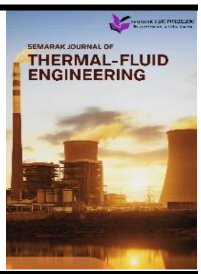
SEMARAK ILMU PUBLISHING

Semarak Journal of Thermal-Fluid Engineering

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



Integrating CFD Tools for the Simulation and Analysis of Turbulent Flow Dynamics in a Y-Junction Pipe

Priya Tharshini Ayasamy[1,*]

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

ARTICLE INFO

Article history:
Received 20 April 2025
Received in revised form 19 May 2025
Accepted 22 May 2025
Available online 26 June 2025

Keywords:
Y-junction pipe; energy loss reduction; velocity profiles; pressure distribution; flow separation

ABSTRACT

The study uses computational fluid dynamics (CFD) to examine turbulent flow behavior in a Y-junction pipe. Y-junctions are frequently used in fluid distribution engineering systems, where performance optimization and energy loss reduction depend on an understanding of flow dynamics. To provide accurate turbulence modelling, the simulation is run in ANSYS Fluent under turbulent circumstances using water as the working fluid and a Reynolds number of 3840. Important elements are examined, including velocity profiles, pressure distribution, flow separation, secondary flow structures, and turbulence intensity. To verify the precision and dependability of the simulation findings, a grid independence test is conducted. The study emphasizes how the shape of the Y-junction affects fluid behavior, including pressure decreases and vortex generation. With possible uses in industrial pipes, HVAC systems, and chemical process equipment, the results offer insights into enhancing Y-junction designs by reducing turbulence and maximizing flow uniformity.

1. Introduction

Y-junction pipes are essential parts of fluid distribution systems and are frequently seen in sectors including chemical processing, oil and gas pipelines, and water treatment. A bifurcation or confluence, where the main flow divides into two branches or where two flows combine to form one, is what defines their shape. The Y-junction pipe is a perfect example for researching turbulent flow behavior because of its distinctive construction, which produces intricate flow dynamics [1,2]. In addition to the abrupt direction changes at the junction, there are major flow disruptions that might result in secondary flows, pressure changes, and possible energy losses. Understanding flow dynamics may have a major impact on performance and energy efficiency in fluid distribution systems, which is why studying turbulent flow behavior in Y-junction pipes is essential [3]. Y-junctions are often used in a variety of technical applications, such as chemical processing equipment, HVAC systems, and industrial pipes. To effectively mimic turbulence and its impact on fluid behavior, sophisticated modeling approaches are required due to the complexity of flow patterns in these

* *Corresponding author.*
E-mail address: priyatharshini257@gmail.com

https://doi.org/10.37934/sjotfe.5.1.4654a

junctions. Mund *et al.,* [4] suggested that analyzing several factors of a system, such as fluid flow, heat transfer, and computational fluid dynamics (CFD), can be used to solve a system of equations with the use of a computer system.

Leschziner [5] point out that, for example, the computational power and methods for numerical and visualization have improved rapidly, but the predictive features of statistical turbulence models are weaker and advance slowly, even though there has been a lot of rigorous work done in the recent past. Therefore, it is very important to employ computational fluid dynamics (CFD) analysis since it gives more accurate results than experimental analysis [6-8]. In many cases, groundwater modelling involves the use of the finite-difference approach to solve the groundwater flow equation. This method is widespread, and many of these models require a relatively fine grid of the computer domain to replicate the specified process in limited areas of interest [9]. The k-ε, k-epsilon, and k-ω, omega turbulence models provide a foundation for accurately estimating energy dissipation and eddy viscosity. They also state that it is necessary to further improve the grid, especially in the locations where the geometry changes suddenly, as it is expected that more comprehensive flow descriptions will occur in those areas.

