INVOTEK

Jurnal Inovasi Vokasional dan Teknologi

http://invotek.ppj.unp.ac.id/index.php/invotek

Vol. 25, No. 1, Feb 2025

ISSN: 1411 – 3411 (p)
ISSN: 2549 – 9815 (e)

CFD Simulation of Pipe Joints Using SimScale: Analysis of the Effect of Different Inlet Velocities on Water Fluid Flow

Haning Hasbiyati[1] *, Audha Fitrah Aulina[2]

[1] Renewable Energy Engineering Study Program, Department of Engineering, Politeknik Negeri Jember
Jl. Mastrip PO BOX 164, Jember, Indonesia-68101

[2] Automotive Engine Study Program, Department of Engineering, Politeknik Negeri Jember
Jl. Mastrip PO BOX 164, Jember, Indonesia-68101

Corresponding author: haning.hasbiyati@polije.ac.id Doi: https://doi.org/10.24036/invotek.v25i1.1263

Abstract

This study aims to analyze the effect of variations in inlet velocity on fluid flow patterns at tee-junction pipe connections using numerical simulations based on Computational Fluid Dynamics (CFD). The SimScale platform was used because it supports cloud-based processing and integration with OpenFOAM solvers. The tee connection model is designed in three dimensions, with the main and branch pipe dimensions each having a diameter of 100 mm. The merging process uses the parametric hexagonal method and local refinement in the joint area to accurately capture turbulent phenomena. Simulations were performed under incompressible and isothermal flow conditions with a standard k–ε turbulence model, using water fluid and the SIMPLE algorithm. The inlet velocity consists of 2 variations A (V_1 = 1 m/s, V_2 = -1 m/s) and variation B (V_1 = -1.5 m/s, V_2 = -3 m/s). Simulation results show that increasing the inlet velocity results in a more turbulent flow, characterized by an increase in the turbulent kinematic viscosity, specific dissipation rate, and turbulent kinetic energy. Conversely, low velocities indicate higher pressure accumulation due to flow resistance. These findings indicate that inlet velocity variations significantly affect flow characteristics, requiring attention in system design to maintain long-term operational efficiency and reliability.

Keywords: CFD, SimScale, Inlet Velocity, Fluid Flow.

1. Introduction

Over the past five years, various studies have been conducted to understand fluid flow problems in T-joints in industrial piping systems [1] through numerical and experimental approaches [2] used unstable fluid simulations to analyze thermal line phenomena resulting from the mixing of molten metal flows at different temperatures, highlighting the influence of momentum ratio and temperature differences on pressure and temperature fluctuations. Research by [3] utilized CFD simulations to evaluate velocity distribution, turbulence, and pressure drop at T-joint intersections [4] developed a new model to calculate pressure loss in compressible gas flow through T-joints using experimental data. Meanwhile, [5] investigated the use of vortex generators as a solution to reduce flow separation and recirculation zones in the joint area, which was found to improve flow efficiency. These findings emphasize the importance of a deeper understanding of flow behavior at T-junctions to support the design of more reliable and energy-efficient piping systems. As the need for efficiency in piping system design increases, numerical simulation-based approaches such as Computational Fluid Dynamics (CFD) are increasingly used to understand flow characteristics in complex geometries. CFD studies can save design time and costs while enhancing understanding of fluid flow systems [6]. Research demonstrates the application of CFD (using ANSYS Fluent) in evaluating pipe geometry, surfaces, and pressure [7],

Received date 13/06/2025, Revised date 01/07/2025, Accepted date 16/07/2025

P-ISSN: 1411-3414
E-ISSN: 2549-9815

[8]. Simulations on downdraft gasification installations describe CFD as a tool for predicting internal fluid flow within pipes, highlighting the relevance of turbulence and internal geometric context [7].

A recent study by [9] shows that calibrating the k-ω SST model using experimental data can improve flow prediction accuracy on free surfaces, such as in mountain slope water channels. This highlights the model's significant potential in various civil and environmental engineering applications. Additionally, a study by [10] investigates adapting k-ω SST model parameters to enhance flow simulation accuracy around ships. By using assimilation data, this study successfully improved flow predictions in the ship's hull area, which is crucial for efficient ship design. By integrating cloud-based CFD and advanced turbulence models like k-ω SST, engineers can perform more accurate and efficient fluid flow analysis. This not only accelerates the design process but also improves the quality and reliability of the designed piping systems. Previous research by [11] showed that variations in inlet velocity can significantly impact pressure distribution and flow patterns at T-shaped pipe joints. They found that increasing the velocity of the second inlet channel improves mixing efficiency but can also cause significant velocity spikes, which could potentially damage the pipe material.

Although previous studies have addressed many important aspects related to fluid flow in T-joints, there are several research gaps that present opportunities for further scientific contributions. The limitations of studies using cloud-based platforms (SimScale) to analyze T-joints in depth, particularly regarding sensitivity to inlet velocity variations. Most studies still use software such as ANSYS Fluent or OpenFOAM. Few studies have systematically compared the effects of inlet velocity variations on pressure distribution and fluid velocity in T-joints using the k-ω SST model on the SimScale platform. There is a lack of integration between simulation results and technical design aspects of piping systems, such as potential material wear due to velocity spikes or uneven pressure, despite their demonstrated impact in study [12] Research is still limited to single-direction or single-inlet scenarios, while industrial scenarios often involve multi-inlet configurations with unbalanced velocities, which can complicate turbulence and mixing phenomena. There is a lack of visualization and quantitative analysis of critical zones such as recirculation zones, separation points, or shear layer development due to inlet velocity variations at T-joints.

Therefore, further research is needed that specifically uses SimScale as a cloud-based platform to simulate the effects of inlet velocity variations at T-shaped pipe connections, focusing on analyzing flow patterns, pressure distribution, and turbulent zones using the k-ω SST model. This research is expected to provide not only theoretical understanding but also practical applications in designing more efficient piping systems. The SimScale platform offers cloud-based solutions for CFD simulation, allowing users to perform fluid flow analysis without the need for expensive hardware. SimScale supports various turbulence models and provides an intuitive user interface, making it suitable for flow analysis at pipe connections with varying inlet velocities. Based on this background, this study aims to perform CFD simulations on T-shaped pipe connections using SimScale, with a focus on analyzing the influence of inlet velocity variations on fluid flow patterns. The results of this study are expected to provide deeper insights into flow behavior in pipe connections and contribute to the body of literature on CFD-based fluid flow simulation studies in the field of engineering.

2. Method

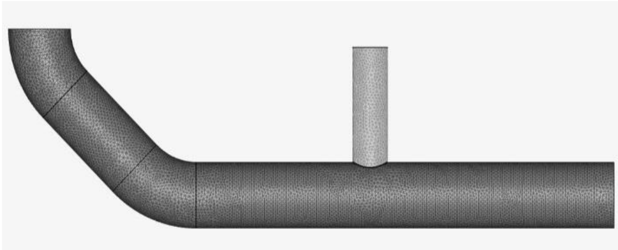
This study uses a Computational Fluid Dynamics (CFD)-based numerical simulation approach to analyze the effect of inlet velocity variations on fluid flow patterns at T-junction pipe connections. The SimScale platform was chosen as the primary tool for the simulation due to its cloud-based accessibility and support for the integrated OpenFOAM solver, which enables web-based processing with good computational accuracy and efficiency [13].

The geometry of the T-joint model was designed in three dimensions with the following specifications: main pipe and branch diameter of 100 mm, main pipe length of 750 mm, and branch pipe length of 125 mm. Pipe wall thickness was ignored under the assumption that the domain is a solid fluid. The model geometry is created directly in SimScale or imported from external CAD software such as SolidWorks in STEP format.

The meshing process is performed using a parametric hexagonal meshing method with local refinement in the joint area to capture turbulence phenomena and recirculation zones more accurately. Mesh quality is checked using skewness and non-orthogonality parameters. The number of mesh

P-ISSN: 1411-3414
E-ISSN: 2549-9815

elements is varied between 1 and 3 million cells to ensure the accuracy of the results without sacrificing computational efficiency [14]. The results of the meshing process for the entire T-junction area are shown in Figure 1a, and the meshing in the inlet area is shown in Figure 1b.



