

Comparative Study of Internal Flow Dynamics Using CFD: T-Junction Pipe Geometry

Muharis Mahbubi^{1,*}

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

ARTICLE INFO	ABSTRACT
<p>Article history: Received 12 April 2025 Received in revised form 1 May 2025 Accepted 15 May 2025 Available online 26 June 2025</p> <p>Keywords: Computational fluid dynamics (CFD); T-Junction pipe; fluid flow</p>	<p>This study investigates the turbulent flow characteristics in a T-junction pipe using computational fluid dynamics (CFD). The objective was to analyze pressure distribution, and velocity distribution under turbulent conditions. The simulation was performed with water at a velocity of 0.297 m/s, corresponding to a Reynolds number of 4,338. A grid independence test was conducted to ensure the accuracy and reliability of the results. The findings revealed significant flow separation at the junction, high turbulence intensity in the side branch, and notable pressure drops across the junction. These insights are crucial for optimizing flow behavior and minimizing energy losses in pipe networks.</p>

1. Introduction

The study of turbulent flow in internal geometries is a fundamental aspect of fluid dynamics, particularly in pipe systems where flow separation, pressure drops, and vortex formation can significantly impact system performance. The T-junction pipe geometry, a common feature in fluid transport systems, presents a unique challenge due to the abrupt changes in flow direction. This study aims to explore the flow behaviour in a T-junction pipe under turbulent conditions, focusing on the effects of the junction on flow rate, pressure distribution, and velocity profiles. The use of computational fluid dynamics (CFD) allows for a detailed and accurate analysis of these complex flow phenomena.

Understanding the flow dynamics in T-junctions is essential for optimizing designs in applications such as HVAC systems, pipelines, and fluid distribution networks. The key areas of interest in this study include identifying regions of flow separation, analyzing the formation of vortices, and assessing the pressure and velocity variations within the pipe system. T-junctions are important fittings in industrial pipe systems, where fluid streams merge and split. The fluid dynamics of T-junctions must be understood for the optimization of flow distribution, minimization of energy losses,

* Corresponding author.
E-mail address: gd230053@student.uthm.edu.my

and prevention of issues such as thermal fatigue. Computational fluid dynamics (CFD) has become an important tool for simulating and predicting the complex flow phenomena in T-junctions.

Nimadge and Chopade [1] submitted a CFD analysis of a T-junction in steady, incompressible fluid flow. The research recognized the significance of both the major and minor energy losses in pipe systems, specifically the role played by T-junctions in these energy losses. The research, conducted with ANSYS Fluent, provided insight into energy dissipation mechanisms. Similarly, Ferede [2] examined the flow of fluid through a 90° branch T-junction connection by employing ANSYS CFX for CFD and numerical simulations. The study focused on pressure loss, head loss, and drag coefficient, demonstrating intricate interactions in the junction and the importance of proper modelling.

Luaibi and Muhammed [3] reported analysis of turbulent incompressible flow in T-junction pipes for loss coefficient prediction and velocity profiles for asymmetrical dividing and combining flows. Using ANSYS Fluent, they simulated different Reynolds numbers (3,000 to 30,000) and found that loss coefficients are Reynolds number independent. Velocity profiles and recirculation regions in the junction were also defined in their paper. Wong *et al.*, [4] investigated thermal mixing in T-junctions with various momentum and Reynolds number ratios using Improved delayed detached eddy simulation (IDDES-SST). The research, which has applications in nuclear power plant safety, compared thermal stratification and striping effects. Their simulations, conducted with CABARET, Conv3D, and Nek5000, emphasized the importance of mesh sensitivity and time integration in accurately resolving unsteady thermal mixing that is crucial in thermal fatigue analysis in piping systems [5].

T-junctions have typical industrial uses in fluid transportation systems where streams merge or separate. Detailed information about the fluid dynamics at such locations is necessary in order to maximize the flow distribution and minimize the energy losses, which may lead to thermal fatigue. CFD has been proven to be an essential tool for investigating and predicting complicated flow phenomena in such systems. Abdul *et al.*, [6] performed CFD simulations for the investigation of flow characteristics and energy loss in T-junctions. Lin and Ferng [7] proposed a 3D CFD method for simulating thermal mixing and reverse flow characteristics in a T-junction, confirming the validity of their method by comparing different steady-state turbulence models with experiments.

Evrin *et al.*, [8] conducted CFD simulation of thermal mixing in T-junctions, analyzing temperature fluctuations and mixing efficiency, with application to the conditions of nuclear power plants. Athulya and Miji [9] examined T-junction multiphase flow with a specific emphasis on gas-liquid interaction and guidelines for model accuracy improvement. Zhang *et al.*, [10] examined the influence of the side arm orientation on stratified flow, where CFD simulations were used to develop an optimized mechanical model for liquid carry-out threshold prediction, which improved T-junction efficiency in phase separation applications.

