FOSSEE, IIT Bombay
OpenFOAM Case Study Project
June 15, 2025

Analysis of flow through an orifice plate of different shapes using OpenFoam

Saarth Shukla

National Institute of Technology, Delhi

Plot No. FA7, Zone P1,

GT Karnal Road, Delhi-110036

231310032@nitdelhi.ac.in

Abstract

Orifice meters are essential for fluid flow measurement due to their durability, simplicity, and reliability. However, the size and shape of the orifice plate significantly influence the fluid dynamics, including pressure drop and velocity distribution. This study investigates the effect of orifice plate geometry using OpenFOAM for Computational Fluid Dynamics (CFD) simulations. Three shapes—circular, curved, and sharp-edged—were examined for three different inlet velocities u = 0.376 m/s, 0.482 m/s, 0.673 m/s, with water as the working fluid in a cylindrical pipe. Simulations provided insights into flow behaviour, focusing on pressure loss, velocity profiles. The results illustrate the effect of shape on flow parameters which can help in designing the orifice plate.

1. Introduction

The orifice plate is a cost-effective flow measurement device, known for its simple construction and ease of installation in pipelines. It operates on the principle of pressure and velocity changes caused by a reduction in the flow area. Orifices are widely adopted in industrial flow measurement due to their robustness, simplicity, and proven advantages. Extensive research exists on the effects of orifice geometry, including studies on discharge coefficients, pressure drops, and vena-contracta dynamics.

CFD tools, particularly OpenFOAM, have emerged as reliable alternatives to experimental methods for analysing orifice flow behaviour. Prior studies have explored conventional orifice geometries, focusing on factors like flow separation and energy losses. Despite their advantages, orifice plates are characterized by significant unrecovered pressure drops, primarily due to form friction.

This study extends previous work (Shah et al.2012) by investigating the influence of orifice shape—circular, curved, and sharp-edged—on flow characteristics. Simulations are performed using OpenFOAM to analyse velocity profiles, pressure drops for water flow in a cylindrical pipe. The outcomes provide insights for optimizing orifice design to enhance performance and accuracy in industrial applications, especially in scenarios where experiments are complex or resource intensive.

2. Problem Statement

Orifice plates are widely used in flow measurement due to their simplicity and cost-effectiveness. However, a significant drawback is the substantial pressure drop across the orifice, which results in energy losses. This pressure drop not only reduces system efficiency but also limits the scope of their application in scenarios where energy conservation is critical. Understanding and optimizing the flow characteristics around orifice plates, particularly the pressure drop, is essential to improve their performance. This study focuses on analysing the impact of different orifice geometries to identify configurations that minimize pressure losses while maintaining measurement accuracy. This study is based on the simulation results reported by Tukiman et al. 2017.

2.1 Objectives and scope of present work

The study evaluates the characteristics of water flow through a cylindrical pipe with an orifice plate using OpenFOAM. Specifically, the velocity and pressure profiles are analysed to understand the flow behaviour. The simulations consider three different inlet velocities 0.376 m/s, 0.482 m/s, 0.673 m/s, representing turbulent regimes Tukiman et al. (2017).

The base geometry considered in the current work is of Tukiman et al. (2017). As part of the work, the geometry of the orifice plate was also varied with curved, and sharp-edged shapes, keeping the thickness of the orifice plate constant. The diameter of the orifice is fixed at 5 mm, while the pipe has a diameter of 12.3 mm.

The inlet velocity of water corresponding to these Reynolds numbers is determined based on the fluid's kinematic viscosity at 25°C, which is 1×10^{-6} m^2/s. The numerical results, including velocity profiles, pressure drops, are validated using experimental data available in Tukiman et al. 2017.

3. Mathematical Model

Steady state 3D Reynolds averaged mass and momentum balance equation was employed to describe the flow through orifice system. The fluid flow was considered to be of constant density. In the present work, standard k-epsilon turbulence model has been used. The k-epsilon turbulence model is simple to use, most widely validated and has excellent performance for many industrially relevant flows (Versteeg and Malalasekera (1995)).

Continuity equation (Mass conservation) -

$$\nabla \cdot u = 0 \tag{3.1}$$

Momentum Conservation Equation -

$$(u \cdot \nabla)u = -(1/\rho)\nabla p + \nu\nabla^2 u \tag{3.2}$$

4. Simulation Procedure

4.1 Geometry and Mesh

The geometry of the pipe consists of an upstream flow length of 246mm and total length of pipe is 494 mm. The orifice plate has a throat with a diameter of 5 mm and the length is 2 mm. The pipe has 12.30 mm inlet diameter. Adequate length at the upstream and downstream of the orifice plate is to avoid the effect of patched boundary

conditions on the flow through the orifice. This also provides fully developed turbulent flow in the upstream of the orifice plate. The model of the orifice is shown in figure 1. The meshing for the simulation was performed using ANSYS Fluent to ensure a high-quality mesh suitable for accurate Computational Fluid Dynamics (CFD) analysis. The mesh sizes for the three cases can be seen in table 1. Key mesh quality parameters such as aspect ratio (maximum: 8.49), skewness (maximum: 0.73), and non-orthogonality (maximum: 70.79, with only one face exceeding 70°) were well within acceptable ranges for CFD simulations. The geometries for circular plate, curved edged and sharp-edged orifice can be seen in figure 2, 3, 4 respectively. And the details of the mesh for the three geometries can be seen in Table 1.

