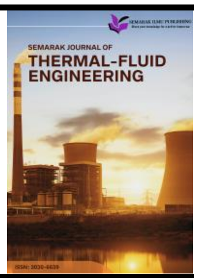
SEMARAK ILMU
PUBLISHING
202103268166(003316878-P)

Semarak Journal of Thermal-Fluid Engineering

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



Hydrodynamic Behaviour in a T-Junction Pipe

Muhammad Hamizan Azli[1,*]

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

ARTICLE INFO

Article history:
Received 15 October 2025
Received in revised form 12 December 2025
Accepted 16 December 2025
Available online 21 December 2025

ABSTRACT

T-junction pipes are commonly used in water supply systems to divide or combine fluid flow. However, the sudden change in flow direction at the junction can cause complex flow behaviour such as high velocity regions, flow separation, recirculation, and pressure loss. Understanding these hydrodynamic characteristics is important to ensure efficient and reliable pipe system design, especially for small pipe sizes commonly used in domestic applications. This study investigates the flow behaviour inside equal T-junction pipes with diameters of 20 mm, 25 mm, and 32 mm using Computational Fluid Dynamics CFD. Three-dimensional models of the T-junction pipes were created and simulated using ANSYS Fluent. Water was selected as the working fluid, and a uniform inlet velocity of 1 m/s was applied to all cases. Turbulent flow conditions were considered using the standard k-epsilon turbulence model. A grid independence test was conducted to select an appropriate mesh size and ensure that the simulation results were not affected by the mesh resolution. Velocity and pressure contours were analysed to observe flow distribution and pressure variation at the junction region. The results show that pipe diameter strongly influences flow behaviour. The 20 mm T-junction produced the highest flow velocity, and the largest pressure drop due to the small diameter, which increases flow resistance. Strong velocity gradients and flow separation were observed at the junction. The 25 mm pipe showed smoother flow patterns with reduced velocity variation and moderate pressure loss. The 32 mm T-junction provided the most uniform flow distribution, and the lowest pressure drop, indicating improved flow stability and lower energy losses. In conclusion, larger pipe diameters result in smoother flow, reduced pressure loss, and better hydrodynamic performance. The findings of this study can help improve the design and selection of T-junction pipes for domestic and light industrial water distribution systems.

Keywords:
T-junction pipe; hydrodynamic behaviour; CFD simulation; ANSYS Fluent; velocity distribution; pressure drop

1. Introduction

Fluid flow through piping systems is fundamental to many engineering applications, such as water supply, HVAC, chemical processing, and heating or cooling networks [1]. Among the common pipe fittings, the T-junction plays a critical role in dividing or merging flow branches, but it also introduces complex hydrodynamic phenomena. Because of the sudden change in flow direction and geometry,

* *Corresponding author.*
E-mail address: dd220028@student.uthm.edu.my

https://doi.org/10.37934/sjotfe.7.1.1724a

the flow in a T-junction is prone to separation, recirculation, and energy losses [2]. The flow inside a T-junction pipe is highly complex due to sudden changes in direction that cause separation, recirculation, turbulence, and pressure losses [3]. Although T-junctions are commonly used in household water systems, especially in pipe sizes of 20 mm, 25 mm, and 32 mm, the hydrodynamic behaviour in these small diameters is not well understood [4]. Most existing studies focus on larger industrial pipes, leaving limited information on how domestic pipe sizes affect velocity distribution, pressure drop, and flow uniformity.

Experimental investigation is difficult because the internal flow structure cannot be easily measured, and repeating tests for different diameters is time-consuming. Therefore, a detailed CFD study is needed to analyse how pipe size influences the internal flow behaviour of T-junctions, so that household piping systems can be designed more efficiently [5]. Understanding flow distribution, pressure variation, and recirculation zones inside T-junctions can help improve the design and performance of piping systems. Numerical simulation provides detailed insight into these behaviours, which is valuable for engineers and designers [6].

Computational Fluid Dynamics (CFD) is a powerful numerical technique used to simulate and analyse the behaviour of fluid flows by solving governing equations, such as the Navier–Stokes equations, through discretization methods like the Finite Volume Method (FVM) [7]. By breaking down complex geometries into smaller computational cells, CFD allows detailed investigation of internal flow characteristics, including velocity distribution, pressure variations, and turbulence effects, which are often difficult or impossible to measure experimentally. Additionally, CFD facilitates mesh independence studies to ensure the accuracy and reliability of numerical results, as well as performance evaluation across different pipe sizes, flow rates, and boundary conditions [8]. This capability is particularly valuable in engineering design and optimization, where it enables engineers to predict system behaviour, improve efficiency, and reduce physical prototyping costs. Overall, CFD has become an essential tool for understanding and optimizing fluid systems in various industrial, domestic, and research applications.

