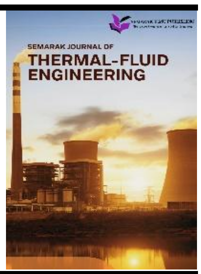
Semarak Journal of Thermal-Fluid Engineering

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



Simulation of Turbulent Flow in Various Internal Flow Geometries of Flow in a Pipe with a Sudden Turn (Sharp 135-Degree Bend)

Muhammad Zakwan Yahaya[1,*]

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

ARTICLE INFO

Article history:
Received 2 June 2025
Received in revised form 5 July 2025
Accepted 8 August 2025
Available online 29 September 2025

Keywords:
Turbulent flow; pipe bend; computational fluid dynamics; flow separation; pressure distribution; secondary flow structures; velocity profile; sudden turn; internal flow analysis

ABSTRACT

Designers of efficient fluid systems must understand turbulent flow in different, complex pipe shapes. Such flow dynamics are best altered by abrupt changes in diameter such as when a pipe bends out of shape. This work concentrates on the difficulties of studying water flow through a pipe that has a sharp bend of 135 degrees. These small-scale changes in geometry are responsible for creating flow separation, building secondary flows and increasing the pressure losses, damaging the performance of the system. Careful prediction of these phenomena is required for engineering uses in both HVAC and process industries. This research mainly aims to understand how abrupt directional changes impact the separation of flow, the pressure lost and the growth of secondary vortices within the pipe. The intention is to observe how turbulence within boundaries changes and to apply those findings to the development of energy-efficient designs. Improvements in aerodynamics were evaluated with CFD simulations in ANSYS Fluent using the standard k-ε model for steady situations. Room temperature water was used for this experiment. With inlet velocities of 0.297 m/s, 0.397 m/s and 0.497 m/s and corresponding Reynolds numbers at 4,338 around each, we confirmed the presence of turbulent flow. Because the diameter of the pipe needed to be comparable, it was always set at 1.50 cm. A grid independence study revealed that the flow results in the bend area were unaffected by changes in mesh. The motion in the model shows definite regions for the breakup of flow at the inner bend and highlighted secondary flow zones caused by strong recirculation. Extra secondary vortices were seen forming beside the inner bend wall, making it clear that the flaws of straight-pipe assumptions are now visible. As a result, this bend disrupts the smooth flow and forms difficult patterns of turbulence. This work points to the importance of using advanced simulations to predict and handle these impacts in fluid systems.

1. Introduction

Many studies in engineering fields such as mechanical, chemical and biomedical agree that understanding turbulent internal flows in complex shapes is important for optimizing fluid machinery [1-3]. Because of its importance in heat exchangers, hydraulic lines and bioreactors, studying how

* *Corresponding author.*
E-mail address: Ad210090@student.uthm.edu.my

https://doi.org/10.37934/sjotfe.6.1.114a

flow behaves at sudden changes in pipes is still highly valuable [4-6]. According to past research, most of the work on turbulent flow in bent pipes has centered around smoother shapes such as pipes bent at 90 or 180 degrees [4-9]. As a result of such studies, we have discovered that flow separation, appearance of Dean vortices and imbalanced velocity in the flow play key roles in reducing efficiency and increasing pressure drop [9-11]. Using CFD, it has been possible to understand in detail the three-dimensional flow patterns in such complex shapes that are often very difficult to observe in laboratory experiments [12-14].

Even so, more work is necessary to understand the impact of 135-degree sharp bends on electrical cables [2]. Difficulties in this geometry are caused by the direct flow changes and high levels of turbulence. As the direction of flow changes quickly, more flow separation, swirls and additional motions appear which add to the energy losses and increase the number of flow types inside [8,15-16]. Although these configurations are very useful, research involving detailed computer studies of 135-degree bends has yet to be completed.

The issue is addressed by carrying out CFD simulations of water flow around a 135-degree bend within a pipe [12]. Analysis of the general flow behavior is carried out at steady-state using ANSYS Fluent with the standard k-ε turbulence model [11,13]. The simulation studies important features of flow using factors such as redistributed velocity, the change in pressure and the turbulence intensity across a change in inlet speed from 0.297 m/s to 0.497 m/s [17]. A grid independence study was run to test the effectiveness of the grid in regions where the flow had high curvature.

The study shows that main flow separation and secondary vortex patterns appear at the inner arc [11,14,18]. The results prove that large bends in pipes alter the internal flow, backed up by earlier studies in simpler geometries and pointing to the importance of research in rarely used pipe shapes [19].

2. Methodology
2.1 Boundary Conditions and Geometry

The simulation used a three-dimensional pipeline with a sharp bend at an angle of 135 degrees as shown in Figure 1 and Figure 2. A diameter of 1.5 cm was chosen which is usual for industrial pipes. The main reason for selecting this geometry was to investigate how sudden directional shifts in fluid confined to channels affect turbulence and pressure losses. The flow domain covered the upstream and downstream areas enough to achieve fully developed flow on all edges which limited the influence of the boundaries on what happened in the bend region [20].

The velocity inlet condition was used as the intake for fluid entering the pipe. Three different velocities, at 0.297 m/s, 0.397 m/s and 0.497 m/s, were examined, growing in both Reynolds number and degree of turbulence. To enable all flow to exit, a pressure condition of zero-gauge pressure was imposed at the outlet. A no-slip treatment was applied to the pipe walls to mimic common interactions between viscous liquids and their surroundings. Most of these conditions appear in study of internal flows because they closely imitate how pipes function in actual systems [21]. The length of the inlet and outlet is 15 cm each.