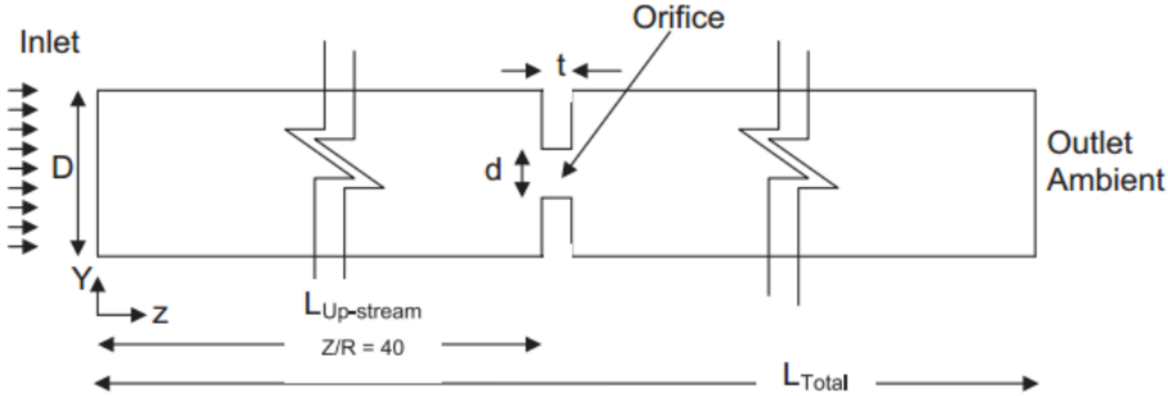
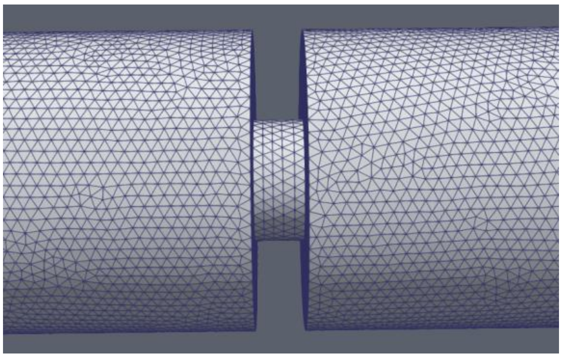


Figure 1. Typical orifice geometry considered for simulation



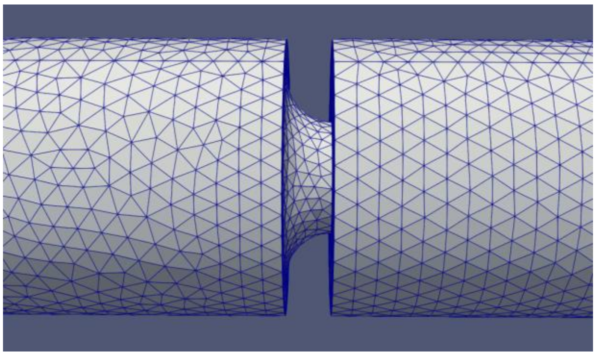


Figure 2. Circular Shaped Orifice

Figure 3. Curved edged Orifice

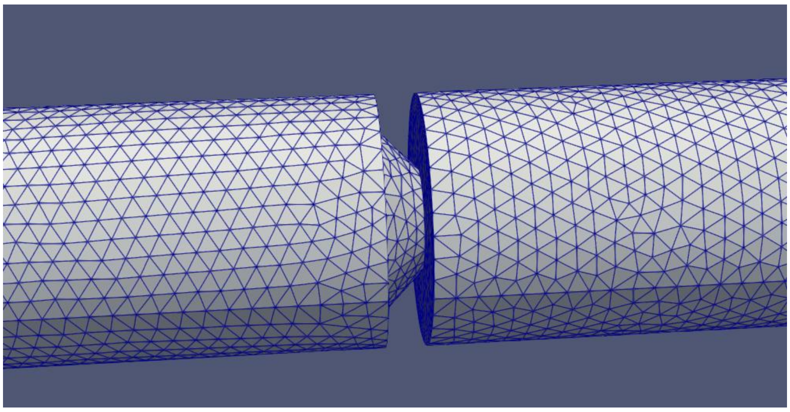


Figure 4. Sharp edged Orifice

Table 1. Mesh Statistics for different shapes

Sr. No.	Orifice shape	Mesh Size
1	Circular	2515204 cells
2	Curved-edged	1424441 cells
3	Sharp-edged	1423298 cells

4.2 Initial and Boundary Conditions

In all the simulations, velocity is set at the inlet of the orifice meter, pressure at outlet is set to 0 Pa and no-slip condition is set at the wall. The standard k-epsilon model has been used as a turbulence model. All the initial and boundary conditions are given in Table 2 for all the three cases.

Table 2. Boundary conditions for CFD simulations

Sr. No.	Velocity (m/s) inlet	Turbulent kinetic energy (m^2/s^2) inlet	Turbulent dissipation rate (m^2/s^3) inlet	Turbulent Viscosity (m^2/s) inlet
1	0.376	0.00053	0.00002	0.001264
2	0.482	0.00087	0.00004	0.001703
3	0.673	0.0017	0.00014	0.0021675

4.3 Solver

The governing equations have been solved for steady state conditions using the open source CFD code OpenFOAM-2406 with simpleFoam solver. All the discretized equations were solved in a segregated manner with the SIMPLE (Semi Implicit Pressure Linked Equation) algorithm. The second-order upwind scheme has been used for discretization. The under-relaxation parameters were set to 0.3 for pressure and 0.7 for momentum equations. Simulations were performed till residuals reduced to value less than 1E-4.

