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

CFD Analysis of Flow Mixing in a Pipe with Two Inlets

Zul Fitry Nazri[1,*]

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

ARTICLE INFO

Article history:
Received 15 October 2025
Received in revised form 12 December 2025
Accepted 15 December 2025
Available online 21 December 2025

Keywords:
Flow; T-junction; steady-state turbulent; computational fluid dynamic

ABSTRACT

The reasons for this importance are that mixing in T-junction pipe systems is essential in many industrial applications, but the influence of the diameter on mixing behaviour has not been well documented. Therefore, this paper discusses the influence of three outlet diameters on velocity distribution, pressure drop and mixing performance using the computational fluid dynamics approach. A two-inlet T-junction was modelled using ANSYS. Steady-state turbulent flow was simulated using the RANS k-ω SST model for three outlet diameters, namely 20 mm, 15 mm, and 10 mm. A Grid Independence Test was carried out to ensure the accuracy of the meshes. Smaller outlet diameters increase the downstream velocity and turbulence intensity, which strengthens the shear layers and enhances mixing. Consequently, these conditions also result in higher pressure losses near the junction. The strongest mixing was obtained with the 10 mm outlet. However, this configuration also produced the greatest pressure drop. Therefore, the outlet diameter is a critical parameter that governs both the mixing effectiveness and the overall hydraulic performance of T-junction flows.

1. Introduction

Flow mixing inside piping systems is a critical process in many engineering applications such as chemical reactors, HVAC systems, water distribution networks, and industrial fluid transport. The interaction of multiple fluid streams within confined geometries produces complex flow patterns, including shear layers, recirculation zones, and turbulence structures. Turbulent mixing in a T-junction system is like a jet in crossflow, which has received widespread attention because of its importance in engineering applications involving shear layers, recirculation zones, and turbulence structures [1-3]. Computational Fluid Dynamics (CFD) provides a powerful method to analyze these flow behaviours in detail, enabling engineers to predict mixing performance and optimize pipe designs without extensive experimentation. T-junction pipe systems typically generate strong shear layers, flow separation, recirculation regions, and turbulence due to abrupt changes in flow direction, all of which significantly influence mixing behaviour [4]. Although CFD is cost-effective for industrial use, complex flow structures remain challenging to model accurately and require expert knowledge

* *Corresponding author.*
E-mail address: cd220120@student.uthm.edu.my

https://doi.org/10.37934/afhme.7.1.1319a

for proper setup, validation, and interpretation [5]. Understanding such complex and highly turbulent flow behaviour is crucial for optimizing system performance and preventing failures [6].

Flow mixing inside internal pipe systems is widely applied in industrial operations such as chemical blending, water distribution, and HVAC systems. In a two-inlet pipe configuration, the mixing effectiveness depends strongly on the geometry of the outlet pipe. Variations in outlet diameter influence downstream velocity, turbulence intensity, pressure drop, and the rate at which the two inlet streams merge. The sudden directional change at the junction also leads to additional pressure losses and flow separation, resulting in a non-uniform velocity profile in the downstream pipe [7]. However, the relationship between outlet diameter and mixing behaviour is not straightforward. Previous studies highlight that geometric configuration strongly influences the flow structure in T-junctions, indicating a need for further investigation on the role of outlet diameter [8]. Experimental studies are also costly, time-consuming, and often limited to specific flow conditions. Therefore, there is a strong need to analyze how different outlet diameters affect mixing characteristics using Computational Fluid Dynamics (CFD), which provides detailed insights into the flow structures within T-junctions [9].

The main objective of this study is to use CFD to investigate the mixing behaviour of two fluid streams merging into a single outlet pipe with different outlet diameters. The specific aims are to model and simulate a two-inlet pipe system using CFD tools, to analyze the effect of three outlet diameters (20 mm, 15 mm, and 10 mm) on flow mixing behaviour, to evaluate the velocity distribution, pressure drop, turbulence intensity, and mixing efficiency for each geometry, to conduct a Grid Independence Test (GIT) to ensure numerical accuracy, and finally to compare the results to determine which outlet diameter provides the best mixing performance and flow characteristics.