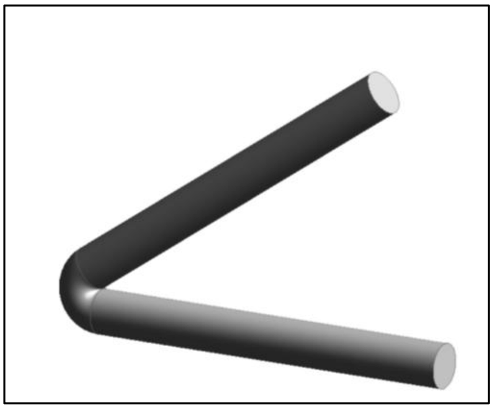


Fig. 1. Model of the pipeline with a sharp bend at an angle of 135 degrees

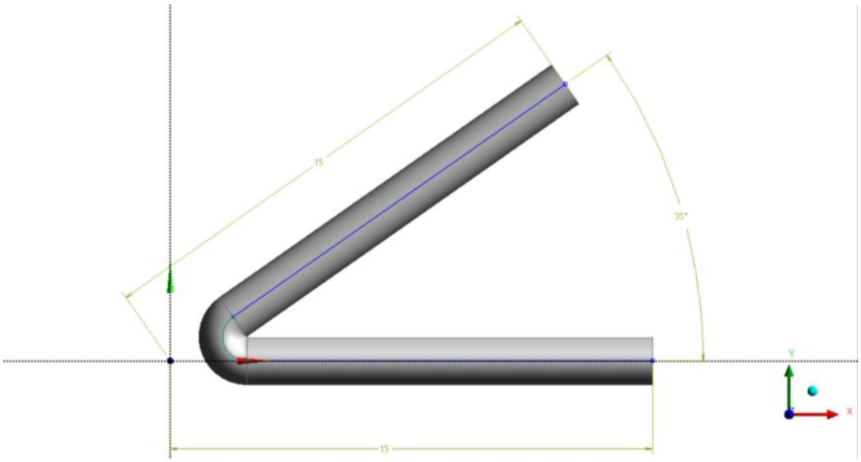
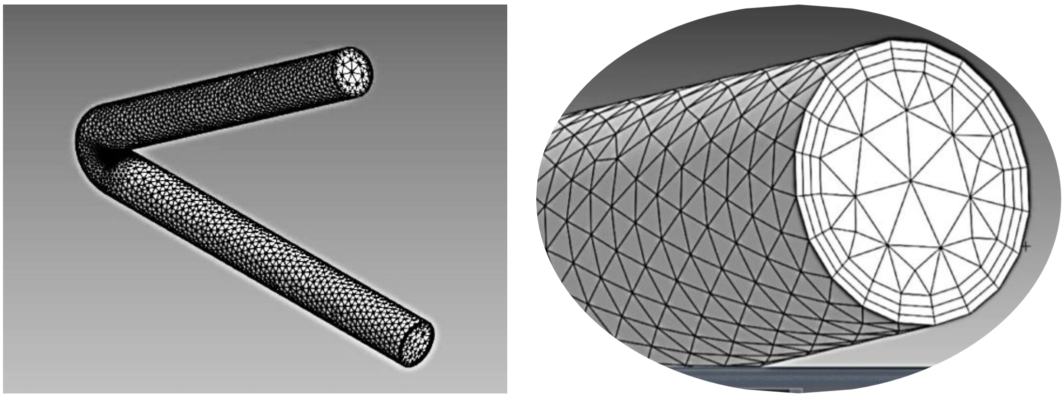


Fig. 2. Geometry of the pipe

2.2 Meshing

Tetrahedral elements on an unstructured mesh were chosen to discretize the computational domain since they handle situations like a 135-degree bend well. With this mesh design, curved and recirculating parts are accurately resolved, mainly at the inner bend point where flow gradients are strongest as shown in Figure 3. In all, there were 25,622 nodes and 88,446 elements built for the domain. The mesh density was increased close to the bend so that key events in the flow such as boundary layer breakaway and secondary vortex generation, could be resolved well. Extra care was taken where the wall met the domain to assure that y^+ had acceptable values for the chosen turbulence model, so that wall shear stresses were reliable. For better results and mesh quality, grid independence studies were run. Results suggested that further improvement in mesh resolution did not strongly affect pressure drop or velocity behavior in the curved bend.



(a) (b)

Fig. 3. Meshing of the pipe (a) Meshing of the pipe (b) Tetrahedral elements

2.3 Governing Equation

The numerical simulation is based on solving the three-dimensional, steady-state Reynolds-Averaged Navier-Stokes (RANS) equations, which describe the conservation of mass and momentum for incompressible, turbulent flow. The continuity and momentum equations can be expressed as follows:

Continuity equation: $\nabla \cdot \vec{v} = 0$ (1)

Momentum equation: $\rho(\vec{v} \cdot \nabla)\, \vec{v} = -\nabla p + \mu \nabla^2 \vec{v} + \nabla \cdot \tau_t$ (2)

where $\vec{v}$ is mean velocity vector, ρ is the fluid density, p is the pressure, μ is the dynamic viscosity and τ_t is the Reynolds stress tensor due to turbulence.

A turbulence model is necessary to complete the RANS equations. This study used the standard k-ε model because it is well established and commonly used in internal flow simulations. According to this model, two transport equations are used, one for turbulent kinetic energy and the other for turbulent dissipation rate.

Turbulent kinetic energy (k): $\frac{\partial k}{\partial t} + \vec{v} \cdot \nabla k = \nabla \cdot \left(\frac{\mu_t}{\sigma_k} \nabla k\right) + P_k - \varepsilon$ (3)

Turbulent dissipation rate (ε): $\frac{\partial \varepsilon}{\partial t} + \vec{v} \cdot \nabla \varepsilon = \nabla \cdot \left(\frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon\right) + C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \frac{\varepsilon^2}{k}$ (4)

where μ_t is the eddy viscosity, P_k is the production of turbulent kinetic energy, σ_k and σ_ε are turbulent Prandtl numbers, and $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are empirical constants. The standard values for these constants, as used in ANSYS Fluent are:

i. $C_\mu = 0.09$
ii. $C_{1\varepsilon} = 1.44$
iii. $C_{2\varepsilon} = 1.92$
iv. $\sigma_k = 1.00$
v. $\sigma_\varepsilon = 1.30$

2.4 Grid Independence Test (GIT)

A Grid Independence Test was conducted to ensure that the simulation results were not significantly affected by mesh refinement. The test involved varying the number of mesh divisions along the edge (P3) from 20 to 100, corresponding to a range of element counts (P1) from 88,446 to 109,048. Key output parameters analyzed included outlet velocity (P4), outlet pressure (P5), and maximum velocity within the domain (P6). As shown in Table 1, increasing mesh resolution resulted in a gradual change in all monitored parameters. However, beyond P3 = 70 (99,663 elements), the variation in outlet velocity and maximum velocity was minimal, with changes less than 1%, indicating convergence. Similarly, outlet pressure stabilized near 33 kPa, suggesting reliable pressure prediction.

