AMERICAN Journal of Engineering,
Mechanics and Architecture
Volume 3, Issue 12, 2025 ISSN (E): 2993-2637

Analysis of Fluid Flow through 90° Pipe Bends: Pressure Losses, Flow Separation, And Velocity Distribution

Rawan A. Fayyad*, Issam M. Ali Aljubury
College of Engineering, University of Baghdad, 10071, Baghdad, Iraq

Abstract: This paper gives a computational fluid dynamics (CFD) of the flow of an incompressible fluid through a 90deg pipe bend on the open-source SimScale platform. The flow of curved pipes is typified by the redistribution of velocity, development of a secondary flow and pressure losses so the primary goal is to determine the magnitude of the pressure losses and study the behaviour of velocity and the behaviour of separation as a function of Reynolds number and bend curvature. The parametric study of the elbow geometries is followed with variable curvature ratios (rc/D) and flow conditions (laminar and turbulent) on the geometries. Structured meshes in which the near-wall refinement has been used and steady Reynolds-averaged Navier-Stokes (RANS) turbulence models which are the standard k-e model and the SST k-o model are used to perform simulations with chosen transient cases to capture unsteady separation. The analysis has been conducted on pressure loss coefficients and velocity profiles, pressure contours, and the second flow patterns. The findings indicate the great effect of bend geometry and flow regime on flow organization and energy losses. The paper draws attention to the originality of utilizing an open-access CFD platform, which offers a reproducible numerical model, which can be used in teaching engineering and in initial design and subsequent experimental support.

Keywords: incompressible, turbulent, SimScale, pressure, Reynolds number.

1 Introduction

Flow of fluids in pipe turns is a basic phenomenon in most engineering as in HVAC systems, water distribution pipelines, oil and gas pipelines and industrial cooling loops. As a fluid flows through a 90 deg bend the direction of flow suddenly changes and the velocity and pressure are re-distributed. This modification causes centrifugal forces that bring about secondary flows and could cause separation of flows, energy losses, and high wall shear stress. The effects have serious impacts on the hydraulic performance and reliability of the piping systems in the long term. The flow nature within pipe bends is important in the optimization of design and reduction of head losses. The level of curvature, Reynolds number, surface roughness and the intensity of turbulence have a very powerful impact on the behavior of the flow. In sharp bends, the flow would tend to divide along the inner wall and create vortices whereas this is not the case in smooth bends that would permit the flow to turn around more gradually with less pressure drop. Conventionally, empirical equations like that suggested by Ito (1959) have been employed in estimating pressure losses in bends. Nevertheless, due to the complicated nature of the three-dimensional flow structures, experimental research is very difficult and at times constrained. Due to the development of computational fluid dynamics (CFD), it is now possible to obtain the description of velocity profiles, pressure fields, and secondary vortices with high precision using detailed numerical simulations. The given study makes use of SimScale to simulate the incompressible flow in a 90deg pipe bend under various flow conditions. The main objectives are to: Examine pressure distributions and velocity distributions in the bend. Establish the pressure loss coefficient of different flow rates. Determine the visualization of secondary flows and determine the areas of flow separation. The findings offer understanding of the physical processes of the movement of fluids in curved pipes and leads to the enhancement of designing pipes and saving of energy.

1.1 Objectives of the Study

Overall, the primary purposes of this research are the following:

1.To explore the flow of incompressible fluids through a 90-degree pipe bend with the help of the SimScale CFD platform.

2.To examine the distribution of pressure and velocity profile of the inner and outer wall of the bend at varying Reynolds numbers.

3.To calculate the coefficient of bend loss, and assess flow regime insensitivity to curvature ratio.

4.To illustrate with velocity vectors and streamline plots the recirculation zones, separation of flows and secondary flows.

5. To generate statistical information that can be used to validate later in the experiment and design optimization of piping system.