2. Methodology

2.1 Geometry Development

The geometry for this study consists of a T-junction pipe system designed using ANSYS DesignModeler. The geometric configuration strongly influences the flow structure in T-junctions [8,10,11] and two identical horizontal inlet pipes merge into a single vertical outlet pipe. The key geometric specifications are shown in Table 1. Outlet length of 400 mm is kept constant to ensure fully developed flow and consistent comparison. The geometry of T-junction pipe is shown in Figure 1.

Table 1
The value of geometry

Sample	Outlet diameter (mm)	Outlet length (mm)	Inlet length (mm)
1	20	400	100
2	15	400	100
3	10	400	100

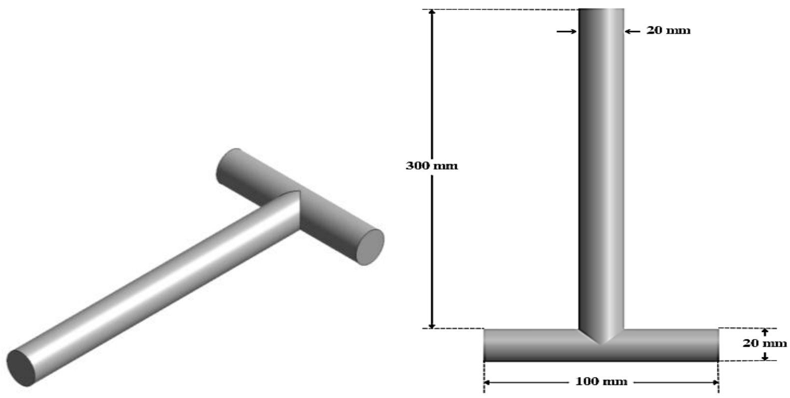


Fig. 1. Geometry of T-pipe

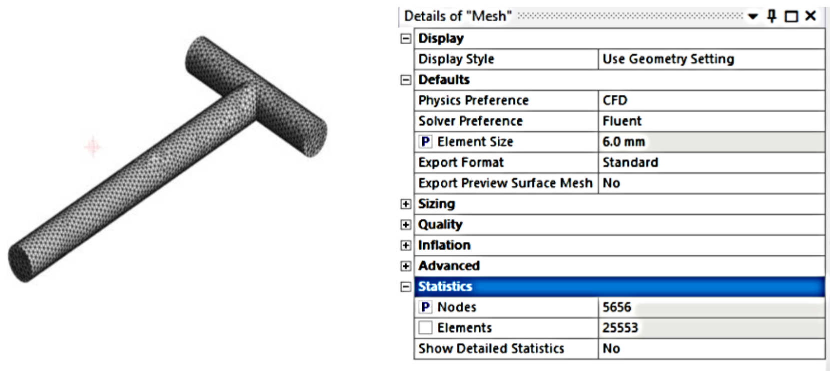
2.2 Discretization Techniques

2.2.1 Meshing generation

The mesh was generated in ANSYS Meshing using a tetrahedral grid. To perform a Grid Independence Test (GIT), only the global element size was varied while keeping all other meshing parameters constant and this will produce different number of nodes based on element size shown in Table 2. Meshing generation and detail of meshing is shown in Figure 2. During meshing, named selections were created for the inlets, outlet, and walls. These allow easy assignment of boundary conditions and help with post-processing and mesh management.

Table 2
The value of nodes for sample 1

Outlet diameter (mm)	Outlet Length (mm)	Inlet Length (mm)	Element size (mm)	Nodes
20	400	100	6	4675
20	400	100	5	7134
20	400	100	4	12448
20	400	100	3	24759



(a) (b)

Fig. 2. (a) Meshing generation (b) Details of meshing

2.2.2 Grid independence test (GIT)

The GIT was performed consistently with recent CFD studies emphasizing mesh effects in T-junctions [2,12] on the 20 mm outlet geometry. To guarantee the accuracy of the calculation results, this study also selected different grid divisions [13] and the goal was to determine the required mesh resolution by observing how the pressure drop (ΔP) varies with mesh refinement.

