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

Computational Fluid Dynamic Modelling on Internal Flow Dynamics Using CFD in 90-Degree Bend Pipe

Mohd Syahir Abd Razak[1,*]

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

ARTICLE INFO

Article history:
Received 4 June 2025
Received in revised form 5 July 2025
Accepted 16 July 2025
Available online 29 September 2025

Keywords:
90-degree bend pipe flow; computational flow dynamic; flow separation; secondary flow; pressure distribution; velocity profiles; turbulent intensity

ABSTRACT

Computational fluid dynamics (CFD) is a powerful method for visualizing and analyzing fluid flow in pipes, helping engineers design more efficient systems. However, many earlier studies focus on just one pipe size or flow speed, which limits understanding of how different conditions affect flow. This study explores how varying pipe diameters (1.5 cm, 3 cm, and 5 cm) and inlet velocities (0.297 m/s, 0.397 m/s, and 0.497 m/s) influence velocity and pressure inside a 90-degree pipe bend. Using ANSYS Fluent, nine scenarios were simulated under steady, incompressible flow conditions by solving the Navier-Stokes equations with velocity inlets and pressure outlets. The results showed that the smallest pipe at the highest velocity had the fastest flow (up to 0.708 m/s) but also the largest pressure drop (254.68 Pa). Larger pipes displayed more uniform velocity distributions and significantly lower pressure losses. These findings clearly demonstrate how pipe size and flow speed affect fluid behaviour, emphasizing the importance of selecting appropriate pipe diameters to optimize flow efficiency and reduce energy loss in piping systems.

1. Introduction

Fluid flow through curved pipes is a fundamental subject in fluid mechanics, with profound implications for the efficiency and reliability of engineering systems ranging from water distribution networks and chemical reactors to heat exchangers and process piping [1]. When pipes bend, the fluid inside experiences centrifugal forces that create swirling flows called Dean vortices. These swirling motions change the speed and pressure of the fluid, making it more challenging to design piping systems that reduce energy loss and prevent early damage [2,3]. Although many studies have looked at how individual factors affect fluid flow, there’s still a gap when it comes to understanding how changing both pipe diameter and inlet velocity together impacts flow separation, turbulence, and pressure loss in 90° pipe bends.

These studies help us gain a clearer picture of how turbulence influences the way fluids move, which is important for systems that need accurate flow control. Prakash and Sumana [4] showed how

* *Corresponding author.*
E-mail address: cd210011@student.uthm.edu.my

https://doi.org/10.37934/afhme.6.1.1425a

the way liquid phases interact in bent pipes depends a lot on how wettable the bends are. Their work reveals that certain pipe coatings can change how two-phase flows behave, giving us valuable insights into controlling flow dynamics. Moreover, using advanced CFD techniques lets researchers see in much more detail how different flow conditions and setups impact the flow, making it easier to understand and improve system designs. They stressed how important it is to thoroughly study flow behavior in different pipe shapes, encouraging a well-rounded approach that looks beyond just curves to include U and C bends as well [5].

Research has improved understanding of how fluids behave in curved pipes. Computer simulations have shown that in pipes with sharp curves, the way the flow separates depend greatly on the pipe's shape [6]. Studied how vibrations interact in U-shaped pipe bends when two different fluids flow through them [7]. Prakash and Samana [4] demonstrated how the surface's ability to attract or repel liquids affects the flow behavior of two fluids moving through small, winding pipes [8]. However, most of these studies usually look at just one pipe size or flow speed, which limits how well their findings apply to different situations. More recent research using advanced turbulence models like LES and RSM has done a better job of capturing detailed flow patterns [8,9], but comprehensive studies that explore a wide range of pipe sizes and flow rates are still rare.

To address this gap, our study will examine how three different pipe sizes (1.5 cm, 3.0 cm, and 5.0 cm) and three different flow speeds (0.297 m/s, 0.397 m/s, and 0.497 m/s) influence key flow characteristics such as fluid movement, pressure loss, and turbulence in 90° bend pipe. We created nine different pipe shapes in ANSYS Fluent and used a finite-volume CFD method to simulate steady, incompressible fluid flow by solving the Navier–Stokes equations. To accurately capture the complex flow in curved pipes where separation can occur, we chose the k–ω SST turbulence model. The simulations used uniform flow speeds at the inlets and set the outlets to zero-gauge pressure [10].

Recent studies have helped us better understand turbulence by using mathematical formulas and computer simulations to model how it behaves. Reetz *et al.,* [11] discuss the emergence of turbulent-laminar stripes, while Yanovych *et al.,* [12] investigate anisotropic effects and vortex shedding in constrained geometries. Additionally, turbulent airflow can affect how large structures like bridges move and respond, influencing their overall behavior [13]. Turbulent flows can sometimes experience rare and intense episodes, like sudden bursts of energy loss or abrupt shifts to a completely different flow pattern [14]. These extreme events, including large dissipation episodes and rapid transitions, are unpredictable and can dramatically alter the behavior of the system for a short time [15]. Understanding these patterns is key to spotting when things might become unstable and helps us improve how we manage and control the flow. Moreover, even in chaotic flow systems, there are organized patterns called coherent structures or turbulent superstructures that can stick around. These play a big role in moving energy and materials over large areas, whether in the atmosphere or in industrial processes [16,17].