5. Results and Discussions

5.1 Validation of simulated results

The result for the circular shaped orifice was validated with the result reported by Tukiman et al 2017, with the inlet velocity of 0.376m/s and water as working fluid. The result of velocity and pressure are shown in Figure 5 and 6 respectively.

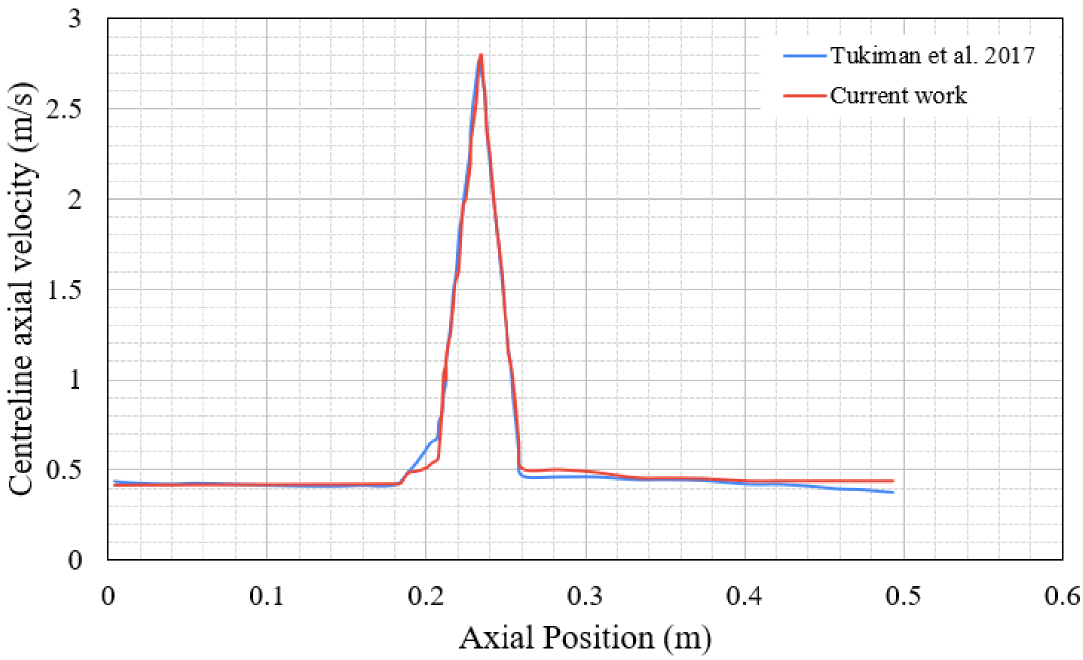


Figure 5. Axial profile of centreline velocity at inlet condition of 0.376 m/s for circular shaped orifice