2.2.3 Governing equation

The fluid flow within the pipe system was modelled as steady, incompressible, and turbulent. The momentum equations are sometimes also referred as Navier-Stokes (NS) equation and commonly applied in recent RANS studies on mixing in pipe junctions [14]. They are most used mathematical equations to describe flow [7]. The governing equations solved in the finite-volume framework are [15,16]:

Continuity equation, $\frac{D\rho}{Dt} + \rho\, \Delta\, .\, V \;=\; 0$ (1)

Momentum equation, $\rho \frac{D\rho}{Dt} = \frac{\partial(\rho u)}{\partial t} + \nabla.\,(\rho u V)$ (2)

2.2.4 Boundary conditions and simulation parameters

The simulation employed a velocity inlet boundary condition with an inlet diameter of 10 mm, where water entered the domain at a uniform velocity of 2 m/s and a turbulence intensity of 5%. At the outlet, a pressure outlet boundary condition was applied with a gauge pressure of 0 Pa, and three outlet configurations were considered, each with diameters of 20 mm, 15 mm, and 10 mm. All walls were treated as no-slip boundaries with smooth surfaces, assuming zero roughness. The working fluid was water, modelled as an incompressible Newtonian fluid with a density of 998 kg/m³ and a dynamic viscosity of 1.002×10^{-3} Pa·s. The working fluid was water, modelled with a density of 998 kg/m³ and dynamic viscosity of 1.002×10^{-3} Pa·s.

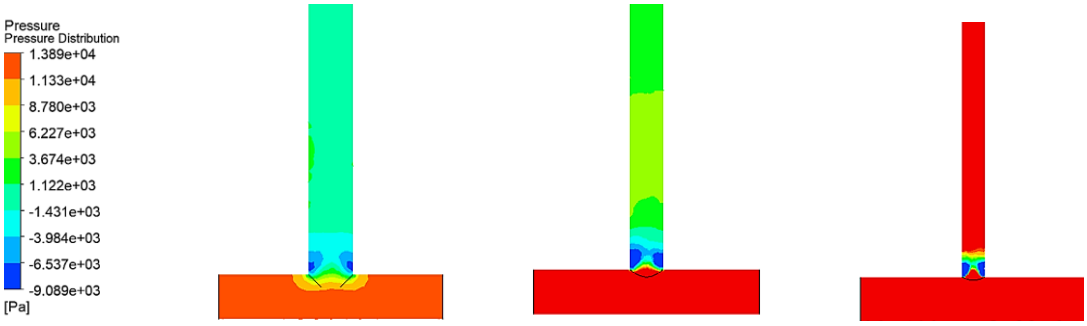
3. Results

3.1 Pressure Distribution

Table 3 shows the results of total pressure. The pressure distribution results show that reducing the outlet diameter increases the overall pressure drop and concentrates the high-pressure region near the T-junction. In the 20 mm outlet, the pressure field is more uniform with the lowest pressure gradient, indicating minimal acceleration as the flow exits the junction. When the outlet is reduced to 15 mm, a stronger pressure buildup forms upstream due to increased flow acceleration through the narrower passage. The smallest diameter of 10 mm produces the highest upstream pressure and the largest pressure drop, as the sharp contraction forces the fluid to accelerate rapidly, generating a steep pressure gradient and stronger flow compression, consistent with previous T-junction studies as shown in Figure 3 [3,17].