(a) The whole area

(b) Inlet

Figure 1. Meshing

Before the simulation process was carried out, boundary conditions were defined for the T-junction area to describe the physical conditions at the boundaries of the system. The boundary conditions applied in this study consisted of two inlets, one outlet, and a wall surrounding the T-junction system. The boundary conditions used in this study are illustrated in Figure 2.

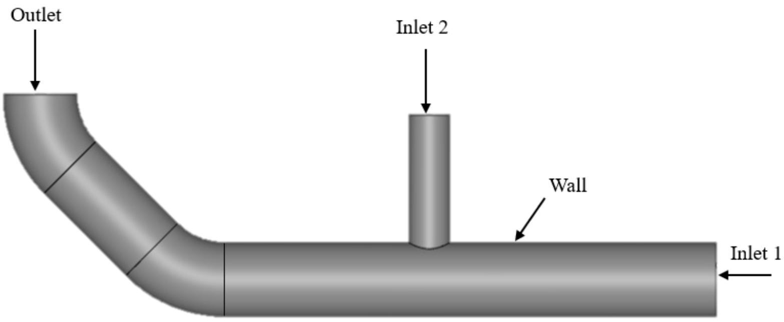


Figure 2. Boundary Condition

The simulation was conducted under steady-state conditions with the assumption of incompressible and isothermal flow. The main simulation parameters include: the working fluid is water with a density of 998.2 kg/m³ and dynamic viscosity of 0.001003 Pa s; the turbulence model used is the standard k–ε model; and the SIMPLE solver algorithm is used to solve the Navier-Stokes equations under incompressible conditions. Turbulence parameters such as turbulent kinetic energy (k) and dissipation rate (ε) are automatically initialized based on inlet conditions using default settings on the SimScale platform. The inlet channel is given a constant pressure of 0 Pa (gauge), and the pipe walls are given no-slip conditions. The inlet velocity used in this study has two variations, namely (Table 1):

Table 1. Variation of Inlet Velocity

Speed Variation	V_1 (m)	V_2 (m)
A	-0.5	-1
B	-1	-3

The CFD simulation procedure is designed in the form of a flow chart shown in Figure 3. The main variables observed in this simulation include fluid velocity distribution (both in contour and vector form), flow patterns (streamlines and recirculation zones), pressure distribution along the pipe axis, turbulent viscosity distribution, and velocity profiles at the outlet and branches. All data is visualized using post-processing features in SimScale, including the creation of cross-sections, velocity profile graphs, and streamlines to detect flow direction and branching.

To validate the accuracy of the CFD simulations, a qualitative validation approach was taken by comparing the simulation results to physical phenomena reported in previous studies, such as the formation of stagnation zones, flow separation, and vortices at T-shaped pipe joints. In addition, mesh independence tests were conducted to ensure the numerical stability of the simulation results. Although

P-ISSN: 1411-3414
E-ISSN: 2549-9815

experimental validation was not performed directly in this study, the simulation results were qualitatively compared with findings from previous literature studies on flow in T-shaped pipe connections. Phenomena such as stagnation zones, flow separation, and vortex formation at the branch were examined to assess the consistency of the results with previously reported physical behavior [15].

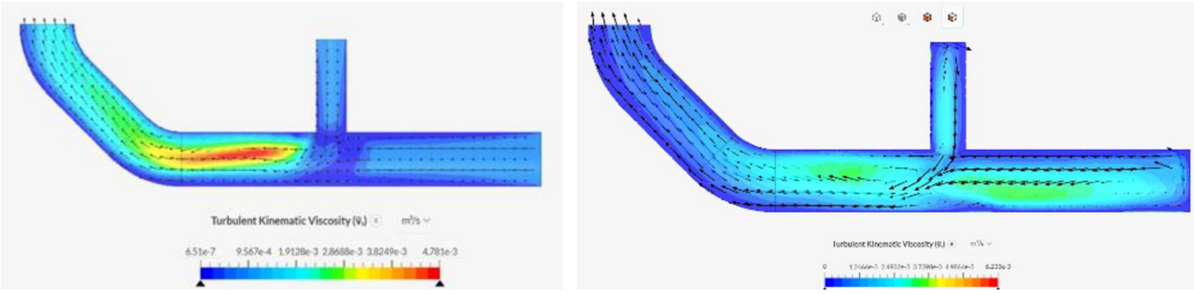
Figure 3. CFD Simulation Procedure

3. Results and Discussion

CFD simulation on T-shaped pipe joints was performed using the SimScale platform to analyze the effects of inlet velocity variations on various fluid flow parameters. T-joints are critical areas in piping networks because sharp flow bends generate high turbulence zones, vortices, and increased dynamic pressure, which cause pipe wall wear and a decrease in flow system efficiency [16]. The simulation results were visualized in the form of contours and graphs depicting parameters such as turbulent kinematic viscosity, specific dissipation rate, pressure, velocity, and turbulent kinetic energy.

3.1 Turbulent Kinematic Viscosity

Based on the results of fluid flow simulations at T-shaped pipe joints (T-junctions) conducted using CFD SimScale software, it can be analyzed that the flow characteristics produced in the branching area are highly complex and critical to the overall performance of the system. Figure 4 presents the results of the simulation of Kinematic Turbulent Viscosity at the T-shaped pipe joint with two variations in inlet velocity, namely velocity variations A and B. A striking difference is observed in the distribution of turbulent viscosity values, which are displayed in a color gradient. In velocity variation A, the area where the main flow and the flow from the vertical branch meet shows yellow to red colors, indicating high turbulent viscosity values. This indicates that the turbulence level in that area is quite significant due to strong interactions between high-speed flows.



(a) Speed Variation A

(b) Speed Variation B

Figure 4. Turbulent Kinematic Viscosity Simulation

P-ISSN: 1411-3414
E-ISSN: 2549-9815

Conversely, in velocity variation B, the dominance of blue to green colors indicates lower turbulent viscosity values, and the flow tends to be calmer and more stable. Additionally, the direction and density of the flow vectors in the image also illustrate the differences in fluid dynamics between the two velocity variations. In variation A, the flow vectors appear denser and change direction sharply around the meeting point, indicating the presence of a strong recirculation and mixing zone between the fluids. This phenomenon causes increased energy dissipation and the possibility of flow separation. Conversely, in velocity variation B, the flow vectors are more uniform and directed, indicating a more stable flow pattern and minimal turbulence. This may be due to the lower velocity ratio in the inlet channel, resulting in insufficient fluid inertial forces to cause aggressive mixing.

From these results, it can be concluded that inlet velocity variation significantly affects turbulence characteristics at pipe joints. The higher the inlet velocity, the higher the turbulent viscosity value formed at the joint area, ultimately affecting flow efficiency and potential energy loss. Therefore, controlling the inlet channel velocity is crucial in pipeline system design, especially to avoid energy loss due to unwanted turbulence. This analysis aligns with findings in modern CFD studies, which state that T-junctions are critical points in flow systems due to their high sensitivity to boundary condition variations and fluid velocity [17].

Previous research [18] shows that the shape and geometry of the branch significantly influence flow characteristics. Y-shaped connections or connections with smooth curvature radii can significantly reduce turbulence and improve pressure distribution. Therefore, the design of the piping system for the turbine must consider reducing sharp changes in fluid direction to maintain system efficiency. Additionally, the flow vector direction from the simulation results indicates that part of the flow diverting upward experiences a decrease in velocity. This can cause an imbalance in energy supply to the turbine section receiving flow from that branch. When designing a turbine prototype, it is important to quantitatively evaluate the distribution of velocity and pressure throughout the flow path so that the position and geometry of the blades can be optimally adjusted. Overall, these simulation results provide important input in the initial design process of a turbine prototype, especially in closed or directed systems such as pipe turbines. The use of CFD simulation enables the early identification of potential energy losses, uneven flow distribution, and turbulent zones that could compromise system performance. Therefore, integrating simulation results with the physical design stage is highly recommended to produce a more efficient, economical, and field-adaptive turbine system.