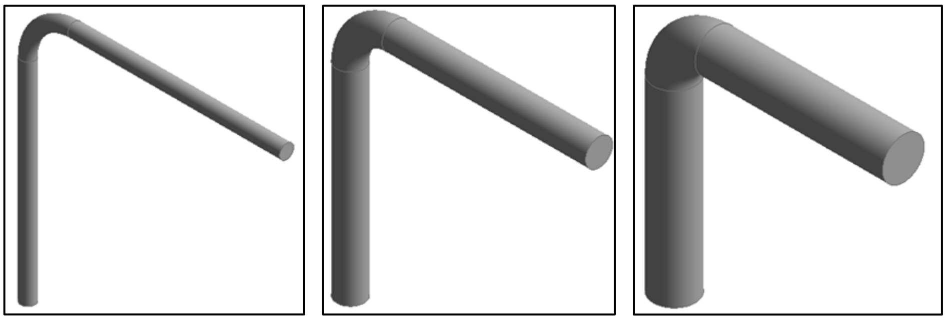
The post-processing involves several steps: first, visually examining overall flow patterns and the swirling Dean vortices using velocity and streamline maps; second, measuring turbulence by looking at turbulent kinetic energy levels; third, analyzing how pressure changes, especially focusing on drops around the bend and the total pressure loss; and finally, comparing centreline velocity profiles and cumulative pressure drops across all cases using line graphs. This approach is based on proven methods from earlier curved-pipe CFD research, helping to guarantee reliable and consistent results [18,19].

2. Methodology

This study uses CFD to investigate the turbulent flow behavior inside a 90-degree bent pipe. The goal is to understand how varying pipe diameters and inlet velocities impact factors like velocity profiles, pressure changes, secondary flow patterns, and flow separation zones. By focusing on these elements, the study aims to reveal how bends shape fluid dynamics in practical piping systems.

2.1 Geometry Construction

In this project, three pipe models with different diameters, 1.5 cm, 3.0 cm, and 5.0 cm were created to study how the size of the pipe affects fluid flow through a 90-degree bend, as shown in Figure 1. Each pipe has a total length of 1 meter, including the inlet and outlet straight sections to ensure fully developed flow. Each pipe was designed to be 1 meter long in total, including straight sections before and after the bend to ensure the flow remains steady as it enters and leaves the curve. We kept the bend's shape consistent across all models so that any differences in the results would come solely from the pipe size. By comparing these three sizes, the study clearly highlights how pipe diameter impacts velocity, pressure, and turbulence within the bend.



(a) (b) (c)

Fig. 1. The geometry of 90-degree bend pipe (a) Diameter 1.5 cm (b) Diameter 3.0 cm (c) Diameter 5.0 cm

2.2 Boundary Conditions

Defining boundary conditions is essential to maintain the accuracy and reliability of the simulation results. Table 1 presents the boundary conditions used in the analysis of the 90-degree bend pipe simulation. To ensure the results weren't affected by the mesh size, we compared velocity and pressure patterns using three different mesh sizes for each pipe diameter (Table 2). The mesh that provided accurate, stable results without requiring too much computing power was chosen for the final simulations.

Table 1
Boundary conditions of 90-degree bend pipe

Boundary type	Location	Condition type	Value
Inlet	Pipe entrance	Velocity inlet	0.297 m/s, 0.397 m/s, 0.497 m/s
Outlet	Pipe exit	Pressure Outlet	0 Pa (gauge pressure)
Wall	Pipe inner wall	No-slip Wall	Velocity = 0 m/s at the surface

Table 2
The mesh sizes for different pipe diameter

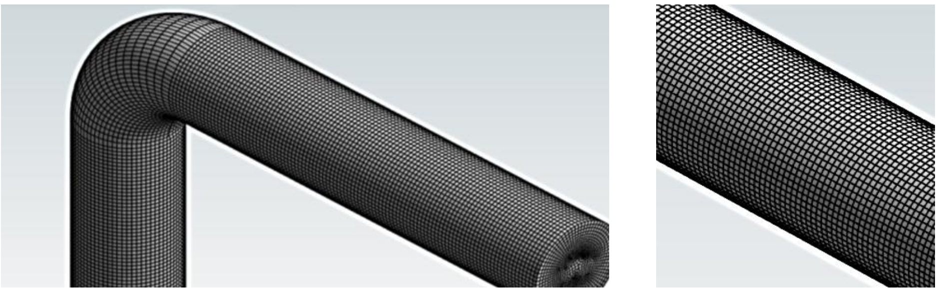
Diameter (m)	Velocity 1 (ms^{-1})	Velocity 2 (ms^{-1})	Velocity 3 (ms^{-1})	Outlet	Turbulence intensity	Turbulence model
0.015	0.297	0.397	0.497	0	5%	$k - \omega$
0.030	0.297	0.397	0.497	0	5%	$k - \omega$
0.050	0.297	0.397	0.497	0	5%	$k - \omega$

2.3 Meshing

Table 3 shows the GIT variables for different diameter. The goal was to create a finely detailed mesh, especially in curved sections where the flow bends, to properly track how boundary layers grow and secondary flows form. Three different mesh sizes were tested for each pipe diameter to ensure mesh independence. Each mesh included finer elements along the pipe walls and in the curved section to resolve velocity gradients and wall effects accurately. Mesh quality metrics such as orthogonal quality and skewness were kept within acceptable limits to ensure numerical stability. Figure 2 shows the meshing image for bend pipe.

