Evaluation of Thermal Mixing in T-Junctions Using Computational Fluid Dynamics (CFD)

Muhammad Usama[1], Zeeshan Khan[2], Faiq Said[2*], Muhammad Ismail[3], Hammad-Ur-Rahman[1]
[1]U.S.-Pakistan Center for Advanced Studies in Energy, University of Engineering & Technology, Peshawar 25124, Khyber Pukhtunkhwa, Pakistan
[2]Laboratory of Fluid Mechanics, Department of Mechanical Engineering, University of Engineering & Technology, Mardan 23200, Khyber Pukhtunkhwa, Pakistan
[3]Department of Technology, Abasyn University, Peshawar 25000, Khyber Pukhtunkhwa, Pakistan
*Correspondence: engr.faiq.said@uetmardan.edu.pk

Citation | Usama. M, Khan. Z, Said. F, Ismail. M, Rahman. H. U, "Evaluation of Thermal Mixing in T-Junctions Using Computational Fluid Dynamics (CFD)", IJIST, Vol. 07 Issue 02, pp 1055-1073, May 2025
Received | May 03, 2025 **Revised |** May 27, 2025 **Accepted |** May 28, 2025 **Published |** May 29, 2025.

The thermal mixing process in T-junctions presents a significant challenge in optimizing heat transfer and temperature distribution, especially in systems involving both hot and cold fluids. The problem addressed in this study was to understand how variations in inlet velocities, pipe diameters, flow rates, and turbulence models affect heat transfer and thermal mixing. The solution was achieved by performing detailed CFD simulations, evaluating these factors under controlled boundary conditions of 40 m/s hot inlet velocity, 30 m/s cold inlet velocity, and a 15 K temperature difference between the main and branch pipes. The results reveal that higher inlet velocities enhance thermal mixing, with outlet temperatures increasing from 223.382 K to 325.975 K as hot inlet velocity increases from 20 m/s to 40 m/s. Increasing the hot inlet diameter from 2 cm to 4 cm improves temperature distribution, raising the outlet temperature from 325.95 K to 329.797 K. The introduction of dual hot inlets further enhances the temperature to 329.797 K. Comparative analysis of turbulence models (k-ω and k-ε) indicates that the k-ω model provides more uniform temperature distribution. Moreover, variations in flow rates show that higher flow rates in the main pipe led to an outlet temperature of 312 K, while higher flow rates in the branch pipe reduced the outlet temperature to 305 K. This research offers critical insights for optimizing T-junction designs, improving thermal mixing, and enhancing heat transfer in industrial applications.

Keywords: T-junction, Thermal Mixing, Computational Fluid Dynamics (CFD), Turbulent Flow, Heat Transfer, ANSYS

Introduction:

Thermal mixing in piping systems is a fundamental phenomenon in many engineering applications involving fluid transport, energy exchange, and process control. It plays a critical role in industries such as nuclear power generation, petrochemical processing, district heating and cooling, and aerospace thermal management. Among various components in these systems, T-junctions, where two fluid streams meet and mix, are particularly significant due to their direct influence on flow distribution, temperature uniformity, and system performance. In nuclear power plants, for example, hot and cold fluid mixing at T-junctions can create steep temperature gradients that may cause thermal fatigue, thermal stratification, and material degradation in pipeline walls. Similarly, in HVAC and industrial fluid systems, improper mixing can lead to inefficient heat transfer, control issues, and localized overheating or undercooling. The complex flow dynamics in T-junctions, often characterized by turbulence, recirculation zones, and unsteady temperature fields, present considerable challenges in accurately predicting thermal behavior. Therefore, a detailed understanding of the heat transfer and fluid mixing mechanisms in T-junction configurations is essential for ensuring both operational safety and thermal efficiency. This study addresses these challenges through computational fluid dynamics (CFD) simulations, focusing on how key factors such as inlet velocity ratios, temperature differentials, and geometric orientation, affect the thermal mixing and flow patterns within a T-junction.

In nuclear power plant cooling systems, the mixing of fluids at different temperatures can result in temperature fluctuations. Incidents such as the pipe crack at the Civaux facility have highlighted how High Cycle Thermal Fatigue (HCTF) can cause significant damage [1] Many studies have explored this issue. The study conducted by the authors [2] on the mixing dynamics in microchannel T-junctions emphasized the importance of fluid dynamics (CFD) in system design. Authors in [3] validated their temperature and velocity predictions through mixing experiments. In another study [4] researchers conducted a study, on mixing in rotating mixers highlighting the impact of rotation speeds, on heat dissipation. Author[5] studied the temperature variances in triple jet setups observing a frequency of temperature fluctuations with differing amplitudes. Author [6] classified jets, in T junctions according to their velocity ratios. A study [7] investigated how temperature variances behave in mixing tees when porous materials are introduced emphasizing the decrease in gradients observed with the inclusion of mediums. Authors in [8] studied the heat dissipation abilities of jets hitting heat sinks showcasing their cooling performance. The combined research efforts improve our knowledge and fine-tune the mixing procedures to ensure the safety of nuclear power plants.

A study [9] established a correlation between measured and simulated coolant temperature variations in a T-junction. Meanwhile, author in a study [10] emphasized the effectiveness of impellers in minimizing temperature fluctuations within T-junctions. Higher velocity in the main pipe reduces backflow in a T-junction, according to research by authors [11]. Authors in [12] looked at how momentum ratios between the main pipe and the branch greatly affect the dynamics of solute mixing in a T junction. Researchers [13] investigated the impact of fluid momentum on thermal mixing behavior at a T junction and the rupture and fatigue that occur in a pipe as a result of hot and cold water mixing. The flow properties and mixing degrees in a T junction were investigated by [14]. An other study [15], on multi-set opposing jets, found a strong agreement between simulation results and experimental measurements. The flow and mixing in a T-junction have been studied. Collectively, these studies contribute significantly to advancing our understanding of thermal mixing behavior in T-junctions is an essential factor for ensuring the safe operation of nuclear power plants and preventing pipe damage caused by thermal fatigue and fluctuations. Authors in [16][17] inquired about the significance of Reynolds number (Re) in inlet flows. In another study, author [18] investigated that the SST turbulence model is good for predicting turbulent flow.

Many researchers worked on thermal mixing and flow dynamics in a T-jsunction which is important for analyzing high-cycle thermal fatigue (HCFT). Moreover, a research [19] explored the working of T-junction in light water reactors and forecasted the anticipated temperature change that arises from turbulence in a T-junction. Three-dimensional turbulent flow analysis is very important for understanding high-cycle thermal fatigue (HCFT) in a T-junction using computational fluid dynamics (CFD).