The combination of the two speed variations A and B was examined at five observation points along the fluid flow path at the pipe connection. Subsequently, data in the form of a combined turbulent kinematic viscosity graph for speed variations A and B was obtained, as shown in Figure 5.

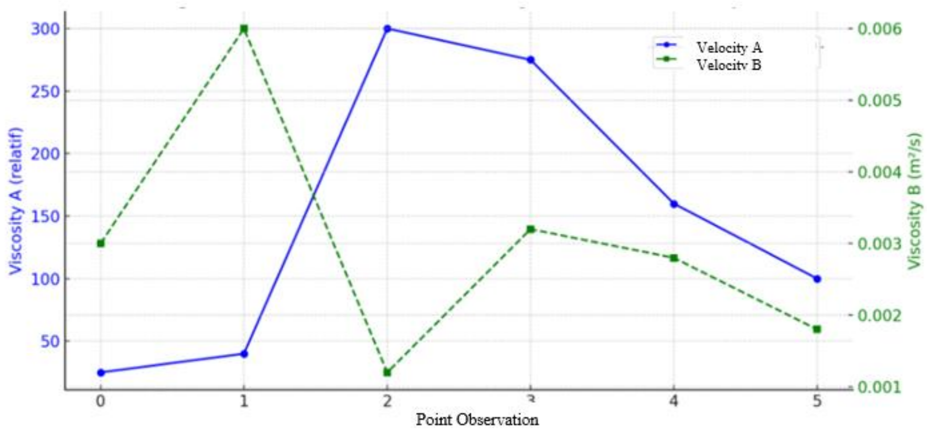


Figure 5. Turbulent Kinematic Viscosity Chart

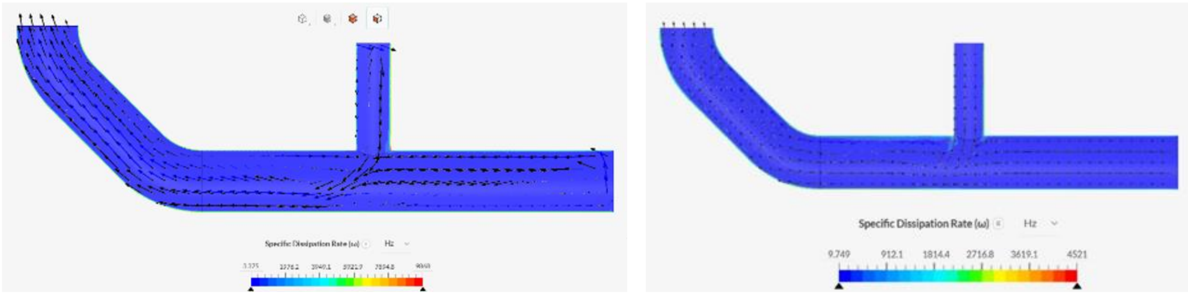
The combined turbulent kinematic viscosity graph for various speeds A and B shows the dynamics of kinematic viscosity at two different paths or measurement points in a turbulent flow system. The blue curve depicts the variation in Viscosity A (relative), showing a gradual increase from point 0 to 1, followed by a sharp spike at point 2 to reach a peak value of approximately 300 relative units. After reaching its peak, this viscosity value slowly decreases at the next point, indicating significant flow disturbance that then begins to subside. Meanwhile, the green curve representing Viscosity B (m^2/s) exhibits fluctuating characteristics, with a sharp spike from point 0 to 1, followed by a drastic drop at

P-ISSN: 1411-3414
E-ISSN: 2549-9815

point 2, then rising again at point 3 before decreasing again. This pattern reflects flow instability in path B, possibly caused by local turbulence effects or changes in channel geometry, such as a T-joint. Interestingly, there is an indirect correlation between the two paths, where the viscosity peak in path A coincides with the lowest point in path B, which may indicate energy compensation or redistribution within the flow system. Overall, this graph provides important insights into fluid behavior under turbulent conditions, which can be utilized for flow planning and control in piping systems or other fluid engineering applications.

3.2 Specific Dissipation Rate

The specific dissipation rate (ω) is used to detect small-scale turbulence in the flow. The following are the simulation results with speed variations:



(a) Speed Variation A (b) Speed Variation B

Figure 6. Specific Dissipation Rate Simulation

Figure 6 shows the simulation results of the Specific Dissipation Rate (SDR) at a T-shaped pipe joint for two conditions, namely velocity variation A and velocity variation B. The Specific Dissipation Rate is a parameter in turbulence models, particularly in the k–ω model, which describes the rate of turbulent energy dissipation per unit volume and time. High SDR values indicate intense turbulent energy release, typically occurring in areas with high velocity gradients or complex flow interactions. In velocity variation A, there is a color gradient from blue to red on the SDR contour map, indicating higher SDR values. This suggests that the flow in this condition experiences significant turbulence, with notable turbulent energy dissipation, particularly around the branch point. The direction of the flow vectors also shows deviations and disturbances at the connection points, reinforcing the likelihood of strong mixing and recirculation zones. Conversely, in velocity variation B, the SDR distribution is predominantly light blue, indicating lower dissipation values and more stable flow conditions. The flow vectors also appear more directed and do not show significant flow disturbances at the connection points.

These differences indicate that inlet velocity plays a crucial role in determining the intensity of turbulent energy dissipation. Variation A, with a higher velocity, causes an increase in kinetic energy, which ultimately increases the SDR value, while variation B results in smaller dissipation due to the smaller amount of flow energy carried. This phenomenon aligns with turbulent flow theory in piping systems, where flow velocity differences influence the interaction patterns between fluids and the transition area from laminar to turbulent flow. These results are consistent with the findings of [19], which show that the orientation of the branch at the “T” joint affects the efficiency of phase separation in gas-liquid flow, with a certain angle of inclination increasing separation efficiency. Additionally, a study by [20] used CFD simulations to analyze flow in a “T” joint and found that changes in geometry and flow conditions can affect velocity and pressure distribution.

Figure 7 shows a combined graph of the Specific Dissipation Rate observed from CFD simulations of water flow for variations in velocities A and B. This graph illustrates changes in the Specific Dissipation Rate values at observation points within the flow domain.

The combined specific dissipation rate graph from CFD simulations of water flow compares the variations in velocities A and B against the specific dissipation rate at five observation points. In the variation of velocity A, a fairly sharp fluctuation pattern is observed, with a significant increase at the second point reaching approximately 900,000 1/s, then decreasing at the third point, and rising again to the highest peak of around 1,000,000 1/s at the fourth point before decreasing at the fifth point. This pattern indicates high turbulent energy fluctuations, which may be caused by sudden changes in velocity

P-ISSN: 1411-3414
E-ISSN: 2549-9815

or flow direction at these points. Meanwhile, velocity variation B shows a more stable and gradual pattern. Its values tend to increase from the first point, reaching a peak of approximately 800,000 1/s at the fourth point, then decreasing slightly at the fifth point. The difference in patterns between velocities A and B indicates that the flow characteristics produced by velocity A are more dynamic and result in more extreme turbulent energy dissipation than velocity B. This information is important for analyzing system flow performance, as points with high dissipation may indicate potential energy loss or areas requiring channel design optimization. A study [21] shows that the use of CFD models can aid in analyzing flow characteristics and energy dissipation in abrasive water jet systems, which is relevant to studies of water flow in pipes. Additionally, research [22] highlights the importance of understanding thermal energy dissipation statistics in turbulent convection, which can provide additional insights into turbulent flow analysis. Thus, a deep understanding of the Specific Dissipation Rate and its application in CFD simulations is crucial for optimizing flow system design and identifying high energy dissipation areas that may cause pressure loss or system inefficiency