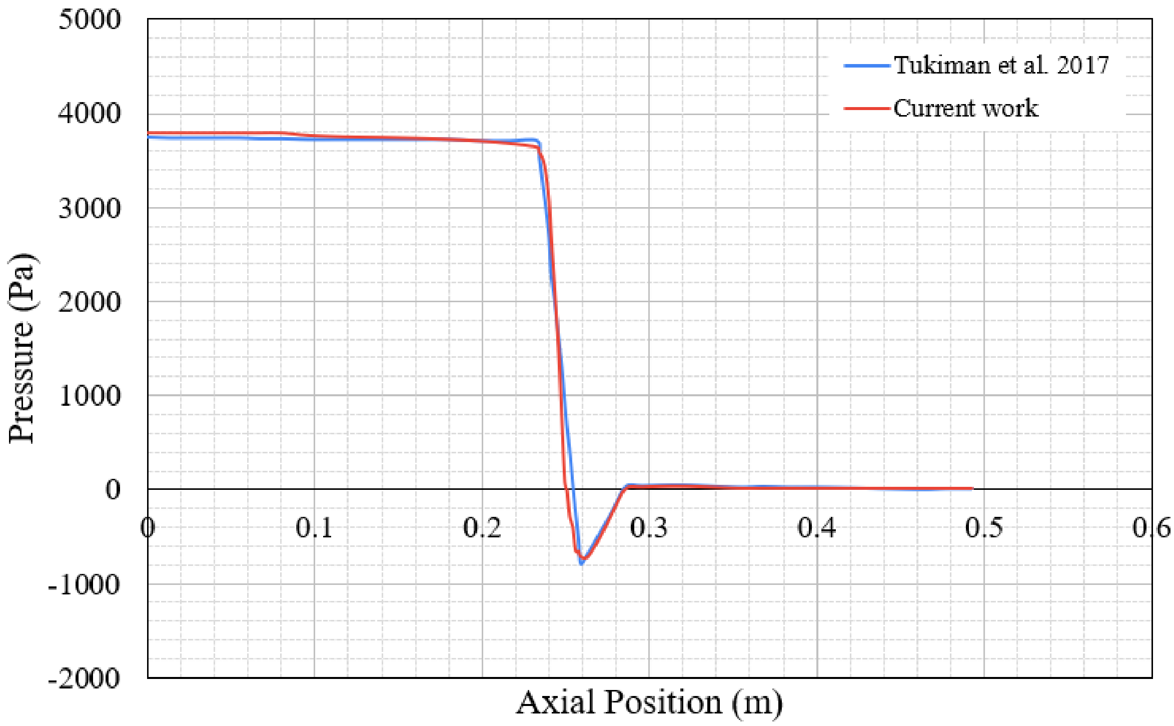


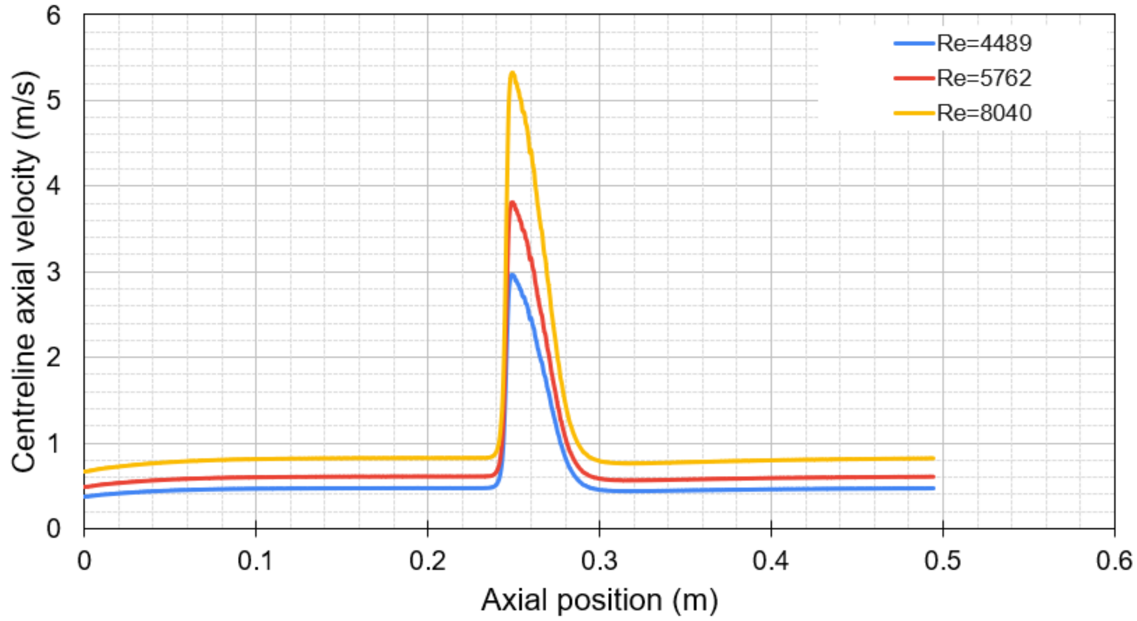
Figure 6. Pressure profile at inlet velocity of 0.376 m/s for circular shaped orifice

.