Table 3
Grid independence test (GIT) variables

Diameter (m)	Element size (m)	Nodes
0.015	0.0022	223,725
	0.0021	226,320
	0.0020	287,885
0.030	0.0033	223,189
	0.0032	244,900
	0.0031	262,581
0.050	0.0045	221,980
	0.0044	242,424
	0.0043	254,648



(a) (b)

Fig. 2. Meshing for 90-degree bend pipe (a) Meshing at 90-degree bend (b) Close-up meshing

2.4 Post Processing

After completing the simulations, ANSYS Fluent was used to review and visualize how fluid moves through curved pipes of various diameters (1.5 cm, 3.0 cm, and 5.0 cm) at three different inlet velocities (0.297 m/s, 0.397 m/s, and 0.497 m/s). Several analysis features were used to better understand the flow patterns. Velocity contour plots were created to show how fluid speed changes throughout the pipe, making it easier to spot areas of high or low velocity, as well as zones where the flow recirculates or separates from the pipe wall. Pressure contour plots were also generated to highlight how pressure drops along the flow path, especially around the curved section where the most significant changes occur.

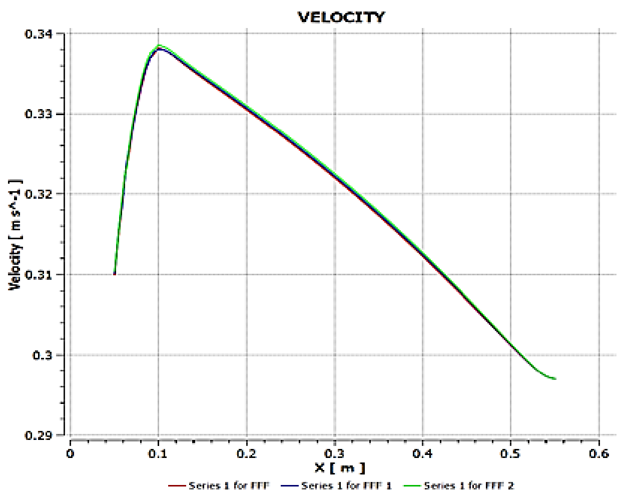
We also looked at velocity vector plots to show both the direction and strength of the fluid flow. These visuals helped us spot secondary flow patterns, like Dean vortices, which often appear in curved pipes. Additionally, we examined turbulent kinetic energy (TKE) contours to get a sense of how intense the turbulence was throughout the system. These contours highlighted the areas with the most turbulence-especially around the bends-and helped explain how energy is lost and where the flow becomes unstable.

To support quantitative analysis, we created line graphs showing how velocity and pressure change along the length of the pipe. Pressure drops charts made it easy to compare the total pressure loss for different pipe sizes and flow speeds. We also used overlay plots to directly compare results from each scenario, allowing us to see how changes in diameter and velocity affect the overall flow. These post-processing steps were key to making sense of the CFD data and drawing clear conclusions about how efficiently and dynamically fluid moves through curved pipes.

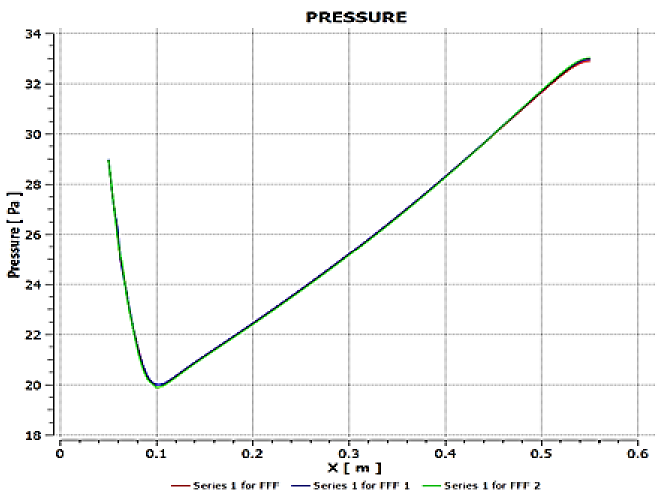
3. Results and Discussion

3.1 Grid Independence Test

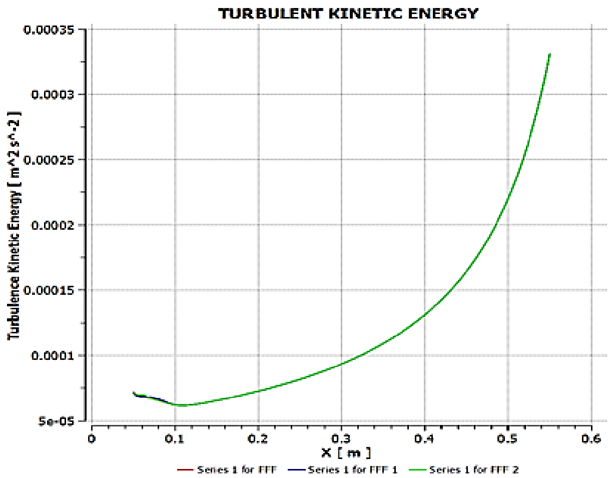
GIT was performed to ensure the convergence of the chart. Table 3 presents the number of iterations nodes for 0.015 m, 0.030 m, and 0.050 m pipe diameters. Figures 3 to 5 show the simulation results for 90-degree bend pipes with internal diameters of 0.015 m, 0.030 m, and 0.050 m, respectively, using the General Initialization Technique (GIT). Each figure includes three types of contour plots: (a) Velocity distribution, which reveals how flow speed changes throughout the bend; (b) Pressure distribution, highlighting the pressure gradients and losses caused by the curve; and (c) Turbulent kinetic energy (TKE) distribution, which pinpoints where turbulence is strongest within the flow. Together, these visualizations make it possible to directly compare how changing the pipe diameter affects key flow features, offering valuable insights into fluid dynamics in curved pipe systems.