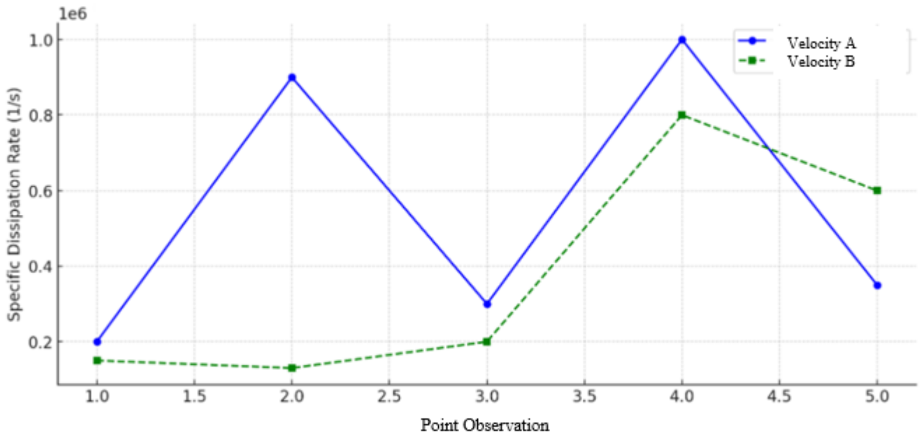
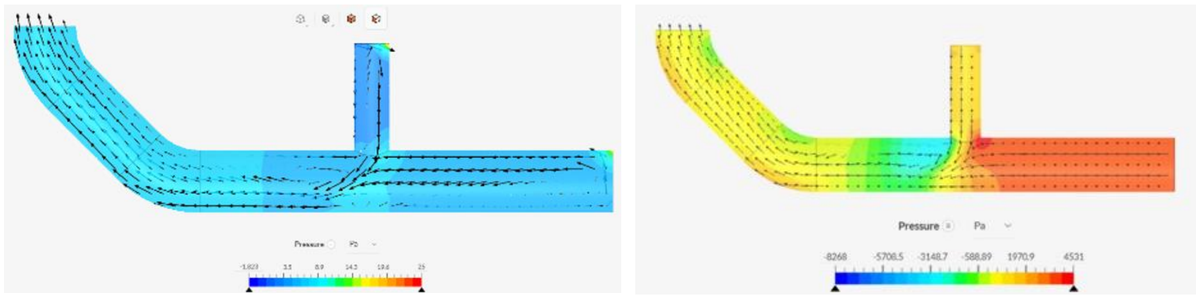


Figure 7. Specific Dissipation Rate Chart

3.3 *Pressure Distribution*

The pressure distribution shows that the highest pressure is at the upstream (before the connection) and decreases dramatically after passing through the connection. This phenomenon is in accordance with Bernoulli's principle, whereby an increase in velocity causes a decrease in pressure. At higher inlet velocity variations, the pressure gradient becomes steeper, which can increase the risk of pipe wall erosion. The following are the simulation results with velocity variations:



(a) Speed Variation A (b) Speed Variation B

Figure 8. Pressure Distribution Simulation

Figure 8 shows the simulation results of pressure distribution at a T-shaped pipe joint for two flow velocity variations, namely velocity variation A and velocity variation B. Pressure distribution is an important parameter in piping systems because it is directly related to the forces acting on the fluid and the overall efficiency of the system. From the color visualization and flow vector direction, significant differences between the two variations can be analyzed. In velocity variation A (Figure 8a), the pressure distribution shows a lower range and tends to be stable along the pipe path. Blue to light green colors dominate, indicating relatively low pressure and not too sharp pressure gradients. This

P-ISSN: 1411-3414
E-ISSN: 2549-9815

suggests that the flow is more uniform, and pressure loss due to friction or changes in fluid direction at the joint area is relatively small. The direction of the flow vectors also shows a smooth pattern without major disturbances, supporting the assumption that the flow is in a more controlled state.

Conversely, in velocity variation B (Figure 8b), the pressure distribution shows a wider color gradient, from blue at the inlet to red at the outlet, indicating significantly higher static pressure in the downstream section. Lower flow velocity causes static pressure to remain high, especially after branching. The yellow to orange color on the right side of the joint indicates pressure accumulation due to slow and less turbulent flow, so that pressure is not converted into kinetic energy.

This comparison shows that inlet velocity has a significant influence on pressure distribution within the piping system. Variation B, which likely has a lower velocity in one of the inlet channels than variation A, results in a more significant imbalance in pressure distribution. This can lead to flow inefficiency and increased risk of system failure due to excessive pressure or extreme negative pressure. Conversely, velocity variation A provides a more even pressure distribution, which is more ideal for piping system applications requiring flow stability. This finding is supported by recent literature emphasizing the importance of velocity and pressure control in pipe joint design to prevent hydrodynamic disturbances and energy loss [23], [24].

The combined pressure distribution graph from the CFD simulation results of water flow shows significant differences in characteristics between velocity variations A and B.

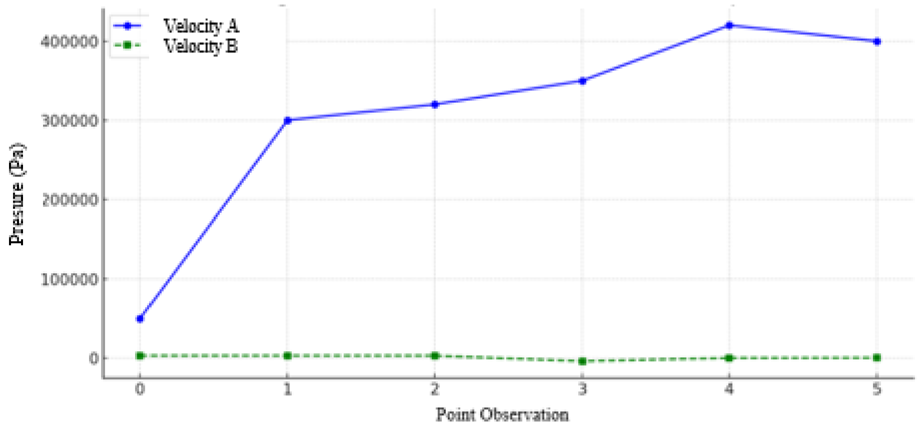


Figure 9. Pressure Distribution Chart

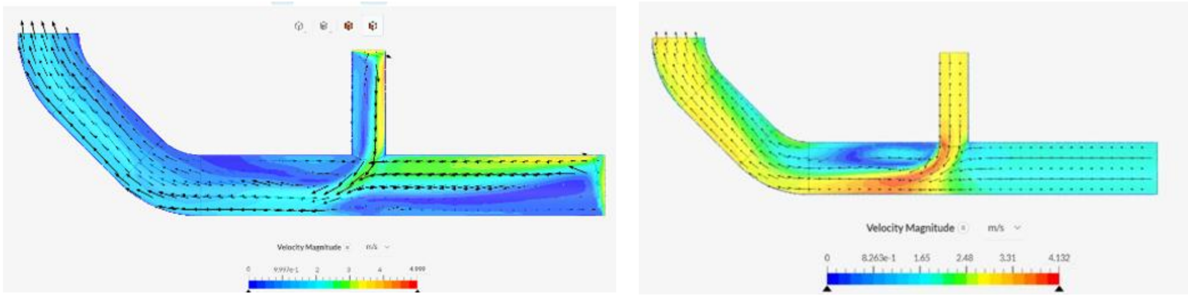
Based on Figure 9, at speed variation A, the pressure increases dramatically from an initial point of about 50,000 Pa to a peak of over 420,000 Pa at the fourth point, before decreasing slightly at the fifth point. This increase indicates that the flow at speed A results in a relatively high pressure buildup along the trajectory, indicating either flow resistance or pressure accumulation due to the high fluid velocity. Meanwhile, at speed variation B, the pressure is relatively very low and tends to fluctuate, even experiencing a negative value at the third point. Negative pressure values indicate the possibility of suction effects or local vacuum zones that can arise due to differences in velocity gradients and turbulence structures in the channel. This phenomenon is in line with fluid theory that pressure in the flow is strongly influenced by acceleration and flow direction. According to [25], in internal flow, such as flow in a pipe or narrow channel, the pressure distribution is closely related to the velocity and type of flow (laminar or turbulent).