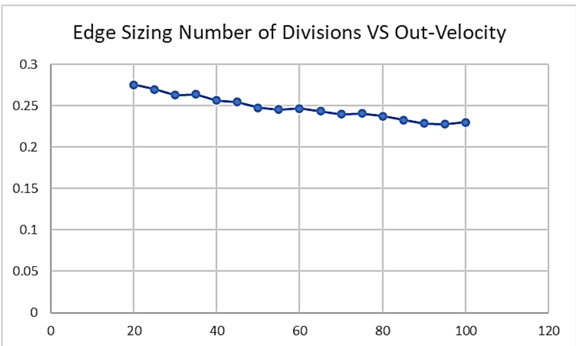
Table 1
Grid independence results summary

Divisions (P3)	Elements (P1)	Nodes (P2)	Out-velocity (P4) (m/s)	Out-pressure (P5) (Pa)	Max velocity (P6) (m/s)
20	88446	25622	0.2753	40.7366	0.3695
70	99663	29364	0.2396	33.1557	0.3785
100	109048	32254	0.2302	31.3717	0.3802

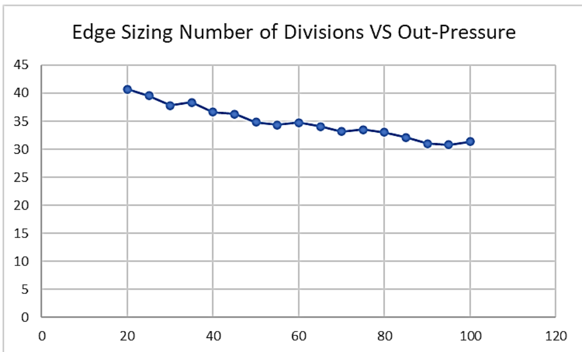
Figure 4 below illustrates the convergence behavior of outlet velocity and pressure as mesh density increases. Based on the results, the mesh with P3 = 70 (approx. 100k elements) was selected for subsequent simulations. This grid achieved a good balance between computational efficiency and result accuracy, with further refinement offering negligible improvements as shown in Figure 5. To quantify convergence, the percentage change in outlet velocity (P4) and maximum velocity (P6) between grid levels was calculated. Between P3 = 70 (99,663 elements) and P3 = 100 (109,048 elements):

i. The outlet velocity (P4) decreased by approximately 3.94%.
ii. The maximum velocity (P6) increased by only 0.46%.

These minimal variations, especially in the maximum velocity, which is more sensitive to mesh density near sharp features, suggest that the solution is effectively grid-independent at P3 = 70. This mesh level provides a reliable result without unnecessary computational overhead.



(a)



(b)

Fig. 4. Convergence behavior at increase of mesh density (a) Outlet velocity (b) Outlet pressure

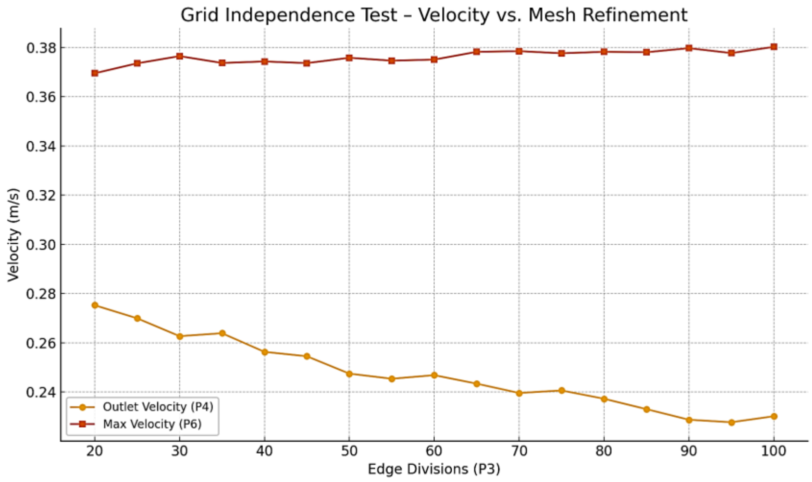


Fig. 5. Grid independence test (GIT)

3. Results and Discussion

The results of the simulation for turbulent water flowing through a 135-degree pipe bend at three different inlet speeds are analyzed in detail here. The main aim was to see how changing the flow rate impacts the speed, pressure and vortex formation in the pipe. Velocity contours, streamlines and volume renderings of velocity and pressure are used to view and interpret the results. Every technique adds details to our knowledge of how the curved pipe influences the number and structure of eddies in the circulating fluid volume. The results from all the visualization approaches are separated to draw attention to the various flow patterns and their changes throughout the experiments and with the changing time step.

Reports of flow in the pipe system indicate a clear evolution in the spread of speeds with every additional iteration and inlet velocity as shown in Figure 6. After 500 iterations, the flow field at an inlet speed of 0.297 m/s is still early in development and reveals minor flow distortion around the inside wall of the bend. The behavior becomes more stable by 1000 iterations and a separation region can be seen along the inner curve, even though its size is not large due to the moderate Reynolds number. The velocity field is steady after 1500 iterations, with a flat appearance downstream of the bend and noticeable but not severe centrifugal effects.

There is a much easier-to-see transition when the given speed is 0.397 m/s than when the initial speed is 0.158 m/s. After 500 iterations, the motion becomes rougher and there are plenty of abrupt directional changes along the curve. With over 1000 iterations, we see the main high-velocity flow beginning to point outward, along with a low-velocity region emerging toward the inner side, suggesting that the flow already begins to rearrange near the boundary layer. After 1500 iterations, the velocity profile becomes steady and clearly shows greater velocity changes on the pipe's surface farther from the center post-bend. Here, the centrifugal action is stronger and the boundary layer's part from the wall easier than in cases with lower inlet velocity.

At the maximum tested inlet velocity of 0.497 m/s, the flow pattern develops even faster. Following 500 trials, the flow field is still disordered and noisy, showing extensive velocity fluctuations within the bend which proves that strong turbulence is being recorded early. Once the simulation has run for 1000 iterations, the high velocity portion of the flow is very close to the outer wall and the inner wall region is seeing increasing separation of the flow. After 1500 iterations, the simulation ends, showing that one main corridor holds high speed, whereas much of the water becomes almost still in regions with recirculation. Strong differences in flow speed and large flow differences suggest there is significant turbulent activity, formation of whirling currents and considerable energy loss.

Overall, the method shows that numerical convergence and different flow regimes play a role in how the velocity is distributed throughout the area. It stresses that when the flow is faster, more complex features form and the simulation needs more passes to entirely resolve those turbulent features and end in convergence. The iterative method further proves that the solver predicts the change from transient to steady flow well in curved pipes.

