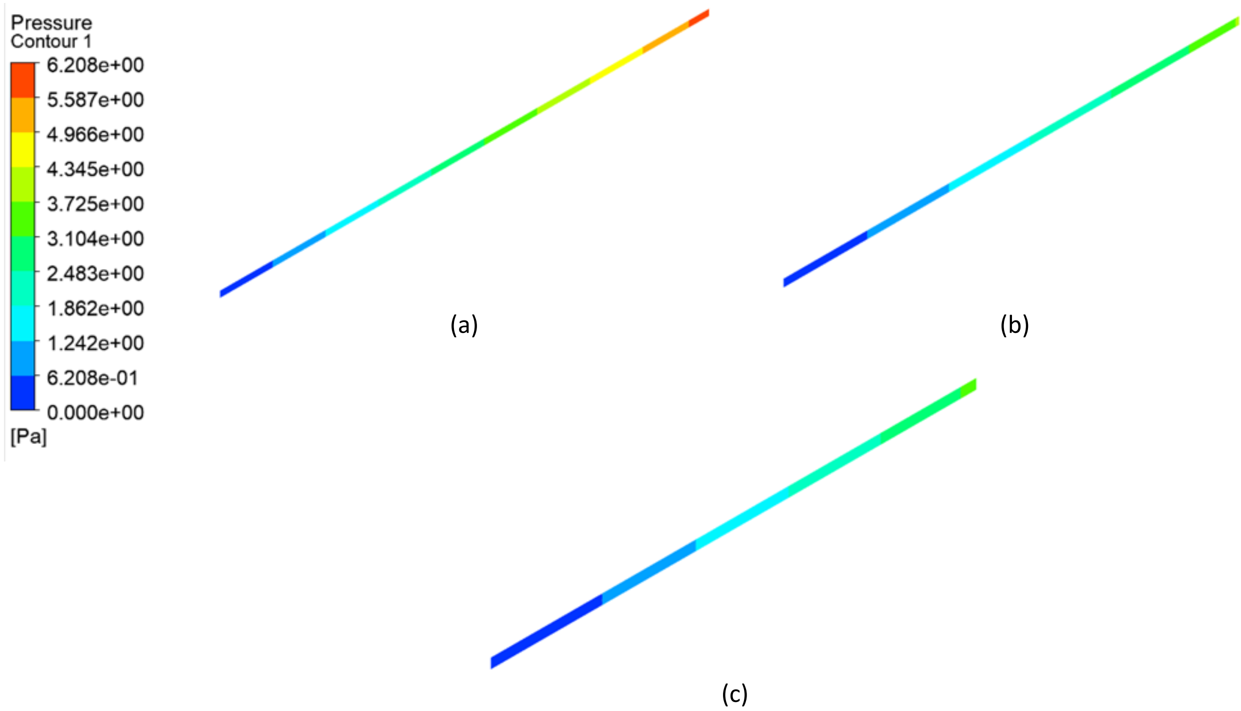


**Fig. 5.** Pressure contour of (a) 0.15 mm (b) 0.20 mm (c) 0.25 mm circular pipe

The pressure contours from ANSYS Fluent also show uniform flow distribution with no signs of turbulence or flow instability. This confirms that the chosen inlet velocity of 0.03 m/s results in a Reynolds number within the laminar regime for all diameters. The simulation results support the theoretical understanding that pressure losses in laminar pipe flow are dominated by viscous effects and follow a linear trend along the pipe wall. Overall, the CFD outputs clearly demonstrate how pipe diameter affects energy losses in laminar flows and validate the use of ANSYS Fluent for modelling Hagen–Poiseuille behaviour.