(a)

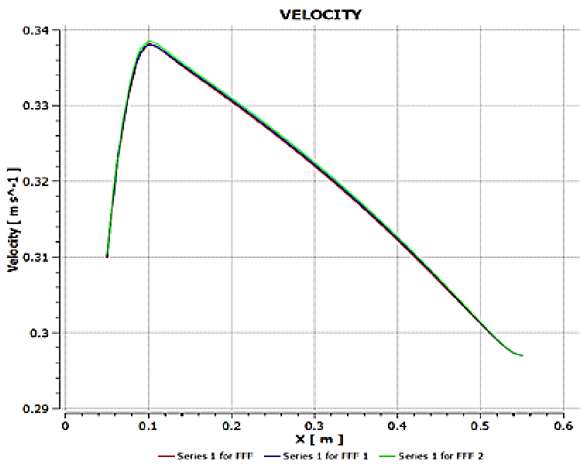


(b)

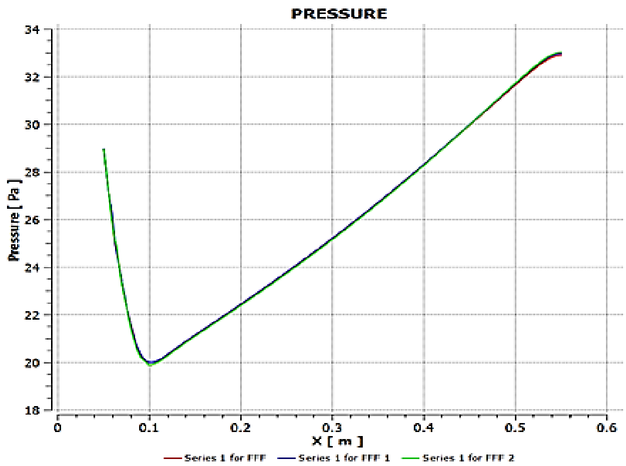


(c)

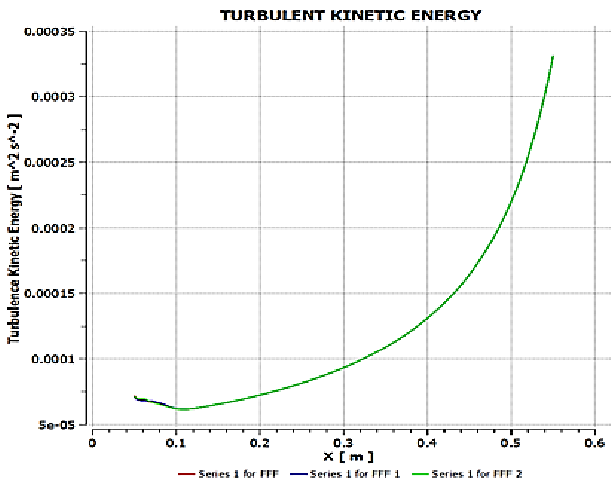
Fig. 3. Result for GIT on 0.015m 90-degree bend pipe (a) Velocity chart (b) Pressure chart (c) Turbulent kinetic energy chart



(a)



(b)



(c)

Fig. 4. Result for GIT on 0.030m 90-degree bend pipe (a) Velocity chart (b) Pressure chart (c) Turbulent kinetic energy chart