Turbulent flow in T-junctions is complex due to sudden changes in direction and flow separation, which generate vortices, recirculation zones, and pressure losses. To accurately simulate such behaviour, turbulence models like k-ε or k-ω are commonly employed [9]. These models account for eddy viscosity, turbulent kinetic energy, and its dissipation, enabling a detailed prediction of turbulent structures within the junction. By capturing these phenomena, turbulence models provide insight into energy losses, flow mixing, and potential regions of low or reversed flow, which are critical for the design and optimization of piping systems in both industrial and domestic applications.

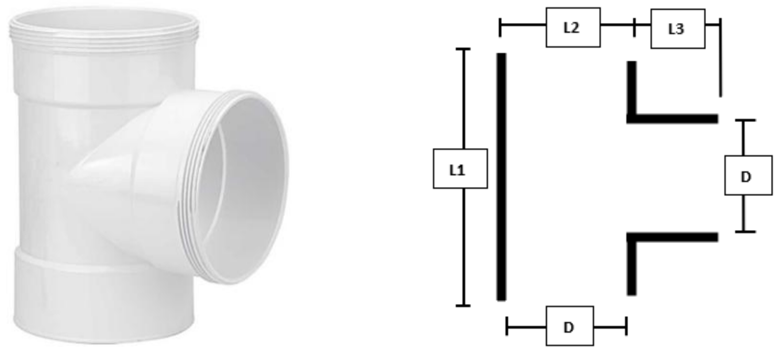
Previous CFD studies reported strong recirculation regions and asymmetric flow patterns inside T-junctions. Numerical investigations also found that pressure drops become less dependent on Reynolds number under certain split ratios, and vortices form inside the branch pipe [10]. This study aims to investigate the hydrodynamic behaviour inside 20 mm, 25 mm, and 32 mm T-junction pipes using ANSYS Fluent. The analysis focuses on velocity distribution and pressure drops. The goal is to understand how pipe diameter affects flow characteristics.

2. Methodology

2.1 Design of T-Junction

The pipes diameters of 20 mm, 25 mm, and 32 mm were selected for this study because they represent three common scales of water distribution systems, ranging from small residential fixtures to larger domestic and light-industrial pipelines. The 20 mm pipe is typically used for sink and tap connections, where higher flow velocity and greater sensitivity to pressure loss make it suitable for analysing small-diameter hydrodynamic effects [11]. The 25 mm pipe is widely applied as a standard household main supply line, providing a medium-scale diameter that reflects typical residential flow behaviour. Meanwhile, the 32 mm pipe is often used in small factories or higher demand building systems, offering a larger diameter where velocity decreases and pressure losses become lower for the same flowrate [12]. By comparing these three diameters within a T-junction, the study can observe how changes in pipe size influence velocity distribution, pressure drop patterns, and flow separation characteristics under different hydraulic conditions.

Figure 1 show the Design of UPVC T-junction under 3 dimensions and 2 dimensions. This image is sourced from BINA Plastic Industry Sdn Bhd and depicts a UPVC which Is Unplasticized Polyv4inyl Chloride T-junction pipe. UPVC pipes are widely used in plumbing and industrial applications due to their high strength, chemical resistance, and durability. The dimension of the equal tee of size 20,25 and 32mm are state in the Table 1.



(a) (b)

Fig. 1. Design (a) 3D design of UPVD T-junction (b) 2D design of UPVC T-junction [13]

Table 1
Detail dimension of equal tee [13]

Size (mm)	D	L1	L2	L3
20	26.7	84.7	33.6	26.3
25	33.5	102.6	40.5	26.8
26.332	42.2	114.6	50.3	27.1

Figure 2 shows the detailed design using Design Modeler in Ansys Fluent. The Length of the pipe is longer than the standard dimension state in Bina Plastic Sdn Bhd. This is because studying minimizes entrance and exit effects on the simulation results.