A research [20], examined the mixing of hot and cold fluids within a T-junction using Particle Image Velocimetry (PIV) and identified an anti-parallel vortex in a horizontal rectangular T-junction. Researcher in [21], studied thermal mixing in a vertical T-junction setup and analyzed three distinct flow patterns. The mixing of fluids with varying velocities and densities inside the T-junction leads to instabilities and fluctuations, causing temperature variations near the pipe walls [22]. Large-scale turbulence is identified as a possible cause of temperature fluctuations that lead to pipe cracking, whereas small-scale turbulence does not significantly affect the pipe material. These findings highlight the need to reduce temperature fluctuations to ensure structural integrity and operational safety.

To measure velocity profiles, the authors [23] utilized LDA and PIV techniques, with temperature data recorded via thermocouples.. Thermal mixing within the T-junction has been observed with a temperature difference of 17 K between the hot and cold fluid streams. Experiments conducted on the Vattenfall T-junction [24][25][26] have revealed that the Reynolds-Averaged Navier-Stokes (RANS) turbulence model is inadequate for accurately predicting the flow behavior within the T-junction. However, even with coarse grid meshes and high Reynolds numbers, Large Eddy Simulation (LES) has been shown to provide accurate predictions of temperature and velocity fields. A detailed review of similar studies has been presented in Table 1.

Objectives:

Based on the identified research gaps, the objectives of this study are as follows:

1. To analyze the effect of varying inlet velocities, inlet diameters, and flow rate ratios on temperature distribution and thermal mixing efficiency in T-junction configurations.
2. Comparing the performance of two widely used turbulence models, k-ε and k-ω, in predicting thermal behavior and flow dynamics within T-junction geometries.
3. To investigate and compare the influence of three-dimensional circular versus rectangular T-junction geometries on heat transfer and fluid mixing under identical boundary conditions.

This study presents an integrated CFD-based investigation that goes beyond isolated parameter analysis, offering a multi-variable exploration of turbulence models, flow conditions, and geometric configurations in T-junctions. A key novelty lies in the direct comparison of the k-ε and k-ω turbulence models under identical operating conditions, revealing the k-ω model's superior ability to capture temperature uniformity and flow detail. Furthermore, the study introduces a 3D comparative analysis of circular and rectangular T-junctions, highlighting the enhanced thermal mixing in rectangular geometries due to increased turbulence and flow separation. Unique to this work is the inclusion of dual hot inlets and parametric variation of hot inlet diameters in conjunction with asymmetric flow rates, addressing a critical research gap in the existing literature. These contributions collectively provide a comprehensive and practically applicable understanding of thermal mixing behavior in complex T-junction flow systems.

Table 1. Similar published studies involving Thermal Mixing and Flow Dynamics in T-junction studies

Author(s)	Methods	Performance	Findings	Limitations	References
Adeosun and Lawal	CFD analysis	Efficient in predicting mixing dynamics in microchannel T-junctions.	Confirmed the significance of CFD in system design.	Limited to microchannel flows; does not explore large-scale applications.	[2]
Zughbi et al.	Experimental validation	Validated temperature and velocity predictions.	Supported CFD predictions with experimental evidence.	Limited exploration of complex flow geometries and conditions.	[3]
Msaad et al.	Experimental study	Effective in analyzing heat dissipation in rotating mixers.	Highlighted the impact of rotation speeds on heat dissipation.	No analysis of non-uniform rotation or complex geometries.	[4]
Cao et al.	Observational study	Measured temperature fluctuations in triple jet setups.	Observed frequency and amplitude of temperature variances.	Focused on specific setups without varying jet configurations.	[5]
Hosseini et al.	Classification study	Categorized jets in t-junctions based on velocity ratios.	Established flow regime classifications for predictable behavior.	No experimental validation of classified regimes.	[6]
Lu et al.	CFD simulations	Reduced temperature gradients using porous materials.	Demonstrated improved thermal mixing efficiency.	Limited to specific porous materials; broader applicability unexplored.	[7]
Rhakasywi et al.	Experimental study	Enhanced cooling performance by optimizing jet placement.	Showcased heat dissipation capabilities of jets hitting heat sinks.	Limited to specific heat sink geometries and flow conditions.	[8]
Hu and Kazimi	Simulation and measurement	Correlation observed between simulated and measured temperature variances.	Validated coolant temperature predictions in T-junctions.	Limited resolution in high-frequency temperature fluctuations.	[9]
Huang et al.	Experimental study	Impellers effectively minimized temperature variations.	Enhanced uniformity in thermal mixing.	Limited scope to specific impeller designs and flow rates.	[10]

Lin and Feng	CFD analysis	Reduced backflow with higher main pipe velocity.	Showed flow stability improvements in T-junctions.	Applicability in multi-phase flows not explored.	[11]
Shao et al.	Simulation-based study	Solute mixing dynamics depended on momentum ratios.	Highlighted the dependence of mixing dynamics on momentum ratios in branch and main pipes.	Did not address non-Newtonian fluid behavior.	[12]
Zhou et al.	Thermal and structural analysis	Identified risks of thermal fatigue and pipe rupture due to mixing.	Demonstrated safety implications for nuclear and industrial piping systems.	No evaluation of advanced materials to mitigate fatigue.	[13]
Sakowitz et al.	Experimental and simulation studies	Improved understanding of mixing degrees and flow characteristics.	Provided insights into thermal mixing behavior.	Does not address high-pressure and high-temperature conditions.	[14]
Wang and Mujumdar	Experimental and simulation studies	Strong agreement between experimental data and simulation results.	Provided key insights for opposing jet configurations in T-junctions.	No exploration of multi-phase or compressible flows.	[15]
Evrim and maurya	Parametric study	Reynolds number significantly affects flow stability in inlet flows.	Highlighted the role of the Reynolds number in controlling turbulence.	Limited analysis of unsteady or transient flow conditions.	[16][17]
Frank et al.	Simulation study	SST turbulence model accurately predicted turbulent flow.	Demonstrated effectiveness in forecasting turbulent behaviors.	Did not evaluate advanced turbulence models for more complex conditions.	[18]
Dahlberg et al.	Study on light water reactors	Explored thermal mixing in T-junctions in nuclear systems.	Forecasted temperature changes caused by turbulence.	Limited to specific reactor configurations; did not address multiphase flow challenges.	[19]
Brückner	Piv and experimental study	Anti-parallel whirlwinds observed in horizontal rectangular T-junctions.	Provided insights into turbulence and mixing behavior.	Focused only on horizontal	[20]

				configurations; limited to specific geometries.	
Kamide et al.	Experimental study	Identified three distinct flow patterns in vertical T-junctions.	Highlighted effects of velocity and density differences on mixing.	Limited to vertical configurations; not generalized for horizontal or angled junctions.	[21]
Large-scale turbulence	Observational and simulation studies	Causes pipe cracking due to temperature fluctuations.	Emphasized the need to minimize temperature fluctuations for safety.	Limited exploration of methods to mitigate large-scale turbulence.	[22]
Smith et al.	Experimental study using PIV and thermocouples	Thermal mixing was observed at a 17K temperature difference.	Validated les as a superior model for predicting behavior over rans.	RANS turbulence model failed to predict realistic behaviors.	[23]