High pressure usually occurs at the inlet or when the flow slows down, while low pressure or even negative pressure may occur in areas of rapid acceleration or behind obstacles. The study [26] also emphasizes that pressure is greatly affected by turbulence modeling and is important in evaluating the hydraulic performance of flow systems. As such, this graph shows how velocity variations can significantly change the pressure distribution in a flow system, which is very useful in the design of piping systems, irrigation channels, or water flow-based equipment. Understanding these pressure distribution graphs is important to avoid pressure loss, cavitation, or potential mechanical damage to the fluid system.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

3.4 Velocity Magnitude

The velocity magnitude shows a predominantly straight flow pattern from the main inlet to the outlet, with a small portion of the fluid entering the T branch. This pattern changes as the inlet velocity increases, where there is greater flow swirling and flow direction deviation. The following are the simulation results with speed variations:



(a) Speed Variation A (b) Speed Variation B

Figure 10. Velocity Magnitude Simulation

Figure 10 shows the simulation results of Velocity Magnitude or the magnitude of fluid flow velocity at the T-shaped pipe connection for two conditions of velocity variation, namely velocity variation A and velocity variation B. This velocity distribution provides important information about flow characteristics, potential stagnation zones, and fluid distribution efficiency in the piping system. In velocity variation A (Figure 10a), the fluid velocity appears high along the main flow, especially in the section after the bifurcation, which is marked with green to yellow color. The flowline pattern shows that the flow from the vertical branch manages to enter and mix with the main flow, forming a fairly sharp velocity gradient. This phenomenon indicates that the higher inlet velocity results in a more directional and dominant flow, and allows for an increase in kinetic energy at the outlet region.

In contrast, in velocity variation B (Figure 10b), the fluid velocity is generally lower, as indicated by the dominance of light blue to yellow colors. The flow pattern shows the presence of a recirculation zone or vortex in the area after the branch, characterized by a circular flow line pattern and low velocity color. This indicates that the lower velocity is not strong enough to overcome the effects of the joint geometry, leading to the formation of stagnant regions or localized turbulence. This comparison shows that velocity variation B produces flow with sharper velocity fluctuations and more complex flow patterns than velocity variation A. Higher inlet velocities or different fluid inlet directions in variation B can lead to large velocity gradients, which implies increased local kinetic energy but also increases the risk of flow irregularities. Meanwhile, velocity variation A shows more uniform flow performance and tends to be more efficient in the context of flow stability.

This is in accordance with the findings in CFD literature studies that emphasize the importance of velocity and inlet direction adjustment in pipe joint design to avoid the negative effects of flow separation and stagnation zones [27]. Research conducted by [28] reinforces these findings, where the flow at the "T" intersection produces complex vortex patterns and significant velocity fluctuations, especially around the inner wall of the flow bifurcation and confluence zone. Furthermore, based on research by [19], areas with sharp changes in flow direction in such joint systems are prone to low flow efficiency and localized stress increases, which, if not addressed, can lead to long-term mechanical damage.

The combined graph of CFD simulation results shown in Figure 11 shows the distribution of velocity magnitude at five observation points for two inlet velocity variations, namely variation A and variation B. In velocity variation A, there is a very high velocity spike at point 1, which then decreases dramatically at subsequent points. This phenomenon can be caused by the high initial velocity, which causes the flow to become unstable and experience a drastic decrease due to the influence of turbulence and internal friction. This follows [26], which states that a sudden increase in kinetic energy at the beginning of the simulation domain can cause significant pressure imbalance and turbulence.

In contrast, velocity variation B shows a more stable and gradually increasing velocity distribution from point 3 to point 5. This pattern indicates that the flow undergoes a more controlled development and exhibits better stability compared to variation A. According to [29] the gradually increasing stability

P-ISSN: 1411-3414
E-ISSN: 2549-9815

of the flow is usually influenced by the domain geometry design and moderate inlet velocity, which allows optimal boundary layer development and reduces pressure loss. Therefore, in designing fluid flow systems such as waterways or piping systems, the selection of inlet velocity variations should consider the stability and efficiency of the flow throughout the domain.

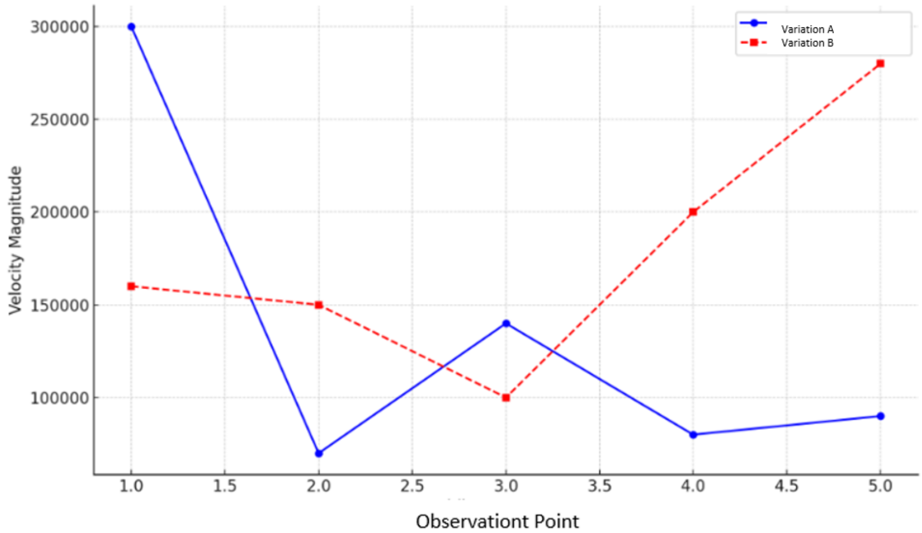
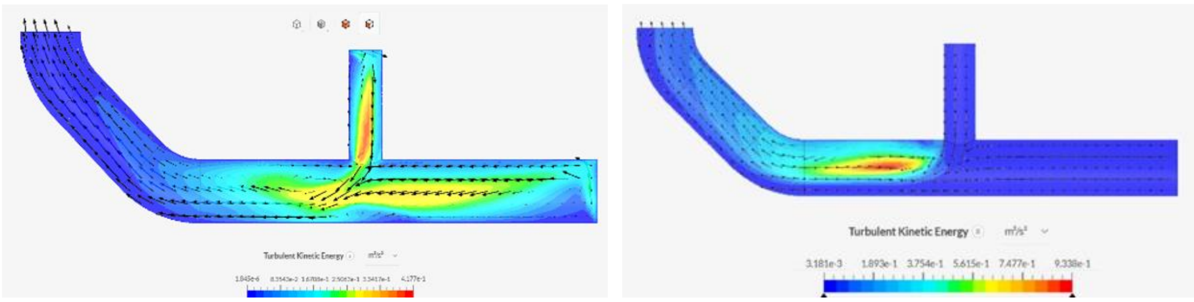


Figure 11. Velocity Magnitude Chart

3.5 *Kinetic Energy*

Turbulent kinetic energy (TKE) is used to describe the intensity of velocity fluctuations in the flow. The following are the simulation results with speed variations:



(a) Speed Variation A (b) Speed Variation B

Figure 12. Turbulent Kinetic Energy Simulation

Figure 12 shows the simulation results of Turbulent Kinetic Energy (TKE) in the water flow at the T-shaped pipe connection for two inlet velocity variations, namely velocity variation A and velocity variation B. Turbulent Kinetic Energy is one of the important parameters in computational fluid dynamics (CFD) that represents the amount of kinetic energy possessed by turbulent velocity fluctuations in the flow. The higher the TKE value, the greater the turbulence intensity, which can affect flow efficiency, fluid mixing, and pressure in the piping system. At speed variation A (Figure 12a), a fairly wide distribution of TKE is seen, with color gradations from blue to green, even reaching yellow and orange in some areas. The highest TKE values appear at the point where the flow meets from the vertical and horizontal directions, as well as along the path to the right outlet. This indicates that velocity A generates high levels of turbulence, especially in the bifurcation and turning areas. The flow vector pattern also shows directional disturbances, which indicate significant velocity fluctuations due to the interaction of flows from two different directions. These conditions are generally generated by high velocities or large differences between the two flow directions.