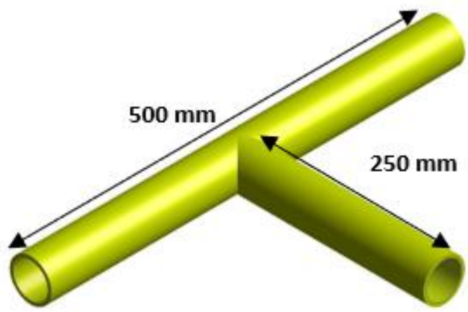


Fig. 2. The geometry of T-junction using Design Modeler

2.2 Discretization

The computational domain of the T-junction pipe was discretized to allow accurate numerical simulation of the flow while maintaining computational efficiency. A structured approach was adopted to generate a high-quality mesh that could capture the complex flow features around the junction. The discretization process ensures that the numerical solution closely represents the actual fluid behaviour inside the pipe [14].

2.2.1 Generate mesh

The mesh generation was performed using the tetrahedron method, which is well-suited for capturing the curved surfaces of the pipe and the geometry of the T-junction. Body sizing techniques were applied to control the element size throughout the computational domain. Figure 3 shows the mesh of fluid domain. Regions where strong gradients were expected such as near the junction, branch, and pipe walls were assigned finer elements to enhance solution accuracy [15].

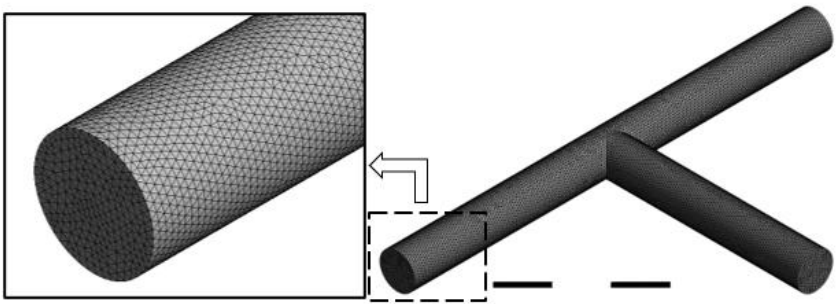


Fig. 3. Mesh of fluid domain of 20mm T-junction pipe

2.3 Governing Equation

In Computational Fluid Dynamics, the governing equations are derived from conservation laws. The three fundamental ones are:

i. Continuity equation (conservation of mass)
ii. Momentum equation (conservation of momentum (Navier-Stokes))
iii. Energy equation (conservation of energy (first law of thermodynamics))

2.3.1 Continuity equation (mass conservation)

Eq. (1) embodies the principle that mass is conserved in a fluid flow: the rate of change of density in a small control volume plus the net mass flux out of that volume must be zero. In practical terms, it means that fluid cannot spontaneously appear or disappear what flows in must either accumulate or flow out. In CFD, the continuity equation ensures that the discretized flow field respects this fundamental physical constraint, preventing artificial sources or sinks of mass that would otherwise distort the solution as state in [16,17].

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) \quad (1)$$

where ρ= fluid density, u= velocity vector t= time

2.3.2 Momentum equation (Navier–Stokes)

Eq. 2 represents conservation of momentum, applying Newton's second law to a fluid element. It states that the change in momentum which is left-hand side arises from forces acting on the fluid: pressure gradients push the fluid, viscous stresses resist motion, and body forces like gravity add or remove momentum. In CFD, this equation is central because it governs how velocity fields evolve in response to forces it captures how pressure and friction such as viscosity shape the flow's behaviour, including acceleration, deceleration, and shear.

$$\rho\left(\frac{\partial\rho}{\partial t} + \nabla \cdot (\rho u)\right) = -\nabla p + \nabla \times \tau + \rho b \tag{2}$$

where p= pressure, τ= viscous stress tensor, b= body force per unit mass

2.4 Boundary Condition

The boundary conditions for this simulation were defined to accurately represent the hydrodynamic behaviour inside the T-junction pipe. Water was selected as the working fluid due to its common application in piping systems, and the inlet velocity was set to 1 m/s [18] to establish a steady and measurable flow entering the domain. Turbulence effects were resolved using the k-ε turbulence model, which is widely used for internal flows because of its stability and reliability in predicting turbulent kinetic energy and dissipation. At the outlet, a zero-gauge pressure condition was applied to allow the fluid to exit freely without imposing additional pressure constraints. All pipe walls were assigned a no-slip condition, ensuring that the fluid velocity at the wall surface is zero, which reflects realistic viscous interactions. The solution was initialized using hybrid initialization to generate a physically reasonable starting field, and the simulation was executed for 1000 iterations to achieve convergence and stable flow predictions throughout the computational domain [19].