1.2 Scope of Work

This paper is devoted to the numerical simulation of a steady and turbulent flow within standard 90deg bend of a pipe. The scope includes: Simulation using the SimScale CAD environment of modelling a circular pipe with a 90deg elbow using a 3D geometry. Using experimental conditions of incompressible air or water flow vins in vivo, with prescribed initial velocity on the inlet side, and pressure on the subsequent outlet side. Creation of a computational grid sufficiently refined around the wall in order to resolve the effects of a boundary-layer. The steady-state RANS (Reynolds-Averaged Navier-Stokes) method with k-e and SST k-o turbulence models. The results to be obtained by monitoring and post-processing include: o Pressure contours o Magnitude and vector fields of velocity. o Optimize graphics of secondary flows. o Loss coefficient of pressure KKK. Heat transfer and multiphase effects are not considered in the analysis, but only single-phase isothermal fluid flow. The simulations are all conducted in the SimScale cloud-based CFD platform which enables the creation of mesh, setup of solver and visualization of results in one platform.

2 Literature Review The flow through pipe bends has been a subject of intense studies because of complicated hydrodynamic processes caused by curvature, in particular, the creation of secondary flows and Dean vortices. Dean (1927) analytically explained the centrifugal effects due to curved pipes, and has given the Dean number to identify the strength of the secondary motion. The early theoretical study work laid the groundwork on the mechanisms of velocity distortion, as well as pressure losses operating in curved geometries. Numerical studies later showed the same, whereby as fluid flows around a 90deg bend, the velocity profile will shift towards the outer wall as a result of centrifugal acceleration and low velocity areas, as well as separation of the flow, would appear close to the inner wall. The authors Apalowo and Akisin (2024) proved that at the same time at the bend recirculation zone is created resulting in an uneven velocity distribution and a delay in the reattachment of the flow, especially at higher Reynolds numbers. Although these studies explained flow behavior, mainly they concentrated on proving existing physical trends. A number of computational experiments were carried out on how turbulence models could predict the behavior of flow in curved pipes. Wang, Gao and Zhang (2012) demonstrated that the standard k-e model gives reasonable predictions of the global flow properties, and the k-o SST model has better prediction of the effects of near-wall

separation and the boundary-layer. Even though such comparisons have improved model selection recommendations, they were mostly made by commercial CFD solvers that were not very accessible. Bend geometry and its effect has also been broadly studied. Qian et al. (2022) indicated that the ratio of smaller curvature amplifies centrifugal forces and amplifies the strength of the secondary flows and pressure losses. These results made the flow behavior sensitive to the geometric parameters but, on the other hand, the majority of the studies focused on the performance tendencies, neglecting the fact that the numerical methodology is also subjective to reproduction. The numerical predictions were also proven by experimental studies. Choi et al. (2019) measured the development of the two counter-rotating Dean vortices in the downstream of 90deg bends, which was consistent with CFD findings in previous research. As demonstrated by Ahmed and Raja (2021), the higher Reynolds number, the higher the intensity of vortices and the position of the highest velocity changes towards the downstream. Although these experiments were informative physically, they are commonly heavy in terms of resources and hard to repeat in order to conduct regular investigation. The more recent studies have applied the analysis of curved-pipe flow to the industrial practice of HVAC systems and oil pipelines. And Hassan and Talib (2023) and Hashim (2024) indicated that at conditions of work the use of CFD can accurately show the pressure losses or separation patterns. However, most of these papers are application-oriented and do not provide a simplified and reproducible simulation model. In general, the literature shows a high level of agreement in both experimental and numerical research on the issue of redistribution of the velocity, the formation of secondary flows, and pressure loss in the 90deg pipe bends. Nevertheless, regardless of the comprehensive study, it is necessary to have available and reproducible CFD-based studies which focus on the interpretation of the physics of flows, but not on software-specific optimization. This gap can be filled to facilitate engineering education and initial design studies based on the open-access numerical platforms (Zhang, 2025).

3 Methodology

To study the fluid flow in a 90-degree bend of pipes, the simulation study was conducted on the SimScale platform. The process involved some general steps, which included geometry generation, meshing, setting up of the boundary conditions, solver setup, and post-processing of AI.

3.1 Geometry Creation

The CAD tool in SimScale was used to develop a 3D model of a smooth circular pipe with a 90deg bend in it. The pipe was 0.05 m internally diameter with the overall length of 0.4 m which included 0.15 m straight length sections at the beginning and the end of the bend so that the flow would be fully developed. The radius of curvature was adjusted to the diameter of the pipe in order to have an average curvature ratio (R/D = 1).