The specific objective is to examine the velocity and pressure fields of flow in tapered pipes of different lengths. This involves examining how tapering affects the flow behavior in terms of velocity enhancement and pressure decrease, as well as looking at other published material and simple models. To determine the number of nodes that are appropriate for simulations or the density of a mesh that strikes the optimal balance between calculation time and precision, it is essential to run a GIT on one geometry [10]. Once the optimal number of nodes for one of the geometries was determined, the mesh refinement technique that was described was applied to the other two geometries to achieve results that were both accurate and reproducible. According to Debtera [11], using ANSYS Fluent software can be the method that provides the most accurate estimates.

In the current investigations, ANSYS Fluent will be utilized to investigate turbulent flow science, determine the difference between the two geometries, and analyze selected flow phenomena such as flow separation, streamwise velocity, and turbulence intensity. CFD has the potential to be used as an advanced design tool, rather than only a forecast tool [12,13]. The first one has to do with the study of the velocity and pressure fields in pipes that converge or diverge and have random lengths. These include looking into how tapering affects the flow dynamics in terms of velocity increase and pressure drop on the models, as well as comparing the computed values with theoretical and published data. One of the main goals is to conduct a GIT on a single geometry to determine the number of nodes or mesh density that is necessary to achieve satisfactory computing performance of the method with the desired accuracy of outputs. This mesh refinement method was also used on the other two geometries to increase accuracy when the optimal number of nodes for attaining the best results was determined.

The fundamental flow characteristics that are considered are pressure and velocity. Bernoulli's theorem states that the parameters of pressure and velocity change in relation to each other as fluid travels through the reduction section of a pipe system [14]. Tapered pipe fluid dynamics is a challenging area of study, and computational fluid dynamics (CFD) is becoming an essential tool for assessing many problems that involve fluid flow in complicated forms in turbulent regimes. Mixed representations are adjusted using a method known as mesh adaptation. This method includes dividing or coarsening groups of cells based on a certain refinement criterion [15]. A criterion must be connected to the flow problem and the turbulence model that is used in the decision-making process. There has been a lot of interest in studying the fluid dynamics of Y-junction pipes because of how it relates to several industrial uses, especially the transfer of heavy oils. Dianita *et al.,* [16] did a study that used computational fluid dynamics (CFD) to simulate core annular flow (CAF) in T- and

Y-pipe junctions. The study focused on how Newtonian and non-Newtonian fluids behave. The research showed that the shape of the junctions is very important for how well the flow works. The results indicated that larger pipe diameters and higher junction angles lead to better oil holdup and lower pressure gradients.

This study emphasizes the significance of optimizing pipe layouts to improve flow stability and efficiency in systems that transport heavy oil. De la *et al.,* [17] conducted additional research on two-phase flows, which included an inquiry of how water-oil mixes change in Y-junctions. Their numerical work highlighted the importance of injection designs on phase behavior. It showed that some settings could cause transitions from stable core annular flow to stratified or slug flows downstream. This shift presents difficulties in maintaining flow stability and efficiency, which emphasizes the need for accurate modelling tools to properly forecast and regulate these behaviors. The findings showed that while some designs enhanced oil holdup, they also caused pressure fluctuations to increase, which meant that design factors needed to be balanced carefully. De la *et al.,* [18] conducted another important study in which they investigated the emulsification process in Y-junction horizontal pipelines. They focused on several injection configurations for mixtures of water and glycerol. The purpose of the research was to investigate the impact of different flow conditions on the effectiveness and stability of emulsification in these junctions. The findings demonstrated that optimizing injection settings might greatly enhance phase interactions and improve overall flow performance, thereby giving useful insight into the design of efficient fluid transport systems.

This work provides additional evidence of the complexity of flow dynamics at junctions and the urgent requirement for better computational fluid dynamics (CFD) approaches to effectively capture these interactions. In general, these studies highlight the complex nature of fluid dynamics in Y-junction pipes and the need to use advanced modelling techniques to improve flow performance in a variety of industrial applications. Continuing study in this subject is vital for developing more efficient fluid transport systems that can efficiently handle complicated multiphase flows while minimizing energy losses and operational issues.