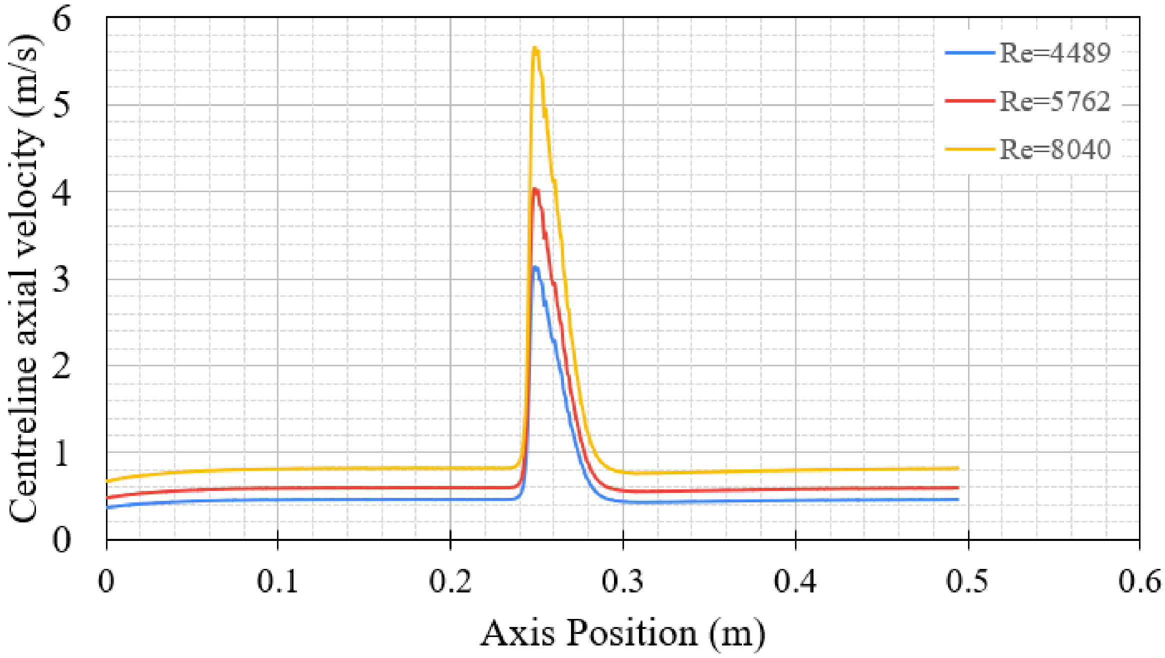
5.2 Velocity Profiles

The centreline axial velocity profiles for water demonstrate distinct behaviour as the flow interacts with the orifice meter. As the flow approaches the orifice, the centreline velocity increases due to the reduction in cross-sectional area. Downstream of the orifice, the velocity reaches its maximum at a point known as the vena-contracta. This localized acceleration is followed by a gradual decrease in velocity as the flow expands and redistributes further downstream. The velocity profile highlights the dynamic nature of flow in the orifice, with significant variations influenced by the geometry of the orifice plate. The velocity profile for circular orifice, curved edge, sharp edged orifice plate for different inlet velocities u = 0.376 m/s, 0.482 m/s, 0.673 m/s are shown in figures 7A to 7C.

A


Re=4489
Re=5762
Re=8040
Centreline axial velocity (m/s)
Axial position (m)
0
1
2
3
4
5
6
0
0.1
0.2
0.3
0.4
0.5
0.6

B


Re=4489
Re=5762
Re=8040
Centreline axial velocity (m/s)
Axis Position (m)
0
1
2
3
4
5
6
0
0.1
0.2
0.3
0.4
0.5
0.6

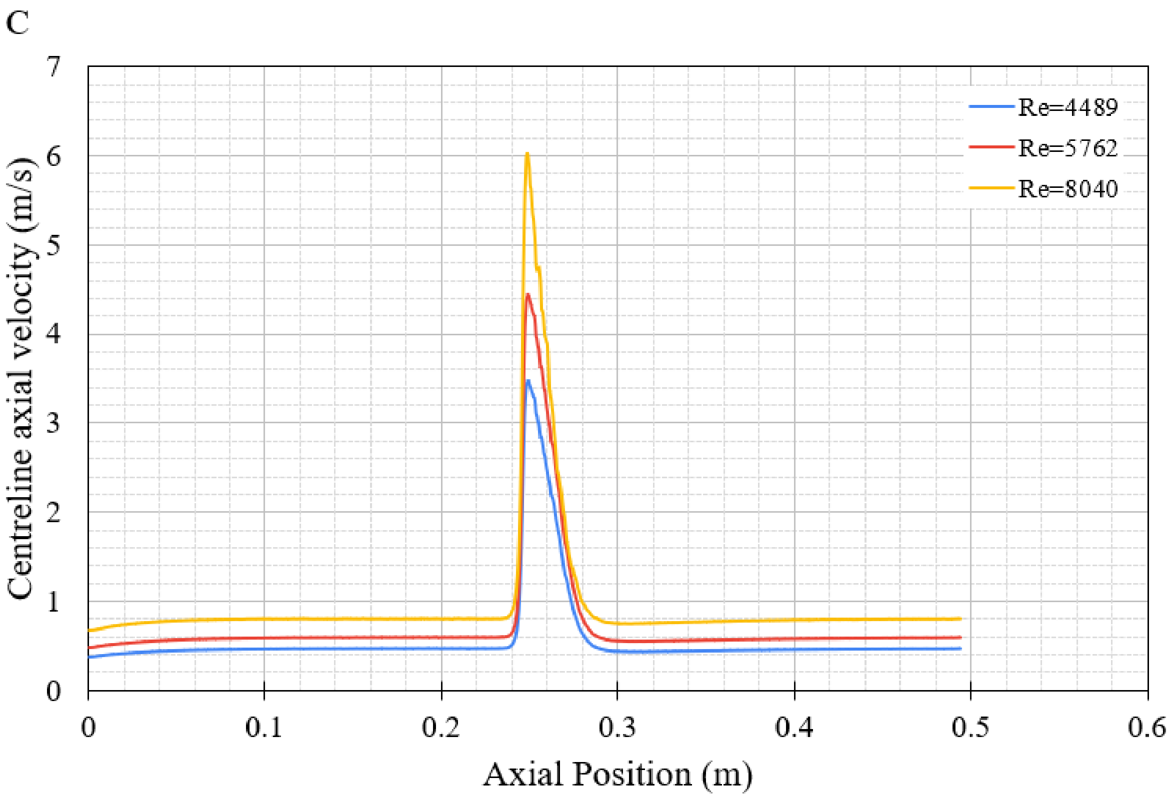
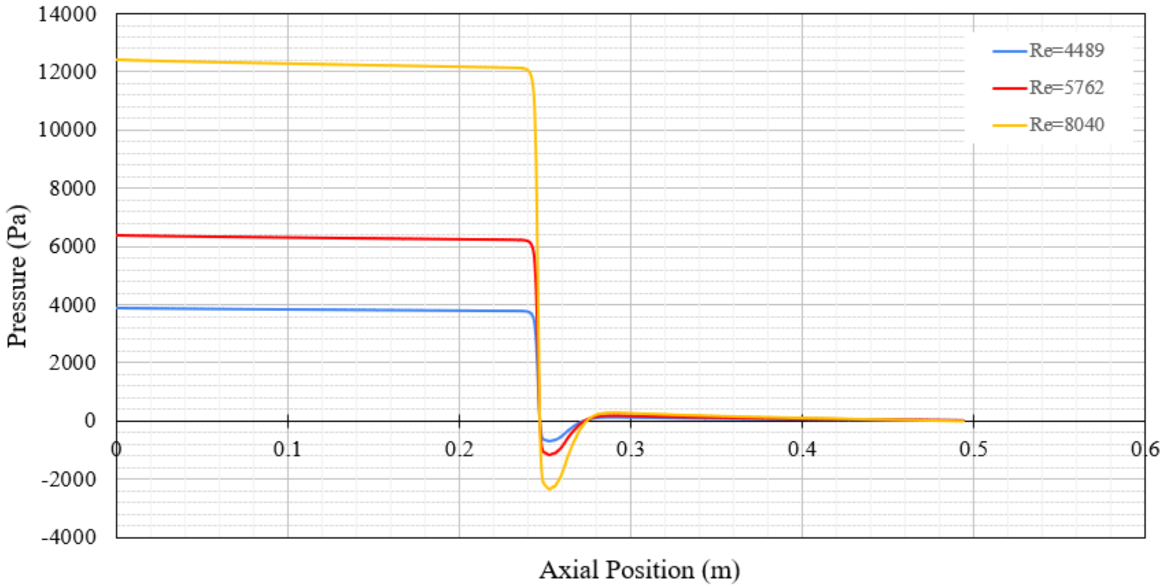