Agbodemegbe *et al.*, [11] examined injection pipe orientation and its effect on mixing and heat transfer downstream of T-junctions. By using CFD analysis, they identified orientations that can enhance mixing and reduce thermal stresses, which is critical in nuclear power plant piping systems. Soto-Francés *et al.*, [12] developed a theoretical model of the diffusive shear work exchange in T-junctions, representing head loss coefficients and giving more insight into energy dissipation mechanisms. Cumulatively, CFD simulations have contributed significantly to the understanding of fluid flow phenomena in T-junctions [13-20]. The studies collectively underline the importance of accurate modelling for the estimation of pressure losses, velocity profile, and thermal behaviour, all of which are essential in the design and safety analysis of industrial pipe systems.

2. Methodology

2.1 Geometry and Grid Setup

The geometry selected for this study consists of a T-junction pipe, with the main pipe having a diameter of 1.30 cm and the side branch forming a 90° angle with the main pipe. The total length of the main pipe is more 10 times than the pipe diameter to ensure that the flow is fully developed before reaching the junction. The side branch's length is changed and act as manipulated variables. This geometry was modelled using ANSYS Design Modeler as shown in Figure 1, and the mesh was created using ANSYS Meshing as shown on Figure 2.

The mesh was carefully generated to capture the complex flow structures that develop near the junction. A structured mesh was applied to the main pipe, while the junction was meshed with unstructured elements to better capture the sharp changes in flow direction and velocity. To ensure that the mesh resolution did not influence the accuracy of the results, a grid independence test was conducted. The mesh resolution was progressively refined, starting with 100,000 cells, and increased until the results showed minimal change. The final mesh contained approximately 500,000 cells, with refinement in regions where flow separation and vortices were expected. Table 1 below shows the values of the geometry used in this case study or stimulation.

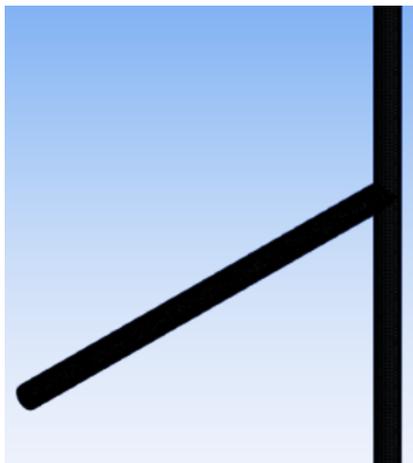


Fig. 1. The geometry used in this study (isometric view)

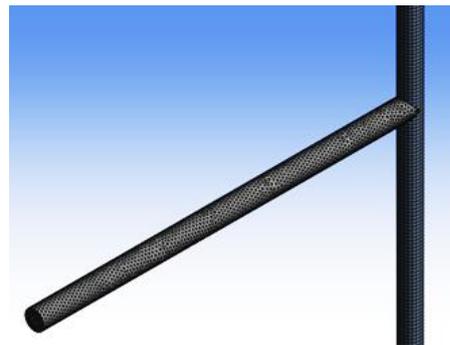


Fig. 2. The mesh setting for this study

Table 1

The values of geometry

Sample	Length 1 (cm)	Length 2 (cm)	Diameter (cm)	Nodes
1	50	100	50	24091
2	75	100	50	26957
3	100	100	50	30021

2.2 Flow Conditions and Boundary Conditions

Water at room temperature (25°C) was chosen as the working fluid. The flow velocity at the inlet was set to 0.297 m/s, resulting in a Reynolds number of 4,338, indicating turbulent flow conditions. The following boundary conditions were applied:

- i. Inlet: A velocity inlet boundary condition was applied with a velocity of 0.297 m/s, simulating a steady-state flow entering the pipe.

- ii. Outlet: A pressure outlet boundary condition was applied at the pipe outlet, with the pressure set to atmospheric conditions.
- iii. Walls: The pipe walls were set to a no-slip boundary condition, assuming smooth walls and zero velocity at the boundary.
- iv. Turbulence model: The standard k- ϵ turbulence model was used, with turbulence intensity set to 5% and the hydraulic diameter used as the turbulence length scale. This model was selected because of its robustness and ability to accurately simulate the turbulence characteristics for a wide range of flows.

2.3 Grid Independent Test

Grid independent test (GIT) is a crucial step in computational fluid dynamics (CFD) simulations to ensure that the numerical solution is mesh size independent. In CFD, the accuracy of the simulation depends on the discretization of the computational space, usually referred to as the grid or mesh. A finer mesh will provide more accurate results but at the cost of increased computational time. The GIT test is performed by successively refining the mesh and verifying significant simulation outputs, such as velocity, pressure, and turbulence characteristics, until the solution converges and is not influenced by further refinement as shown in Figure 3 and Figure 4. Table 2 shows the values of geometry using 100 cm pipe length.