Methodology: The flow of methodology adopted in this study is as under:

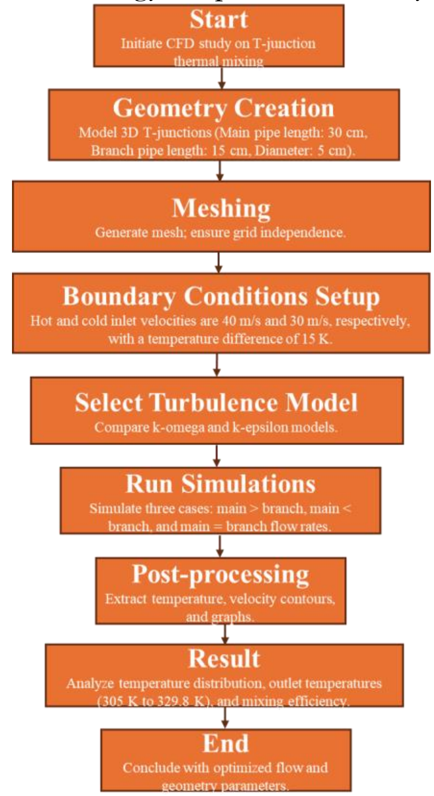


Figure 1. Flow diagram of the computational methodology used in the thermal mixing analysis of T-junctions under varying flow and geometric conditions.

Geometry generation for simulations was carried out using Space Claim and Design Modeler. Space Claim enables sketching, meshing, and modeling, all of which are essential for creating an accurate and efficient design. While on the other hand, Design Modular allowed us to make 3D and 2D geometry methodologically.

In this study, the diameter of the branch pipe, its orientation relative to the main pipe, and the velocity variation in the branch pipe were key geometric factors analyzed to assess thermal performance in a T-junction. To determine the temperature distribution along the main pipe's diameter within the T-junction, both two-dimensional (2D) and three-dimensional (3D) models were utilized. In the 2D simulations, a reference geometry was developed for comparison with other 2D configurations. The reference 2D geometry shown in figure 2 comprises both the main and branch pipes, where the main pipe has a length of 50 cm and a diameter of 5 cm, while the branch pipe measures 15 cm in length with a diameter of 2 cm. Water, serving as the working fluid, flows at a velocity of 30 m/s, which remains consistent in both pipes. Modifications were made to these specifications for comparison with the reference T-junction, such as altering branch pipe velocities, changing branch pipe diameters, incorporating two branch pipes, and varying the turbulence models. Additionally, a 3D analysis was conducted to compare rectangular and circular 3D T-junction configurations.

Numerical Techniques and Empirical Equations:

This study analyzes fluid flow dynamics and thermal mixing in T-junctions using Computational Fluid Dynamics (CFD) simulations with ANSYS Fluent 19.2 R1. The Shear Stress Transport (SST) $k-\omega$ turbulence model has been utilized to accurately forecast turbulent mixing, separation, and recirculation, while the SIMPLE method was employed for pressure-velocity coupling. The inlet velocity profiles in the computational domain are set to match the experimental mass flow rate, and the boundary conditions are assumed to be adiabatic and non-slip. For increased precision, a second-order upwind technique was used to discretize convective and diffusive fluxes. To guarantee accurate residual decay, convergence requirements were chosen at $< 10^{-4}$ for the momentum and continuity equations and $< 10^{-7}$ for the energy equation [27].

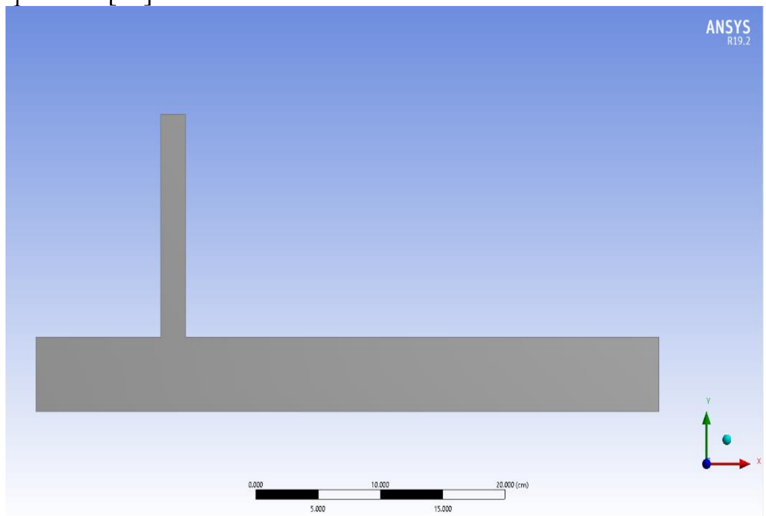


Figure 1. Geometry of basic T-junction (2D)

Governing Assumptions:

To model the three-dimensional turbulent flow within the T-junction, several assumptions have been applied. The flow is treated as a continuum, with no-slip boundary conditions imposed at the walls. It is assumed to be incompressible and characterized by a high Reynolds number; representative of turbulent mixing commonly observed in T-junctions. The fluid flow is considered steady-state, and the walls of the T-junction are assumed to be hydraulically smooth.

Governing Equations:

These simulations are based on the following equations:

Continuity Equation:

$$\nabla \cdot \vec{V} = 0 \qquad (1)$$

Where $\vec{V}$ = (u, v, w) denotes the velocity components along the x, y, and z directions, respectively. This equation enforces the principle of mass conservation by ensuring incompressible flow, which implies no accumulation or depletion of mass within the control volume.

Momentum Equations:

For the x-direction:

$$\rho(u\, \partial u/\partial x + v\, \partial u/\partial y + w\, \partial u/\partial z) = -\partial p/\partial x + \mu \nabla^2 u \qquad (2)$$

Where ρ is the fluid density, p is the pressure, and μ is the dynamic viscosity. Similar equations apply for the y and z directions. These equations describe the momentum transport due to fluid motion and viscous effects.

Energy Equation:

$$\rho C_p\, (u\, \partial T/\partial x + v\, \partial T/\partial y + w\, \partial T/\partial z) = k \nabla^2 T \qquad (3)$$

Where T is temperature, Cp is specific heat capacity at constant pressure, and k is thermal conductivity. This equation captures heat transfer via convection (fluid motion) and conduction (thermal diffusion) within the domain.

These governing equations form the mathematical basis for simulating the flow and thermal mixing in T-junctions under steady-state incompressible conditions.

