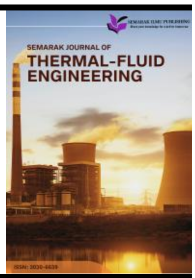
Semarak Journal of Thermal-Fluid Engineering

Journal homepage: https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639



Comparative Study of Internal Flow Dynamics Using CFD: Flow in a Curved Elbow with Variable Radius

Muhammad Amarfazlin Mohd Din[1,*]

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

ARTICLE INFO

Article history:
Received 20 June 2025
Received in revised form 26 July 2025
Accepted 8 August 2025
Available online 29 September 2025

Keywords:
Computational fluid dynamics (CFD); velocity profile; pressure distribution; turbulence intensity

ABSTRACT

The efficiency and quality of many industrial processes depend on the design and handling of piping systems, as changes in pipe diameter and speed of flow can greatly affect fluid movement. Since these factors impact the way flow operates, they are important to understand to ensure the system runs well, wastes less energy and stays reliable. However, properly predicting pressure, velocity and turbulence levels in pipelines with turns or irregular shapes is challenging because of how fluid dynamics and the pipe's structure interact. Frequently, approaches used before finding it difficult to handle the complexities; therefore, improvements in numerical techniques are required. We will investigate how reducing pipe diameter and increasing inlet velocity changes the flow characteristics in two independent tubes of 150 mm and 200 mm. Using Fluent software from the ANSYS platform, simulations were done at inlet velocities of 0.247 m/s, 0.397 m/s and 0.497 m/s. The analysis examined important characteristics of flow, including the pressure distribution, the velocity profile and the strength of turbulence, mostly in the regions just after pipe bends. According to simulation results, higher inlet velocity results in more pressure drops, sharper velocity gradients and larger turbulence levels in both pipe shapes. Higher levels of pressure and velocity changes, together with increased turbulence, were seen in the 150 mm diameter pipe at higher velocities. When we used the bigger size pipe (200 mm), the results indicated less pressure decrease and less change in the flow, showing the value of greater pipe size in saving energy and keeping the flow consistent. Overall, this research points out that pipe diameter and flow velocity play a major role in how well piping systems perform. The study results give helpful guidance for developing and improving stable and reliable piping systems in factories and plants.

1. Introduction

The motion of fluids through pipes is a basic topic in fluid mechanics that finds use in industries, water distribution and the medical field. While uniform pipe diameters make flow predictions straightforward, some systems include pipes whose radius varies, so the area they carry changes as

* *Corresponding author.*
E-mail address: AD210097@student.uthm.edu.my

https://doi.org/10.37934/sjotfe.6.1.5769a

fluid moves through them. Due to this variation, the flow properties become more complex when turbulent, leading to unusual velocity profiles, changes in pressure and turbulent intensity. Because of these complexities, engineering issues in the oil and gas industry which depend on fluid transport, are very relevant. Because oil and gas pipelines encounter a range of ground levels and depths, pipeline design must be modified to maintain both efficiency and safety as products are moved upwards from the sea. Proper operation and improved design in variable-radius pipes require understanding how fluids flow through them [1-3].

Different pipe diameters have different complex behaviours such as flow separation, the formation of recirculation areas and higher degrees of turbulence. When studying variable radius pipe flows, FEA tools such as ANSYS Fluent are needed to manage the nonlinear relationships between geometry, air or water turbulence and the presence of multiple phases. Important approaches include tuning the way diameters meet, checking the results with analytical methods and applying innovative turbulence models. Using these methods, we can accurately estimate pressure loss, pipe velocity patterns and the temperature levels in complex piping.

In our study, CFD analysis is performed using ANSYS Fluent to examine fluid flow in two different diameter pipes of 150 mm and 200 mm. The approach is developed to find how critical flows are affected by changes in pipeline diameter and flow rate. Three velocities at the pipe start (0.297 m/s, 0.397 m/s and 0.497 m/s) were simulated to understand a variety of conditions. I examine the velocity, pressure and turbulence data within each pipe. Using the same limits and improved simulations for turbulence, the method deals well with the link between flow and changes in diameter or the speed of flow. By simulating these conditions in ANSYS Fluent, the study captures the detailed hydrodynamic behaviour that cannot be accurately predicted by conventional analytical methods, especially in regions where diameter transitions cause significant deviations from ideal flow assumptions [4].

Numerous researchers have investigated pipe flow dynamics in systems with bends and elbows, focusing on aspects such as pressure drop, velocity patterns, and turbulence characteristics. Bilal *et al.,* [5] conducted CFD analyses of two-phase (air-water) flow in elbows with pipe diameters of 6.35 mm and 12.7 mm, using r/D ratios of 1.5 and 3, and a range of air and water velocities. His study presented detailed pressure drop profiles and cross-sectional pressure contour maps, demonstrating the complex behaviours present in multiphase flows and confirming the reliability of CFD predictions. Similarly, Mazumder [6] analysed both single- and two-phase flow in a 12.7 mm elbow, also using FLUENT, and compared the CFD results with empirical models, finding reasonable agreement for pressure drop and flow regime characterization. Taibi *et al.,* [1] focused on laminar flow in elbow-shaped pipes with varying curvatures and Reynolds numbers from 200 to 2000. His work included a mesh convergence study to ensure simulation accuracy and revealed the presence of Dean vortices secondary swirling flows that become stronger as the Dean number increases. Pressure contour plots from his simulations showed low-pressure regions near the inner curve and high-pressure zones near the outer curve, with the location of these zones shifting downstream as curvature increased.

Other studies, such as those by Ellis and Joubert [7], and Hunt and Joubert [8], addressed turbulent flow in curved ducts, comparing the results to straight duct flows. These investigations revealed that curved geometries significantly affect turbulence intensity distributions, mean velocity profiles, and streamwise energy, highlighting the complex interplay between geometry and flow behaviour. Collectively, these studies demonstrate that CFD tools like ANSYS Fluent, when validated with empirical models and supported by careful meshing strategies, provide reliable insights into the effects of pipe geometry on flow dynamics, especially in industrial applications involving elbows and bends.