Figure 7. Plots for centre line axial velocity vs axial position A. for circular plate B. for curved edged plate C. for sharp-edged plate

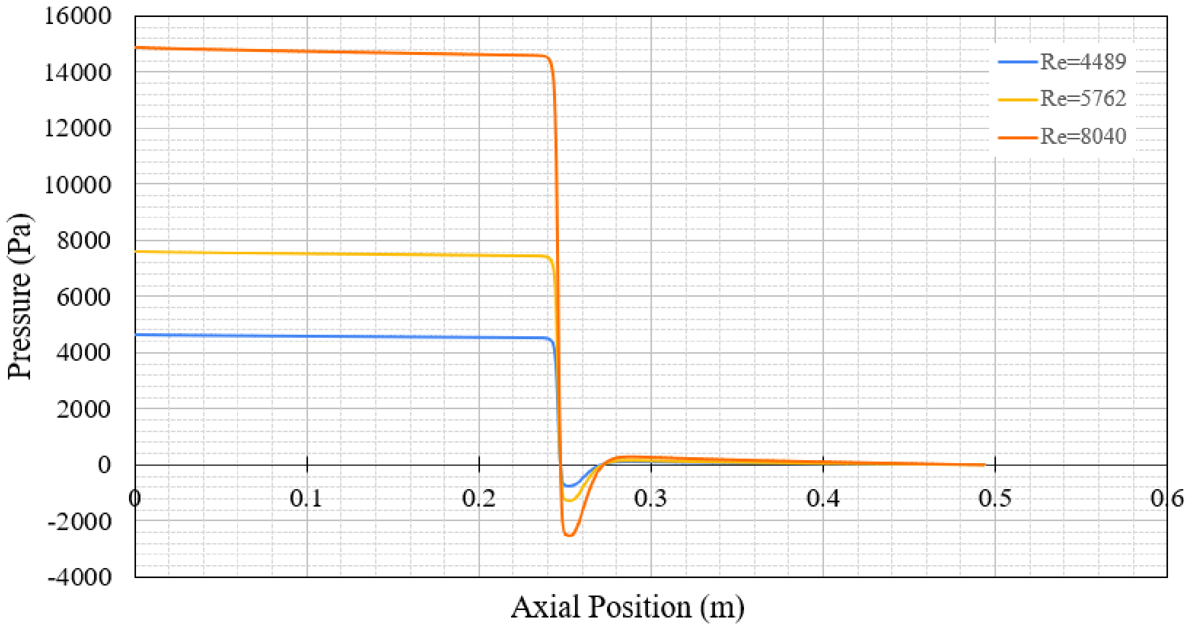
5.2 Pressure Profiles

The pressure profile across the orifice meter reflects the principle of energy conservation. As the flow approaches the orifice, pressure decreases due to the acceleration of the fluid. The minimum pressure is observed at the vena-contracta, where the velocity is at its peak. Beyond this point, as the flow decelerates and regains a larger cross-sectional area, pressure begins to recover. However, complete recovery is not achieved due to energy losses from factors like turbulence and friction. This pressure behaviour underscores the critical influence of the orifice geometry on energy distribution within the system. The pressure profile for circular orifice, sharp edged, curved edge orifice plate for different inlet velocities u = 0.376 m/s, 0.482 m/s, 0.673 m/s are shown in figures 8A to 8C.