Grid Independence Analysis and Mesh Resolution Selection:

Grid independence for a 2D T-junction was explored using mesh sizes from 1mm to 3mm to ensure the accuracy of results and minimize mesh-dependent effects. The study-maintained consistency across varying mesh resolutions by keeping the velocity constant at 30 m/s in both the main and branch pipes. The analysis focused on fluid temperature and pipe diameter, particularly along a vertical line located 48 cm from the inlet of the main pipe. ANSYS Fluent simulations confirmed mesh independence, enabling reliable use of any selected mesh size. Consequently, a 2mm mesh size was chosen for continuity in the study.

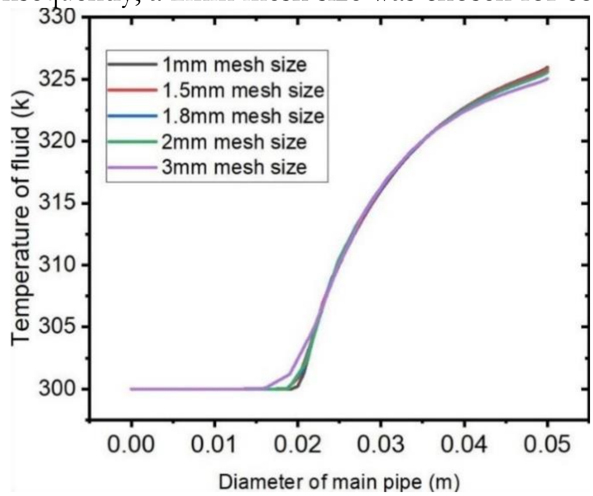


Figure 2. Graph of Grid Independence Analysis

Figure 3 illustrates the relationship between the fluid temperature (K) and the diameter of the main pipe (m) for various mesh sizes. Around the horizontal axis at a diameter of 0.02 m, it is observed that all mesh size curves except the purple curve (representing the 3 mm mesh size) converge closely. Among the remaining curves, corresponding to mesh sizes of 1 mm, 1.5 mm, and 1.8 mm, the differences are negligible. Consequently, a 2 mm mesh size is selected as it balances computational efficiency and accuracy. Selecting finer mesh sizes (1 mm, 1.5 mm, or 1.8 mm) was found to increase simulation time without yielding significant improvements in the results, as the solution had become independent of the mesh size at this level.

Justification for Selected Flow Rate Cases:

The three flow rate cases, Case 1, Case 2, and Case 3, were strategically selected to investigate the influence of momentum ratio on thermal mixing performance within a T-junction, a key factor contributing to high-cycle thermal fatigue (HCTF) in piping systems.

- **Case 1** (Main: 1 m/s, Branch: 0.5 m/s) represents a low momentum ratio scenario in which the main flow dominates. This condition increases the likelihood of branch flow back penetration, resulting in unstable mixing and elevated near-wall temperature fluctuations, phenomena known to accelerate HCTF.
- **Case 2** (Main: 1 m/s, Branch: 1 m/s) reflects a balanced momentum ratio, enabling the analysis of symmetric thermal and hydrodynamic interactions. This setup allows for the evaluation of mixing dynamics under idealized, evenly matched inlet conditions and supports comparisons with the symmetric behavior observed in previous studies such as Hosseini et al. [6].
- **Case 3** (Main: 1 m/s, Branch: 2 m/s) simulates a high momentum ratio dominated by the branch flow. In this configuration, the injected jet significantly alters the flow structure, promoting flow separation, recirculation, and potential hot spot formation, behavior consistent with patterns reported by Zhou et al. [13] and Lin and Ferng [11].

By encompassing low, moderate, and high momentum ratio conditions, these cases were chosen to capture a comprehensive range of thermal mixing regimes and to serve as validation scenarios for comparison with established experimental and numerical findings in the literature.

Computational Resources:

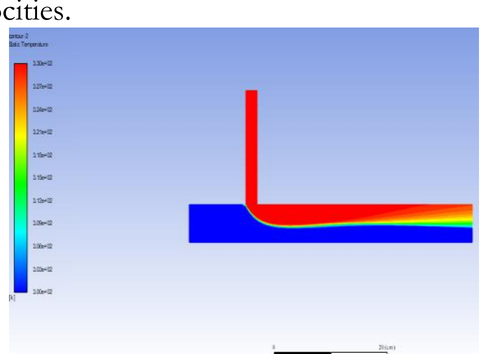
All CFD simulations were performed using ANSYS Fluent 19.2 at the U.S.-Pakistan Center for Advanced Studies in Energy (USPCAS-E), University of Engineering and Technology, Peshawar, Pakistan. The computations were conducted on a high-performance computing (HPC) cluster equipped with dual Intel® Xeon® processors, 128 GB RAM, and parallel processing enabled across 16 cores. Double precision was activated to ensure numerical accuracy, and the residual convergence criterion was set to 1×10^{-6} for all governing equations. Depending on the mesh resolution, flow conditions, and turbulence models applied, typical simulation runtimes ranged from 6 to 10 hours.

Results and Analysis:

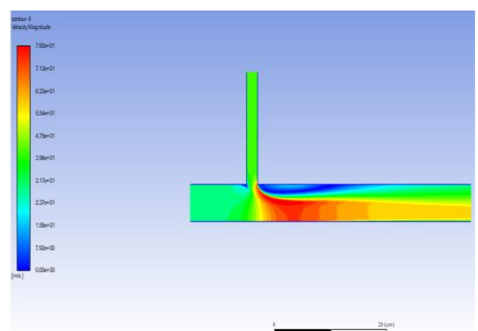
This investigation explored the temperature and velocity profiles within the flow field by analyzing data from the thermal interaction of heated and chilled fluid streams. A comparative study was carried out to evaluate the influence of the Mass Ratio (Mr) on the thermal blending properties and flow dynamics within a T-junction. To thoroughly examine the relationship between the mixing jet, flow rate ratio, and their impact on thermal blending and mixing length, the Thermal Mixing Degree (TMD) was used as a quantitative indicator to assess the efficiency of thermal mixing.

Exploring Turbulent Flow Dynamics: Differential Inlet Velocities and Temperature Distribution in T-Junction Cases:

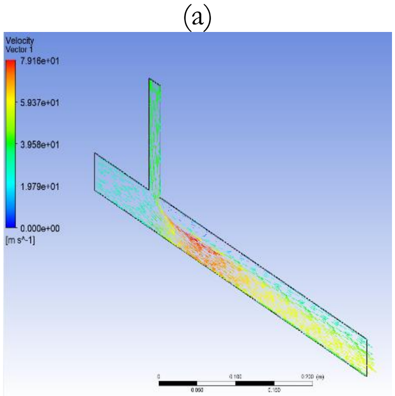
In the T-junction analysis, with the hot inlet velocity set at 40 m/s and the cold inlet at 30 m/s, the k-omega turbulence model illustrated diverse temperature and velocity contours, demonstrating the effects of varying inlet velocities (Figure 4). Higher velocity at the hot inlet expanded temperature distribution due to enhanced turbulence, unlike equal velocities.