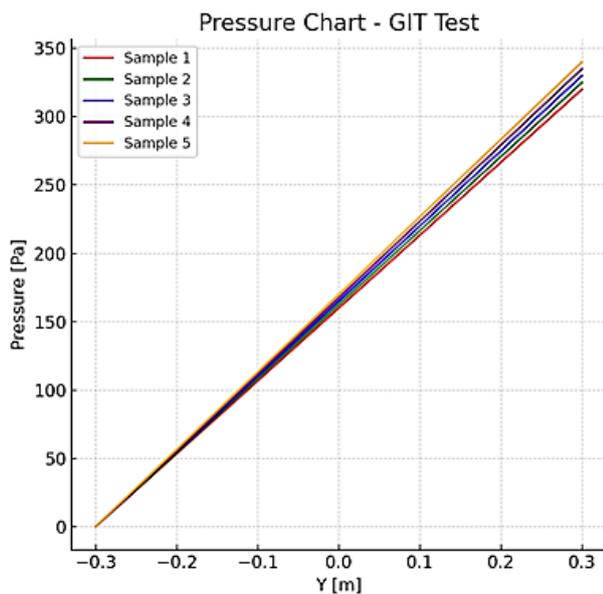


Fig. 3. The pressure chart for GIT test

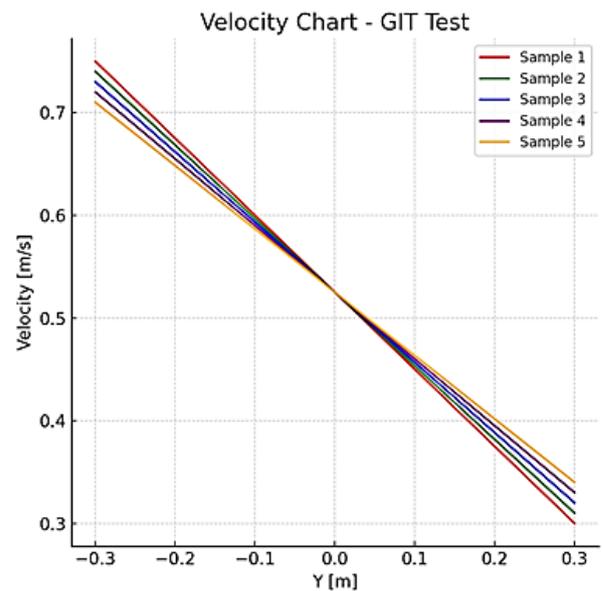


Fig. 4. The velocity chart for GIT test

Table 2

The values of geometry using 100 cm pipe length

Sample	Length 1 (cm)	Length 2 (cm)	Diameter (cm)	Nodes	Element size (cm)
1-1	100	100	50	8651	0.02
1-2	100	100	50	8673	0.04
1-3	100	100	50	8654	0.06
1-4	100	100	50	8667	0.08
1-5	100	100	50	8641	0.10

2.4 Evidence of Convergence and Accuracy

Figures 5 to 7 are a confirmation of convergence within the CFD simulation on ANSYS Fluent. Convergence is a crucial need in numerical modelling to ensure stability in the solution such that iterative iterations will no longer produce significant variations in core flow parameters like velocity, pressure, and turbulence parameters. For Figure 5 and Figure 7 (residual plots - short iterations), the residual plots in the figures show a sharp decline in residual values initially followed by a less steep decline. The continuity, velocity components (x, y, z), and turbulence dissipation rate (epsilon) are all declining with progressing iterations. The ultimate residual values of most of the parameters are within the typical convergence criterion (10^{-3} to 10^{-5}), indicating a converged solution. The console messages at the bottom confirm that the simulation converged in a reasonable number of iterations.

For Figure 6 (residual plot - long iterations), the second plot provides a closer examination of the residual plot for a larger iteration number (2000 iterations). The residual behavior confirms that after around 500-1000 iterations, the main flow parameters have reached a steady state, with extremely small oscillations. This confirms that the solution has achieved numerical stability, and further iterations don't alter the results significantly. This is an ideal sign of a well-converged solution. The terminal outputs of the first and third figures clearly state, "Solution is converged", which verifies that the solver has reached its predefined stopping criteria. This implies that the flow properties, pressure, velocity, and turbulence parameters no longer exhibit noticeable changes, and the results are reliable.