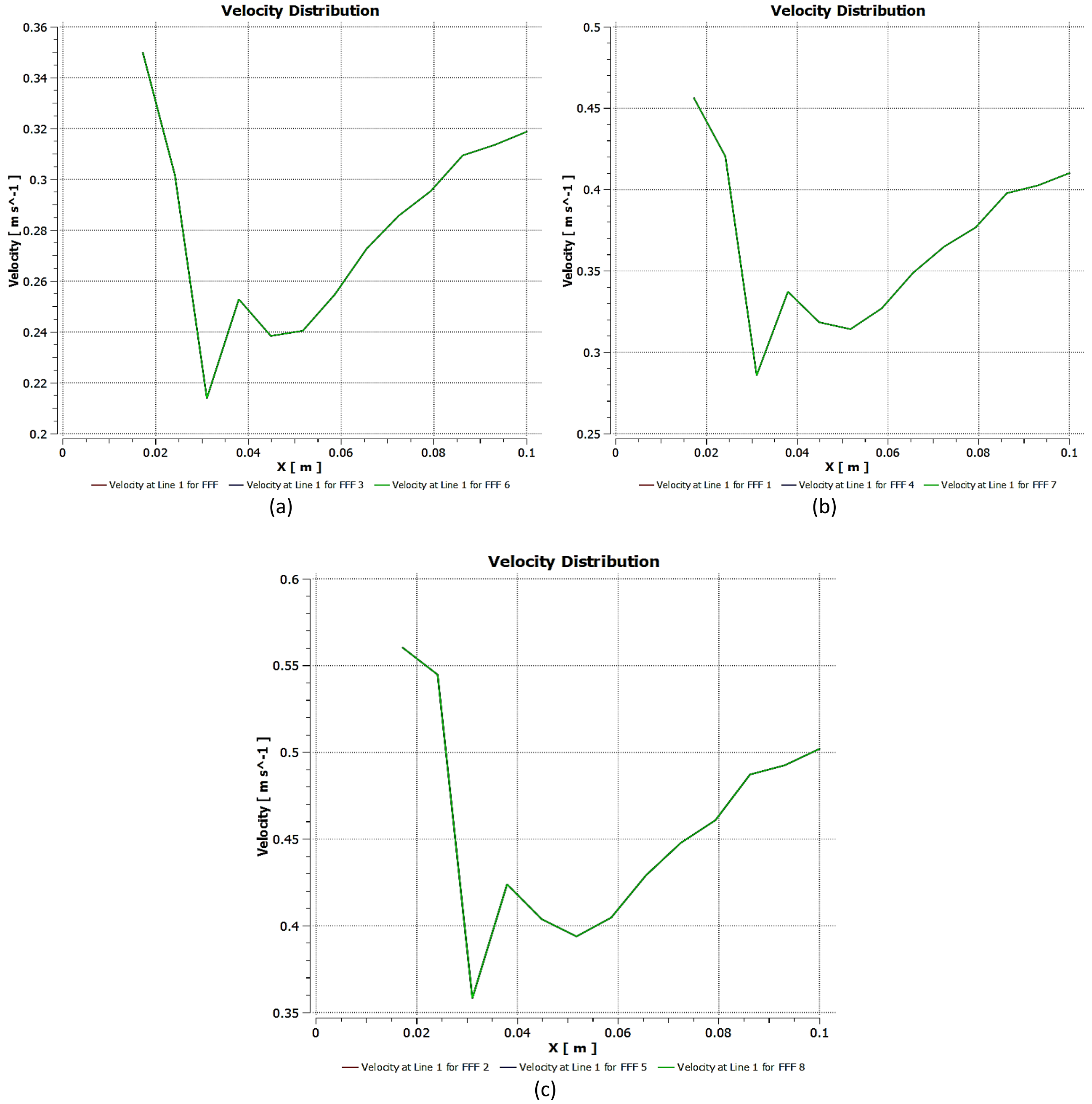


Fig. 6. Velocity distribution for 500, 1000, and 1500 iterations at velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

3.2 Analysis

3.2.1 Velocity analysis of contour

Data from simulation at multiple velocities and iterations allows us to draw key conclusions about the flow of the pipe bend as shown in Figure 7 until Figure 9. When inlet velocity is 0.297 m/s, the flow is quite symmetrical and a low, moderate separation occurs on the inner end of the bend. Between 500 and 1500 iterations in the simulation, the velocity curve becomes nicely defined and there is a smoother transition and easy identification of slower zones at the inner curve. When reducing the height from 27.6 to 27.3 mm, a higher inlet velocity of 0.397 m/s causes the velocity profile to become more uneven. A major speeding-up current is visible near the outer edge of the bend, proving that centrifugal force pushes the flow to the outside. At the same time, a recirculating area starts near the wall, where fluid speeds decline sharply due to the effect of boundary layer separation.

When the inlet velocity reaches a maximum of 0.497 m/s, the graph demonstrates fast movement of the water by the outer bend and a noticeable slowdown of flow plus reversal inside the bend. Once the model reaches 1500 iterations, the contours become less active and reveal where there is strong shear and a growing region of slower flow after the bend. The results confirm that, as prescribed by classical fluid dynamics, raising velocity increases inertia, detaches boundary layers faster and enhances the mixing of turbulence, especially in places with abrupt changes in shapes.

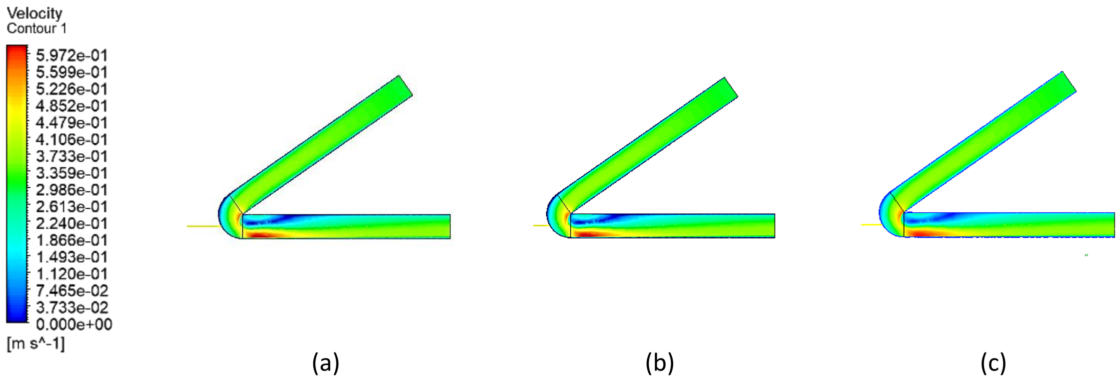


Fig. 7. Velocity analysis of contour for velocity = 0.297 m/s for iterations of (a) 500 (b) 1000 (c) 1500