A


14000
12000
10000
8000
6000
4000
2000
0
-2000
-4000
Pressure (Pa)
0
0.1
0.2
0.3
0.4
0.5
0.6
Axial Position (m)
Re=4489
Re=5762
Re=8040

B


16000
14000
12000
10000
8000
6000
4000
2000
0
-2000
-4000
Pressure (Pa)
0
0.1
0.2
0.3
0.4
0.5
0.6
Axial Position (m)
Re=4489
Re=5762
Re=8040

C

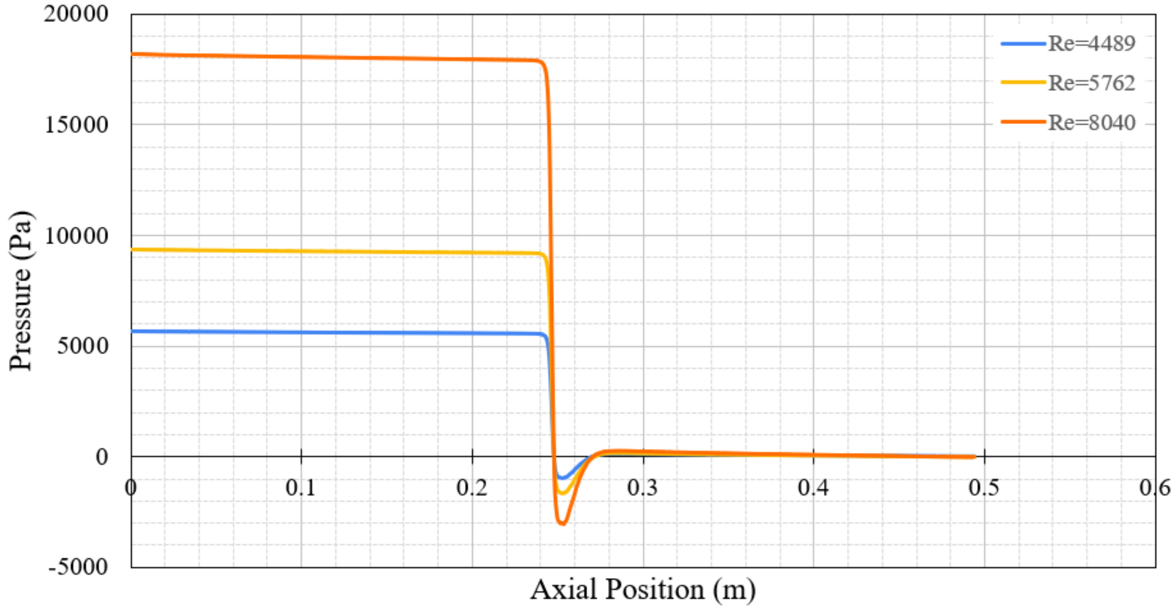


Figure 8. Plots for Pressure vs axial position A. for circular plate B. for curved edged plate C. for sharp-edged plate

5.3 Pressure Profiles and Coefficient of Discharge

The pressure drop and discharge coefficient in the three different shaped orifice plates for different Reynolds numbers is shown in the Figure 9 and 10 under the same boundary conditions. The coefficient of discharge clearly shows that we get higher value for coefficient of discharge in the case of circular orifice and its value remain almost constant as the Reynolds Number is increased while in the case of curved and sharp-edged orifice gives a rather irregular and smaller value of coefficient of discharge for various Reynolds Number.

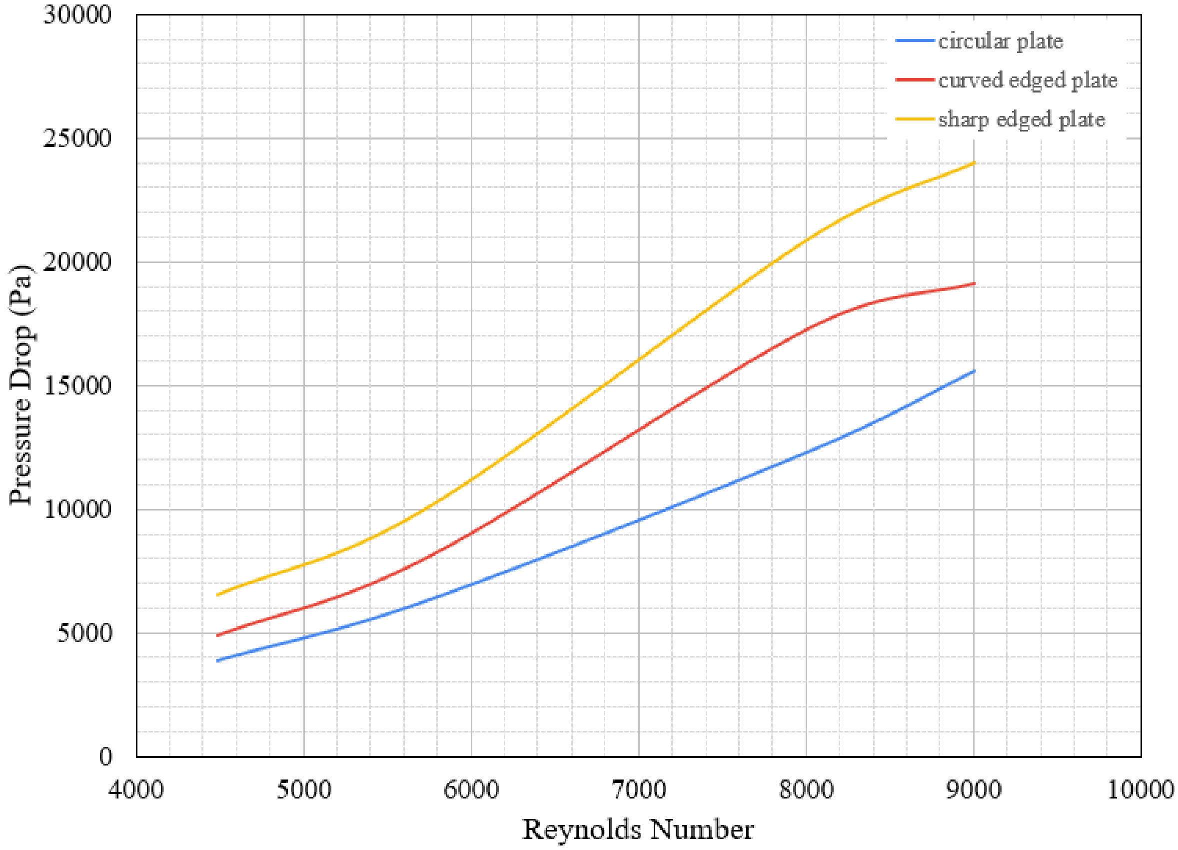


Figure 9. Plot of Pressure drop for three shapes vs Reynolds Number for circular shaped, curved edged and sharp-edged orifice plates