2. Methodology

2.1 System Description

With a focus on comprehending how changes in pipe shape and mesh size affect flow dynamics, the system presented employs computational fluid dynamics (CFD) simulations carried out in ANSYS to analyze a Y-junction pipe in depth. To start the investigation, the Y-junction geometry was duplicated and modified to produce three separate models, each with a different pipe radius. The first model has a radius of 0.65 cm. While the second model significantly extended the radius 0.95 cm to assess the effect of an enlarged cross-section on flow behavior, the first model kept the baseline radius as a reference. The third model, which concentrated on the consequences of narrower flow channels, increased to a radius of 1.25 cm.

In every CFD simulation, a grid independence test is carried out to discover the optimum grid size that has the smallest number of grids without significantly changing the numerical results based on the evaluation of various grid sizes. The purpose of these changes was to provide light on how the radius influences variables including pressure drop, turbulence intensity, and velocity distribution. To guarantee the precision and dependability of the simulations, mesh refinement and duplication were carried out after the geometric adjustments. Five copies of each of the three geometries were made, each with an increasingly fine-tuned mesh. This stage was essential for doing a grid independence study, a procedure meant to determine the ideal mesh size at which the stimulation results become stable and insensitive to additional fine-tuning. It was possible to assess how the distribution of nodes and elements affected the computational results since the mesh sizes most likely varied from coarse to fine.

2.2 Geometry Details

The geometry setup for the Y-junction pipe in ANSYS was designed to systematically study the effect of varying pipe radius on fluid flow behavior as shown in Figure 1. The Y-junction geometry was carefully modeled using the Design Modeler tool within ANSYS, with three different configurations, each having a unique pipe radius, while maintaining consistent lengths for the pipes. The Y-junction consisted of a main inlet pipe that bifurcates into two outlet branches at an angle of 30° each. The lengths of the pipes were kept constant across all configurations to ensure comparability of results. Specifically, the main inlet pipe was modeled to have a length of 120 cm, while each of the two outlet branches was designed with a length of 100 cm. This consistent dimensional setup enabled a focus on the effects of varying the pipe radius without introducing inconsistencies in flow path length as shown in Table 1.