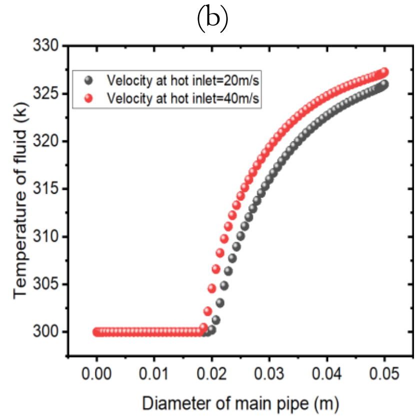
(a)



(b)



(c)



(d)

Figure 3. Results for case 1 a). Temperature Distribution contour b). Velocity Distribution contour c). Velocity Vector d). Temperature Distribution graph

When the hot inlet velocity was reduced to 20 m/s while the cold inlet remained at 30 m/s, the temperature distribution decreased accordingly. (Figure 4). At 20 m/s, the main pipe's outlet temperature dropped to 223.382 K compared to 325.975 K at 30m/s, highlighting the sensitivity of temperature distribution to velocity changes. This analysis underscores the critical role of inlet velocities in system dynamics.

The comparison reveals the complex link between inlet velocities and temperature distribution in our T-junction study. Detailed contour plots and temperature graphs highlight the importance of variable inlet velocities for understanding thermal dynamics. Visualization illustrates how velocity adjustments affect temperature distribution.

Exploring Thermal Effects: Impact of Hot Inlet Diameter Variation on Temperature Distribution in T-Junction Flow:

In the investigation, the hot inlet diameter was increased from 2 cm to 4 cm while maintaining a constant velocity of 30 m/s in both the hot and cold inlets, to analyze its impact on the temperature distribution at the outlet of the main pipe. Figure 5 depicts temperature and velocity contours, revealing notable changes due to the modification. The temperature distribution graph demonstrated a substantial rise along the main pipe's outlet vertical line with the increase in diameter, from 325.95 K to 329.797 K, attributed to improved mixing and heat transfer in larger diameter pipes. This exploration underscores the direct relationship between inlet diameter adjustments and temperature variations, emphasizing the significance of geometric parameters in shaping T-junction system thermal characteristics.

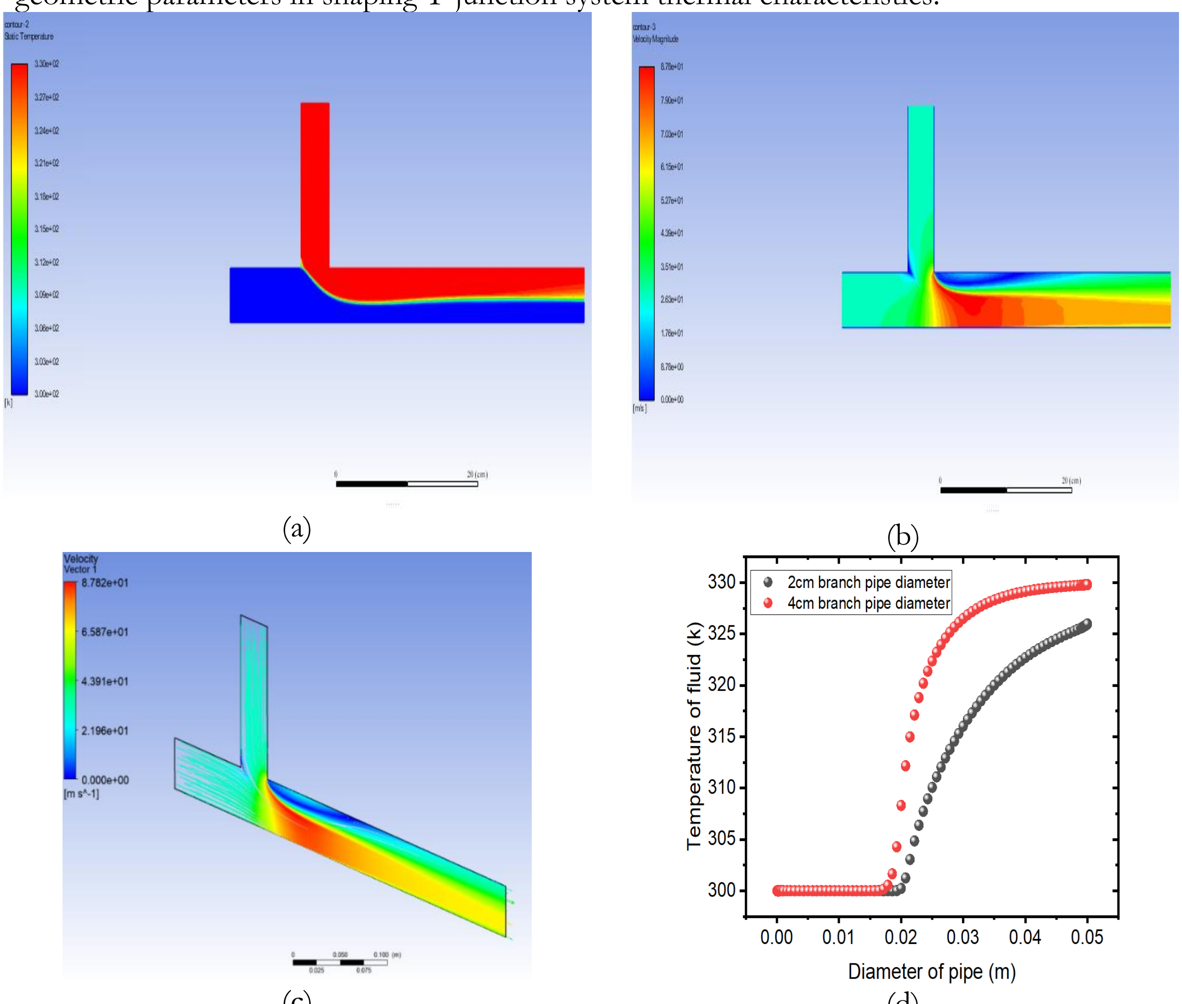


Figure 4. Results for case 2 a). Temperature Distribution contour b). Velocity Distribution contour c). Velocity Vector d). Temperature Distribution graph

Enhancing Heat Transfer: Impact of Dual Hot Inlets on Temperature Distribution in T-Junction Flow:

In the study of heat transfer enhancement in T-junction flows, a critical adjustment involves installing two hot inlets, departing from the previous single inlet configuration. Figure 6 compares temperature and velocity contours, highlighting the distinctive characteristics introduced by dual hot inlets. The temperature distribution graph compellingly contrasts T-junctions with two hot inlets to those with a single inlet, showing a notable increase in outlet temperature from 325.975 K to 329.797 K with dual inlets. This enhancement resulted from the increased heat input, highlighting the significant impact of dual hot inlets on temperature dynamics and demonstrating the potential for improved heat transfer in the context of our research.