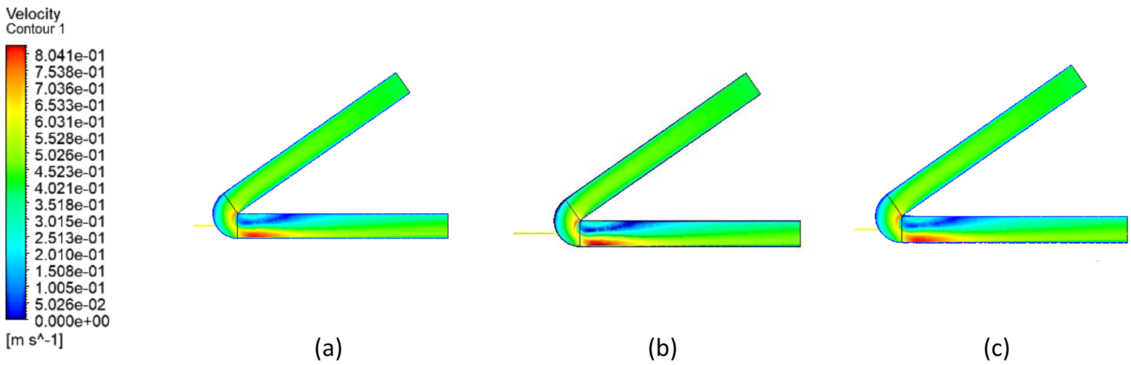


Fig. 8. Velocity analysis of contour for velocity = 0.397 m/s for iterations of (a) 500 (b) 1000 (b) 1500

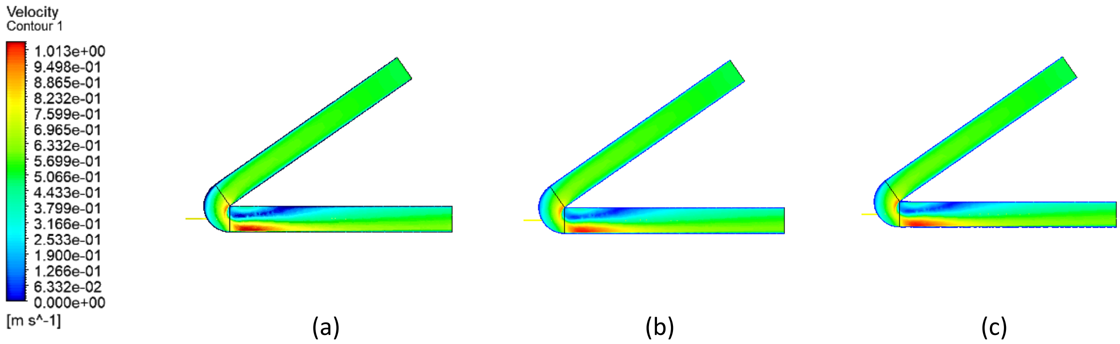


Fig. 9. Velocity analysis of contour for velocity = 0.497 m/s for iterations of (a) 500 (b) 1000 (c) 1500

3.*2.2 Velocity analysis of streamline*

As shown from Figure 10 to Figure 12, streamline visualizations help explain how fluid particles move in the pipe, with a strong emphasis on disruptions caused by pipe curvature. At this speed, the streamlines continue in a mainly laminar way and are only slightly curved in the nearby bend. Flow separation is low and there are nominal disruptions close to the center of the channel. As the velocity rises to 0.397 m/s, the streamlines start to diverge more widely near the bend and develop small curly patterns which confirm the presence of vortices inside the bend. The combination of flow and vortex effects really stand out at 0.497 m/s which is visible on the distorted streamlines. Because of centrifugal force, the flow in the core gets pushed to the outer edges, resulting in large secondary flows with several vortices that keep existing downstream.

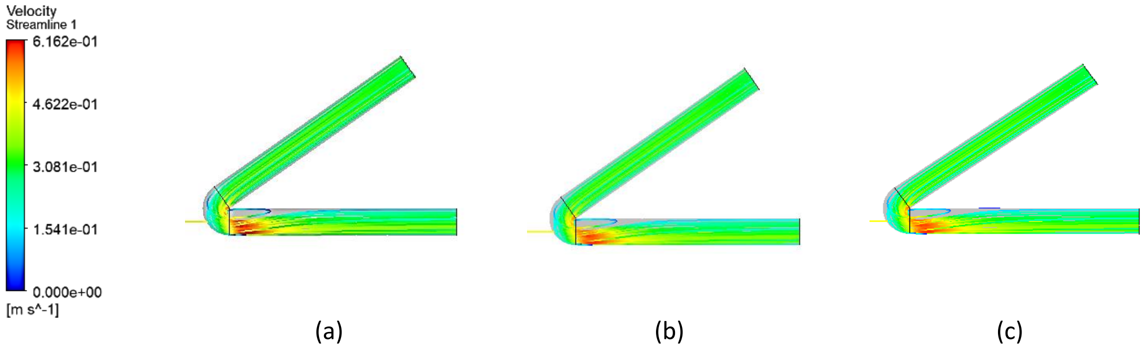


Fig. 10. Velocity analysis of streamline for velocity = 0.297 m/s for iterations of (a) 500 (b) 1000 (c) 1500