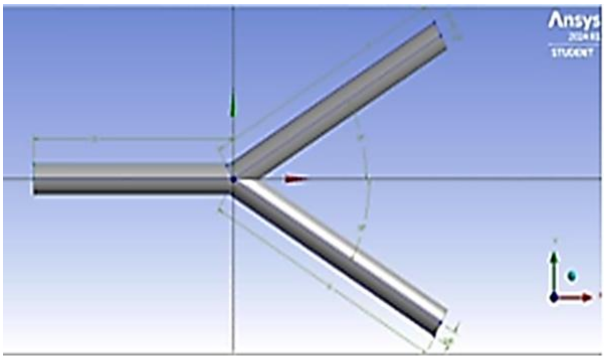


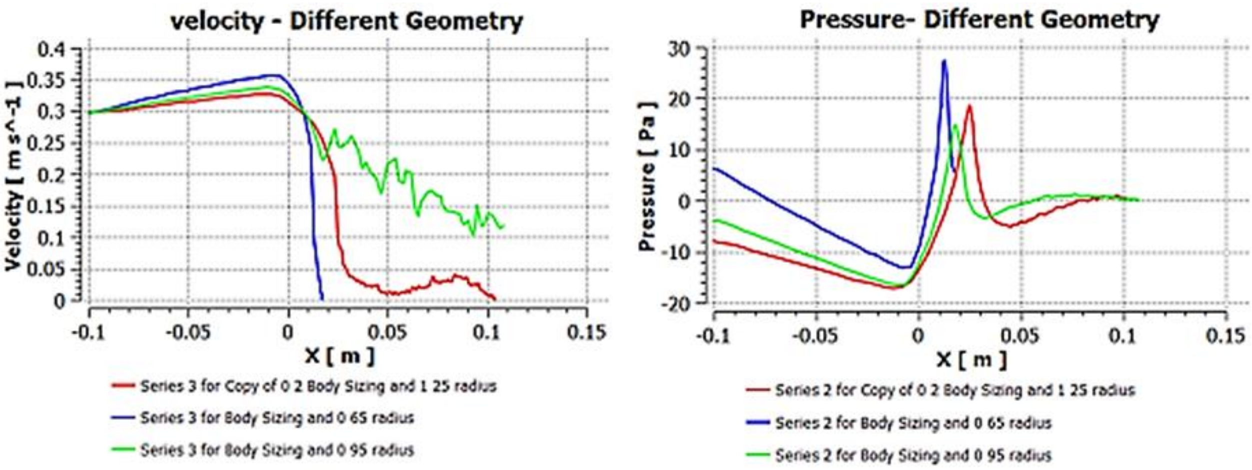
Fig. 1. Y Junction pipe

Table 1
Geometry parameter

Case	External radius (cm)	Length (cm)	Element (cm)	Element	Nodes
1	0.65			48585	10367
2	0.95	100 and 120	0.2	100472	19974
3	1.25			170007	32507

The modeling process involved creating 2D sketches for the pipe layout on the XY plane. The sketches included straight lines and arcs to represent the inlet and bifurcated branches. After completing the sketches, the Revolve tool was used to generate 3D cylindrical pipes by rotating the 2D profile around the central axis. Careful attention was given to ensure smooth transitions at the junction and a realistic representation of the bifurcation. Each geometry configuration was saved as a separate model, ensuring that only the pipe radius was varied while the length and overall design remained identical. This approach allowed for consistent boundary conditions and mesh setups, facilitating a direct comparison of simulation results for different pipe radius. The detailed geometry creation ensured that the Y-junction models accurately represented real-world applications and provided a solid foundation for analyzing fluid flow dynamics in the subsequent CFD simulations.

Significant impacts on velocity and pressure distributions are shown by the examination of the Y-junction pipe with different radius, as the graphs in Figure 2 demonstrate. The confined flow route in the small- radius design (0.65 cm) led to higher flow acceleration and the highest velocity values. The red curve of the pressure graph, which shows energy losses because of increased flow resistance and possible vortex formation at the junction, indicates that this also caused the biggest pressure reduction. On the other hand, the baseline medium-radius arrangement (0.95 cm) displayed moderate velocity and pressure profiles, achieving a balance between pressure loss and flow rate. In comparison to the small radius instance, the velocity distribution was smoother, and the pressure drop was less severe, suggesting increased flow efficiency. Due to the extended flow path's ability to lower flow resistance and promote more uniform distribution, the large radius design (1.25 cm) showed the lowest velocity and pressure drop. However, in applications that call for dynamic flow modifications, this design may result in longer reaction times and decreased mixing efficiency. When building Y-junctions with varying pipe radius, the comparison often emphasizes the tradeoffs between velocity profiles, pressure losses, and flow efficiency. The results highlight how crucial it is to adjust pipe diameters to satisfy operating needs while striking a balance between flow dynamics and energy economy.



(a) (b)

Fig. 2. Different geometry (a) Velocity (b) Pressure drop chart based on different radius