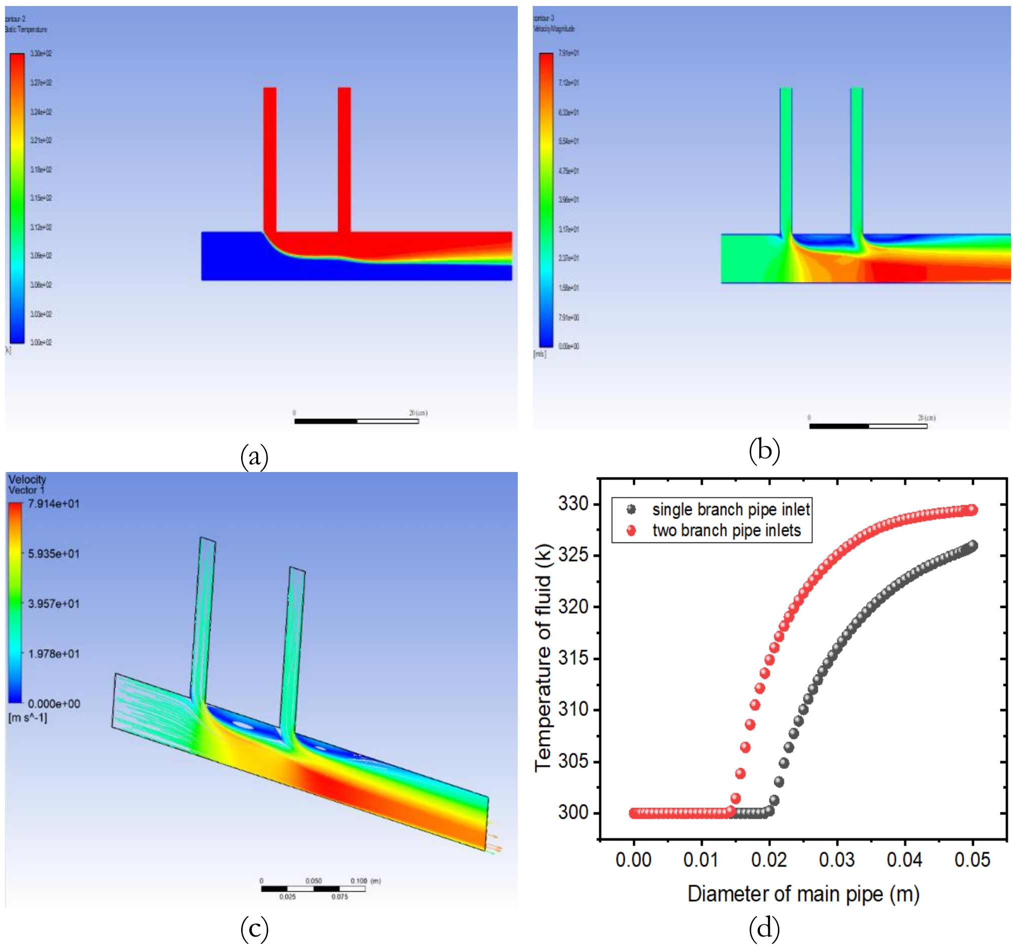


Figure 5. Results for case 3 a). Temperature Distribution contour b). Velocity Distribution contour c). Velocity Vector d). Temperature Distribution graph

Unveiling Turbulence Dynamics: Comparative Analysis of Temperature Distribution with k-epsilon and k-omega Models in T-Junction Flow:

In an extensive examination of turbulence dynamics in T-junction flows, the fourth phase introduces a pivotal shift from the k-omega to the k-epsilon turbulence model. This deliberate alteration aims to unveil the impact on temperature distribution at the main pipe's outlet, specifically along the vertical direction within the T-junction configuration. Figure 7 visually captures temperature and velocity contours under the influence of the k-epsilon model, offering a comparative analysis with the k-omega model. Figure 7d, which displays the temperature distribution graph, clearly showed a more uniform temperature profile with the k-omega model, underscoring its effectiveness in capturing turbulence where hot and cold fluids interact. This finding highlighted the crucial role of turbulence model selection in accurately simulating temperature dynamics in T-junctions.