2. Methodology
2.1 Geometry Creation

Figure 1 shows the flowchart of process. The geometry was created using ANSYS DesignModeler, which provides tools for building clean and parametric 3D models suitable for CFD simulation. In this study, the model consists of two pipes that have various radius with different diameters which is diameter geometry 1 is 150 mm and diameter geometry 2 is 200 mm, representing a basic internal flow system. Figure 2 show that the axis that being use to Sweep all over to make the solid fluid domain, which was later used in the meshing and simulation steps. The axis dimension is selected according to the Table 1. While, Figure 3 shows the isometric view of pipe.

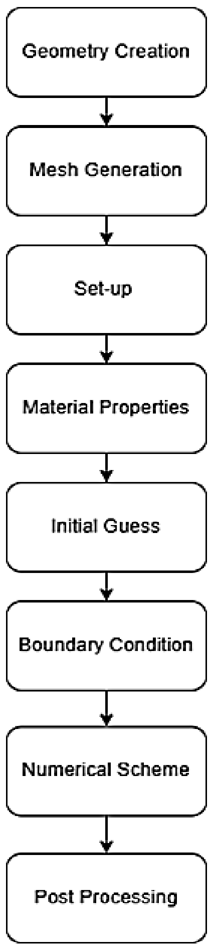


Fig. 1. Process flowchart

Table 1
Axis dimension

Label	Unit (mm)
H10	300
H12	600
R13	100
R16	150
R17	200
R18	400
V11	300
V19	200

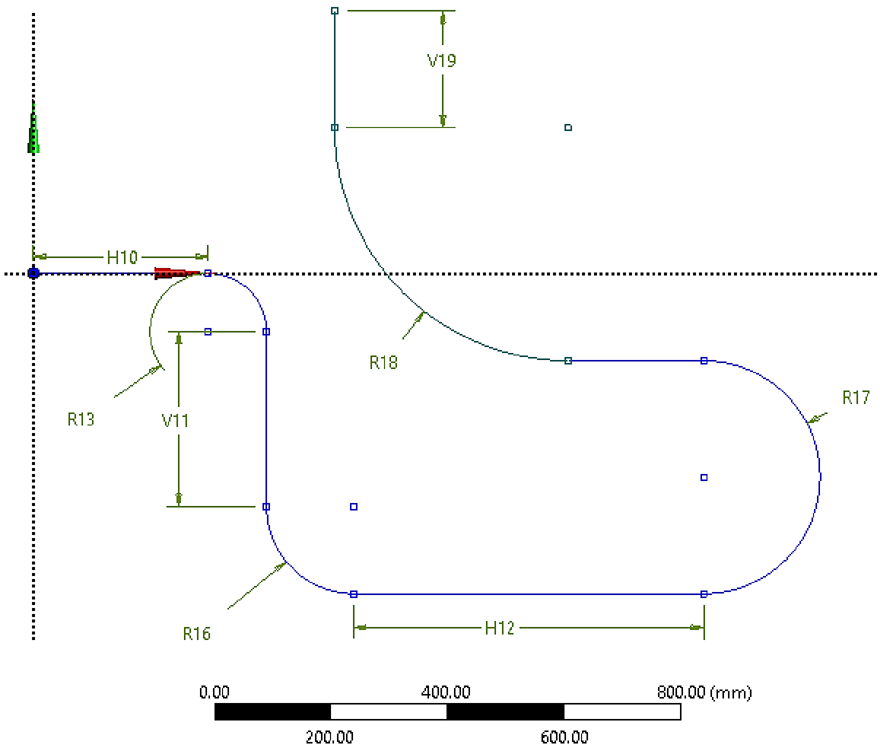
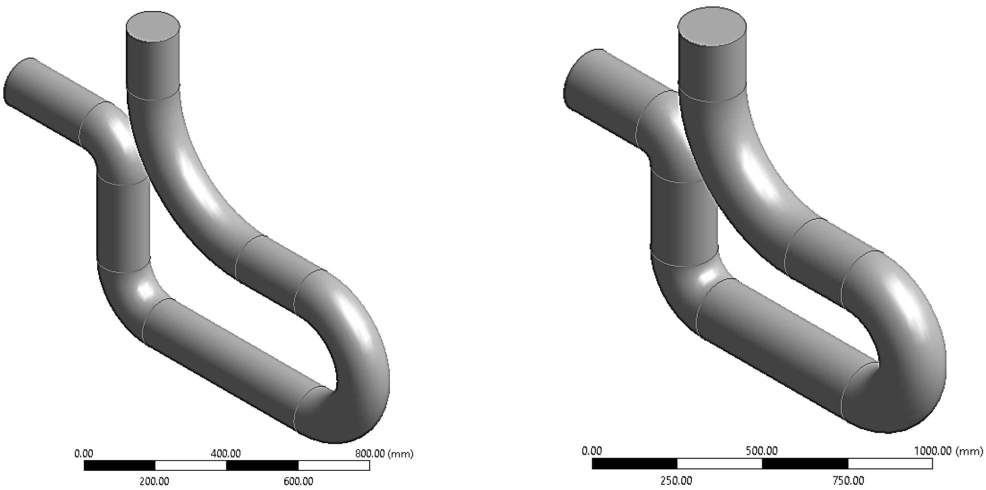