Pressure and velocity are important parameters in this study because they describe how the fluid behaves when flowing through the T-junction pipe. Velocity helps to show how the flow changes direction at the junction and how evenly the water is distributed between the main pipe and the branch [20]. Areas of high or low velocity indicate flow acceleration, deceleration, and possible recirculation, which can affect flow efficiency and cause energy losses. Pressure is important because changes in pressure reflect resistance to flow and losses caused by the junction geometry. A high pressure drop across the T-junction indicates larger energy losses and poorer hydraulic performance. By analysing both velocity and pressure together, the relationship between flow distribution and energy loss can be better understood. This comparison allows the effect of pipe diameter on flow stability, pressure reduction, and overall hydrodynamic performance of the T-junction to be clearly evaluated [21].

3. Result

3.1 Grid Independence Test

To verify that the simulation results were not affected by the mesh resolution, a grid independence test was conducted. Several meshes with different numbers of nodes were generated, and the pressure drop was recorded show in the Table 2. The truncation error was calculated using Eq. (3) for each mesh configuration shown in Table 2, and the mesh that provided the lowest error while still maintaining reasonable computational efficiency was selected for the final simulations which is element size 1.7 is selected. This step ensures that the numerical results reflect the true physical behaviour of the flow, rather than artifacts introduced by the discretization process.

$$\left|\frac{A-B}{A}\right| \times 100 \tag{3}$$

where A is The Pressure different for node 1 and B is pressure different for node 2

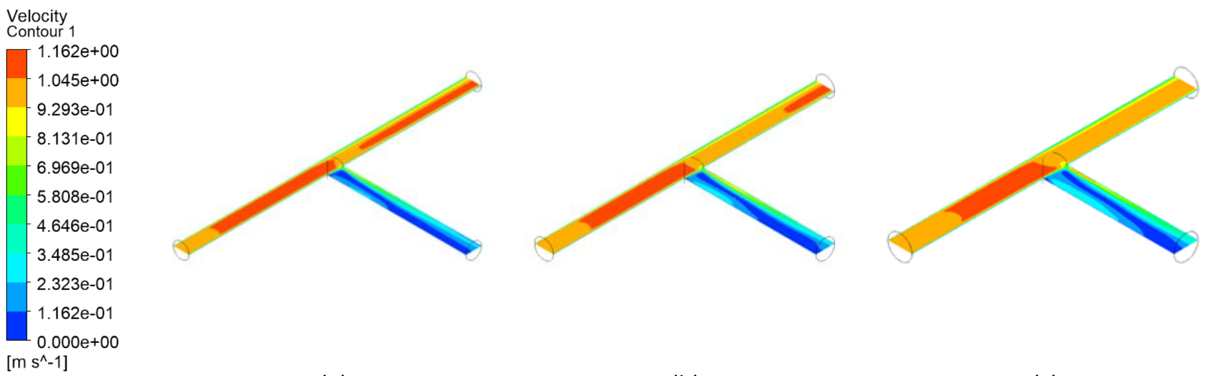
Table 2
Summary of grid independence test

Nodes	Element size (mm)	Pressure inlet (Pa)	Pressure outlet	Pressure different	Error (%)
10580	3.0	104.47210	0	104.47210	18.172
52573	1.7	127.67248	0	127.67248	3.4870
91327	1.4	132.28521	0	132.28521	3.9493
185641	1.1	127.25933	0	127.25933	4.68849
244473	1.0	121.56000	0	121.56000	-

3.2 Velocity Contour of T-Junction Pipe

The velocity and pressure contours obtained from the CFD simulation provide a detailed visual representation of the hydrodynamic behaviour inside the 20 mm, 25 mm, and 32 mm equal T-junction pipes. These contour plots illustrate how the fluid accelerates, decelerates, and redistributes as it flows through the junction, allowing the identification of high-velocity regions, recirculation zones, and areas of significant pressure variation. By comparing the three pipe diameters, the effect of geometric scaling on flow uniformity and energy loss can be evaluated more clearly.

Figure 4 show the velocity contours for the 20 mm, 25 mm and 32 mm T-junction pipes show clear differences in how the flow moves through the system. In Figure 4(a), the 20 mm pipe has the highest velocity because the small pipe size forces the water to move faster when it enters the junction. the flow from the main pipe speeds up and pushes strongly into the branch, creating a fast central jet and a slower area along the branch walls. This causes a bigger difference between fast and slow regions and more mixing near the junction. In Figure 4(b) at the 25 mm T junction the jet into the branch is weaker and the slow region along the walls is smaller, so the flow turns more smoothly. In Figure 4(c) at the 32 mm T junction the flow at the junction is the gentlest, with a wide, slower flow into the branch and less sharp change in speed. Overall, the smaller the pipe the stronger and narrower the jet at the T, and the larger the pipe the smoother and more even the flow through the junction.