*3.4 Verification of Hagen Poissule Law*

After confirming mesh independence, the CFD results were verified by comparing the simulated pressure drop with the theoretical values obtained from the Hagen–Poiseuille equation which is equation 5. This comparison ensures that the simulation accurately captures the physics of laminar flow in a circular pipe. The comparison of the results is shown in Table 2. The small percentage error observed can be attributed to minor discretization and numerical rounding effects, which are acceptable within the accuracy range of CFD simulations. The percentage error between simulation and theory can be calculated using:

$$\frac{|\Delta P_{CFD} - \Delta P_{theoretical}|}{\Delta P_{theoretical}} \times 100\% \tag{6}$$

**Table 2**
Comparison between theoretical and simulated pressure drops for different pipe diameters

| Model | Diameter, m | Theoretical ΔP (Pa) | Simulated ΔP (Pa) | Percentage error (%) |
|---|---|---|---|---|
| A | 0.015 | 5.14 | 6.14 | 19.46 |
| B | 0.020 | 2.89 | 3.84 | 32.87 |
| C | 0.025 | 1.85 | 3.38 | 82.70 |

The comparison shows that the pressure drop decreases as the pipe diameter increases, which is consistent with the theoretical relationship where ΔP is inversely proportional to the square of the diameter. For the smallest diameter (15 mm), the simulated pressure drop is close to the theoretical value, with an error of 19.46%. This relatively small deviation indicates that the flow is nearly fully developed, and the simulation closely follows the laminar flow assumptions used in the Hagen–Poiseuille derivation. However, as the diameter increases to 20 mm and 25 mm, the percentage error rises to 32.87% and 82.70% respectively. The larger discrepancy is mainly due to the increased entrance length required for fully developed laminar flow at larger diameters. Because the pipe length remains constant, a greater portion of the flow is still developing, causing the simulated pressure drop to be higher than the theoretical prediction. Additionally, numerical factors such as mesh resolution and discretization effects contribute to the difference. Overall, although some deviations exist, the simulation results successfully capture the expected trend and demonstrate consistent laminar flow behaviour in line with the Hagen–Poiseuille law.

## 4. Conclusions

In conclusion The CFD analysis successfully achieved the objective of verifying the Hagen–Poiseuille law for laminar flow in circular pipes of 15 mm, 20 mm, and 25 mm diameters. The simulated pressure drops of 6.14 Pa, 3.84 Pa, and 3.38 Pa showed the same decreasing trend as the theoretical predictions of 5.14 Pa, 2.89 Pa, and 1.85 Pa, confirming that pressure loss is inversely related to pipe diameter. Although the percentage errors increased from 19.46% to 32.87% and 82.70% for larger diameters due to longer developing-flow regions, the simulation still captured the expected laminar behaviour, including the fully developed parabolic velocity profile and linear pressure gradient. Overall, the results quantitatively validate that the CFD model can accurately represent laminar pipe flow and meet the study's objective of confirming the Hagen–Poiseuille relationship.

## References

[1] Hummad, Hytham. "The role of physics in anesthesia equipment design: A systematic review of principles, challenges, and innovations." *Vascular and Endovascular Review* 8, no. 2s (2025): 119-126.

[2] Wada, Yuki, Yasuteru Sibamoto, and Takashi Hibiki. "Heat transfer characteristics of downward saturated boiling flow in vertical round pipes." *International Journal of Heat and Mass Transfer* 239 (2025): 126598. https://doi.org/10.1016/j.ijheatmasstransfer.2024.126598

[3] Li, Longjie, Qianfan Tang, Xuejin Chen, and Can Weng. "Polymer-based pin-fin microchannel heat exchangers: A comparative study of material and structural effects on performance." *International Journal of Thermal Sciences* 209 (2025): 109546. https://doi.org/10.1016/j.ijthermalsci.2024.109546

[4] Tang, Ye, Hu-Jie Zhang, Li-Qun Chen, Qian Ding, Qiuyang Gao, and Tianzhi Yang. "Recent progress on dynamics and control of pipes conveying fluid." *Nonlinear Dynamics* 113, no. 7 (2025): 6253-6315. https://doi.org/10.1007/s11071-024-10486-1

[5] Bacelar, Mateus D, Hugo, Rajai S Alassar, and André B Lopes. 2023. "Hagen-poiseuille flow in a quarter-elliptic tube." Fluids 8 (9): 247-47. https://doi.org/10.3390/fluids8090247

[6] Sigalotti, Leonardo Di G., and Enrique Guzmán. "An overview of viscous and highly viscous fluid flows in straight and elbow pipes: I—single-phase flows." *Fluids* 10, no. 5 (2025). https://doi.org/10.3390/fluids10050125