Fig. 2. Dimension of the sketch axis

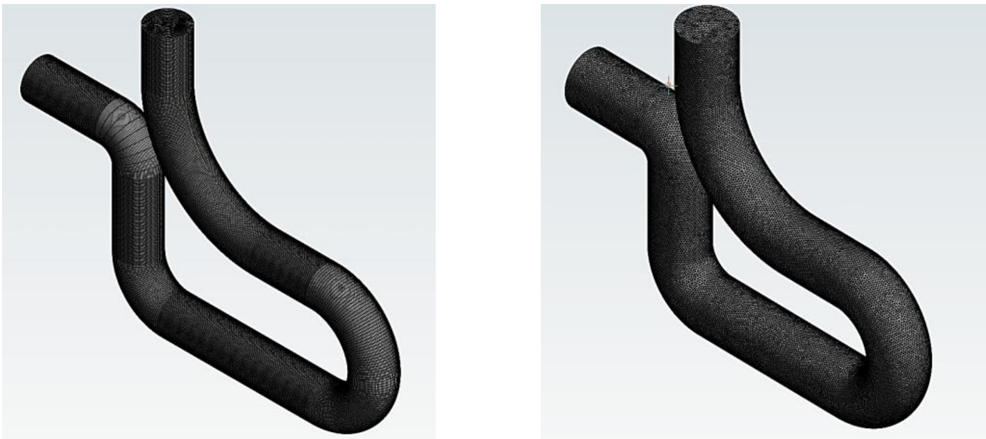


(a) (b)

Fig. 3. Isometric view of pipe (a) Geometry 1 with diameter 150 mm (b) Geometry 2 with diameter 200 mm

2.2 Meshing

The geometries were imported into a meshing tool in ANSYS R1 2024 where meshes were generated. A mesh independence study was optionally conducted to ensure that the simulation results are not sensitive to the mesh density. Mesh for geometry 1 is structured hexahedral mesh. While, geometry 2 is unstructured tetrahedral mesh. Number of nodes and element for geometry 1 is 1077984 and 1038160. Number of nodes and element for geometry 2 is 187138 and 981513. The element size for geometry 1 is 3.9 mm and element size for geometry 2 is 7.8 mm, as shown in Figure 4.



(a) (b)
Fig. 4. Mesh of the geometry (a) Mesh of the geometry 1 (b) Mesh of the geometry 2

2.2.1 Grid independence test

To make certain the CFD results are accurate, the GIT was carried out for each of the pipe geometries (Figures 5 and 6). The examination consisted of building three different mesh densities coarse, medium and fine, by making the mesh finer around the walls and changes in pipe diameter. Every mesh was analysed with all three inlet velocities by checking the pressure drop, velocities and turbulence over various cross-sections. After comparing results from multiple meshes, the solution is believed to be grid-independent if no more than a 2% change is measured by increasing the mesh further. The selection of the mesh was done to reach this requirement and still allow the CFD analysis to remain computationally efficient and supply accurate and reliable predictions in the 150 mm and 200 mm pipes.