(a) (b) (c)

Fig. 4. Velocity contour (a) 20 mm (b) 25 mm (c) 32 mm T-junction pipe

3.3 Pressure *Contour of T-Junction Pipe*

The corresponding pressure contours in 5 also show strong influence from pipe diameter. The pressure contours for the 20 mm, 25 mm and 32 mm T-junction pipes show how the pressure changes when the flow reaches the junction. In Figure 5(a), the 20 mm pipe shows higher pressure near the inlet and at the T junction because the smaller pipe size increases resistance and causes pressure to build up before the flow splits. In Figure 5(b), the 25 mm pipe has slightly lower pressure at the junction compared to the 20 mm pipe, and the pressure drop into the branch is smoother, showing that the larger size reduces flow resistance. In Figure 5(c), the 32 mm pipe has the lowest pressure change at the junction, with a wider low-pressure region forming in the branch because the larger pipe allows the flow to expand more easily. Overall, the pressure at the T section becomes lower and more evenly distributed as the pipe size increases, showing that larger pipes reduce pressure losses and create a smoother pressure transition through the junction.

Overall, the comparison across 20 mm, 25 mm, and 32 mm pipes clearly shows that increasing pipe diameter reduces velocity gradients, moderates pressure drops and improves flow stability. This trend highlights the important role of pipe size in controlling hydrodynamic behaviour within T-junction configurations

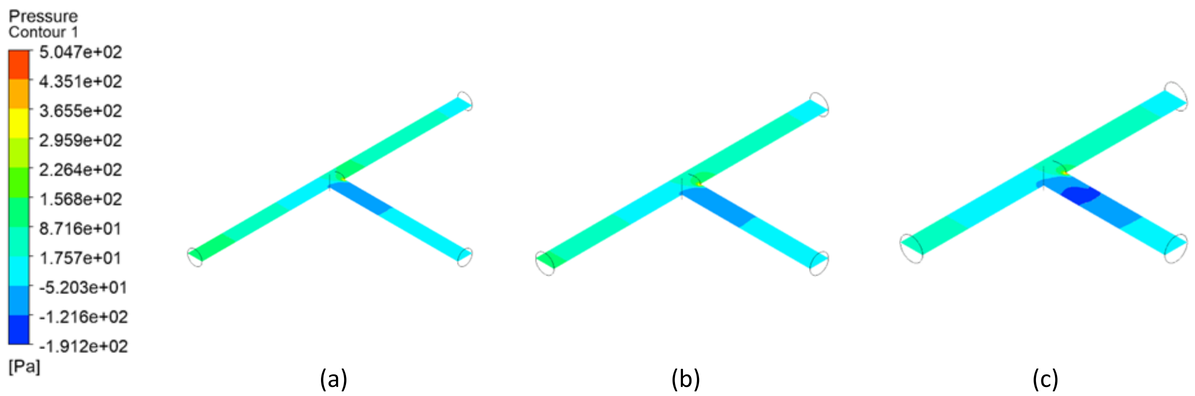


Fig. 5. Pressure *contour* of T-junction pipe (a) 20mm (b) 25mm (c) 32mm

4. Conclusions

The CFD analysis of T-junction flow in 20 mm, 25 mm, and 32 mm pipes demonstrates that increasing the diameter significantly improves hydrodynamic performance. The 20 mm tee experienced steep velocity gradients, strong flow separation, and a large pressure drop, indicating high energy loss. The 25 mm configuration showed moderated velocity distributions and more gradual pressure variation, suggesting reduced turbulence and improved efficiency. The 32 mm tee delivered the smoothest flow, with more uniform velocities and minimal pressure losses, highlighting its superior flow stability. These results corroborate prior CFD studies of junctions which found that larger diameters reduce recirculation regions, lower pressure losses, and promote better flow uniformity. Optimizing junction geometry and diameter is therefore crucial in reducing hydraulic losses in piping systems.