In contrast, in velocity variation B (Figure 12b), the TKE value is much lower, indicated by the dominance of blue and a little light green. Turbulent energy concentration points only appear in the bend area, and do not spread widely as in variation A. The flow vector direction also appears more regular

P-ISSN: 1411-3414
E-ISSN: 2549-9815

and does not show many abrupt changes. This suggests that at speed B, the flow tends to be more stable with less turbulent disturbance, resulting in higher efficiency in mass flow but possibly less favorable in the fluid mixing process. This comparison shows that varying speed A results in higher turbulence intensity, which is useful in applications that require rapid mixing (such as fluid reactors or chemical mixing), but can lead to greater pressure losses. Meanwhile, velocity variation B produces a more stable and efficient flow for clear fluid transportation without much pressure loss. These findings are in line with the research of [28] who used CFD simulations to observe the flow behavior at a “T” junction. They found that the bifurcation creates a region of maximum turbulence fluctuation due to changes in flow direction and geometry. In addition, [30] showed that the highest turbulent kinetic energy appears around the branch point due to the interaction of the inflow and outflow, which causes significant disturbances in the fluid boundary layer. This is reinforced by the results of research [19] which states that the effect of branch direction and position on TKE is very large, especially in two-phase flow applications and under variable pressure conditions.

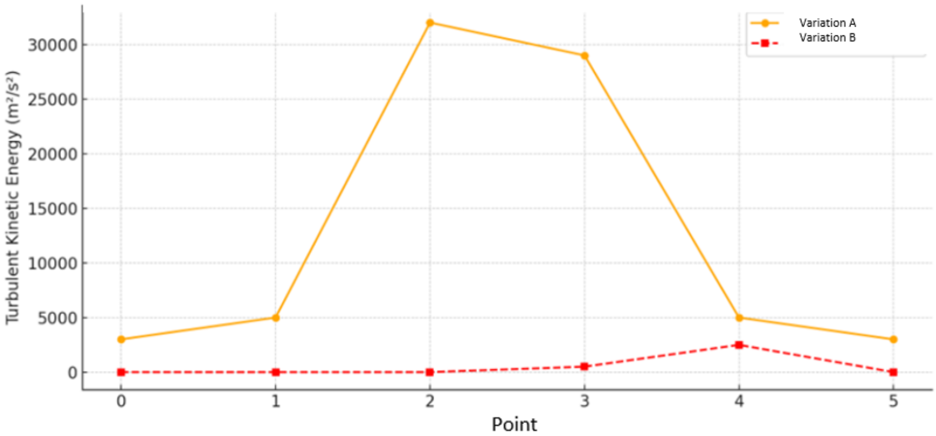


Figure 13. Turbulent Kinetic Energy Chart

Figure 13 shows the comparison of Turbulent Kinetic Energy (TKE) from CFD simulation results of water flow at two speed variations, namely speed A and speed B, along the observation points in the pipe system. It can be seen that at speed variation A, the TKE value increases significantly until it reaches its peak at points 2 and 3 (about 32,000 and 29,000 m^2/s^2), before decreasing dramatically. Meanwhile, in velocity variation B, the turbulent kinetic energy is relatively low, with a maximum value of only about 2,500 m^2/s^2 occurring at point 4.

The striking difference between these two variations indicates that increasing the inlet velocity significantly increases the level of flow turbulence. This is in line with the research of [31] who stated that increasing the fluid velocity tends to increase the velocity gradient and produce higher turbulent energy, especially in areas with geometry changes or branches such as T-shaped pipe joints. Thus, inlet velocity variations in CFD simulations not only affect the turbulent kinetic energy distribution but also provide an overview of potential critical zones in piping systems that need to be considered in the design of engineering fluid systems.

In velocity variation A a comparison between the main parameters (velocity, pressure, TKE, and turbulent viscosity) for each inlet velocity variation. It can be seen that all parameters increase as the inlet velocity increases, indicating that the flow becomes more turbulent, unstable, and more complex fluid interactions occur. This shows that variations in inlet velocity greatly affect the performance of the piping system, both in terms of efficiency and potential long-term damage.

Figure 14 shows that all parameters have increased values at a certain point, especially when there is a change in flow direction and interaction between pipe branches (e.g., at points 1 and 3). The Specific Dissipation Rate shows the highest fluctuation, indicating a location with maximum turbulence intensity. Velocity and Pressure tend to increase gradually as the inlet velocity increases. The TKE and Turbulent Viscosity parameters follow a similar trend, indicating that the greater the inlet velocity, the greater the turbulent energy occurring in the system.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

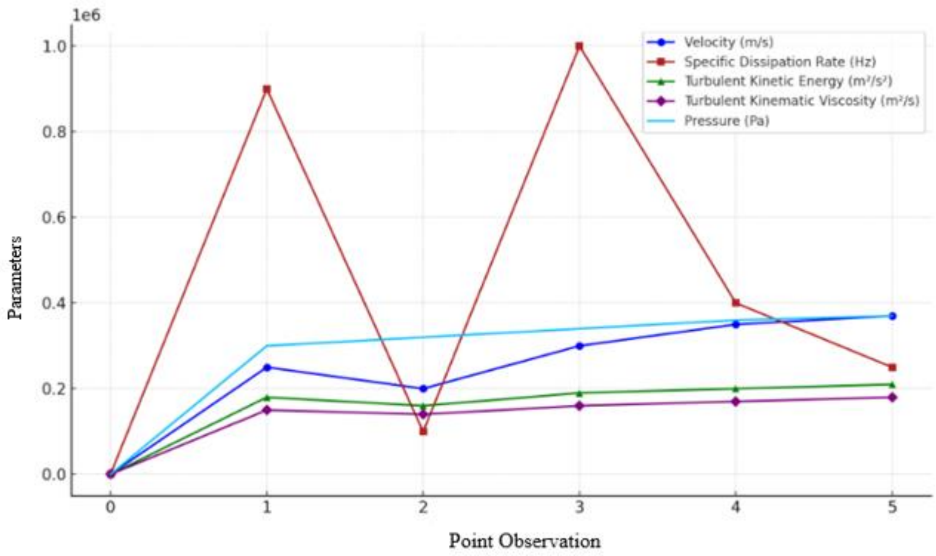


Figure 14. Comparison Chart of Main Parameters of Speed Variation A

This finding is in line with the results of a recent study [19] which states that flow bifurcation at the "T" junction produces flow irregularities in the form of vortices, jet collisions, and wall separation that trigger an increase in SDR and TKE. Research [28] also confirms that the peak values of SDR and TKE are usually at bends or fluid collision points, and are strongly influenced by pipe geometry and boundary conditions. Overall, these graphs demonstrate the importance of a thorough understanding of the interactions between hydrodynamic parameters in the design of pipe joints, especially to reduce energy loss and the risk of damage due to extreme turbulence.