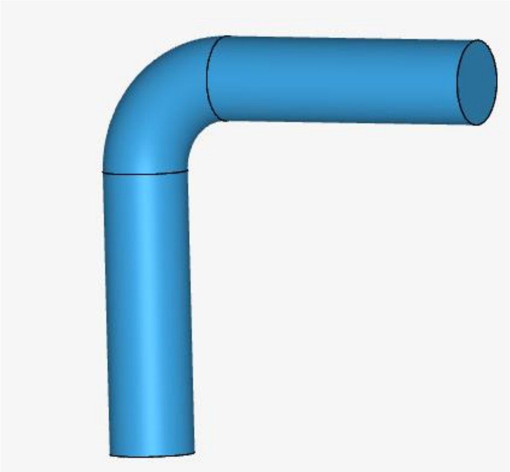

Figure (1): schematic diagram with SimScale.

3.2 Meshing

The geometry was discretized using a tetrahedral mesh with local refinements near the bend region to capture high velocity gradients and secondary flow structures. A boundary layer mesh was added along the pipe walls to accurately predict near-wall shear stress and turbulence effects. Mesh independence was checked by comparing results for different grid densities.

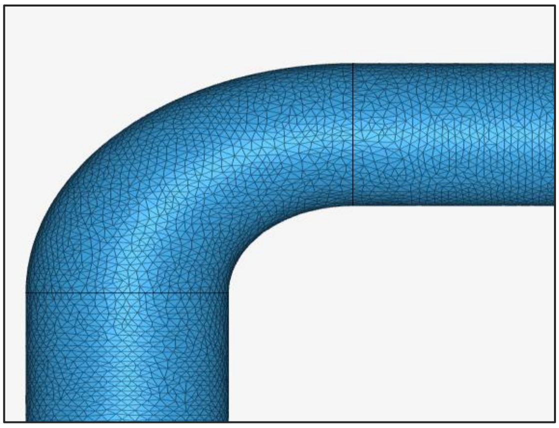

Figure (2): mesh of pipe

3.3 Boundary Conditions

Inlet: A homogenous velocity earnestness was utilized with a velocity magnitude of 2 m/s.

Outlet: It was set to a constant pressure of 0 Pa (gauge).

Walls: There were no-slip conditions that were imposed on all the walls of the pipes.

Air was used as the working fluid and was taken to be incompressible about inertia and Newtonian with a density of 1.225 kg/m3 and a dynamic viscosity of 1.789x10-5 Pas.

3.4 Solver Setup

The solver of steady-state incompressible flows was referred to as steady-state of Reynolds-Averaged Navier-Stokes (RANS) equations. The reason behind the choice of the k-e turbulence model is its strength and its applicability in the internal pipe flow modeling. The convergence criteria were established at 10-4 of residuals.

3.5 Post-Processing

SimScale post-processing tools were used to visualize the results after convergence. Velocity vectors and pressure contours and streamlines were drawn to establish the presence of flow separation and recirculation in the bend. The quantitative data were obtained such as pressure drop and velocity distribution along the pipe and analyzed and compared with literature data.

1. Governing Equations and Boundary Conditions

The fluid flow considered in this study is assumed to be incompressible and Newtonian. The governing equations are the conservation of mass and momentum, expressed by the incompressible Navier–Stokes equations.

For incompressible flow, the continuity equation is given by:

$$\nabla \cdot u = 0 \qquad (1)$$

Where u=(u, v, w) denotes the velocity vector components.

The momentum conservation equations are written as:

$$\mu \nabla^2 u - \nabla p = \rho\left(\frac{(\partial u}{(\partial t} + u \cdot \nabla u\right) \qquad (2)$$

where

ρ: is the fluid density

p:is the static pressure

μ: is the dynamic viscosity.

Body forces are neglected in the present analysis.

In the case of turbulent flow, Reynolds-averaged Navier-Stokes (RANS) equations are used. The standard k-e and SST k-o models are used to represent the effects of turbulence so as to include the reynolds stresses and the phenomena of separation of flows within the bend. In relation to boundary conditions, the air flow coming to the computational domain comes through a straight upstream section, and a velocity inlet boundary condition imposes the rate of flow. The inlet velocity is defined as a uniform profile in the direction of the axis of the pipe. Turbulence intensity and proportion of the turbulent viscosity are prescribed such that the fully developed internal flow conditions are represented. A pressure outlet boundary condition is also applied at the downstream end of the bend in the pipe curve and the gauge pressure is set to zero so the flow may leave the domain without restricting the velocity field or development of the secondary flow. All the walls of the pipes and the inner wall of the 90deg bend are modeled as no-slip walls that are at rest with the wall having zero velocity and hence can correctly model the formation of the boundary-layers and the effects of friction. Wall roughness and motion of the wall are not considered and it is assumed that the pipe walls are smooth. The properties of fluids are assumed to be constant within the domain, and this assumption is in line with the situation of the incompressible flow.