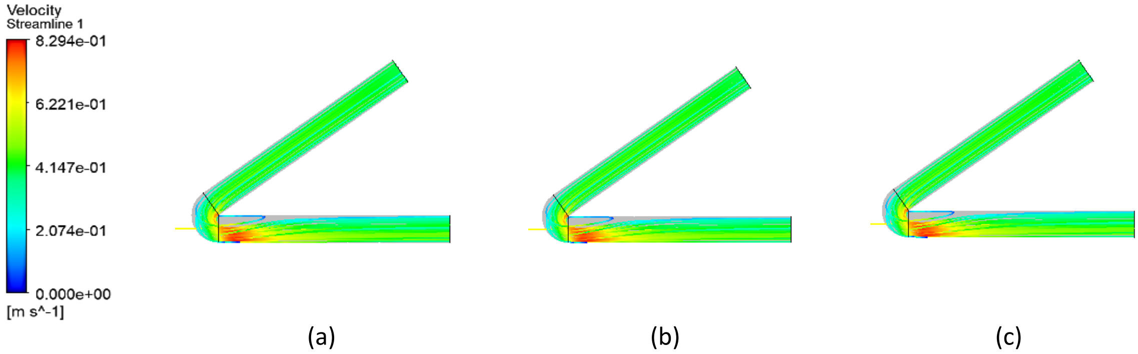
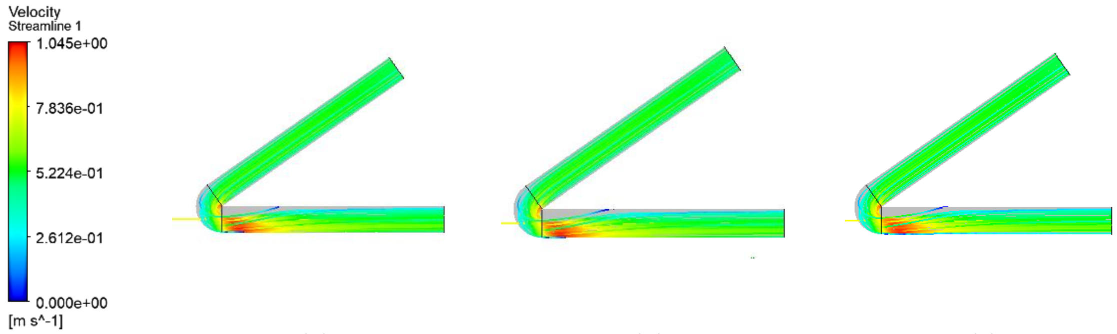


Fig. 11. Velocity analysis of streamline for velocity = 0.397 m/s for iterations of (a) 500 (b) 1000 (c) 1500



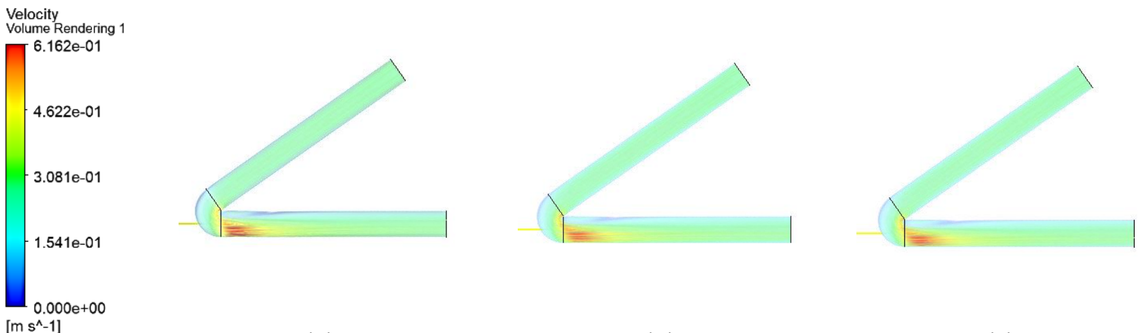
(a) (b) (c)

Fig. 12. Velocity analysis of streamline for velocity = 0.497 m/s for iterations of (a) 500 (b) 1000 (c) 1500

Once the system has gone through 1500 iterations, the fast-moving streamlines show a regime where the flow is extremely chaotic and unpredictable. Such swirls come about when centrifugal effects and viscous shear interact as fluids flow around a curve within a pipe. Having vortices in the boundary layer increases mixing and exchange of momentum, although it counteractively increases turbulence and areas of recirculation.

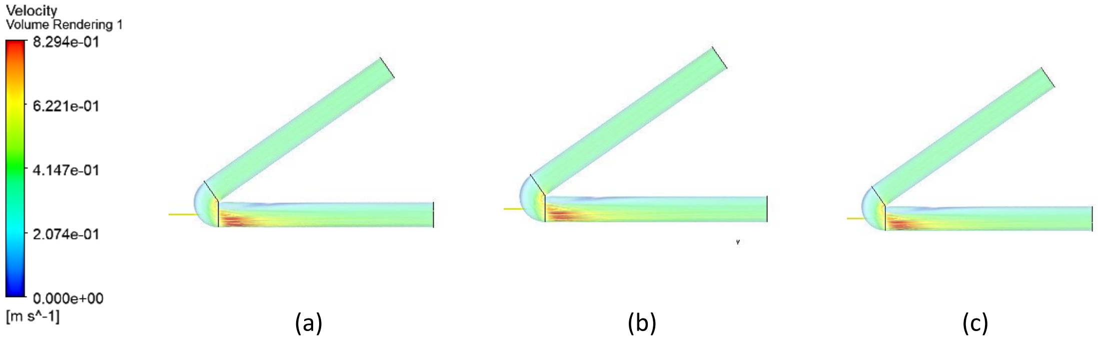
3.2.3 Velocity analysis of volume rendering

The three-dimensional look of the velocity field through volume rendering reveals where and how the speed of the flow changes in the pipe as shown from Figure 13 to Figure 15. If the speed is low enough, the simulation demonstrates that particles move at similar speeds as they switch smoothly into the bend. At a velocity of 0.397 m/s, it becomes clear in the simulation that the outer wall is dominated by fast-moving liquid, while the section that rounds the inner bend exhibits slower motion. Visualization clearly brings out the big difference in speed between the fast-moving zones and the areas where fluid is spinning. It is evident that the fluid moves very fast on the outer side of the curve and reaches a large velocity decrease inside the bend region because of the separation bubble. 3D representations confirm the observations of contours and streamline by adding spatial depth to what is seen in the flow as represented by Figure 7 to Figure 15. From the outer curves at high speed and the turbulence near the outlet, it is easy to tell that sharper bends at faster speeds change the velocity pattern, making the flow more unstable and less efficient.



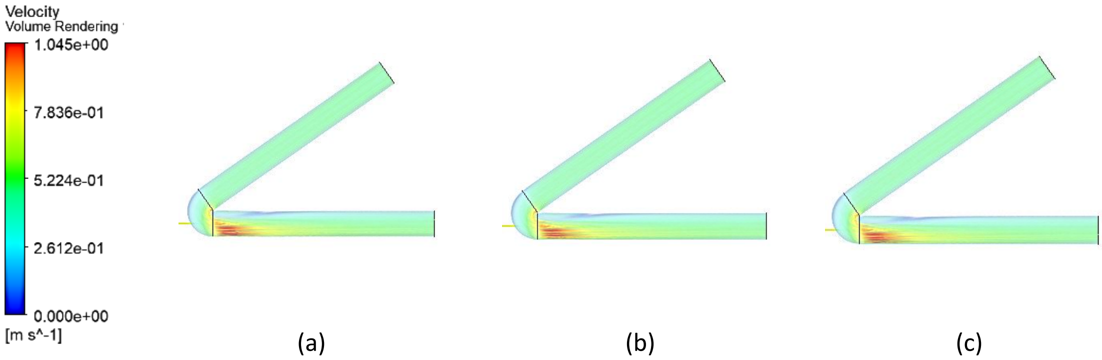
(a) (b) (c)