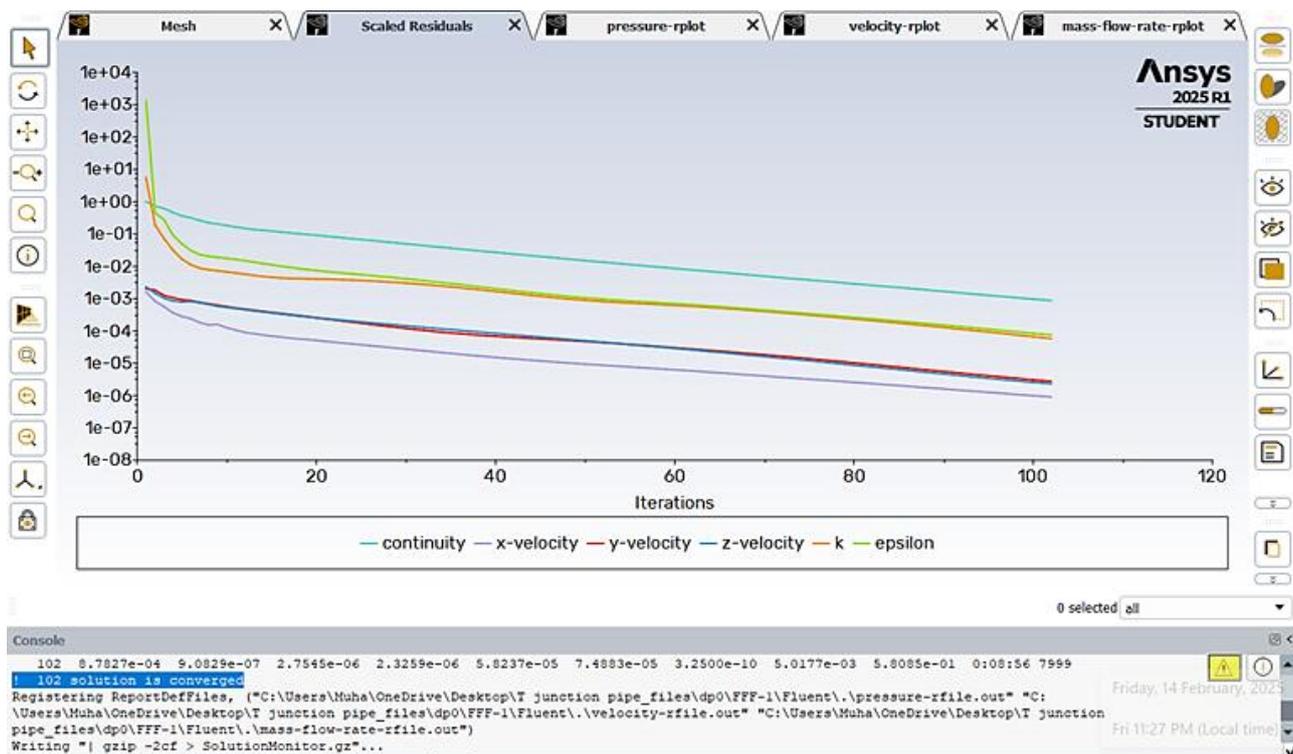


Fig. 5. Evidence of convergence for 50 cm length of pipe

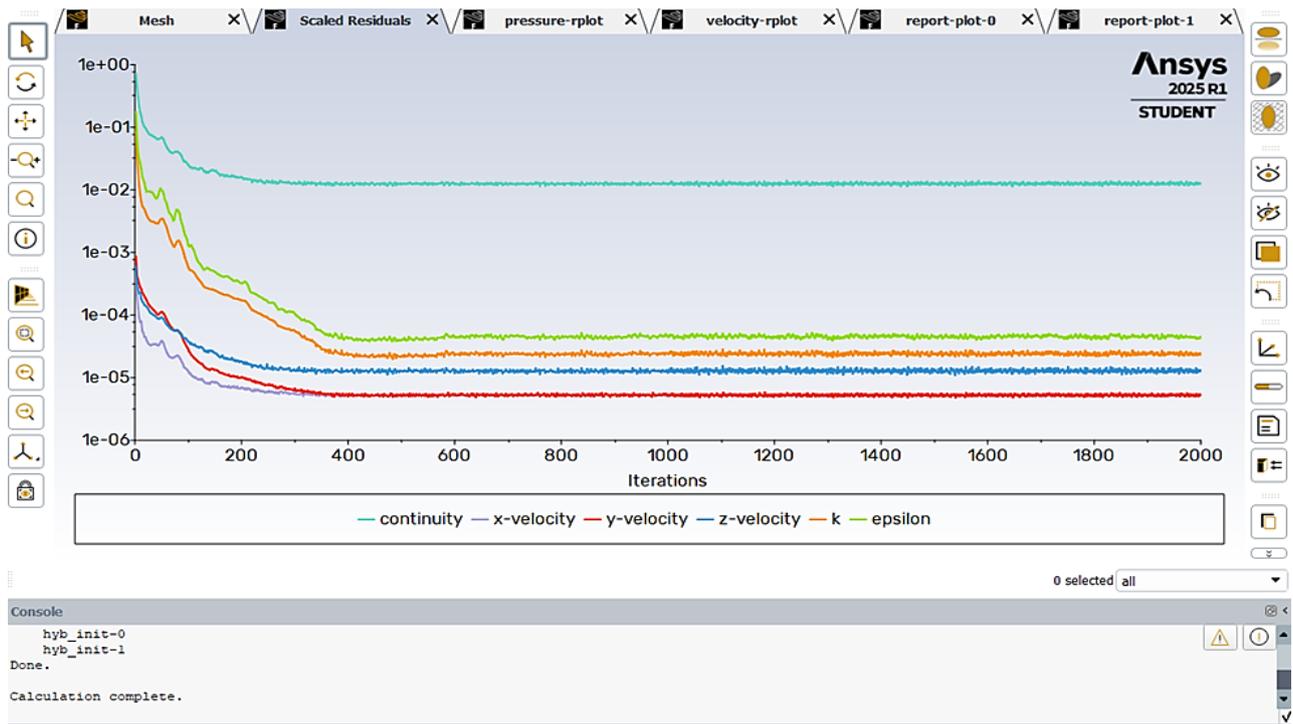


Fig. 6. Evidence of convergence for 75 cm length of pipe

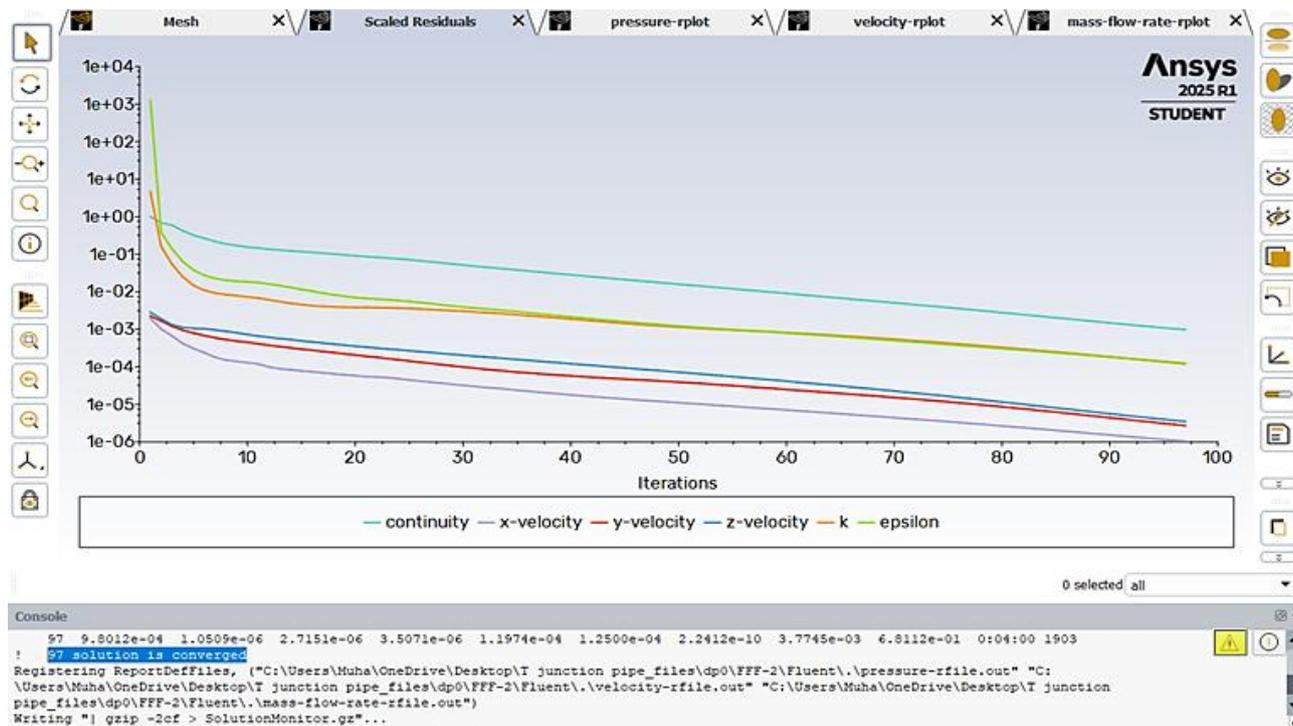


Fig. 7. Evidence of convergence for 100 cm length of pipe

3. Results

3.1 Pressure Distribution

Figure 8 shows the pressure gradient in a T-junction with pressure indicated in Pascals (Pa) through a colour map of red (highest pressure) to blue (lowest pressure). Pressure is highest near the inlet and the location where the junction makes an abrupt change in direction, with the fluid

experiencing an immediate change in direction of flow. This rise in pressure is due to the stagnation effect, where the kinetic energy of the incoming fluid is converted into pressure energy as the fluid is compelled to split between the diverging sections. As fluid passes through the junction and into the outlet branches, the pressure further decreases. This decrease in pressure is anticipated due to energy loss, wall friction, and momentum redistribution. The pressure drops across cases (a), (b), and (c) suggests variations in flow resistance, possibly due to variations in orientation of branches or flow distribution at the junction. Case (c) has the minimum varying pressure drop, which can indicate that it can possess an optimal junction configuration which allows for easier deflection of flow with less energy loss. In comparison, the pressure changes of cases (a) and (b) are a bit larger, which could indicate the effect of increased flow separation, stagnation areas, or turbulence effects.

An effective T-junction should be able to minimize sudden pressure drop to prevent unwanted energy loss. Large pressure drops in industrial piping systems can cause reduced efficiency and increased power demand for pumping. If pressure distribution in these scenarios is analysed, it would be observed that the design should be such that pressure gets distributed more uniformly and stagnation points are minimized, which are the causes of energy wastage.