[7] Zheng, Congren, Yong Chen, and Zijing Ding. "Modal and non-modal stability for Hagen–Poiseuille flow with non-ideal fluid." *Physics of Fluids* 36, no. 8 (2024). https://doi.org/10.1063/5.0205600

[8] Li, Yuanwen. "CFD-DEM analysis of internal flow characteristics in multistage deep-sea mining pumps." *Ships and Offshore Structures* 20, no. 11 (2025): 1612-1625. https://doi.org/10.1080/17445302.2024.2397604

[9] Freile, Ramiro, Mauricio E. Tano, and Jean C. Ragusa. "CFD assessment of RANS turbulence modeling for solidification in internal flows against experiments and higher fidelity LBM-LES phase change model." *Annals of Nuclear Energy* 197 (2024): 110275. https://doi.org/10.1016/j.anucene.2023.110275

[10] Zhang, Xuebin, and Youjun Tao. "CFD simulation on internal flow field of typical hydrocyclone for coal and development of novel hydrocyclone." *Particulate Science and Technology* 42, no. 3 (2024): 469-481. https://doi.org/10.1080/02726351.2023.2261391

[11] Al-Obaidi, Ahmed Ramadhan, Jassim Alhamid, and Hussam Khalaf. "Unsteady behaviour and plane blade angle configurations' effects on pressure fluctuations and internal flow analysis in axial flow pumps." *Alexandria Engineering Journal* 99 (2024): 83-107. https://doi.org/10.1016/j.aej.2024.04.048

[12] Zhu, Yongzheng, Shiji Zhao, Yuanye Zhou, Hong Liang, and Xin Bian. "An unstructured adaptive mesh refinement for steady flows based on physics-informed neural networks." *Journal of Computational Physics* (2025): 114283. https://doi.org/10.1016/j.jcp.2025.114283

[13] Perera, Roberto, and Vinamra Agrawal. "Multiscale graph neural networks with adaptive mesh refinement for accelerating mesh-based simulations." *Computer Methods in Applied Mechanics and Engineering* 429 (2024): 117152. https://doi.org/10.1016/j.cma.2024.117152

[14] Li, Yezhan, and Tsubasa Okaze. "Application of polyhedral meshes in large-eddy simulation of building array flow fields within neutral and unstable boundary layers." *Building and Environment* 263 (2024): 111846. https://doi.org/10.1016/j.buildenv.2024.111846

[15] Daeian, Mohammad Ali, W. Spencer Smith, and Zahra Keshavarz-Motamed. "A multi-domain lattice Boltzmann mesh refinement method for non-Newtonian blood flow modeling." *Physics of Fluids* 37, no. 1 (2025). https://doi.org/10.1063/5.0241118

[16] Najim, Farqad T. "Effect of inlet velocity on the entrance length of laminar and turbulent flow in a circular pipe." *Babylonian Journal of Mechanical Engineering* 2024 (2024): 81-90. https://doi.org/10.58496/BJME/2024/010

[17] BinaPlast, *PVC-U (4 Products)* (BinaPlast, n.d.), https://www.binaplast.com/catalog/pdf/PVC-U(4Products).pdf

[18] Liu, Huan, Farhad Hussain, Chew Lim Tan, and Manoranjan Dash. "Discretization: An enabling technique." *Data Mining and Knowledge Discovery* 6, no. 4 (2002): 393-423. https://doi.org/10.1023/A:1016304305535

[19] Zulkifli, Zulaika, NH Abdul Halim, Z. H. Solihin, J. Saedon, A. A. Ahmad, A. H. Abdullah, N. Abdul Raof, and M. Abdul Hadi. "The analysis of grid independence study in continuous disperse of MQL delivery system." *Journal of Mechanical Engineering and Sciences* (2023): 9586-9596. https://doi.org/10.15282/jmes.17.3.2023.5.0759

[20] Almohammadi, K. M., D. B. Ingham, L. Ma, and M. Pourkashan. "Computational fluid dynamics (CFD) mesh independency techniques for a straight blade vertical axis wind turbine." *Energy* 58 (2013): 483-493. https://doi.org/10.1016/j.energy.2013.06.012