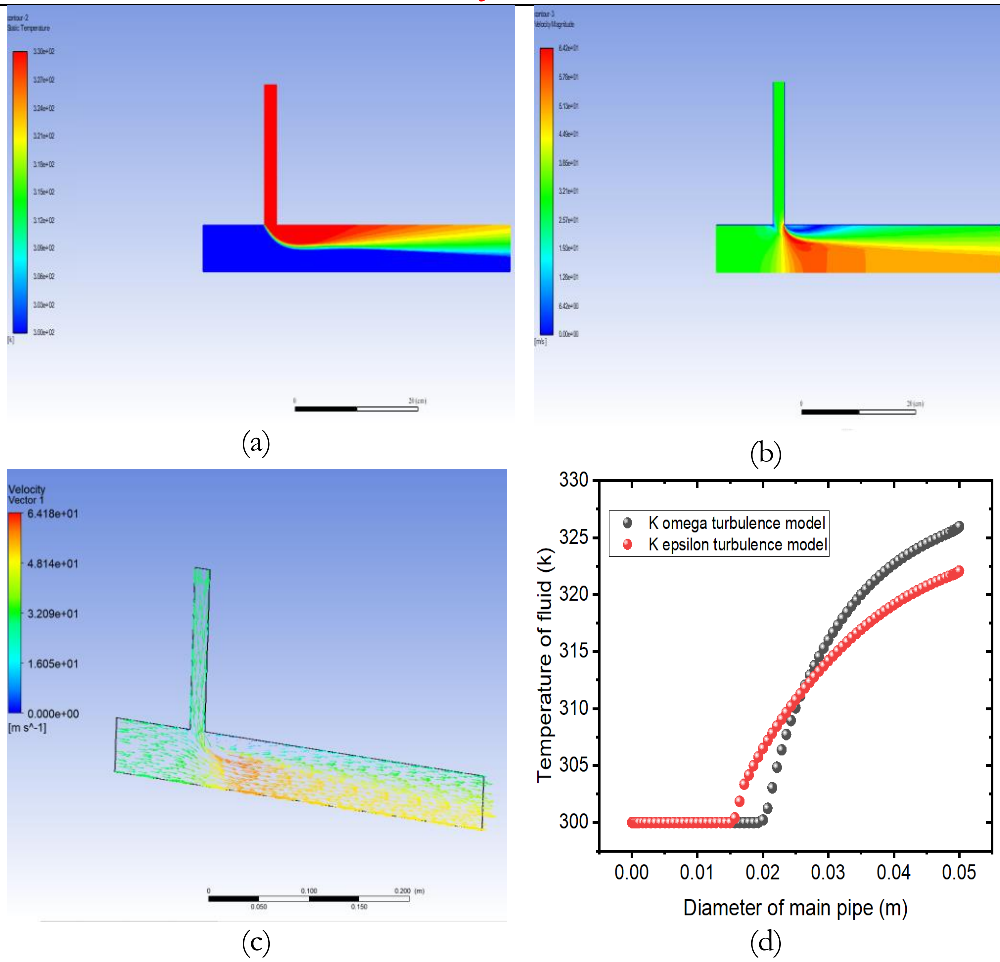


Figure 6. Results for case 4 a). Temperature Distribution contour b). Velocity Distribution contour c). Velocity Vector d). Temperature Distribution graph

Comparative Analysis of Circular vs. Rectangular T-Junctions in 3D Flow and Heat Transfer:

Figures 8 and 9 show that the circular T-junction produced a centralized thermal plume with relatively uniform velocity distribution. In contrast, Figure 10 reveals that the rectangular T-junction led to greater thermal stratification and asymmetrical mixing, particularly near the outlet region, due to sharp corner-induced flow separation. These differences highlight the influence of junction geometry on thermal and hydrodynamic behavior under identical inlet conditions.

Different flow rates in T-Junction (when the branch pipe is below the main pipe):

To gain a deeper understanding of the temperature distribution in a T-junction, three different flow rates were tested. The T-junction geometry was created using a modeling tool for this analysis. The main pipe had a length of 30 cm, while the branch pipe was 15 cm long, and both pipes had a diameter of 5 cm. In the first case, the flow rate in the main pipe exceeded that in the branch pipe. In the second case, the flow rate in the main pipe was lower than in the branch pipe. In the third case, the flow rates in both pipes were identical. The temperature difference between the hot fluid entering the main pipe and the cold fluid entering the branch pipe was maintained at 15 K.