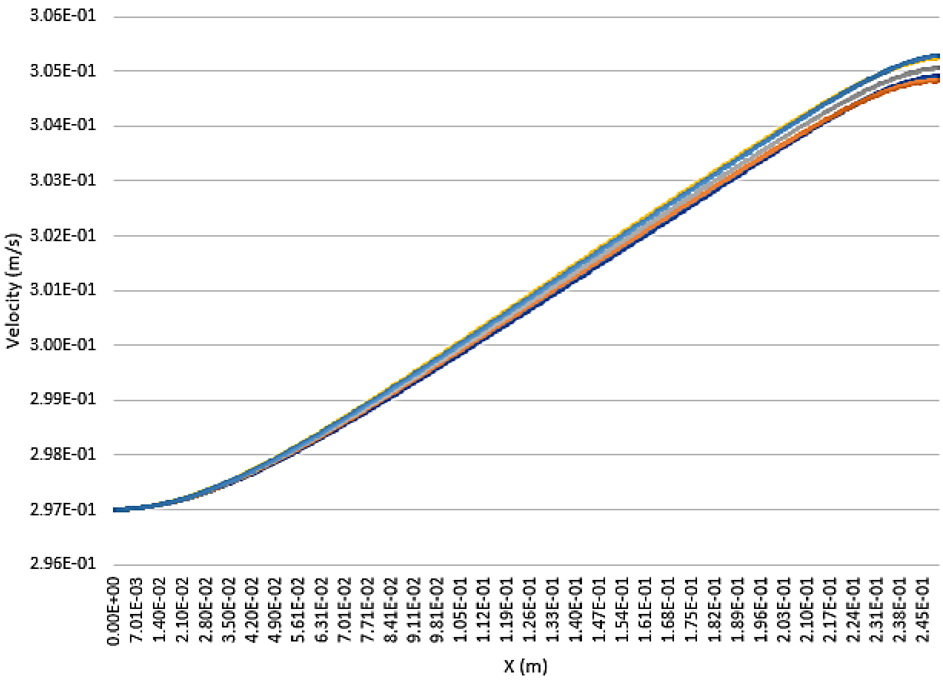


Fig. 5. GIT for geometry 1

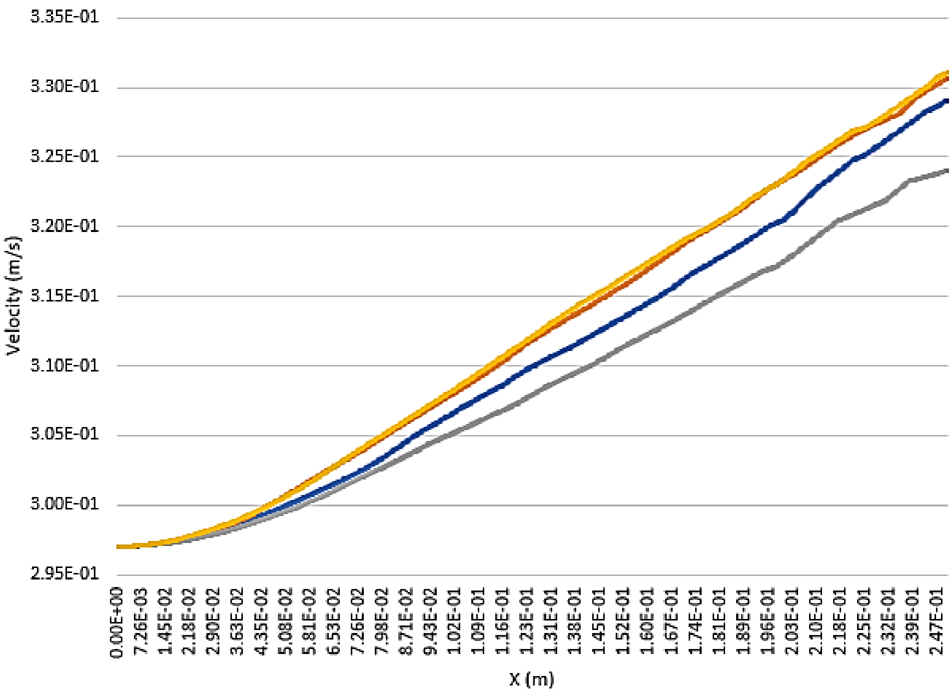


Fig. 6. GIT for geometry 2

2.3 Governing Equation

The principles from thermal fluid mechanics guide the functioning of pipes made with different diameter sizes. The Navier-Stokes equations describe how fluids behave while they conserve momentum together with mass and energy. The study deals with steady-state turbulent flow of an incompressible Newtonian fluid. The governing equations get solved through ANSYS Fluent CFD simulations for pipes which display different radii [9].

2.3.1 Continuity equation

For incompressible flow, the continuity equation ensures mass conservation and is expressed as:

$$\nabla \cdot V = 0 \tag{1}$$

where V is the velocity vector and $\nabla \cdot V$ is the divergence of the velocity field.

2.3.2 Momentum equation

The momentum equation in fluid dynamics is based on Newton's Second Law, which states that the rate of change of momentum equals the sum of forces acting on a fluid element. In CFD, this is represented by the Navier–Stokes equations. For incompressible, steady-state, Newtonian flow, the momentum equation in vector form is:

$$\rho(V\cdot\nabla)V= -\nabla p+\mu\nabla^2 V+F \tag{2}$$

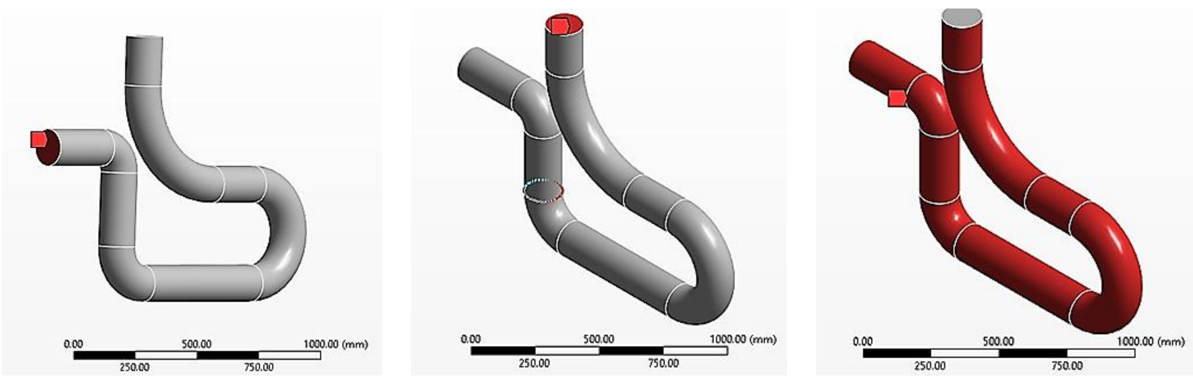
where ρ is density of the fluid (kg/m³), $\vec{V}$ is velocity vector (m/s), p is pressure (Pa), μ = dynamic viscosity (Pa·s), $\nabla^2\vec{V}$ = viscous diffusion term and $\vec{F}$ = body forces (e.g., gravity).

Fully turbulent conditions prevail in pipes with radius variations because they maintain high Reynolds numbers. The turbulent flow analysis relies on the standard k–ε turbulence model because it demonstrates both high accuracies along with efficient computational capabilities for internal flow analysis. The model employs two more transport equations for turbulent kinetic energy (k) and its dissipation rate (ε) because they help analyse the impact of turbulence on fluid dynamics. The application of k-ε model delivers accurate results regarding secondary flow patterns and improved mixing when pipe radii change in various engineering scenarios. ANSYS Fluent requires suitable near-wall treatment before implementing the model to properly capture turbulence near the pipe walls.

2.4 Boundary Condition and Parameter Selection
2.4.1 Named selection on geometry

A total of three inlet velocity settings 0.297 m/s, 0.397 m/s and 0.497 m/s were used for pipes of different radii to study velocity-induced flow changes. The researchers chose these velocity values to achieve a Reynolds number of 4338 at the highest speed in their largest pipe which corresponded to a 2.0 cm diameter. The models followed a uniform velocity profile at the inlet points using velocity inlet boundary conditions. Linking all pipes under similar flow conditions led to straightforward comparisons between their flow characteristics. For the wall condition, the pipe walls were treated no-slip boundary and stationary wall, the fluid velocity at the wall surface is assumed to be zero. This is a realistic behaviour of internal flow in enclosed pipes.

Finally, both geometries use a pressure outlet boundary condition at their exit points. In CFD simulations, applying a pressure outlet allows fluid to leave the domain freely, ensuring stability of the numerical solution [10]. The free-exit condition enables the outlet pressure to be set at zero-gauge pressure, representing a common assumption for open-ended flow where backflow is negligible [11]. This setting mimics a fully developed flow condition at the outlet, minimizing artificial reflections and promoting numerical convergence [12]. According to Versteeg [13], using a zero-gauge outlet pressure is especially effective in incompressible flows to simplify downstream boundary treatment without compromising accuracy. This assumption ensures that pressure gradients at the outlet do not disrupt upstream flow characteristics, making it a widely accepted boundary setup in steady-state flow simulations [11].