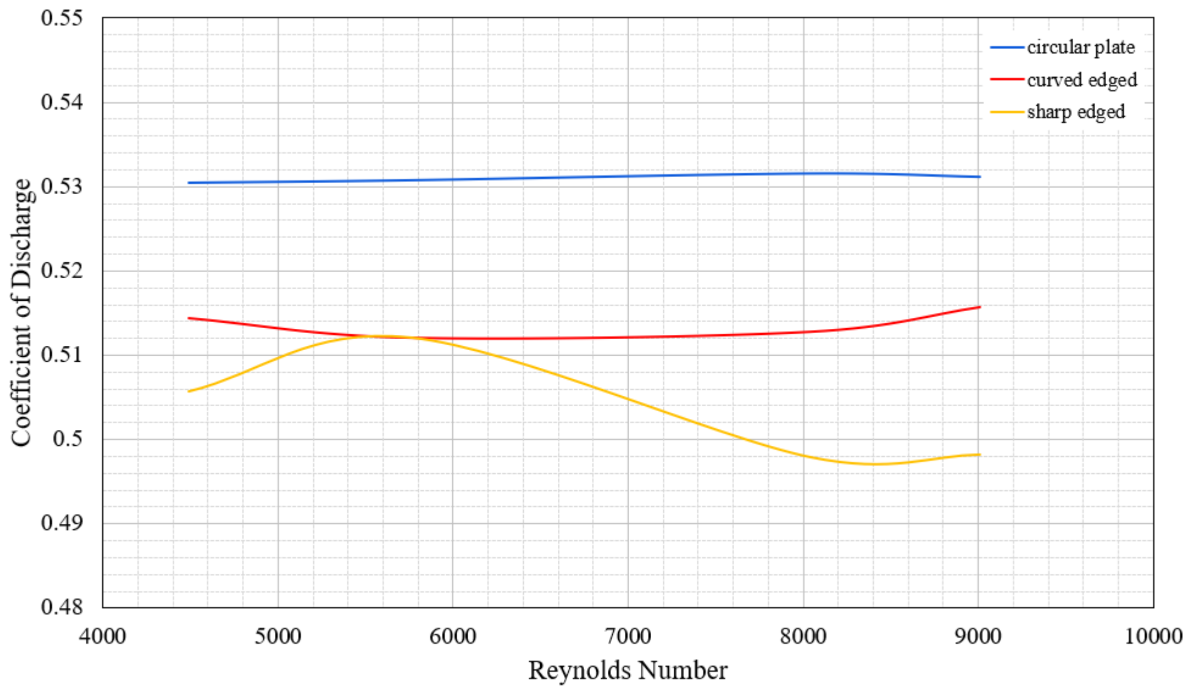


Figure 10. Plot of Coefficient of Discharge vs Reynolds Number for circular shaped, curved edged and sharp-edged orifice plates

6. Conclusion

The simulation through a circular orifice was successfully performed and validated against available experimental data Tukiman et al 2017. Using the same data, simulations for two additional orifice shapes were conducted, providing a comparative analysis of flow characteristics. The pressure and velocity variations observed across the different geometries offered valuable insights into the influence of orifice shape on flow behaviour. These findings underscore the potential for tailored orifice geometries to improve performance in industries including fluid metering, chemical processing, and pipeline systems. The study further validates the use of free and open source CFD tools for simulating flow through orifice systems.

Notations

D	pipe diameter (m)
d	orifice diameter (m)
L	length of pipe (m)
Re	Reynolds Number (dimensionless)
t	orifice plate thickness (m)
u	velocity of fluid (m/s)
∇	divergence operator
ν	kinematic viscosity of fluid (m^2/s)
ρ	density of fluid (kg/m^3)

References

Sanghani, C. R., Jayani, D. C., Dobariya, C. R., and Sabhadiya, G. K. 2016. Comparative Analysis of Different Orifice Geometries for Pressure Drop. International Journal of Science Technology & Engineering 2: 2349–784.

Shah, M. S., Joshi, J. B., Kalsi, A. S., Prasad, C. S. R., and Shukla, D. S. 2012. Analysis of Flow Through an Orifice Meter: CFD Simulation. Chemical Engineering Science 71: 300–309.

Tukiman, M. M., Ghazali, M. N. M., Sadikin, A., Nasir, N. F., Nordin, N., Sapit, A., and Razali, M. A. 2017. CFD Simulation of Flow Through an Orifice Plate. IOP Conference Series: Materials Science and Engineering 243: 012036.

Versteeg, H. K., and Malalasekera, W. 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education Limited, England.