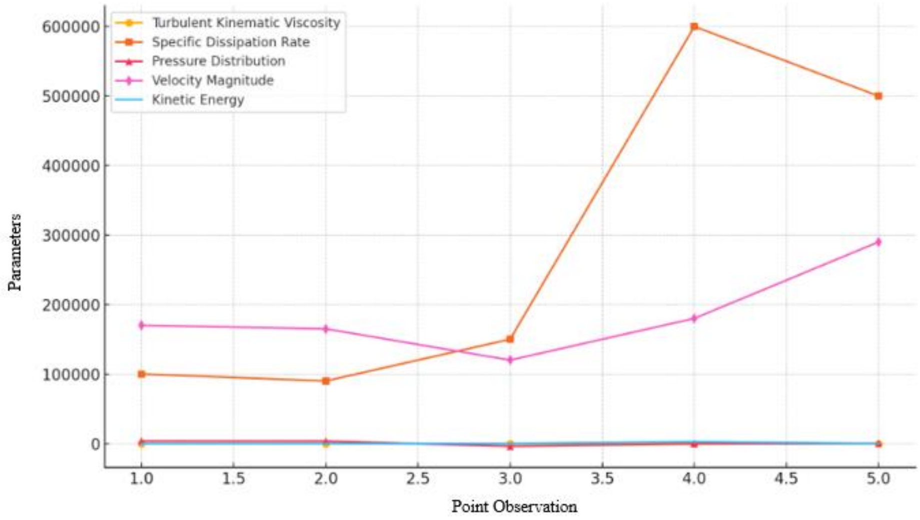


Figure 15. Comparison Chart of Main Parameters of Speed Variation B

Figure 15 shows a comparison of five key parameters-Turbulent Kinematic Viscosity, Specific Dissipation Rate, Pressure Distribution, Velocity Magnitude, and Kinetic Energy-at five observation points for speed variation A. From the graph, it can be seen that the Turbulent Kinematic Viscosity value (orange line) experiences a drastic spike at the 4th observation point, reaching a peak value of over 600,000 before decreasing again at the 5th point. This increase indicates the possible occurrence of locally strong turbulence at that point, which may also trigger an increase in kinetic energy and significant changes in the pressure distribution.

The Specific Dissipation Rate parameter (brown color) shows small fluctuations, indicating that the turbulent energy dissipation rate tends to be stable at all observation points. In the Velocity Magnitude parameter (purple color), a gradual increasing trend is seen up to point 5, which indicates the acceleration of the fluid flow or the influence of external acceleration. Meanwhile, the Kinetic Energy

P-ISSN: 1411-3414
E-ISSN: 2549-9815

(light blue line) has increased consistently up to point 5, which is consistent with an increase in velocity and a possible increase in turbulence. This trend indicates a close relationship between kinetic energy, fluid velocity, and turbulence intensity in the system, as described in a study by [26] on computational fluid dynamics. Overall, this graphical analysis reflects the presence of complex fluid flow dynamics at varying A velocities, with significant interactions between velocity, pressure, and turbulence parameters. A good understanding of these patterns is important in the design of fluid flow systems such as air ducts, piping systems, or aerodynamic devices.

Based on Figure 16, it can be observed that each parameter experiences significant fluctuations along the observation points. The viscosity parameter in both variations shows an increasing trend from point 1 to point 4, which indicates an increase in fluid viscosity due to temperature changes or local turbulence effects. Meanwhile, the fluid pressure shows a significant increase, especially at points 4 and 5, which can be attributed to stagnation or compression zones along the path [26]. For the flow velocity, the highest peak is seen at point 4, which indicates an acceleration of the flow, most likely due to channel constriction or nozzle effects. Turbulent kinetic energy increases along the observation points, with a maximum value at point 5, indicating an increase in turbulence intensity downstream of the flow [32]. In general, the parameters of variation B tend to show higher values, indicating that the flow conditions of B are more complex or more influenced by turbulent factors than those of variation A

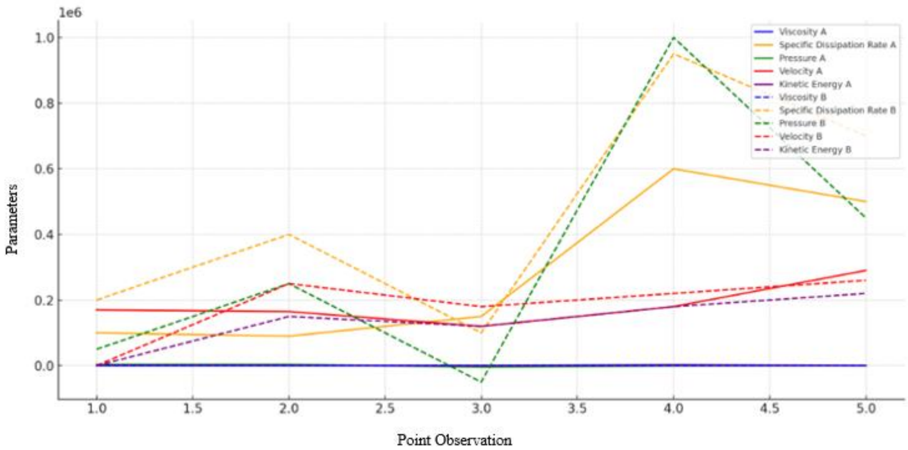


Figure 16. Comparison Chart of Main Parameters of Speed Variations A and B

4. Conclusion

Based on the CFD simulation results displayed through five main parameters (Turbulent Kinematic Viscosity, Specific Dissipation Rate, Pressure Distribution, Velocity Magnitude, and Turbulent Kinetic Energy), it can be concluded that Velocity Variation A produces more turbulent flow conditions compared to Velocity Variation B. This is indicated by the higher values of turbulent kinematic viscosity and specific dissipation rate in Variation A, indicating greater turbulence intensity. This is indicated by higher values of turbulent kinematic viscosity and specific dissipation rate in Variation A, indicating greater turbulence intensity. In addition, the flow velocity (magnitude of velocity) and turbulent kinetic energy were also higher in Variation A, indicating that more flow energy was utilized to form vortices and complex turbulence patterns around the pipe joints. On the other hand, in Velocity Variation B, there is greater pressure accumulation, indicated by the higher pressure distribution, which indicates a decrease in velocity due to flow resistance or possible flow separation. Velocity Variation A excels in almost all turbulent flow parameters, except in the pressure aspect, where Velocity Variation B shows more dominant values. This indicates that higher inlet velocities (in Variation A) accelerate the flow but also increase the turbulence complexity and potential kinetic energy in the pipe system. The variation of inlet velocity greatly affects the distribution and intensity of all flow parameters. Increasing speed not only increases the efficiency of fluid transport but also has the potential to cause stronger turbulence and uneven pressure distribution, so an optimal system design is needed to avoid long-term damage.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

References

[1] B. Sun, Q. Liu, H. Fang, C. Zhang, Y. Lu, and S. Zhu, "Numerical and Experimental Study of Turbulent Mixing Characteristics in a T-Junction System," *Applied Sciences 2020, Vol. 10, Page 3899*, vol. 10, no. 11, p. 3899, Jun. 2020, doi: 10.3390/APP10113899.

[2] J. Duan and X. Huang, "An unsteady RANS study of thermal striping in a T-junction with sodium streams mixing at different temperatures," *Front Energy Res*, vol. 10, Jan. 2023, doi: 10.3389/FENRG.2022.991763.

[3] E. S. Taha, M. A. Abdulwahid, A. M. A. Morad, and Q. A. Maatooq, "Computational Fluid Dynamic Analysis of the Flow though T-junction and Venturi Meter," *Thermal Mechanical Engineering and Fuel Energy Deartmentt*, vol. 1, p. 4, 2022, doi: 10.4108/eai.7-9-2021.2314880.

[4] K. Zhang, Z. Hu, S. Zhu, Y. Wang, W. Wang, and K. Deng, "Study on a new pressure loss model of T-junction for compressible flow with particle image velocimetry test," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 236, no. 2, pp. 273–292, Mar. 2022, doi: 10.1177/09576509211037638.

[5] A. Hamad, S. Mohammed, A. Aftab, and K. A. Ahmad, "Reducing Flow Separation in T-Junction Pipe Using Vortex Generator: CFD Study," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences Journal homepage*, vol. 44, pp. 36–46, 2018.

[6] M. Dafa and F. Labik, "Computer Simulation in Fluid Movement Analysis" *Journal of Science and Mathematics Education*, vol. 1, no. 1, pp. 21–25, Mar. 2025, doi: 10.70716/JOSME.V1I1.153.

[7] P. P. Jati, and D. A. Widyaparaga, "CFD Simulation of Wave Velocity Dynamics of Two-Phase Oil-Water Stratified Wavy Flow in Horizontal Pipe" *Journal of Mechanical Design and Testing*, vol. 3, no. 1, pp. 1–11, Jun. 2021, doi: 10.22146/JMDT.56417.