(a) (b) (c)

Fig. 7. (a) Inlet (b) Outlet (c) Wall

2.4.2 Operating condition

It was assumed that the flow was turbulent, steady state, and incompressible. The turbulence was modelled using the standard k–ε model, which works well for simulating interior pipe flow. It can be used to capture turbulence generalities without complex modelling and produces a fair trade-off between accuracy and computing expense.

2.4.3 Working fluid

For both cases the working fluid was water at room temperature. The fluid was assumed to be incompressible and Newtonian. The simulations used the following physical properties such as density (ρ) = 997 kg/m³ and dynamic viscosity (μ) = 0.001003 Pa·s. These constant properties had been applied to all velocity cases and geometries.

2.5 Analysis
2.5.1 Flow separation

A boundary layer separates from surfaces because of opposing pressure changes that create circulating turbulent regions [14]. The research design includes distinct simulation runs using 150 mm and 200 mm pipe diameters with equivalent initial conditions provided. The size variation of the pipe determines how the velocity shape behaves along with wall shear conditions and creates altered pressure distribution patterns [15]. The flow speed rises when pipes have smaller diameters, but the fluid boundary separates earlier in pipes with larger diameters under specific Reynolds numbers. Turbulence intensity and velocity field observations between the two pipe conditions will help researchers understand how pipe diameter impacts internal flow separation behavior [16].

2.5.2 Secondary flow structures (dean vortices)

Non-uniform curved pipes generate Dean vortices as secondary flow structures which occur when centrifugal forces meet pressure gradients in specific locations. Dean vortices occur in curved pipes and channels because the fluid experiences unbalanced interaction between centrifugal force and radial pressure gradient. The pipe develops two opposite spiral vortex patterns which carry momentum while blending fluid between its center area and its external walls. These secondary flow patterns have an equivalent effect when flow experiences abrupt geometry changes or encounters inlet misalignment in the system such as between a pipe offset or larger diameter connection. CFD studies need to observe these flow structures because they improve mixing and enhance turbulence intensity while affecting pressure loss behaviour. Scientists use velocity vector plots and vorticity contours in the cross-sectional view of the flow domain to examine Dean-like vortices.

According to Li *et al.,* [17] The behavior of secondary flow in a curved duct with continuously changing curvature is studied, emphasizing the development of various types of Dean vortices throughout the duct. This behavior contrasts with previous studies that concentrated on ducts having a fixed curvature. It is noted that a decrease in flow rate, leading to a reduced Reynolds number, promotes the formation of Dean instabilities—especially the split base vortex and the inner counter-rotating wall (ICW) Dean vortex. This trend differs from the usual behavior seen in laminar flow within ducts.

2.5.3 Velocity profiles

The velocity profile demonstrates the pattern of how fluid velocity spreads through a cross-section of the pipe while flowing through [10]. The research compared velocity profile patterns in 150 mm and 200 mm pipes when they received identical inlet conditions while pipe diameter was the only tested factor. Laminar flow typically develops a parabolic profile, which flattens in the center and steepens near the walls in turbulent conditions [18]. The flow core will possess greater average velocity within the smaller diameter pipe since its smaller cross-sectional area produces a compact and high-energy flow pattern. The slower more dispersed velocity profile develops in the larger diameter pipe. Velocity field distortions because of pipe entrance separation caused by misalignment or expansion variations can be detected through CFD analysis by examining velocity contours and vector field visualizations. The comparison of velocity profiles helps determine the relationship between pipe diameter and its effects upon momentum distribution and boundary layer growth as well as turbulence characteristics.

2.5.4 Pressure distribution

Pressure distribution shows the pressure changes across the complete flow area serving as a vital sign for both energy loss and flow conditions. This research evaluates pressure distribution independently for 150 mm and 200 mm diameter pipes which receive identical inlet conditions. The flow resistance of smaller diameter pipes causes elevated pressure gradient, but larger diameter pipes experience reduced pressure drop through similar lengths because they have reduced wall shear and velocity. A pressure drop occurs suddenly only when flow separates like when it enters an unconnected downstream pipe but recovers or becomes stagnant depending on attachment to the flow. Computational results generate pressure contour maps in addition to line graphs that follow the flow path to demonstrate how diameter differences affect pressure decreases while assessing flow stability patterns.

2.5.5 Turbulence intensity

Turbulence intensity (TI) is a key parameter used to quantify the level of turbulence in fluid flows, especially in pipes. It is defined as the ratio of the fluctuating velocity to the mean flow velocity. Specifically, TI is expressed as a percentage ratio between the fluctuating velocity components and the mean velocity. The study examined turbulence intensity in two pipe dimensions, 150 mm and 200 mm, exposed to identical inlet conditions. The results showed that higher flow velocities in smaller diameter pipes lead to increased shear forces at the pipe wall, resulting in higher turbulence levels at both the entrance and downstream areas [19]. In contrast, the larger diameter pipe exhibited lower turbulence intensity due to lower flow velocities and larger areas for the flow to stabilize [20]. Additionally, turbulence intensity can spike in regions where flow detachment occurs, such as in pipe elbows or at divergent pipe junctions, where recirculation zones and turbulent wakes form [21]. These regions are critical for understanding flow instability, and computational fluid dynamics (CFD) simulations, including post-processing contour plots, provide detailed insights into these areas of flow separation [22].

3. Results

3.1 Pressure Distribution

The pressure contour plots for two pipe geometries (150 mm and 200 mm diameters) at three velocities (0.247 m/s, 0.397 m/s, 0.497 m/s) demonstrate how velocity and diameter influence pressure distribution in bent pipes, as shown in Figures 8 and 9. Increasing velocity amplifies pressure drops in both geometries, with distinct high-pressure zones upstream of bends and low-pressure regions downstream. geometry 1 (150 mm) shows steeper pressure gradients and stronger turbulence at higher velocities, particularly at 0.497 m/s, where rapid deceleration and recirculation zones occur. In contrast, geometry 2 (200 mm) exhibits milder pressure changes due to reduced frictional resistance, though high-pressure zones still form at bends. These results confirm that smaller diameters and higher velocities intensify energy losses and flow disturbances, emphasizing the need for optimized bend design in piping systems to mitigate wear and inefficiency.