References

[1] Sigalotti, Leonardo Di G. "Pipe Flow: Research and Applications." *Fluids* 10, no. 6 (2025): 149. https://doi.org/10.3390/fluids10060149

[2] 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

[3] Enas S. Taha, Mohammed A. Abdulwahid, and Akeel M. Ali Morad, “Computational fluid dynamic analysis of the flow though T-junction and Venturi meter,” In *Proceedings of the 2nd International Multi-Disciplinary Conference (IMDC-IST)*, 2021, https://doi.org/10.4108/eai.7-9-2021.2314880
[4] Nuruzzaman, Md, William Pao, Hamdan Ya, Md Ragibul Islam, Mohammad Ayub Adar, and Faheem Ejaz. "Simulation analysis of thermal mixing characteristics of fluids flowing through a converging T-junction." *CFD Letters* 13, no. 9 (2021): 28-41. https://doi.org/10.37934/cfdl.13.9.2841
[5] Wang, Wenhui, Xiaolu Lu, Yi Cui, and Kangyao Deng. "Modified pressure loss model for T-junctions of engine exhaust manifold." *Chinese Journal of Mechanical Engineering* 27, no. 6 (2014): 1232-1239. https://doi.org/10.3901/CJME.2014.0904.143
[6] Smyk, Emil, Michał Stopel, and Mikołaj Szyca. "Simulation of flow and pressure loss in the example of the elbow." *Water* 16, no. 13 (2024): 1875. https://doi.org/10.3390/w16131875
[7] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
[8] Ferziger, Joel H., Milovan Perić, and Robert L. Street. *Computational methods for fluid dynamics*. springer, 2019. https://doi.org/10.1007/978-3-319-99693-6
[9] Ma, Haoran, and Hamn-Ching Chen. "Enhancing the two-layer k-epsilon turbulence model through rough wall modification." *Physics of Fluids* 36, no. 10 (2024). https://doi.org/10.1063/5.0232725
[10] 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
[11] Baranova, Tatyana A., Yulia V. Zhukova, Andrei D. Chorny, Artem Skrypnik, and Igor A. Popov. "Non-isothermal vortex flow in the T-junction pipe." *Energies* 14, no. 21 (2021): 7002. https://doi.org/10.3390/en14217002
[12] Zhao, Zhiling, Lu Wang, Wenhang Shi, Cong Li, and Guozijian Wei. "Motion adsorption characteristics of particulate matter in water supply network." *Water* 14, no. 21 (2022): 3550. https://doi.org/10.3390/w14213550
[13] Binaplast Sdn. Bhd. PVC-U (4 Products) Catalogue. Petaling Jaya, Selangor: Binaplast Sdn. Bhd.
[14] Zhao, Zhiling, Lu Wang, Wenhang Shi, Cong Li, and Guozijian Wei. "Motion adsorption characteristics of particulate matter in water supply network." *Water* 14, no. 21 (2022): 3550. https://doi.org/10.3390/w14213550
[15] ElCheikh, Amne, and Michel ElKhoury. "Effect of local grid refinement on performance of scale-resolving models for simulation of complex external flows." *Aerospace* 6, no. 8 (2019): 86. https://doi.org/10.3390/aerospace6080086
[16] Ji, Guozhao, Meng Zhang, Yongming Lu, and Jingliang Dong. "The basic theory of CFD governing equations and the numerical solution methods for reactive flows." (2023): 1-30. https://doi.org/10.5772/intechopen.113253
[17] Anderson, John David, and John Wendt. *Computational Fluid Dynamics*. Vol. 206. New York: McGraw-hill, 1995. https://doi.org/10.1007/978-3-662-11350-9_2
[18] Yoo, Do Guen, Ho Min Lee, Ali Sadollah, and Joong Hoon Kim. "Optimal pipe size design for looped irrigation water supply system using harmony search: Saemangeum project area." *The Scientific World Journal* 2015, no. 1 (2015): 651763. https://doi.org/10.1155/2015/651763
[19] Liu, Yilang, Wenbo Cao, Weiwei Zhang, and Zhenhua Xia. "Analysis on numerical stability and convergence of Reynolds averaged Navier–Stokes simulations from the perspective of coupling modes." *Physics of Fluids* 34, no. 1 (2022). https://doi.org/10.1063/5.0076273
[20] Taha, Enas Salman, Mohammed A. Abdulwahid, Akeel M. Ali Morad, and Qusay A. Maatooq. "Computational fluid dynamic analysis of the flow through T-junction and venturi meter." *IMDC-IST* (2022). https://doi.org/10.4108/eai.7-9-2021.2314880
[21] 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