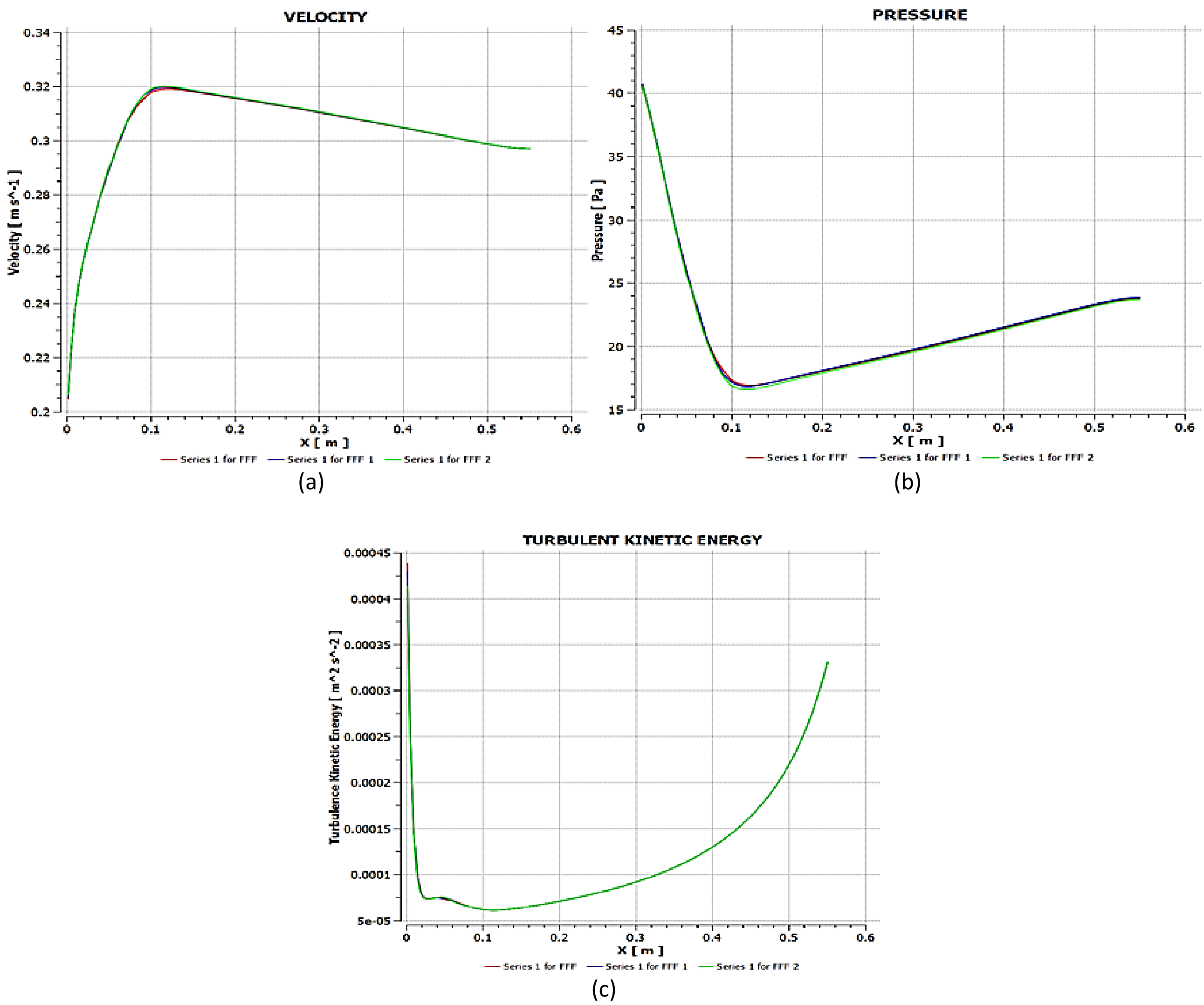
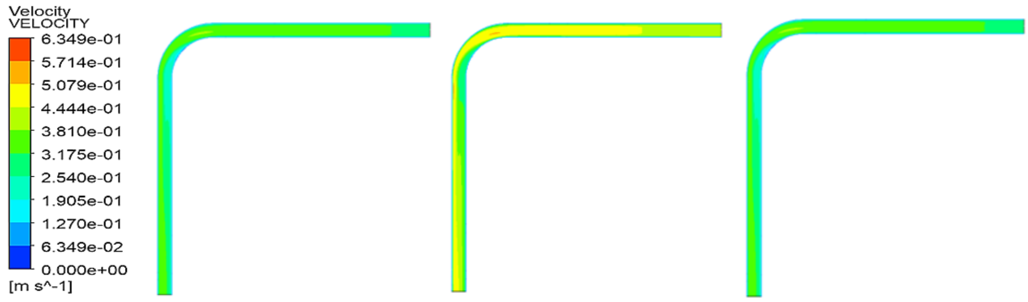


Fig. 5. Result for GIT on 0.050m 90-degree bend pipe (a) Velocity chart (b) Pressure chart (c) Turbulent kinetic energy chart

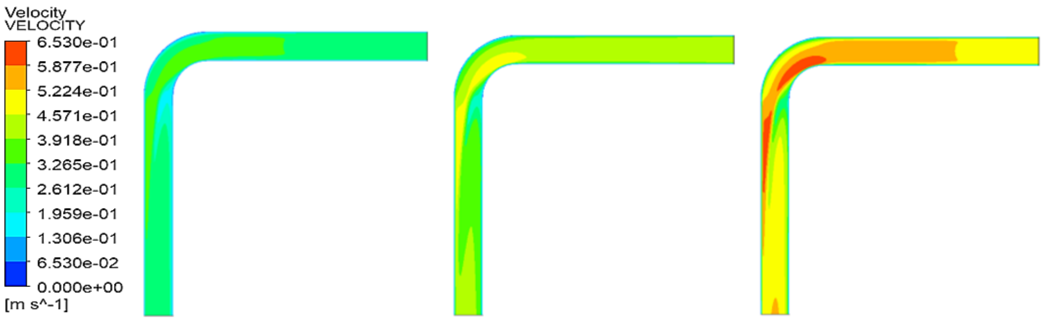
3.2 Velocity

Figures 6, 7, and 8 show how fluid flows through pipes of different sizes (0.015 m, 0.030 m, and 0.050 m) at various speeds (0.297 m/s, 0.397 m/s, and 0.497 m/s). In all cases, the fluid moves fastest along the inside curve of the 90-degree bend due to centrifugal force, while the outside curve slows down and creates a swirling, turbulent area. As the flow speed increases, these fast-moving zones become more noticeable, especially in smaller pipes, which experience sharper changes in velocity and more turbulence. Larger pipes, on the other hand, have a smoother flow with less variation in speed through the bend. This shows that both pipe size and flow speed play important roles in how fluid behaves when turning.