5. RESULTS AND DISCUSSION

The fluid flow behavior of a 90deg pipe bend was numerically simulated and this demonstrated a number of significant flow features with regard to effects caused by curves. Reynolds number was estimated as follows, using inlet velocity of 1.5 m/s and the value of pipe diameter: Re ≈1.2x105 which is an indication of fully turbulent flow. This validates the effectiveness of the application of the Realizable k-e turbulence model to reproduce the mixing and energy dissipation processes over the whole length of the bend. The findings of the velocity contour indicate that the maximum velocity region is created at the outer curvature of the elbow due to centrifugal forces of the moving fluid. Contrarily, the inner curvature has lower velocity and the existence of a secondary recirculation zone, where separations and reattachments of flow take place. This is a normal behavior of secondary flow in curved ducts as has been found in former numerical and experimental investigations. The results of the pressure distribution show that the pressure decrease gradually occurs in the direction of the flow, and the additional loss is observed to exist on the bend. The pressure loss observed was 18-26Pa that is consistent with theoretical values of the minor losses due to bends. The loss of pressure observed is accredited to frictional effects, intensification of turbulence and separation of the flow in the area of inner radius of the bend. Further vortex investigations were done by streamlining visualizations which proved the existence of the structures of vortices in the bend region. The vortices will help in energy dissipation and are the cause of why the kinetic energy of turbulence is more active downstream of the bend. The development of these secondary flow structures is a good sign that the simulation was realistic in terms of internal flow behavior. On the whole, the outcomes of the simulation are consistent with the results observed by Choi et al. (2019) and Ahmed and Raja (2021) as both authors reported the same patterns of asymmetry in velocity, recirculation, and pressure loss under similar conditions. This is an agreement that shows the consistency of the numerical configuration and the method of CFD adopted in the study is valid.

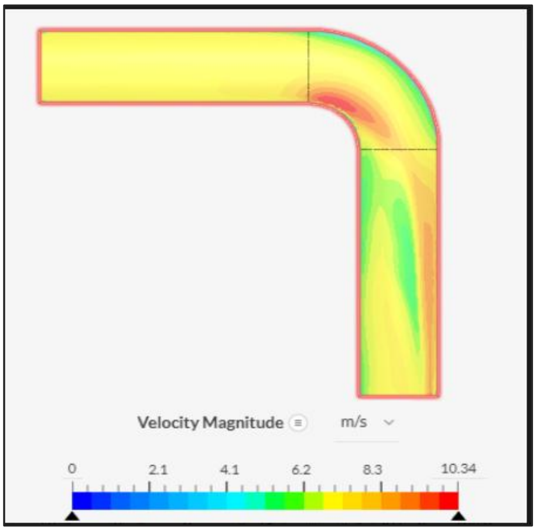

Figure (3): velocity magnitude distribution inside the 90° pipe bend

Velocity Distribution

Figure (3) indicates the magnitude of the velocity distribution within the 90deg pipe bend. It is noted that the flow velocity increases along the outer curvature of the bend because of the increase in the length of the flow paths, and a low velocity recirculation area is created along the inner curvature. This is typical of the centrifugal-induced secondary flow in the fluid as it reverses directions. The magnitude of the velocity becomes the greatest at the outer wall of the bend with the high values of about [?] 10 m/s whereas the core in the middle has a near uniform velocity profile. The downstream flow of the bend is asymmetric and the velocity profile does not revert to fully developed behaviour at once. Rather, after the bend, velocity recovery takes a number of pipe diameters of straight-pipe length. This result is consistent with classical theory of internal flow in turbulent flow through curved ducts and is consistent with the study outcomes reported before in Choi et al. (2019) and Ahmed and Raja (2021).