Table 3
The result of total pressure at inlet and outlet

Mesh element size (mm)	Mesh nodes	Total pressure inlet (Pa)	Total pressure outlet (Pa)	Pressure drop (ΔP)	Truncation error (%)
6	4675	13784.21	8086.31	0.413	0.72
5	7134	13928.46	8070.96	0.421	1.20
4	12448	13722.89	8053.64	0.413	0.72
3	24759	13739.88	8030.12	0.416	0

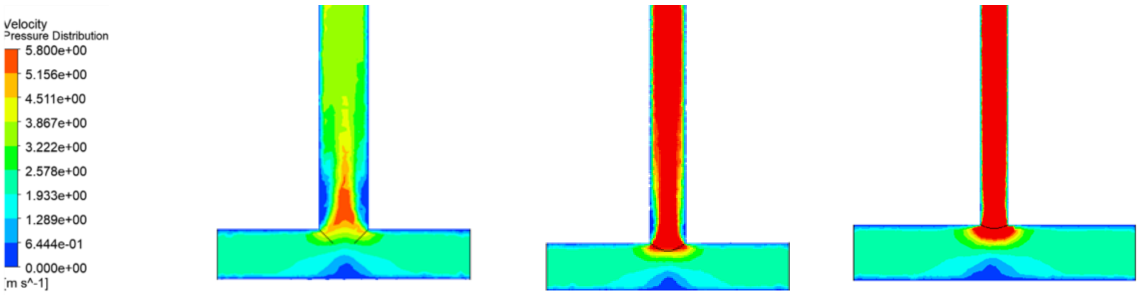


(a) (b) (c)

Fig. 3. Pressure distribution on T-junction pipe with outlet diameter of (a) 20 mm (b) 15 mm (c) 10 mm

3.2 Velocity Distribution

The velocity distribution also varies significantly with outlet diameter, increasing as the outlet becomes smaller. For the 20 mm outlet, the velocity remains moderate with a broad, uniform flow profile downstream. At 15 mm, the flow accelerates more strongly and forms a narrower high-velocity core region, indicating more intense shear and mixing. The 10 mm outlet produces the highest velocity, as the flow is tightly constricted and rapidly accelerated through the reduced area, forming a fast, jet-like stream that enhances turbulence and mixing effectiveness as shown in Figure 4 This behaviour aligns with reported findings that geometric contraction strengthens velocity gradients and mixing intensity in T-junctions [18,20].



(a) (b) (c)

Fig. 4. Velocity Distribution on T-Junction pipe with outlet diameter of (a) 20 mm (b) 15 mm (c) 10 mm

4. Conclusions

This study shows that outlet diameter plays a critical role in governing mixing performance within a two-inlet T-junction. Comparing the three outlet diameters (20 mm, 15 mm, 10 mm) shows that smaller outlets increase mixing but also raise pressure losses. The 20 mm outlet had the lowest pressure drop and smoothest flow but weak mixing. Reducing the diameter to 15 mm increased velocity and shear, improving mixing with moderate pressure loss. The 10 mm outlet caused the highest pressure drop and velocity, generating the strongest turbulence and most vigorous mixing. In summary, smaller outlets enhance mixing through higher velocity gradients and turbulence, while larger outlets minimize energy loss. The 10 mm maximizes mixing, the 20 mm minimizes pressure drop, and 15 mm balances both. These outcomes agree with recent findings showing that geometric contraction strengthens turbulence production and accelerates mixing in junction flows [18], while also increasing hydraulic losses [19]. The results further support observations that localized acceleration near reduced outlets improves fluid entrainment and mixing uniformity [20]. Overall, the 10 mm outlet provides the strongest mixing performance, making it the most effective configuration among the tested designs.

References

[1] Sun, Bin, Quan Liu, Hongyuan Fang, Chao Zhang, Yuanbo Lu, and Shun Zhu. "Numerical and experimental study of turbulent mixing characteristics in a T-junction system." *Applied Sciences* 10, no. 11 (2020): 3899. https://doi.org/10.3390/app10113899

[2] 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

[3] Rimza, Sandeep, Paritosh Chaudhuri, Brijesh Kumar Yadav, and Sayantan Mukherjee. "A case study of thermal mixing behavior of hot and cold fluid in T-junction with/without mixing jets." *Case Studies in Thermal Engineering* 50 (2023): 103417. https://doi.org/10.1016/j.csite.2023.103417

[4] Mahbubi, Muharis. "Comparative study of internal flow dynamics using CFD: T-junction pipe geometry." *Semarak Journal of Thermal Fluid Engineering* 5, no. 1 (2025): 11-20. https://doi.org/10.37934/sjotfe.5.1.1120a