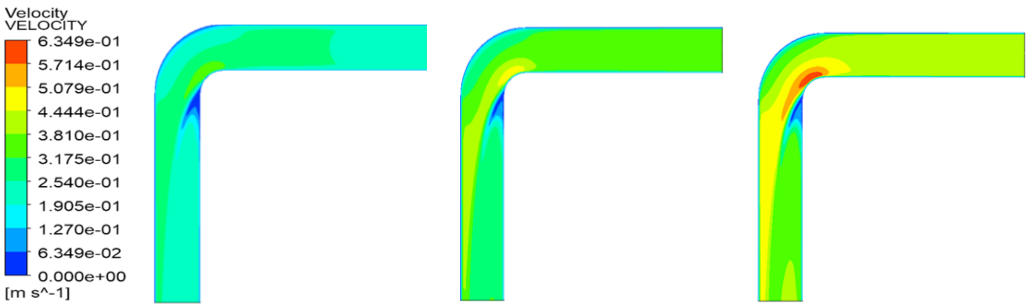
(a) (b) (c)

Fig. 6. Contour velocity on 0.015m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s



(a) (b) (c)

Fig. 7. Contour velocity on 0.030m 90-degree bend pipe (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

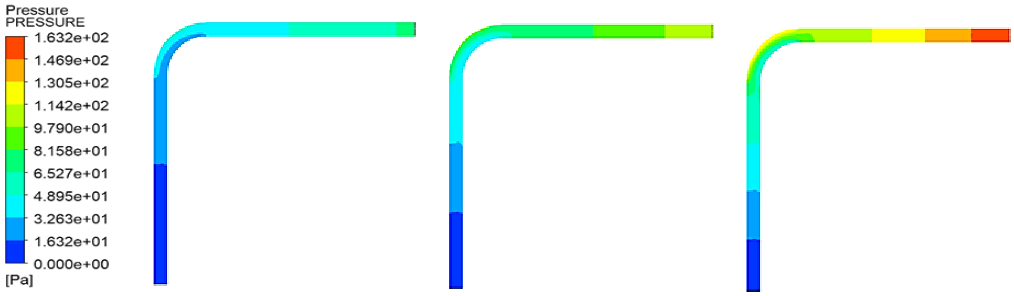


(a) (b) (c)

Fig. 8. Contour velocity on 0.050m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

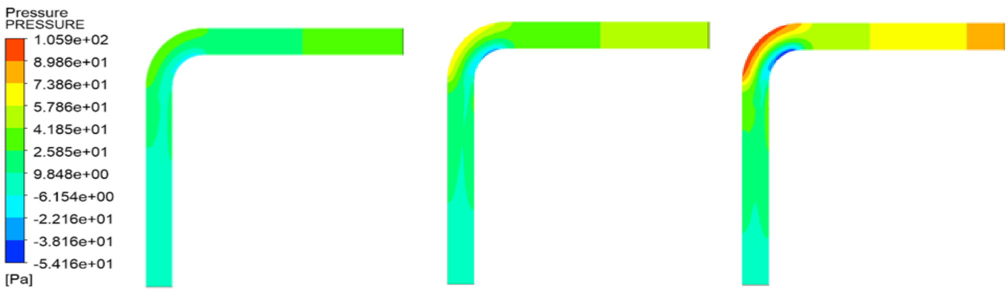
3.3 Pressure

Figures 9, 10, and 11 show how pressure changes in pipes of different sizes (0.015 m, 0.030 m, and 0.050 m) at various flow speeds (0.297 m/s, 0.397 m/s, and 0.497 m/s). Pressure is highest at the pipe's entrance and drops as the fluid moves, with a sharp decrease at the 90-degree bend, especially near the inside curve where the fluid speeds up. Smaller pipes lose more pressure because their narrow size causes more friction and resistance. When the flow speed increases, the pressure drop becomes bigger. Larger pipes have a smoother pressure change, which helps reduce pressure loss during bends. This shows that pipe size is important for keeping pressure steady in a system.



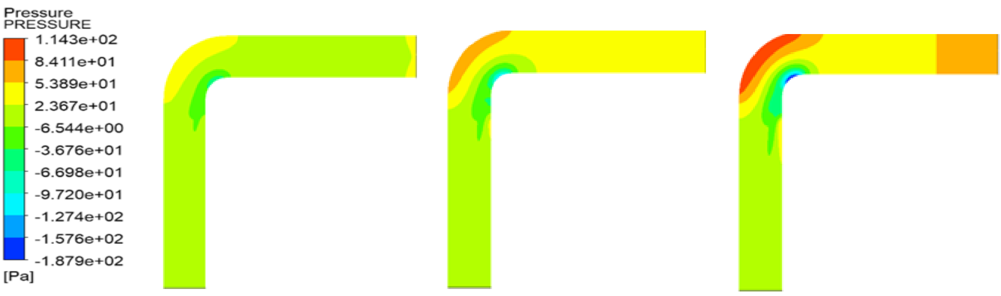
(a) (b) (c)

Fig. 9. Contour pressure on 0.015m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s



(a) (b) (c)

Fig. 10. Contour pressure on 0.030m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s



(a) (b) (c)

Fig. 11. Contour pressure on 0.050m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

3.4 Turbulence Kinetic Energy

Figures 12, 13, and 14 show how turbulence behaves inside pipes of different sizes (0.015 m, 0.030 m, and 0.050 m) at different flow speeds (0.297 m/s, 0.397 m/s, and 0.497 m/s). Turbulence is strongest near the inside curve of the 90-degree bend. In the smallest pipe, turbulence is very focused in the bend because the fluid changes direction sharply. When the flow speed increases, turbulence gets stronger. The medium pipe shows similar turbulence but spread out more, while the largest pipe has less intense and more even turbulence. This means smaller pipes and higher speeds cause more turbulence, while bigger pipes help reduce it.