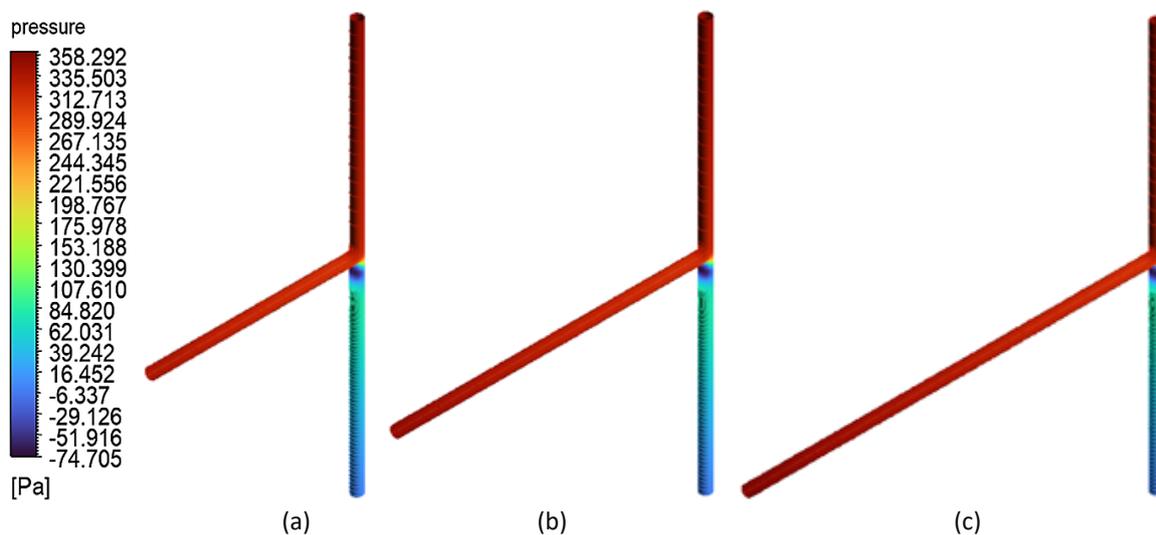


Fig. 8. Pressure distribution on T-Junction pipe with (a) 50 cm length (b) 75 cm length (c) 100 cm length

3.2 Velocity Distribution

Figure 9 indicates the velocity distribution at the T-junction in units of meters per second (m/s) as depicted on a colour map from low velocity (blue/yellow) to high velocity (red/cyan). Maximum velocity areas are observed near the outlet branches since fluid velocities rise as the fluid moves along the downstream direction due to pressure reduction in the first image. In contrast, low-velocity regions exist near pipe walls and the area surrounding the junction, where there is high resistance to flow. Flow separation and probable recirculation zones occur in one of the most characteristic velocity distribution features near the junction. These effects are created when the fluid is exposed to a sudden direction change, resulting in regions of low velocity and even backflow. Cases (a), (b), and (c) being compared, the velocity profiles can be observed to be different, most likely due to differences in the flow division and branch angle of the branch connections.

The velocity distribution for case (c) appears more uniform, so that the junction flow is less prone to extreme turbulence and is more stable. On the other hand, cases (a) and (b) differ considerably, suggesting that they might have greater velocity fluctuations and potential vortices at the junction.

A properly designed T-junction should provide a smooth transition of flow across the branches to minimize velocity imbalances that cause turbulence and pressure fluctuations. High turbulence within the junction may increase energy losses, cause mechanical vibrations, and result in pipe deterioration over time. From the velocity profile of these simulations, it is clear that an optimal junction should have a smooth transition of flow without abrupt transitions causing high velocity gradients.