Fig. 13. Velocity analysis of volume rendering for velocity = 0.297 m/s for iterations of (a) 500 (b) 1000 (c) 1500



(a) (b) (c)

Fig. 14. Velocity analysis of volume rendering for velocity = 0.397 m/s for iterations of (a) 500 (b) 1000 (c) 1500



(a) (b) (c)

Fig. 15. Velocity analysis of volume rendering for velocity = 0.497 m/s for iterations of (a) 500 (b) 1000 (c) 1500

3.2.4 Pressure analysis of volume rendering

Energy transformation inside the flow is shown clearly in the pressure distribution as result of volume rendering as shown in Figure 16 to Figure 18. Since the pressure decline across the pipeline bend is moderate at 0.297 m/s, the head loss is low. Rising inlet velocity strengthens the pressure drop, especially at 0.497 m/s, when the pressure very close to the inner wall drops a lot. Pressure is greatest upstream of the curve, but it reduces sharply downstream as the water separates from the curved surface and becomes more turbulent. It stands for losses in momentum connected to how the jet changes direction and turbulence. A faster inlet velocity leads to a greater loss of pressure in the pipe bend, confirming that pressure and inlet velocity are connected in turbulent systems. By using illustrations, these findings show that pressure losses in bent piping need to be considered, especially when the flow rate is high.

From the CFD analysis, we discovered that that a 135-degree bend in the pipe makes a major difference to the flow field at high velocities. Both the velocity contours and streamline plots demonstrated that there was more asymmetry at the higher inlet velocity, separation at the boundaries and increased speed in some regions, while secondary flows, including Dean vortices, were strongest due to centrifugal forces. Thanks to volume renderings, it could be clearly observed that the gradient of fluid speeds was very high at the bend and that a lot of pressure was being lost there. Generally, the findings suggest that a higher inlet velocity results in more troublesome turbulence, flow distortion and energy dissipation. For this reason, considering how fluids go through sharp bends should become a key focus when building and studying piping systems. They not only confirm the predictions theory makes but also prove that CFD tools can handle and show turbulent flow details accurately.

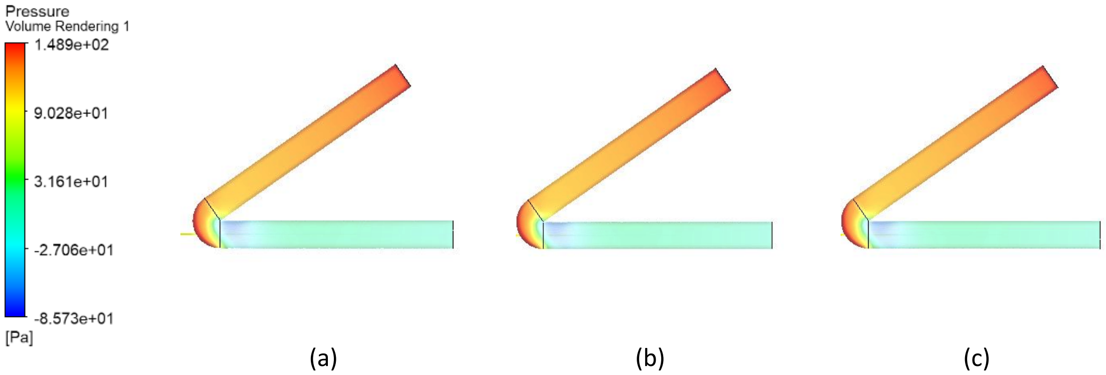


Fig. 16. Pressure analysis of volume rendering for velocity = 0.297 m/s for iterations of (a) 500 (b) 1000 (c) 1500

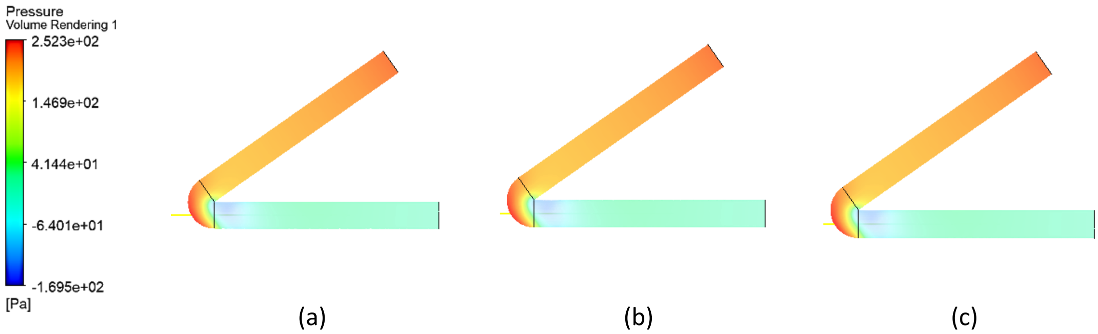


Fig. 17. Pressure analysis of volume rendering for velocity = 0.397 m/s for iterations of (a) 500 (b) 1000 (c) 1500

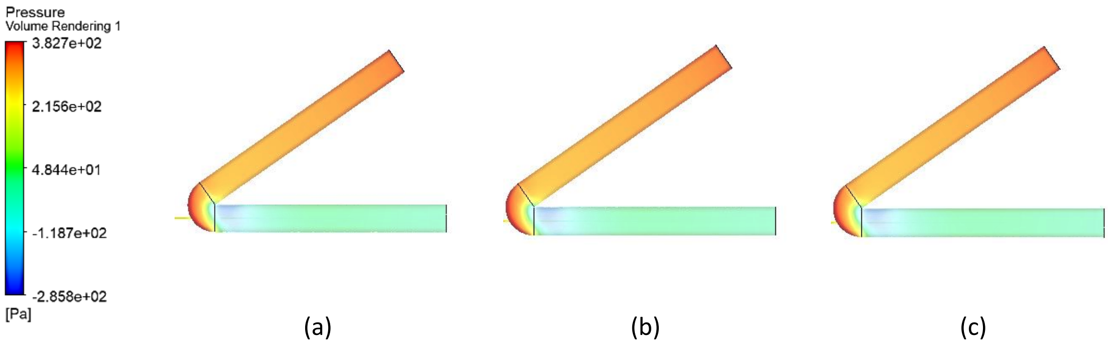


Fig. 18. Pressure analysis of volume rendering for velocity = 0.497 m/s for iterations of (a) 500 (b) 1000 (c) 1500

4. Conclusions

By using CFD, the study met its aim of studying the internal flow behavior in the complex sharp-bend geometry of the pipe. Studying different inlet velocities allowed the researchers to learn about the influence of quick changes in the momentum boundary layer on velocity, pressure loss and turbulence content. The simulation outcomes make it clear that the bend causes major disruptions in the fluid flow. Among important observations are unequal flow speeds, areas where the flow detaches and forms recirculation zones, as well as Dean vortices. Rising inlet velocities made these effects stronger since the mixture was thoroughly mixed, even while causing more pressure drop across the bend.