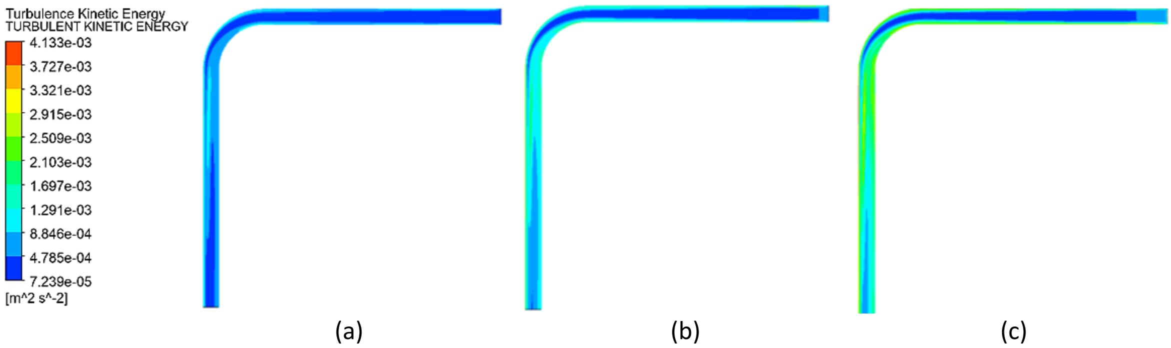


Fig. 12. Contour turbulence kinetic energy on 0.015m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

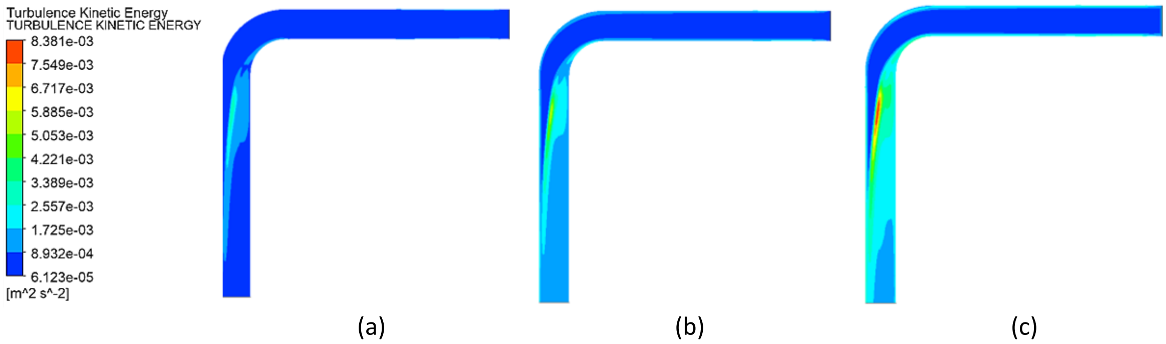


Fig. 13. Contour turbulence kinetic energy on 0.030 m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

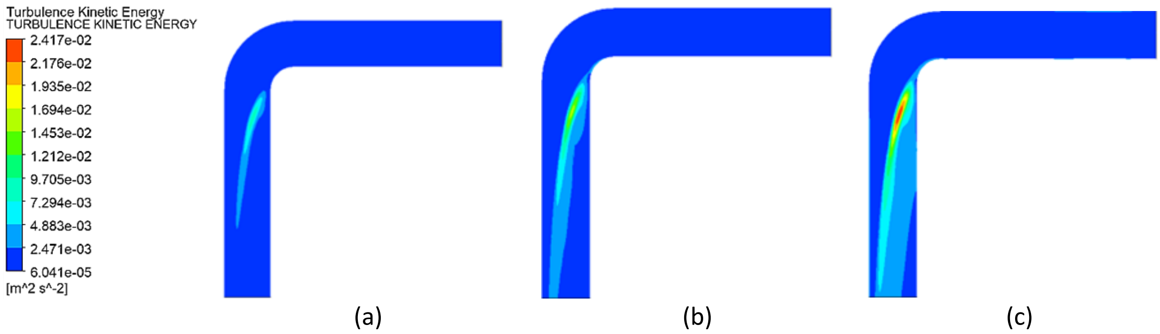


Fig. 14. Contour turbulence kinetic energy on 0.050 m 90-degree bend pipe at different velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

4. Conclusions

The CFD analysis of fluid flow through 90-degree pipe bends of different sizes (0.015 m, 0.030 m, and 0.050 m) at varying inlet speeds (0.297 m/s, 0.397 m/s, and 0.497 m/s) offers valuable insights into how velocity, pressure, and turbulence behave in these curved pipes. The simulations show that fluid moves fastest along the inside curve of the bend because of centrifugal forces, while the flow slows down along the outer curve, creating swirling recirculation zones. This effect is stronger in smaller pipes, where the tighter space causes sharper velocity changes. Pressure drops significantly at the bend, especially in smaller pipes where friction and resistance are greater, and these pressure losses increase with higher flow speeds. Larger pipes, however, show a smoother pressure drop, indicating they better handle changes in flow direction. When it comes to turbulence, the highest intensity is near the inner curve, particularly in smaller pipes where the fluid's sudden change in direction and acceleration cause more chaotic motion. Larger pipes distribute turbulence more evenly, resulting in gentler flow transitions. Overall, this study highlights how both pipe size and flow speed strongly influence fluid behavior in bends. Understanding these effects through CFD helps engineers design piping systems that maintain stable flow, reduce pressure loss, and save energy.