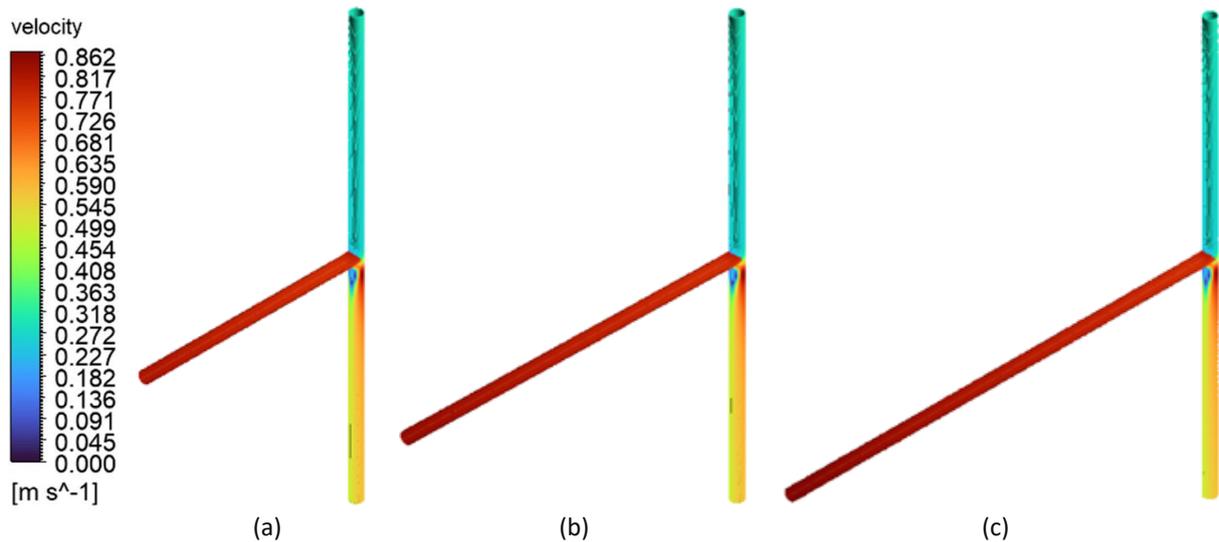


Fig. 9. Velocity distribution on T-Junction pipe with (a) 50 cm length (b) 75 cm length (c) 100 cm length

In the research "CFD Simulations and flow analysis through a T-Junction Pipe" by Nimadge and Chopade [1], researchers studied steady, incompressible fluid flow through a T-junction using Computational Fluid Dynamics (CFD). The research focused on pressure and velocity distributions in the T-junction for determining the associated energy losses. The results showed that pressure and velocity profiles are significantly influenced by the angle of the 90° T-junction, causing pressure reductions and velocity fluctuations which can be seen as fluid moves through the junction as shown in Figure 10 and Figure 11. The result is not far different between the result obtained in this study, where equivalent pressure and velocity differences are realized due to junction geometry.

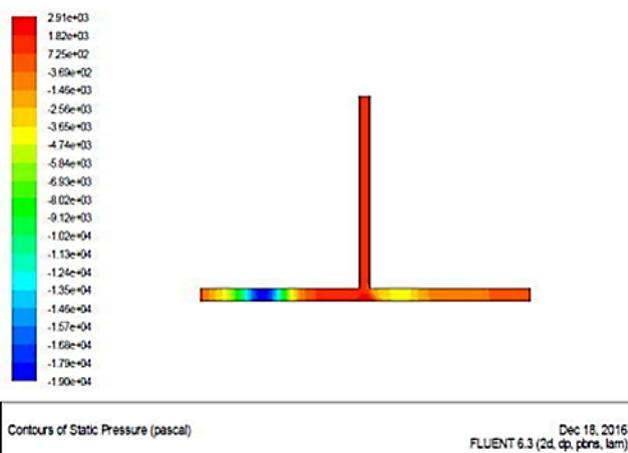


Fig. 10. The pressure contour obtained in the research

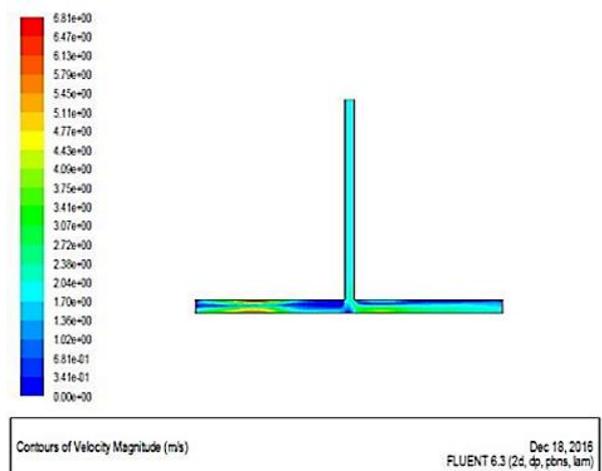


Fig. 11. The velocity contour obtained in the research

4. Conclusions

This CFD analysis of turbulent flow in a T-junction pipe indicates that pressure is maximum at the inlet and reduces along the branches of the outlet, influenced by momentum redistribution and stagnation. Velocity distribution reflects high velocity at the outlet branches and low velocity near the junction, where there is recirculation and flow separation. The k- ϵ turbulence model guaranteed stable convergence, and the grid independence test guaranteed the reliability of the results. Comparison with Nimadge and Chopade [1] indicates cognate trends, observing the way junction geometry affects pressure drop and velocity fluctuations. The findings validate the role of CFD in the design of optimal pipe junctions in a bid to realize minimum energy loss and maximum efficiency in industrial use.