2.3 Grid Independence Test

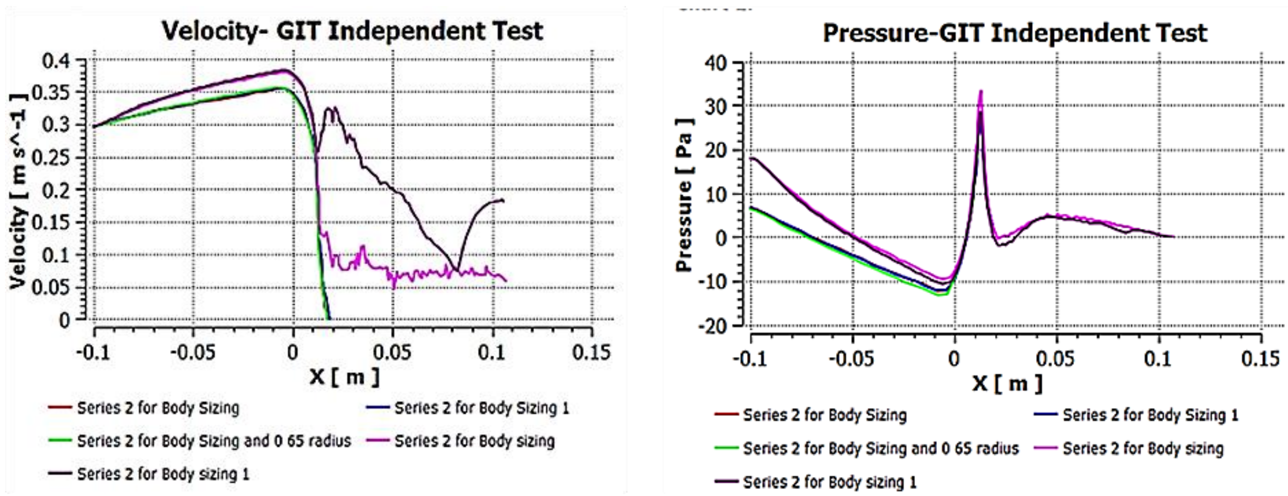
The Grid Independence Test is a crucial step in computational fluid dynamics (CFD) simulations to ensure that the results are not significantly influenced by the mesh size but are instead reflective of the actual physical phenomena being modeled [19]. In the analysis of a Y-junction pipe using ANSYS, the test was performed by creating five different meshes with varying element sizes 0.09 cm, 0.1 cm, 0.2 cm, 0.5 cm, and 1.1 cm as shown in Table 2. The purpose of this was to evaluate the influence of mesh refinement on the simulation results, particularly on the accuracy and convergence of key parameters such as velocity, pressure, and flow distribution. For each mesh size, a corresponding grid for the Y- junction geometry was generated, ensuring that each mesh was sufficiently fine to capture important flow features, such as turbulence, vortices, and flow separation, but not so fine that it would unnecessarily increase the computational cost. The five mesh sizes chosen ranged from the coarser mesh 1.1 cm to the finer mesh 0.09 cm, which allowed observation on how the resolution of the mesh impacted the accuracy of the flow characteristics in the junction.

The analysis of nodes and elements for each mesh size was critical. As the mesh size decreased from 1.1 cm to 0.09 cm, the number of elements and nodes increased, resulting in a more detailed representation of the flow field. A finer mesh can capture smaller-scale features, such as turbulence eddies and boundary layer behavior, which are critical for accurate predictions of pressure drop, velocity profiles, and vortex formation. The convergence of key parameters such as velocity and pressure drop with decreasing mesh size were observed. With the course meshes 1.1 cm, it can be noticed that the flow characteristics were not as well-defined, with larger discrepancies in pressure and velocity profiles compared to the finer meshes. As the mesh was refined, the results became more stable, showing less fluctuation in values, particularly for parameters like the pressure drop across the junction and the velocity distribution. However, once the mesh size of 0.09 cm was reached, the results showed diminishing returns, where further refinement had little effect on the overall outcome. This indicated that the mesh has achieved independence, meaning the results were no longer significantly affected by further mesh refinement. The grid independence test was done to select an optimal mesh size that provided a balance between computational cost and result accuracy, ensuring reliable and efficient simulations for the Y-junction pipe.