References

[1] Doan, Nguyen Anh Khoa, Alberto Racca, and Luca Magri. "Convolutional autoencoder for the spatiotemporal latent representation of turbulence." In *International Conference on Computational Science*, pp. 328-335. Cham: Springer Nature Switzerland, 2023. https://doi.org/10.1007/978-3-031-36027-5_24

[2] Dean, Wo R. "XVI. Note on the motion of fluid in a curved pipe." *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science* 4, no. 20 (1927): 208-223. https://doi.org/10.1080/14786440708564324

[3] Ayegba, Paul O., Lawrence C. Edomwonyi-Otu, Nurudeen Yusuf, and Abdulkareem Abubakar. "A review of drag reduction by additives in curved pipes for single-phase liquid and two-phase flows." *Engineering Reports* 3, no. 3 (2021): e12294. https://doi.org/10.1002/eng2.12294

[4] Prakash, Ravi, and Sumana Ghosh. "Effect of bend wettability on hydrodynamics of liquid–liquid two-phase flow in serpentine mini geometry." *Industrial & Engineering Chemistry Research* 60, no. 7 (2021): 3142-3155. https://doi.org/10.1021/acs.iecr.0c05279

[5] Pham, Thinh Quy Duc, Jichan Jeon, Daeseong Jo, and Sanghun Choi. "Two-phase flow simulations using 1D centerline-based C-and U-shaped pipe meshes." *Applied Sciences* 11, no. 5 (2021): 2020. https://doi.org/10.3390/app11052020

[6] Wang, Yan, Quanlin Dong, and Pengfei Wang. "Numerical investigation on fluid flow in a 90-degree curved pipe with large curvature ratio." *Mathematical Problems in Engineering* 2015, no. 1 (2015): 548262. https://doi.org/10.1155/2015/548262

[7] De Moerloose, Laurent, and Joris Degroote. "A study of the vibration of a horizontal U-bend subjected to an internal upwards flowing air–water mixture." *Journal of Fluids and Structures* 93 (2020): 102883. https://doi.org/10.1016/j.jfluidstructs.2020.102883

[8] Reetz, Florian, Tobias Kreilos, and Tobias M. Schneider. "Invariant solution underlying oblique stripe patterns in plane Couette flow." *arXiv preprint arXiv:1809.02877* (2018). https://doi.org/10.1038/s41467-019-10208-x

[9] Yanovych, Vitalii, Daniel Duda, Vaclav Uruba, and Pavel Antoš. "Anisotropy of turbulent flow behind an asymmetric airfoil." *SN Applied Sciences* 3 (2021): 1-16. https://doi.org/10.1007/s42452-021-04872-2

[10] 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

[11] Reetz, Florian, Tobias Kreilos, and Tobias M. Schneider. "Invariant solution underlying oblique stripe patterns in plane Couette flow." *arXiv preprint arXiv:1809.02877* (2018). https://doi.org/10.1038/s41467-019-10208-x

[12] Yanovych, Vitalii, Daniel Duda, Vaclav Uruba, and Pavel Antoš. "Anisotropy of turbulent flow behind an asymmetric airfoil." *SN Applied Sciences* 3 (2021): 1-16. https://doi.org/10.1007/s42452-021-04872-2

[13] Wu, Buchen, Geng Xue, Jie Feng, and Shujin Laima. "The effects of aerodynamic interference on the aerodynamic characteristics of a twin-box girder." *Applied Sciences* 11, no. 20 (2021): 9517. https://doi.org/10.3390/app11209517

[14] Blonigan, Patrick J., Mohammad Farazmand, and Themistoklis P. Sapsis. "Are extreme dissipation events predictable in turbulent fluid flows?." *Physical Review Fluids* 4, no. 4 (2019): 044606. https://doi.org/10.1103/PhysRevFluids.4.044606

[15] Morón, Daniel, Alberto Vela-Martín, and Marc Avila. "Predictability of decay events in transitional wall-bounded flows." In *Journal of Physics: Conference Series*, vol. 2753, no. 1, p. 012009. IOP Publishing, 2024. https://doi.org/10.1088/1742-6596/2753/1/012009

[16] Green, Gerrit, Dimitar G. Vlaykov, Juan Pedro Mellado, and Michael Wilczek. "Resolved energy budget of superstructures in Rayleigh–Bénard convection." *Journal of Fluid Mechanics* 887 (2020): A21. https://doi.org/10.1017/jfm.2019.1008

[17] Pandey, Ambrish, Janet D. Scheel, and Jörg Schumacher. "Turbulent superstructures in Rayleigh-Bénard convection." *Nature Communications* 9, no. 1 (2018): 2118. https://doi.org/10.1038/s41467-018-04478-0

[18] Guo, Guanming, Masaya Kamigaki, Qiwei Zhang, Yuuya Inoue, Keiya Nishida, Hitoshi Hongou, Masanobu Koutoku, Ryo Yamamoto ,Hieaki Yokohata ,Shinji Sumi, and Yoichi Ogata "Experimental study and conjugate heat transfer simulation of turbulent flow in a 90° curved square pipe." *Energies* 14, no. 1 (2020): 94. https://doi.org/10.3390/en14010094

[19] Muhammad, Noor, Maha MA Lashin, and Soliman Alkhatib. "Simulation of turbulence flow in OpenFOAM using the large eddy simulation model." *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering* 236, no. 5 (2022): 2252-2265. https://doi.org/10.1177/09544089221109736