The system was able to accurately estimate these thorny phenomena when the standard k-ε turbulence model was used in combination with a validated grid from a grid independence check. The results demonstrate that even small differences in inlet velocity can bring about noticeable changes in the flow, demonstrating how sensitive sharp bends are to turbulent conditions. In short, the research underlines the need for proper design around sharp bends in piping systems. Making turbulence-reducing or smooth-flowing changes to the pipe's surface could save energy and enhance performance. Further studies should consider running unsteady simulations, using advanced turbulence models (k-ω SST or LES) and validating the results with experiments which would give new insights into flow in unusual geometries.

References

[1] Shao, Qing, Weihua Hui, and Futing Bao. "Numerical and experimental investigation on turbulent flow and secondary flow in a 180° bend--Circular Cross-section." In *IOP Conference Series: Earth and Environmental Science*, vol. 189, no. 2, p. 022061. IOP Publishing, 2018. https://doi.org/10.1088/1755-1315/189/2/022061

[2] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA Journal* 32, no. 8 (1994): 1598-1605. https://doi.org/10.2514/3.12149

[3] Hüttl, T. J., and R. Friedrich. "Direct numerical simulation of turbulent flows in curved and helically coiled pipes." *Computers & Fluids* 30, no. 5 (2001): 591-605. https://doi.org/10.1016/S0045-7930(01)00008-1

[4] Guo, Guanming, Masaya Kamigaki, Qiwei Zhang, Yuuya Inoue, Keiya Nishida, Hitoshi Hongou, Masanobu Koutoku, Ryo Yamamoto, 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

[5] Kalpakli, Athanasia. "Experimental study of turbulent flows through pipe bends." PhD diss., KTH Royal Institute of Technology, 2012.

[6] Apalowo, Rilwan Kayode, and Cletus John Akisin. "CFD-based investigation of turbulent flow behavior in 90-deg pipe bends." *Journal of Applied Research in Technology & Engineering* 5, no. 2 (2024): 53-62. https://doi.org/10.4995/jarte.2024.20665

[7] Azzola, J., J. A. C. Humphrey, H. Iacovides, and B. E. Launder. "Developing turbulent flow in a U-bend of circular cross-section: measurement and computation." *Journal of Fluids Engineering* 108, no. 2 (1986): 214-221. https://doi.org/10.1115/1.3242565

[8] Sudo, K., M. Sumida, and HJEiF Hibara. "Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend." *Experiments in Fluids* 25, no. 1 (1998): 42-49. https://doi.org/10.1007/s003480050206

[9] Enayet, M. M., M. M. Gibson, A. M. K. P. Taylor, and M. Yianneskis. "Laser-Doppler measurements of laminar and turbulent flow in a pipe bend." *International Journal of Heat and Fluid Flow* 3, no. 4 (1982): 213-219. https://doi.org/10.1016/0142-727X(82)90024-8

[10] Dean, W. R. "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

[11] Sudo, K., M. Sumida, and H. Hibara. "Experimental investigation on turbulent flow in a square-sectioned 90-degree bend." *Experiments in Fluids* 30, no. 3 (2001): 246-252.https://doi.org/10.1007/s003480000157

[12] Patankar, Suhas V. *Numerical Heat Transfer and Fluid Flow*. Series in Computational Methods in Mechanics and Thermal Sciences. Washington, D.C.: Hemisphere Publishing Corporation, 1980. https://doi.org/10.1201/9781482234213

[13] Dutta, Prasun, Himadri Chattopadhyay, and Nityananda Nandi. "Numerical studies on turbulent flow field in a 90 deg pipe bend." *Journal of Fluids Engineering* 144, no. 6 (2022): 061104. https://doi.org/10.1115/1.4053547

[14] Patel, Virendra C., Wolfgang Rodi, and Georg Scheuerer. "Turbulence models for near-wall and low Reynolds number flows-a review." *AIAA Journal* 23, no. 9 (1985): 1308-1319. https://doi.org/10.2514/3.9086

[15] Anwer, M., and R. M. C. So. "Swirling turbulent flow through a curved pipe: Part I: Effect of swirl and bend curvature." *Experiments in Fluids* 14, no. 1 (1993): 85-96. https://doi.org/10.1007/BF00196992

[16] Spedding, P. L., E. Benard, and G. M. McNally. "Fluid flow through 90 degree bends." *Developments in Chemical Engineering and Mineral Processing* 12, no. 1-2 (2004): 107-128. https://doi.org/10.1002/apj.5500120109

[17] Petrakis, M. A., G. T. Karahalios, and S. Kaplanis. "Steady flow in a curved pipe with circular cross-section. comparison of numerical and experimental results." *Open Fuels & Energy Science Journal* 2 (2009): 20-26. https://doi.org/10.2174/1876973X01002010020

[18] Dutta, P., S. K. Saha, and N. Nandi. "Numerical study of curvature effect on turbulent flow in 90 pipe bend." *Dep. of Aerospace Eng. and Applied Mechanics, ICTACEM-2014/028* (2014).

[19] Sudo, K., M. Sumida, and HJEiF Hibara. "Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend." *Experiments in Fluids* 25, no. 1 (1998): 42-49. https://doi.org/10.1007/s003480050206

[20] Dutta, Prasun, Sumit Kumar Saha, Nityananda Nandi, and Nairit Pal. "Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling." *Engineering Science and Technology, an International Journal* 19, no. 2 (2016): 904-910. https://doi.org/10.1016/j.jestch.2015.12.005

[21] Kalpakli, Athanasia, and Ramis Örlü. "Turbulent pipe flow downstream a 90 pipe bend with and without superimposed swirl." *International Journal of Heat and Fluid Flow* 41 (2013): 103-111. https://doi.org/10.1016/j.ijheatfluidflow.2013.01.003