Table 2
Grid independent test parameters

Case	Element size (cm)	Length (cm)	External radius (cm)	Element	Nodes
1	0.09			513960	96420
2	0.1			380419	72348
3	0.2	100 and 120	0.65	48585	10367
4	0.5			48452	10508
5	1.1			48742	10558

Figure 3 shows how various element sizes affect a GIT's (grid independent test) velocity and pressure profiles. Every line represents a distinct configuration with differences in radius and body size that impact flow behavior. A bigger element size is shown by the red line, which exhibits smoother transitions with a consistent rise in velocity, a rapid decrease close to X = 0, and modest stabilization, which suggests less turbulence. The pressure profile that corresponds to it shows a clear increase followed by a gradual decline until stabilizing at a higher level. The pressure profile of the green line, which represents a smaller element with a smaller radius (0.65 cm), displays softer fall and recovery, indicating better flow management, smoother flow transitions, with a swift velocity climb and fewer abrupt decreases.



(a) (b)

Fig. 3. Independent test for (a) Velocity (b) Pressure drop chart test

On the other hand, the purple and pink lines, which represent intermediate element sizes, show notable variations in pressure and velocity, emphasizing flow instability and turbulence. These findings highlight how crucial it is to optimize element size and shape to decrease turbulence, provide smooth flow profiles, and improve system performance. While intermediate sizes tend to worsen flow disruptions, smaller components with a smaller radius often offer the optimum equilibrium.

3. Comparison with Published Value

In the published study by Dianita *et al.,* [20], a grid independence test was performed to identify the optimal computational grid size for accurate CFD simulations of flow behavior in a horizontal pipe with sudden expansion. The analysis revealed that using 328,700 computational cells provided a balance between accuracy and computational efficiency. This grid size achieved upstream and downstream pressure gradient values of 9.39 kPa/m and 1.67 kPa/m, respectively, with relative errors of 1% and 9% compared to the experimental results of 9.29 kPa/m (upstream) and 1.54 kPa/m (downstream). While the finer grid size of 509,212 cells showed slightly improved accuracy (relative errors of 2% and 12%), the differences were deemed insignificant, leading to the selection of 328,700 cells for further simulations. The study under comparison investigates the effects of varying pipe radius on velocity and pressure distributions in Y-junctions, using computational fluid dynamics (CFD) simulations, while this conducted study explores similar phenomena but with additional emphasis on mesh refinement through a detailed grid independence test. Both studies focus on the interaction between geometry, flow characteristics, and computational accuracy, employing systematic methodologies to ensure reliable results.

The compared study demonstrates that smaller pipe radius (0.65 cm) results in higher flow velocity and significant pressure drops due to increased flow resistance and vortex formation. In contrast, a medium radius (0.95 cm) offers a balanced performance, minimizing pressure loss while maintaining efficient flow. The larger radius (1.25 cm) achieves the lowest pressure drop and velocity but at the expense of reduced dynamic flow response and mixing efficiency. These findings highlight the trade-offs between velocity profiles, pressure losses, and flow efficiency, underscoring the importance of tailoring pipe geometry to operational requirements. In this conducted study, a Grid Independence Test further validates the accuracy of CFD simulations, focusing on how mesh refinement impacts key parameters such as velocity, pressure, and flow distribution. By testing five different mesh sizes (0.09 cm to 1.1 cm), this ensured that the results were independent of mesh resolution. Finer meshes captured intricate flow features like turbulence and vortices, leading to more precise predictions. However, beyond the mesh size of 0.09 cm, additional refinement yielded diminishing returns, confirming grid independence. This approach not only optimized computational efficiency but also enhanced the reliability of simulation results [16,20].

Both studies emphasize the importance of optimizing geometric and computational parameters to balance flow efficiency, energy economy, and simulation accuracy. However, the incorporation of a systematic Grid Independence Test in this conducted study provides an additional layer of validation, ensuring that the observed flow characteristics are reflective of real-world dynamics and not artifacts of numerical modelling. This robust the methodology of this study and well-suited for practical applications where both accuracy and efficiency are critical.