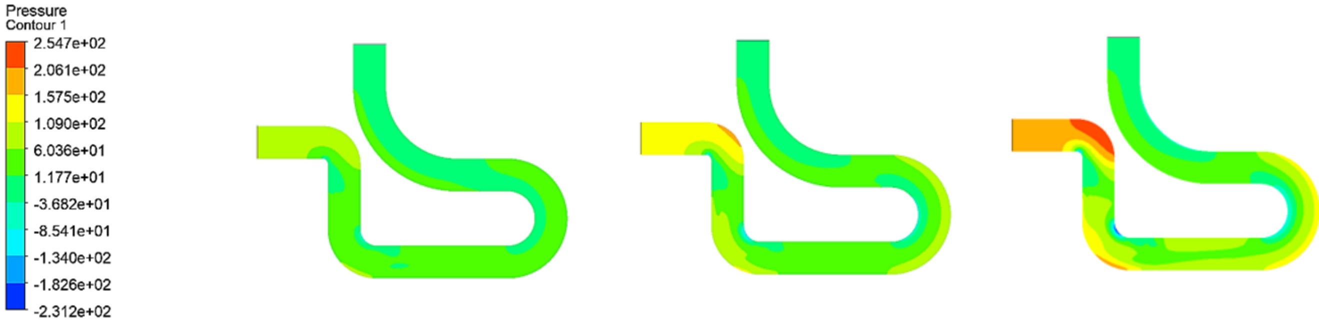


Fig. 8. Pressure distribution for geometry 1 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

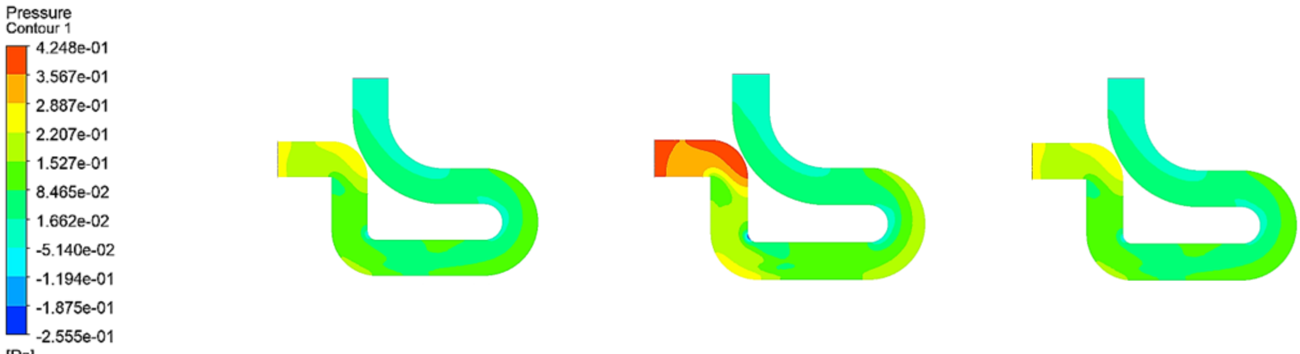


Fig. 9. Pressure distribution for geometry 2 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

3.2 Velocity Profiles

In the velocity contour results, boosting inlet velocity strengthens the high-velocity areas, mainly around the outer curves of each pipe design. These results suggest that the smaller pipe, with its more noticeable velocity increases, is unstable and more prone to high velocity clumps. However, with the larger 200 mm pipe, we notice that the velocity profile is spread further, and the gradients are lower which helps prevent flow separation and recirculation. Generally, both as velocities rise and pipes become smaller, the problems of irregular flows and problems become more serious;

however, bigger pipelines reduce these challenges, stressing that pipe size and rate of flow impact the system's design, as shown in Figures 10 and 11.

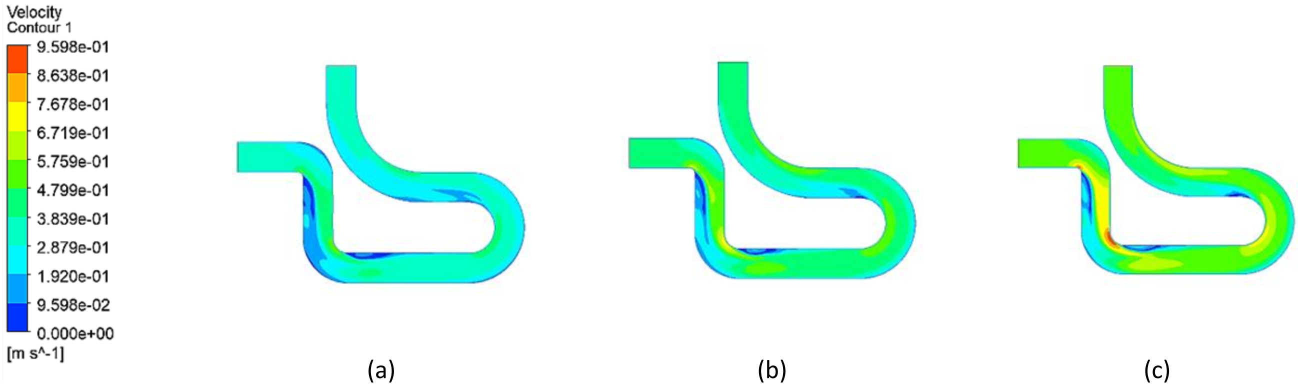


Fig. 10. Velocity profiles for geometry 1 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