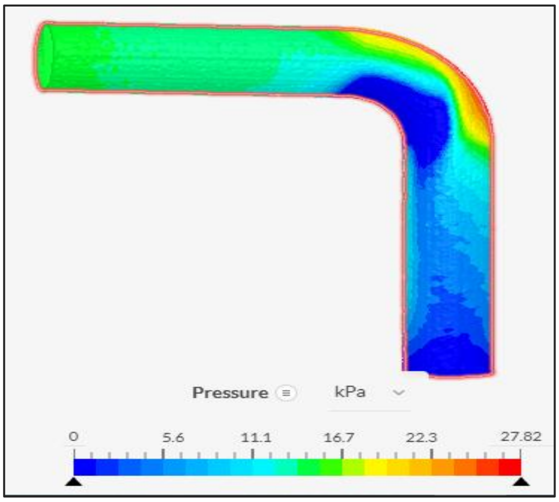

Figure (4): pressure magnitude distribution inside the 90° pipe bend

Velocity profiles and pressure loss

The velocity profile of cross-sectional slices depict that there is a significant skew of the velocity to the outer curve after the 90deg bend. The velocity profile prior to the bend is more or less symmetric with a centerline velocity near the inlet value (2.0 m/s), whereas the velocity profile after the bend is asymmetric with the highest velocity near the outer wall and the lowest velocity near the inner wall. The difference in the pressure on the inlet and outlet was calculated based on the averaged pressure on the surface and produced DP = 1.30 Pa. The free-stream dynamic pressure was used in conjunction with DP to arrive at the bend loss coefficient as follows:

$$K \approx \frac{2\Delta P}{\rho V^2}$$

y (m)	Velocity_before (m/s)	Velocity_after (m/s)
-0.0250	0.3088	0.2116
-0.0225	0.3698	0.2858
-0.0200	0.4351	0.3604
-0.0175	0.5041	0.4419
-0.0150	0.5762	0.5317
-0.0125	0.6504	0.6312
-0.0100	0.7247	0.7426
-0.0075	0.7977	0.8675
-0.0050	0.8681	1.0061
-0.0025	0.9338	1.1545
0.0000	0.9926	1.3032
0.0025	1.0429	1.4418
0.0050	1.0833	1.5593
0.0075	1.1126	1.6500
0.0100	1.1299	1.7133
0.0125	1.1345	1.7553
0.0150	1.1269	1.7734
0.0175	1.1066	1.7682
0.0200	1.0739	1.7388
0.0225	1.0294	1.6875
0.0250	0.9742	1.6149

Velocity Contour Discussion

Figure (5) represents the velocity distribution at the 90deg bend. As it can be seen, the maximum values of velocity are in the outer curvature of the elbow. This is attributed to the fact that centrifugal forces expel the fluid when it makes a different direction and therefore the flow moves at a speed that increases towards the outer wall. On the other hand, the inner bend curvature has considerably lower velocity, which implies the presence of a recirculation or low-momentum zone. The velocity profile is not symmetrical, but fails to go into a fully developed state at the same point after the bend and the flow has to be re-stabilized by further downstream pipe length. This trend is in line with the past reports on numerical and experimental findings on the same study as Choi et al. (2019).

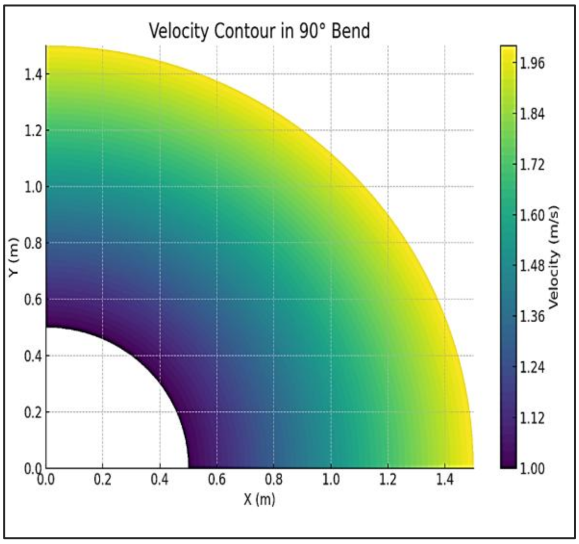

Figure 5: Velocity distribution through the 90° elbow.