3.1 Findings Analysis vs Published Data and CFD Model Parameters Accuracy

The results of this investigation validate the chosen CFD model parameters and guarantee that the simulations faithfully depict the flow dynamics in Y-junction pipes in the real world since they closely match with the reported values in literature material. The observation in this study of the variance in velocity and pressure distributions over various pipe radii are in good agreement with trends that have been previously documented. For example, the larger pressure drops and increased flow acceleration in designs with smaller radius (0.65 cm) are in line with well-established fluid dynamics principles, which state that limited flow regions result in increased resistance and energy losses. These results are further supported by the medium-radius designs' modest performance (0.95 cm) and the larger- radius configurations' decreased velocity and pressure drop (1.25 cm).

The apparent accuracy of the findings was greatly influenced by the model parameters chosen, including mesh refinement levels, boundary conditions, and suitable turbulence models. The study made sure that the numerical predictions were representative of the actual processes and unaffected by mesh size by conducting a Grid Independence Test. The durability of the CFD setup was shown by the convergence of important parameters, including velocity and pressure drop, at a mesh size of 0.09 cm.

4. Conclusions

To sum up, the investigation of fluid flow behavior in a Y-junction pipe with different radii has shed light on the connection between flow efficiency, pressure drop, velocity distribution, and pipe shape. Three alternative Y-junction pipe designs with varying radii, small (0.65 cm), medium (0.95 cm), and large (1.25 cm) were carefully modelled and simulated in ANSYS to investigate their influence on the flow dynamics. The findings showed that although a smaller radius increased flow acceleration and velocity, it also increased pressure losses and the possibility of vortex formation at the junction. While the large radius showed lower velocities and pressure decreases, indicating increased flow efficiency but possibly at the expense of slower responses and less mixing, the medium radius provided a balanced flow with moderate velocity and pressure drop.

Additionally, the Grid Independence Test verified that the mesh size significantly affected the accuracy of simulation results, with finer meshes enhancing the predictions of flow characteristics up to a point of diminishing returns. The correctness of the simulations was confirmed by the convergence analysis of residuals over all three configurations, which showed stable and dependable solutions. These results highlight how crucial it is to choose the best pipe radius for a given application, balancing flow efficiency, pressure, and velocity to achieve the intended operational goals. Accurate CFD modeling is also ensured by meticulous mesh refinement and convergence monitoring. This study provides helpful recommendations for Y-junction pipe design optimization in a range of fluid flow applications.

References

[1] Decaix, Jean, Mathieu Mettille, Jean-Louis Drommi, Nicolas Hugo, and Cécile Münch-Alligné. "Computation fluid dynamics investigation of the flow in junctions: application to hydraulic short circuit operating mode." *LHB* 109, no. 1 (2023): 2290025. https://doi.org/10.1080/27678490.2023.2290025

[2] Kharoua, Nabil, Mohamed Alshehhi, and Lyes Khezzar. "Effects of fluid flow split on black powder distribution in pipe junctions." *Advanced Powder Technology* 27, no. 1 (2016): 42-52. https://doi.org/10.1016/j.apt.2015.10.023

[3] 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

[4] Mund, Chinmaya, Sushil Kumar Rathore, and Ranjit Kumar Sahoo. "A review of solar air collectors about various modifications for performance enhancement." *Solar Energy* 228 (2021): 140-167. https://doi.org/10.1016/j.solener.2021.08.040

[5] Leschziner, M. A. "Modelling turbulent separated flow in the context of aerodynamic applications." *Fluid Dynamics Research* 38, no. 2-3 (2006): 174-210. https://doi.org/10.1016/j.fluiddyn.2004.11.004

[6] Trong Bui and Bogdan-Alexandru Belega. "CFD analysis of flow in convergent-divergent nozzle." In *International Conference of Scientific Paper AFASES*. 2015.