[5] Bhaskaran, Rajesh, and Lance Collins. "Introduction to CFD basics." *Cornell University-Sibley School of Mechanical and Aerospace Engineering* (2002): 1-21. https://doi.org/10.1615/hedhme.a.000102

[6] Ismail, Muhammad Dzul Imran. "Simulation of Internal Flow Behaviour on the Different Diameter of Y-Junction Pipe." *Journal of Advances in Fluid, Heat, and Materials Engineering* 6, no. 1 (2025): 50-62. https://doi.org/10.1063/5.0189282

[7] 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. https://doi.org/10.59743/jbs.v25i.156

[8] 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. https://doi.org/10.31142/ijtsrd23381

[9] Lin, Mei, Bo Su, Shicong Li, and Qiuwang Wang. "Numerical study of flow and mixing characteristics in a T-junction under inflow pulsation." *Energy Procedia* 158 (2019): 5238-5244. https://doi.org/10.1016/j.egypro.2019.01.658

[10] Kok, Jacobus BW, and S. Van der Wal. "Mixing in T-junctions." Applied mathematical modelling 20, no. 3 (1996): 232-243. https://doi.org/10.1016/0307-904X(95)00151-9

[11] Bradshaw, Peter. *An introduction to turbulence and its measurement: thermodynamics and fluid mechanics series*. Elsevier, 2013.

[12] Dianita, Cindy, Ratchanon Piemjaiswang, and Benjapon Chalermsinsuwan. "CFD simulation and statistical experimental design analysis of core annular flow in T-junction and Y-junction for oil-water system." *Chemical Engineering Research and Design* 176 (2021): 279-295. https://doi.org/10.1016/j.cherd.2021.10.011

[13] Gao, Ran, Wei Guo, Changqing Yang, Meng Wang, Sikai Zhang, Hang Zhou, and Angui Li. "Truncation method for calculating the resistance of ventilation air-conditioning duct systems under nonfully developed flow boundary conditions." In *Building Simulation*, 14, no. 4, pp. 1237-1249. Beijing: Tsinghua University Press, 2021. https://doi.org/10.1007/s12273-020-0734-y

[14] Duan, Jinping, and Xiaoxue Huang. "An unsteady RANS study of thermal striping in a T-junction with sodium streams mixing at different temperatures." *Frontiers in Energy Research* 10 (2023): 991763. https://doi.org/10.3389/fenrg.2022.991763

[15] Anderson Jr, John D. "Governing equations of fluid dynamics." In *Computational fluid dynamics: an introduction*, pp. 15-51. Berlin, Heidelberg: Springer Berlin Heidelberg, 1992. https://doi.org/10.1007/978-3-662-11350-9_2

[16] Lloyd, John R. *McGraw-Hill series in mechanical engineering*. Edited by Jack P. Holman. MacGraw-Hill, 1955. https://doi.org/10.1002/aic.690010406

[17] 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

[18] Ejaz, Faheem, William Pao, Mohammad Shakir Nasif, Ahmed Saieed, Zeeshan Q. Memon, and Md Nuruzzaman. "A review: Evolution of branching T-junction geometry in terms of diameter ratio, to improve phase separation." *Engineering Science and Technology, an International Journal* 24, no. 5 (2021): 1211-1223. https://doi.org/10.1016/j.jestch.2021.02.003

[19] Gajbhiye, Bhavesh D., Harshawardhan A. Kulkarni, Shashank S. Tiwari, and Channamallikarjun S. Mathpati. "Teaching turbulent flow through pipe fittings using computational fluid dynamics approach." *Engineering Reports* 2, no. 1 (2020): e12093. https://doi.org/10.1002/eng2.12093

[20] Sakowitz, Alexander, Mihai Mihaescu, and Laszlo Fuchs. "Turbulent flow mechanisms in mixing T-junctions by Large Eddy Simulations." *International Journal of Heat and Fluid Flow* 45 (2014): 135-146. https://doi.org/10.1016/j.ijheatfluidflow.2013.06.014