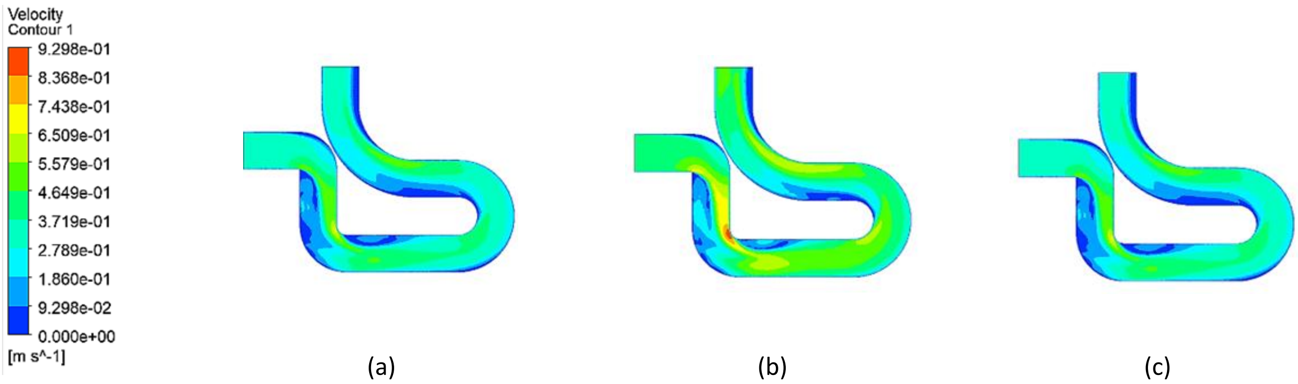


Fig. 11. Velocity profiles for geometry 2 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

3.3 Turbulence Intensity

For both sized pipes with three flow velocities, the measured turbulence intensity revealed that higher velocities always increase the quantity of turbulence kinetic energy. The zones in both test cases with strong turbulence (green to red areas) are mainly found along the bends, where separation and recirculation are most likely. Both central and outer regions of the loop are calm at a speed of 0.247 m/s, aside from some turbulence near the bend. When the velocity speed reaches 0.397 m/s or 0.497 m/s, areas with intense turbulence appear and nearly double in size, largely in the region downstream of the bend. Geometry 1 shows stronger maximum turbulence values than geometry 2 at the same velocity, although geometry 2's turbulence levels are more evenly spread and lower on average, as shown in Figures 12 and 13. Generally, the study confirms that a combination of high velocities and narrow pipes creates more turbulence than normal, mainly at bends, making the flow unstable and leading to more energy lost in the system.

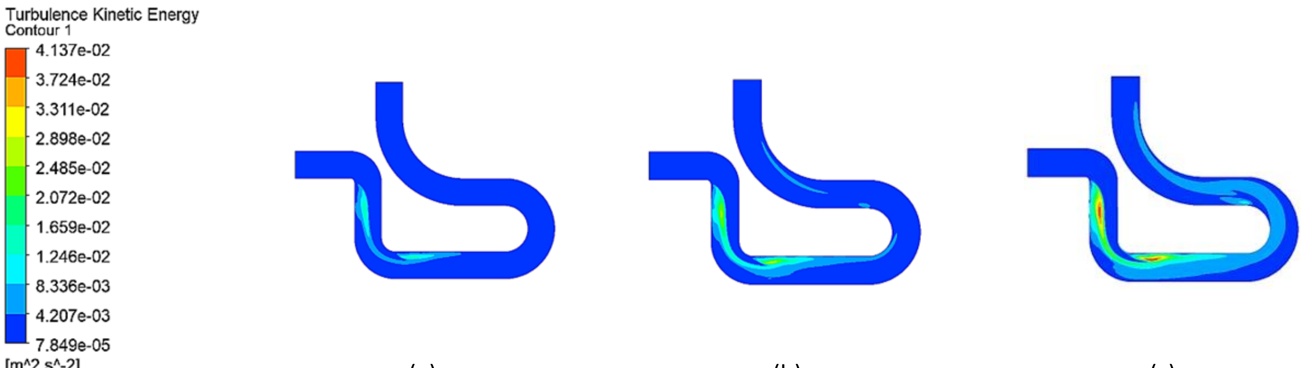


Fig. 12. Turbulence intensity for geometry 1 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

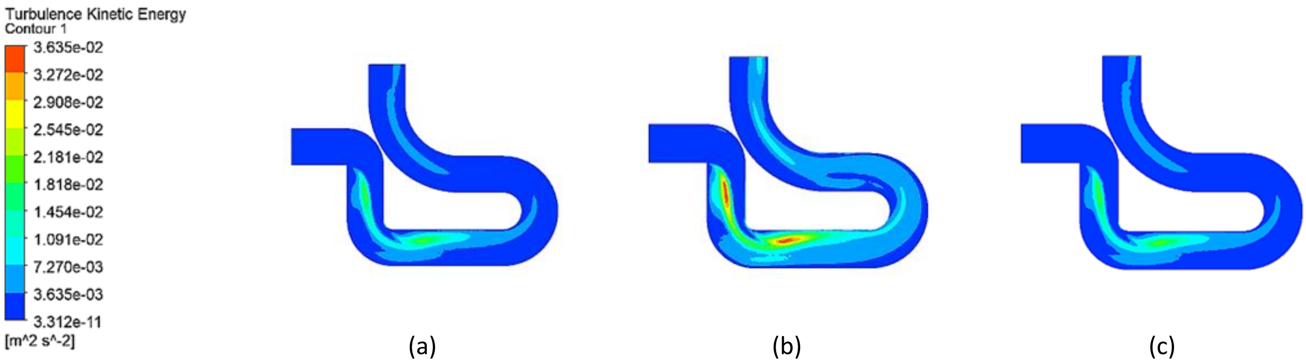


Fig. 13. Turbulence intensity for geometry 2 (a) Velocity = 0.247 m/s (b) Velocity = 0.397 m/s (c) Velocity = 0.497 m/s

4. Conclusions

This study utilizes advantage of CFD and worked with pipe sizes of 150 mm and 200 mm to examine how inlet velocities affect the flow in each geometry. Higher inlet velocities are seen to cause greater pressure losses, steeper rises in fluid velocity and more turbulence in both pipe designs. We found that, with both pipes, the 150 mm pipe was more likely to show stronger changes in pressure, velocity and turbulence, especially at faster rates. Unlike the 150 mm diameter, the 200 mm diameter pipe experienced lower energy losses and less extreme changes in flow, suggesting that larger pipes are better for controlling these problems. All in all, these results stress that the design of efficient and stable piping relies on considering the pipe diameter and its operating velocity.