References

- [1] Nimadge, G. B., and S. V. Chopade. "CFD analysis of flow through T-junction of pipe." *International Research Journal of Engineering and Technology (IRJET)* 4, no. 2 (2017): 906-911.
- [2] Ferede, N. A. "Numerical analysis and CFD simulation of fluid flow in T-Junction pipe By using ANSYS CFX." *International Journal of Scientific & Engineering Research* 10, no. 10 (2019): 504-513. <https://doi.org/10.13140/RG.2.2.35259.23845/1>
- [3] Luaibi, Marwa S., and Mohammed A. Abdulwahid. "Numerical analysis by computational fluid dynamic simulation of fluid flow in a T-junction." In *Proceedings of 2nd International Multi-Disciplinary Conference Theme: Integrated Sciences and Technologies, IMDC-IST*, pp. 7-9. 2021. <https://doi.org/10.4108/eai.7-9-2021.2314888>
- [4] Wong, Y. H., L. Lampunio, Y. Duan, M. D. Eaton, and M. J. Bluck. "Effects of different momentum ratios and Reynolds number in a T-junction with an upstream elbow." *Nuclear Engineering and Design* 428 (2024): 113523. <https://doi.org/10.1016/j.nucengdes.2024.113523>
- [5] Obabko, A. V., P. F. Fischer, T. J. Tautges, S. Karabasov, V. M. Goloviznin, M. A. Zaytsev, V. V. Chudanov, V. A. Pervichko, and A. E. Aksenova. *CFD validation in OECD/NEA t-junction benchmark*. No. ANL/NE-11/25. Argonne National Lab.(ANL), Argonne, IL (United States), 2011. <https://doi.org/10.2172/1024601>
- [6] Abdulwahida, Mohammed Abdulwahhab, Niranjana Kumar Injetib, and Ass Prof Sadoun Fahad Dakhilc. "CFD simulations and flow analysis through a T-junction pipe." *Governing* 2 (2012): v3.
- [7] Lin, C. H., and Y. M. Ferng. "Investigating thermal mixing and reverse flow characteristics in a T-junction using CFD methodology." *Applied Thermal Engineering* 102 (2016): 733-741. <https://doi.org/10.1016/j.applthermaleng.2016.03.124>
- [8] Evrim, Cenk, and Eckart Laurien. "Numerical study of thermal mixing mechanisms in T-junctions." *Applied Thermal Engineering* 183 (2021): 116155. <https://doi.org/10.1016/j.applthermaleng.2020.116155>
- [9] Athulya, A. S., and R. Miji Cherian. "CFD modelling of multiphase flow through T junction." *Procedia Technology* 24 (2016): 325-331. <https://doi.org/10.1016/j.protcy.2016.05.043>
- [10] Zhang, Ming, Yuehong Cui, Weizheng An, Haiyan Wang, Lisong Wang, and Shuo Liu. "Investigation of the effect of side arm orientation of the T-junction on gas-liquid stratified flow." *Processes* 11, no. 10 (2023). <https://doi.org/10.3390/pr11102949>
- [11] Agbodemegbe, Vincent Yao, Seth Kofi Debrah, Afia Boatemaa, and Edward Shitsi. "Investigating the effects of injection pipe orientation on mixing and heat transfer for fluid flow downstream a T-junction." *Journal of Power and Energy Engineering* 12, no. 10 (2024): 1-30. <https://doi.org/10.4236/jpee.2024.1210001>
- [12] Soto-Francés, Víctor-Manuel, José-Manuel Pinazo-Ojer, Emilio-José Sarabia-Escrivá, and Pedro-Juan Martínez-Beltrán. "Characteristic differential equation of a T-junction: diffusive shear work exchange from its head loss coefficients." *arXiv preprint arXiv:2112.15394* (2021). <https://doi.org/10.48550/arXiv.2112.15394>
- [13] Okafor, C. V., O. R. Ononye, and A. U. Okeke. "Analysis of water flow in tee-junction pipes using CFD." *Journal of Multidisciplinary Engineering Science and Technology (JMEST)* 7, no. 3 (2020): 11550-11553.
- [14] Garde, Ramchandra J. *Fluid Mechanics Through Problems*. New Age International, 2006.
- [15] Oertel, Herbert, ed. *Prandtl's essentials of fluid mechanics*. New York, NY: Springer New York, 2004. <https://doi.org/10.1007/978-1-4419-1564-1>
- [16] Balakrishnan, Dinesh. *Computational fluid dynamics study on t-junction separator for liquid-gas separation*. Final Year Project Diss., University of Technology Petronas, 2013.

- [17] Monrós-Andreu, G., R. Martínez-Cuenca, S. Torró, and S. Chiva. "Local parameters of air–water two-phase flow at a vertical T-junction." *Nuclear Engineering and Design* 312 (2017): 303-316. <https://doi.org/10.1016/j.nucengdes.2016.08.033>
- [18] Zhang, Jian, Qi-lin Wu, Shuo Liu, and Jing-yu Xu. "Investigation of the gas–liquid two-phase flow and separation behaviors at inclined T-junction pipelines." *ACS Omega* 5, no. 34 (2020): 21443-21450. <https://doi.org/10.1021/acsomega.0c01805>
- [19] Smith, B. L., J. H. Mahaffy, and K. Angele. "A CFD benchmarking exercise based on flow mixing in a T-junction." *Nuclear Engineering and Design* 264 (2013): 80-88. <https://doi.org/10.1016/j.nucengdes.2013.02.030>
- [20] Wrzesień, Sylwia, Paweł Madejski, and Paweł Ziółkowski. "Computational fluid dynamics simulation of gas–liquid multiphase flow in T-junction for CO₂ separation." *Zeszyty Energetyczne* 7 (2020): 403-414.