[7] Deshpande, Nikhil D., Suyash S. Vidwans. "Theoretical and CFD analysis of De-Laval nozzle." *International Journal of Mechanical and Production Engineering* 2, no. 4 (2014).

[8] Sudhakar, B. V. V. N., B. Purna Chandra Sekhar, P. Narendra Mohan, and Md Touseef Ahmad. "Modeling and simulation of convergent-divergent nozzle using computational fluid dynamics." *International Research Journal of Engineering and Technology* 3, no. 08 (2016): 346-350.

[9] Mehl, Steffen, Mary C. Hill, and Stanley A. Leake. "Comparison of local grid refinement methods for mod flow." *Groundwater* 44, no. 6 (2006): 792-796. https://doi.org/10.1111/j.1745-6584.2006.00192.x

[10] Lee, Minhyung, Gwanyong Park, Changyoung Park, and Changmin Kim. "Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution." *Advances in Civil Engineering* 2020, no. 1 (2020): 8827936. https://doi.org/10.1155/2020/8827936

[11] Debtera, Baru. "Computational fluid dynamics simulation and analysis of fluid flow in pipe: Effect of fluid viscosity." *Journal of Computational and Theoretical Nanoscience* 18, no. 3 (2021): 805-810. https://doi.org/10.2139/ssrn.4201717

[12] Tominaga, Yoshihide. "CFD simulations of turbulent flow and dispersion in built environment: A perspective review." *Journal of Wind Engineering and Industrial Aerodynamics* 249 (2024): 105741. https://doi.org/10.1016/j.jweia.2024.105741

[13] Shrestha, Ujjwal, and Young-Do Choi. "A CFD-based shape design optimization process of fixed flow passages in a Francis hydro turbine." *Processes* 8, no. 11 (2020): 1392. https://doi.org/10.3390/pr8111392

[14] Dang Le, Quang, Riccardo Mereu, Giorgio Besagni, Vincenzo Dossena, and Fabio Inzoli. "Computational fluid dynamics modeling of flashing flow in convergent-divergent nozzle." *Journal of Fluids Engineering* 140, no. 10 (2018): 101102. https://doi.org/10.1115/1.4039908

[15] Antepara, Oscar, O. Lehmkuhl, R. Borrell, J. Chiva, and A. Oliva. "Parallel adaptive mesh refinement for large-eddy simulations of turbulent flows." *Computers & Fluids* 110 (2015): 48-61. https://doi.org/10.1016/j.compfluid.2014.09.050

[16] Dianita, Cindy, Ratchanon Piemjaiswang, and Benjapon Chalermsinsuwan. "Effect of T-and Y-pipes on core annular flow of Newtonian/Non-Newtonian Carreau fluid using computational fluid dynamics and statistical experimental design analysis." *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering* 47, no. 3 (2023): 941-958. https://doi.org/10.1007/s40997-022-00568-z

[17] De la Cruz-Ávila, M., I. Carvajal-Mariscal, Leonardo Di G. Sigalotti, and Jaime Klapp. "Numerical study of water-oil two-phase flow evolution in a Y-junction horizontal pipeline." *Water* 14, no. 21 (2022): 3451. https://doi.org/10.3390/w14213451

[18] De la Cruz-Ávila, M., I. Carvajal-Mariscal, J. Klapp, and J. E. V. Guzmán. "Numerical study of multiphase water–glycerol emulsification process in a Y-Junction horizontal pipeline." *Energies* 15, no. 8 (2022): 2723. https://doi.org/10.3390/en15082723

[19] Dehshiri, Seyyed Shahabaddin Hosseini, and Bahar Firoozabadi. "A grid independence study to select computational parameters in dust storm prediction models: A sensitive analysis." *Urban Climate* 49 (2023): 101534. https://doi.org/10.1016/j.uclim.2023.101534

[20] 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