References

[1] Taibi, R., G. Yin, and M. C. Ong. "CFD investigation of internal elbow pipe flows in laminar regime." In *IOP Conference Series: Materials Science and Engineering*, vol. 1201, no. 1, p. 012012. IOP Publishing, 2021. https://doi.org/10.1088/1757-899X/1201/1/012012

[2] Traetow, M. A. *Simulation of Turbulent Pipe Flow.* The University of Iowa, 1999.

[3] Dogan, T., Conger, M., Kim, D. H., Park, S. T., Milano, C., Mousaviraad, M., Xing, T., Stern, F. *Verification of laminar and validation of turbulent pipe flows*. The University of Iowa, C. Maxwell Stanley Hydraulics Laboratory, 2022.

[4] Gebler, Malte, Anton JM Schoot Uiterkamp, and Cindy Visser. "A global sustainability perspective on 3D printing technologies." *Energy policy* 74 (2014): 158-167. https://doi.org/10.1016/j.enpol.2014.08.033

[5] Bilal, Faris S., Thiana A. Sedrez, and Siamack A. Shirazi. "Experimental and CFD investigations of 45 and 90 degrees bends and various elbow curvature radii effects on solid particle erosion." *Wear* 476 (2021): 203646. https://doi.org/10.1016/j.wear.2021.203646

[6] Mazumder, Quamrul H. "CFD analysis of single and multiphase flow characteristics in elbow." *Engineering* 4, no. 4 (2012): 210-214. https://doi.org/10.4236/eng.2012.44028
[7] Ellis, Lindsay Bruce, and P. N. Joubert. "Turbulent shear flow in a curved duct." *Journal of Fluid Mechanics* 62, no. 1 (1974): 65-84. https://doi.org/10.1017/S0022112074000589
[8] Hunt, I. A., and P. N. Joubert. "Effects of small streamline curvature on turbulent duct flow." *Journal of Fluid Mechanics* 91, no. 4 (1979): 633-659. https://doi.org/10.1017/S0022112079000380
[9] Mohamed Khairi, Mohamed Ikram. "Computational fluid dynamic (CFD) simulation of slug flow within pipe bend and pipe elbow which induce vibration." IRC, 2020.
[10] Launder, Brian Edward, and Dudley Brian Spalding. "The numerical computation of turbulent flows." In *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pp. 96-116. Pergamon, 1983. https://doi.org/10.1016/B978-0-08-030937-8.50016-7
[11] Papanastasiou, Tasos C., Nikos Malamataris, and Kevin Ellwood. "A new outflow boundary condition." *International Journal for Numerical Methods in Fluids* 14, no. 5 (1992): 587-608.
[12] Ferziger, Joel H., Milovan Perić, and Robert L. Street. *Computational methods for fluid dynamics*. springer, 2019. https://doi.org/10.1007/978-3-319-99693-6
[13] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
[14] Dróżdż, Artur, Paweł Niegodajew, Mathias Romańczyk, and Witold Elsner. "Effect of Reynolds number on turbulent boundary layer approaching separation." *Experimental Thermal and Fluid Science* 125 (2021): 110377. https://doi.org/10.1016/j.expthermflusci.2021.110377
[15] Blazek, Jiri. *Computational fluid dynamics: principles and applications*. Butterworth-Heinemann, 2015.
[16] Liang, Chang-Hai, Ming-Bo Sun, Yu-hui Huang, Da-Peng Xiong, Jiang-fei Yu, Yi-Xin Yang, Hong-Bo Wang, Yong-chao Sun, and Guang-Wei Ma. "Mechanism of development of turbulent boundary layer in a curved circular pipe under supersonic conditions." *AIP Advances* 12, no. 3 (2022). https://doi.org/10.1063/5.0085586
[17] Li, Yalin, Xikun Wang, Bo Zhou, Shouqi Yuan, and Soon Keat Tan. "Dean instability and secondary flow structure in curved rectangular ducts." *International Journal of Heat and Fluid Flow* 68 (2017): 189-202. https://doi.org/10.1016/j.ijheatfluidflow.2017.10.011
[18] Pope, Stephen B. "Turbulent flows." *Measurement Science and Technology* 12, no. 11 (2001): 2020-2021. https://doi.org/10.1017/CBO9780511840531
[19] Russo, Francesco, and Nils T. Basse. "Scaling of turbulence intensity for low-speed flow in smooth pipes." *Flow Measurement and Instrumentation* 52 (2016): 101-114. https://doi.org/10.1016/j.flowmeasinst.2016.09.012
[20] Chinenye-Kanu[1], Nkemjika Mirian, Mamdud Hossain, Ghazi Mohamad Droubi, and Sheikh Zahidul Islam. "CFD investigation of the effect of pipe diameter on multiphase flow induced vibration." In *32nd Scottish Fluid Mechanics Meeting Book of Abstracts*, p. 37. https://doi.org/10.13140/RG.2.2.17823.20643
[21] Basse, Nils T. "Turbulence intensity and the friction factor for smooth-and rough-wall pipe flow." *Fluids* 2, no. 2 (2017): 30. https://doi.org/10.3390/fluids2020030
[22] Abraham, J. P., E. M. Sparrow, and J. C. K. Tong. "Heat transfer in all pipe flow regimes: laminar, transitional/intermittent, and turbulent." *International Journal of Heat and Mass Transfer* 52, no. 3-4 (2009): 557-563. https://doi.org/10.1016/j.ijheatmasstransfer.2008.07.009