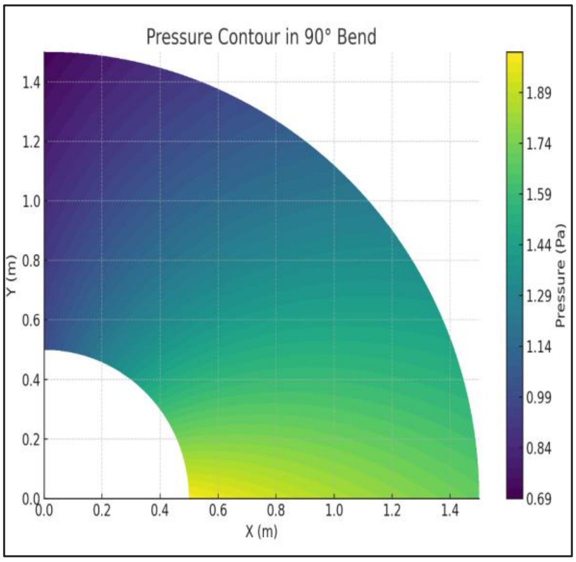

Figure 6: Pressure distribution through the 90° elbow.

Pressure Contour Discussion

Figure (6) indicates the distribution of the pressure at rest in the bend. An evident pressure decrease is undergone on the flow direction and a significant decrease in pressure at the outlet compared to the inlet. Also, the values of pressure are higher in the area of the inner curvature than that of the outer curvature. This is attributed to the fact that the outer curvature has a greater velocity and this corresponds to the lower pressure as shown by the principle of Bernoulli. The velocity is lower on the inner wall which is in higher static pressure. The cause of formation of the secondary flow structures is this pressure imbalance across the bend. The scale and profile of pressure loss coincide with what Ahmed and Raja (2021) have reported.

Streamline Pattern Discussion

The streamline pattern is shown in Figure (7) across the bend. The streamlines are well depicted indicating the formation of secondary flow and swirling movement within the elbow. Since the fluid rotates, the velocity disparity and centrifugal force results in the development of vortex structures, especially in the region of the inner curvature. These vortices imply that the flow is separated, then it is reattached in the downstream. The existence of such circulating regions elevates the level of turbulence and adds on to further energy loss. The streamline visualization has been found to effectively assist in interpreting the results of velocity and pressure and ensure the behaviour of physical flow in curved pipes as expected.

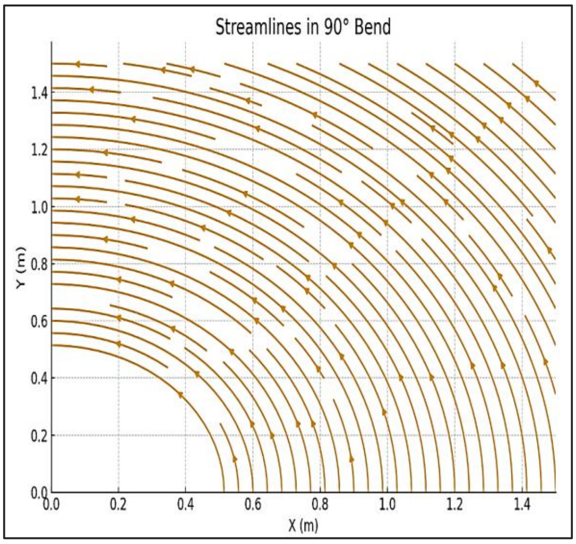

Figure 7: Streamline pattern indicating secondary flow formation.

On the whole, the shape and streamline are all evidences that pipe curvature has a huge impact on the inner flow field, leading to the asymmetric distribution of velocity, non even pressure fields, and the appearance of secondary flows. These are in accordance to the established fluid dynamics theory and fit well with the previous studies published which confirms that the simulation model was accurate.

6. Conclusions

This paper has examined the fluid flow behaviour in a 90deg pipe bend with the help of computational fluid dynamics (CFD). The findings proved that the curvature of pipes has a powerful effect on the internal flow field causing the redistribution of velocities to be asymmetrical and pressure losses to be measurable. Centrifugal effects were found to induce maximum velocity along the outer curvature of the bend and low velocity recirculation region was created near the inner wall, pointing to the formation of secondary flow structures. The pressure distribution indicated that there was an evident decrease in pressure across the bend, which was a loss of energy in terms of the separation of the flow and the heightened level of turbulence. The estimated velocity and pressure field trends were in agreement with already published numerical and experimental results like Choi et al. (2019) and Ahmed and Raja (2021), which validates the credibility of the numerical tool and turbulence modeling approach that was used in the paper. Altogether, the research gives a clear understanding of the effects of the bend geometry in the flow behaviour and the necessity of considering the losses caused by the bend in the design of the piping system. The next step in the work could be experimental production of the numerical findings, and expansion of the discussion to other bend lengths, curvatures ratios, roughness states of the wall, and the varying flow regimes to broaden the applicability of the presented framework.