[8] M. Fathonah Muvariz *et al.*, "Strength Analysis of Elbow 450 Pipe Against Fuel Oil Fluid Flow on Tugboats," *INOVTEK Polbeng*, vol. 14, no. 02, pp. 117–129, Nov. 2024, doi: 10.35314/YVK2EH69.

[9] D. Romanova *et al.*, "Calibration of the k-ω SST Turbulence Model for Free Surface Flows on Mountain Slopes Using an Experiment," *Fluids 2022, Vol. 7, Page 111*, vol. 7, no. 3, p. 111, Mar. 2022, doi: 10.3390/FLUIDS7030111.

[10] N. Sakamoto, T. Hino, H. Kobayashi, and K. Ohashi, "Parameter adaptation of $k-\omega$ SST turbulence model for improving resolution of moderately separated flows around 2D wing and 3D ship hulls via EnKF data assimilation," *Journal of Marine Science and Technology (Japan)*, vol. 29, no. 4, pp. 885–909, Dec. 2024, doi: 10.1007/S00773-024-01026-Y/FIGURES/31.

[11] B. D. Gajbhiye, H. A. Kulkarni, S. S. Tiwari, and C. S. Mathpati, "Teaching turbulent flow through pipe fittings using computational fluid dynamics approach," *Engineering Reports*, vol. 2, no. 1, p. e12093, Jan. 2020, doi: 10.1002/ENG2.12093;WEBSITE:WEBSITE:PERICLES;WGROUP:STRING:PUBLICATION.

[12] P. T. Ndiaye and O. N. Thiam, "Computational fluid dynamics analysis of the influence of velocity at inlet 2 on heat transfer and fluid flow in the mixing elbow," *Journal of Scientific and Engineering Research*, vol. 2024, no. 2, pp. 93–101, Accessed: Jun. 11, 2025. [Online]. Available: www.jsaer.com

[13] Simulation Software Engineering AI in the Cloud SimScale. Accessed: Jun. 11, 2025. [Online]. Available: https://www.simscale.com/

[14] F. Moukalled, L. Mangani, and M. Darwish, "The Finite Volume Method in Computational Fluid Dynamics," vol. 113, 2016, doi: 10.1007/978-3-319-16874-6.

[15] Z. Robison, J. P. Mosele, A. Gross, and S. Lynch, "Numerical investigation of turbulent junction flows," *AIAA Journal*, vol. 59, no. 11, pp. 4642–4659, 2021, doi: 10.2514/1.J059468.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

[16] M. J. Mahanta, A. Gupta, S. S. Bodda, S. G. Cho, and G. So, “Characterizing the cyclic behavior of piping T-joint connections,” *International Journal of Pressure Vessels and Piping*, vol. 211, p. 105284, Oct. 2024, doi: 10.1016/J.IJPVP.2024.105284.

[17] M. Nuruzzaman, W. Pao, H. Ya, M. R. Islam, M. A. Adar, and F. Ejaz, “Simulation Analysis of Thermal Mixing Characteristics of Fluids Flowing through a Converging T-junction,” *CFD Letters*, vol. 13, no. 9, pp. 28–41, Sep. 2021, doi: 10.37934/CFDL.13.9.2841.

[18] C. Dianita, R. Piemjaiswang, and B. 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*, vol. 176, pp. 279–295, Dec. 2021, doi: 10.1016/J.CHERD.2021.10.011.

[19] M. Zhang, Y. Cui, W. An, H. Wang, L. Wang, and S. Liu, “Investigation of the Effect of Side Arm Orientation of the T-Junction on Gas–Liquid Stratified Flow,” *Processes 2023, Vol. 11, Page 2949*, vol. 11, no. 10, p. 2949, Oct. 2023, doi: 10.3390/PR11102949.

[20] M. Luaibi and M. Abdulwahid, “Numerical Analysis by Computational Fluid Dynamic Simulation of Fluid Flow in A T- Junction,” Mar. 2022, doi: 10.4108/EAI.7-9-2021.2314888.

[21] D. Deepak, D. Anjaiah, K. V. Karanth, and N. Y. Sharma, “CFD Simulation of Flow in an Abrasive Water Suspension Jet: The Effect of Inlet Operating Pressure and Volume Fraction on Skin Friction and Exit Kinetic Energy,” *Advances in Mechanical Engineering*, vol. 2012, 2012, doi: 10.1155/2012/186430.

[22] Z. Wang *et al.*, “Statistics of kinetic and thermal energy dissipation rates in vibrational turbulent Rayleigh–Bénard convection with rough surface,” *Numeri Heat Transf A Appl*, 2024, doi: 10.1080/10407782.2024.2379621;WGROUP:STRING:PUBLICATION.

[23] R. Rasooli, O. Dur, and K. Pekkan, “Estimation of pulsatile energy dissipation in intersecting pipe junctions using inflow pulsatility indices,” *AIP Adv*, vol. 11, no. 1, Jan. 2021, doi: 10.1063/5.0014450/1071240.

[24] M. D. Bassett, R. J. Pearson, and D. E. Winterbone, “Calculation of steady flow pressure loss coefficients for pipe junctions,” *Proc Inst Mech Eng C J Mech Eng Sci*, vol. 215, no. 8, pp. 861–882, 2001, doi: 10.1243/0954406011524199.

[25] M. Al Amin, “CFD analysis of velocity distribution and pressure drop in laminar pipe flow,” 2024, Accessed: Jun. 11, 2025. [Online]. Available: http://www.theseus.fi/handle/10024/877564

[26] G. Siqueira de Aquino, R. Silva Martins, M. Ferreira Martins, and R. Ramos, “An Overview of Computational Fluid Dynamics as a Tool to Support Ultrasonic Flow Measurements,” *Metrology 2025, Vol. 5, Page 11*, vol. 5, no. 1, p. 11, Feb. 2025, doi: 10.3390/METROLOGY5010011.

[27] S. Safaruddin, M. Mahmuddin, and A. Tando, “Pressure characteristics of flow passing through vertical pipe bends in radial and tangential directions” *Sultra Journal of Mechanical Engineering (SJME)*, vol. 1, no. 1, pp. 25–32, Oct. 2022, doi: 10.54297/SJME.V1I1.306.

[28] E. S. Taha, M. A. Abdulwahid, A. M. A. Morad, and Q. A. Maatooq, “Computational Fluid Dynamic Analysis of the Flow though T-junction and Venturi Meter,” *Thermal Mechanical Engineering and Fuel Energy Deartmentt*, vol. 1, p. 4, 2022, doi: 10.4108/eai.7-9-2021.2314880.

[29] Computational Fluid Dynamics: A Practical Approach, Jiyuan Tu, Guan Heng Yeoh, Chaoqun Liu, Yao Tao, Google Books. Accessed: Jun. 11, 2025. [Online]. Available: https://books.google.co.id/books?hl=en&lr=&id=_3OyEAAAQBAJ&oi=fnd&pg=PP1&dq=Computational+Fluid+Dynamics:+A+Practical+Approach+(3rd+ed.)&ots=IRZOJQl-fr&sig=HFrMxy8vwp6vYvu77WKbj_ffIVE&redir_esc=y#v=onepage&q=Computational%20Fluid%20Dynamics%3A%20A%20Practical%20Approach%20(3rd%20ed.)&f=false

[30] M. Zhou, J. Li, Z. Qiu, and N. Zhang, “Numerical investigation of thermal-mixing characteristics at vertically oriented T-junction pipelines,” *Int J Heat Fluid Flow*, vol. 106, p. 109292, Apr. 2024, doi: 10.1016/J.IJHEATFLUIDFLOW.2024.109292.

P-ISSN: 1411-3414
E-ISSN: 2549-9815

[31] L. Zhao, J. Chen, and G. Duan, "Turbulent flow in an I-L junction: Impacts of the pipe diameter ratio," *Physics of Fluids*, vol. 36, no. 2, Feb. 2024, doi: 10.1063/5.0189282/3262862.

[32] ANSYS FLUENT 12.0 Theory Guide. Accessed: Jun. 08, 2025. [Online]. Available: https://www.afs.enea.it/project/*neptunius*/docs/fluent/html/th/main_pre.htm