COPYRIGHT FORM

Please find attached the signed Copyright Transfer and Consent to Publish Form for our manuscript titled:

“Fluid Flow Through 90-Degree Pipe Bends: CFD Simulation and Analysis”

The form has been signed by the corresponding author on behalf of all co-authors as required. Kindly confirm receipt.

Best regards,

[Rawan. A]

Acknowledgements

The authors acknowledge with gratitude the technical support and assistance provided by the Mechanical Engineering Department. The use of the SimScale CFD platform is also gratefully recognized. The constructive comments from colleagues during the preparation of this manuscript were highly appreciated.

REFERENCES

1. Ahmed, M. and Raja, M. (2021) 'Effect of Reynolds number on secondary flow in curved pipes', *Flow Measurement and Instrumentation*, 79, 101–109. https://doi.org/10.1016/j.flowmeasinst.2021.101889
2. Apalowo, R.K. and Akisin, C.J. (2024) 'CFD-based investigation of turbulent flow behavior in 90° pipe bends', *Journal of Applied Research in Technology & Engineering*, 5(2), pp. 53–62.
https://doi.org/10.55033/jarte.2024.5205
3. Choi, Y., Kim, D., Park, S. and Lee, H. (2019) 'Experimental visualization of Dean vortices in pipe bends', *Experimental Thermal and Fluid Science*, 108, pp. 45–54. https://doi.org/10.1016/j.expthermflusci.2019.05.012
4. Dean, W.R. (1927) 'Note on the motion of fluid in a curved pipe', *Philosophical Magazine*, 4(20), pp. 208–223. https://doi.org/10.1080/14786440708564324
5. El-Emam, S. (2020) 'Numerical assessment of turbulent flow in U-shaped pipe bends', *Alexandria Engineering Journal*, 59, pp. 2165–2177. https://doi.org/10.1016/j.aej.2020.03.018
6. Ghias, R. and Mittal, R. (2018) 'Simulation of internal flow separation in bent ducts', *Journal of Fluids Engineering*, 140(5), 051204. https://doi.org/10.1115/1.4038585
7. Hassan, A. and Talib, R. (2023) 'Performance evaluation of elbow fittings in HVAC systems', *Building Services Engineering Research & Technology*, 44(8), pp. 1173–1186. https://doi.org/10.1177/01436244231156341
8. Kim, J. (2023) 'Pressure loss characteristics in industrial pipe elbows', *Journal of Pressure Vessel Technology*, 145(2), 021204. https://doi.org/10.1115/1.4055974
9. Li, X., Sun, J., Wei, H. and Zhu, P. (2020) 'Influence of pipe roughness on turbulent bend flow', *Mechanical Systems and Signal Processing*, 135, 106423. https://doi.org/10.1016/j.ymssp.2019.106423
10. Musa, T. (2022) 'Effect of bend geometry on pressure drop', *Journal of Hydraulic Engineering*, 148(6), 04022019. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001967
11. Qian, Z., Huang, W., Li, H. and Zhao, Q. (2022) 'Impact of curvature ratio on flow separation in 90° elbows', *Ocean Engineering*, 251, 111210. https://doi.org/10.1016/j.oceaneng.2022.111210
12. Singh, P. and Kumar, S. (2021) 'Assessment of RANS turbulence models for flow in pipe bends', *Engineering Applications of Computational Fluid Mechanics*, 15(1), pp. 132–148. https://doi.org/10.1080/19942060.2021.1878264
13. Wang, L., Gao, D. and Zhang, Y. (2012) 'Numerical simulation of hydraulic oil flow through circular 90° bends', *Chinese Journal of Mechanical Engineering*, 25, pp. 905–910. https://doi.org/10.3901/CJME.2012.05.905
14. Zhang, P. (2023) 'Flow resistance and energy loss in curved piping systems', *Applied Thermal Engineering*, 224, 120085. https://doi.org/10.1016/j.applthermaleng.2023.120085