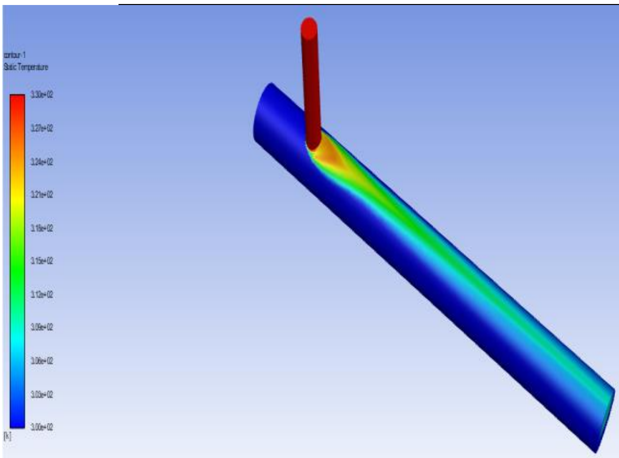


Figure 7. Circular T junction temperature contour in 3D analysis

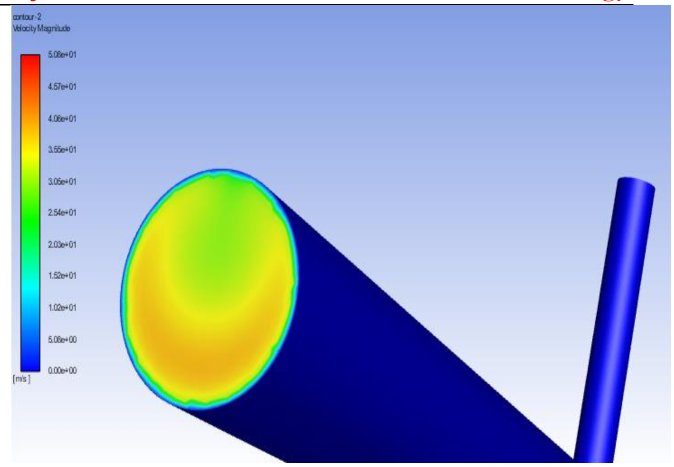


Figure 8. The circular T junction's velocity contour

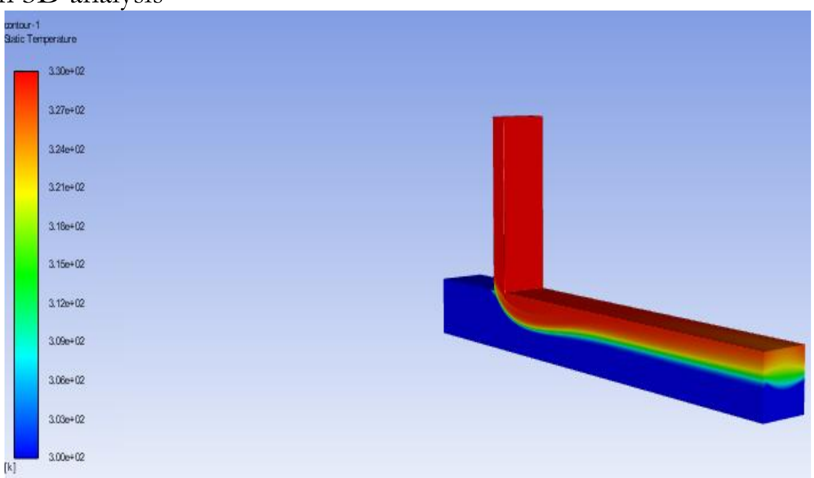


Figure 9. 3D Temperature Contour Analysis of Rectangular T-Junction

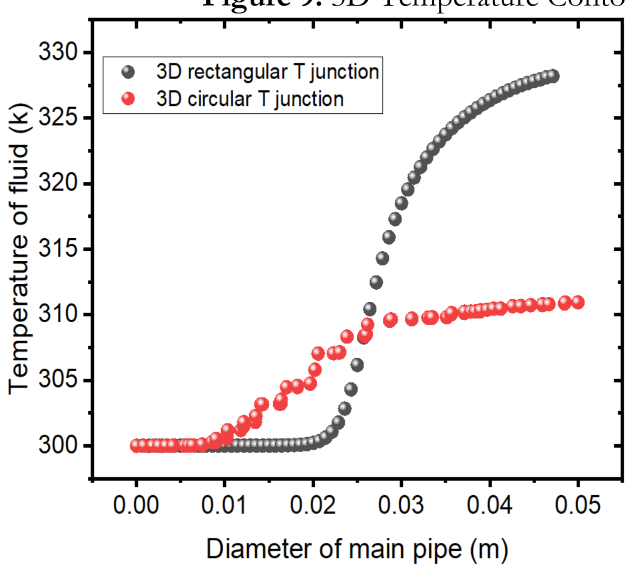


Figure 11. Diameter of a Pipe

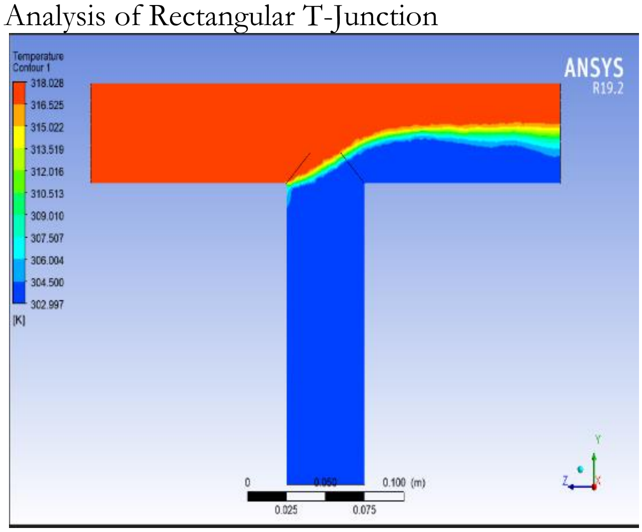


Figure 10. Temperature distribution contour for case 1

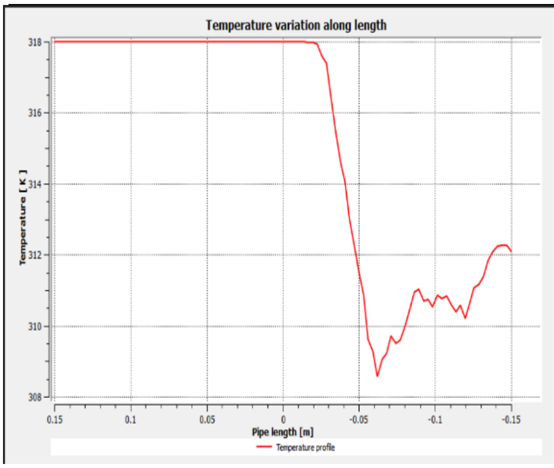


Figure 11. Temperature distribution graph for case 1

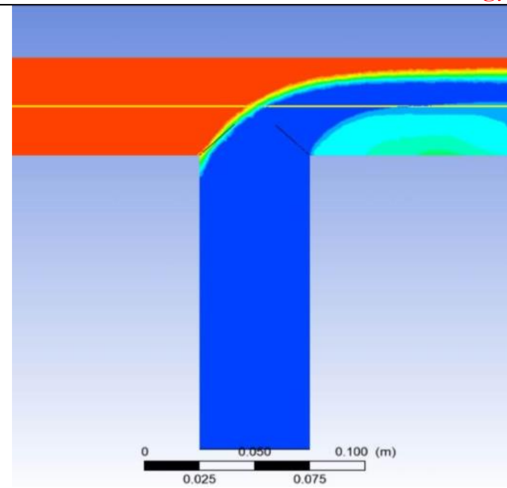


Figure 12. shows that the temperature is 312K at the outlet, after mixing hot and cold fluid

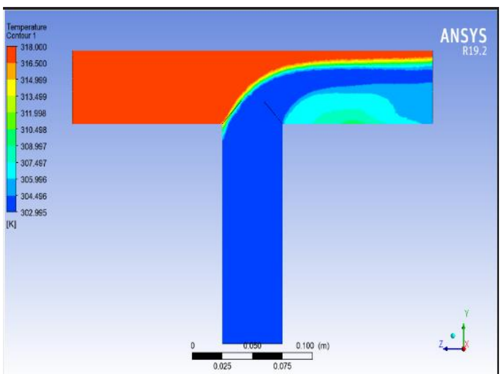


Figure 13. Temperature distribution contour for case 2

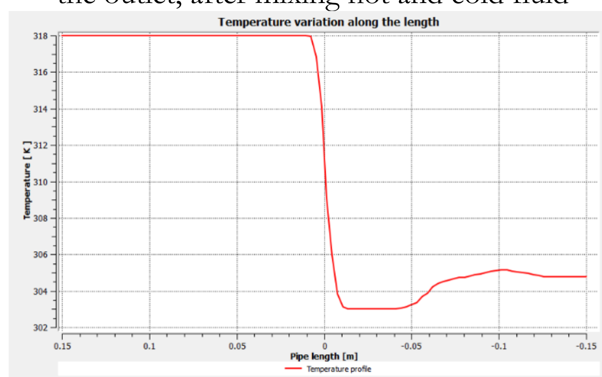


Figure 14. Temperature distribution graph for case 2

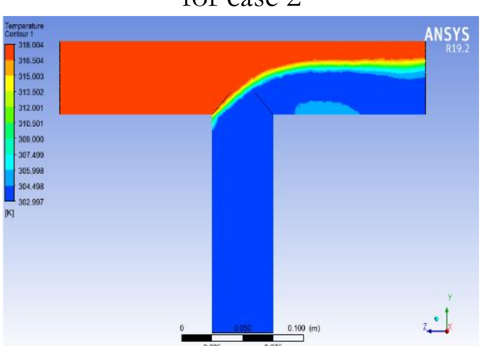


Figure 15. Temperature distribution contour for case 3

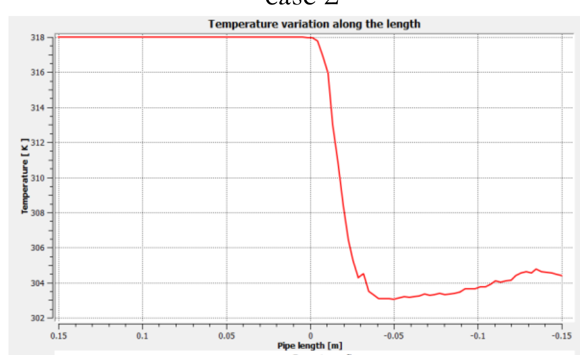


Figure 16. Temperature Profile for Case 3

Case 1: In case 1 flow rate in the main pipe was higher than the branch pipe of a T-Junction. An effect on temperature distribution along the length of the main pipe is given in Figure 12 and Figure 13.

From these figures, the temperature distribution along the length of the main pipe was clarified. The temperature distribution was investigated along the centerline (yellow line) of the main pipe in the T-junction, as shown in Figure 14.

Case 2:

In Case 2, the flow rate in the main pipe was lower than that in the branch pipe of the T-junction. The impact on temperature distribution along the main pipe is illustrated in Figures 15 and 16. The temperature at the outlet is 305 K along the centerline of the main pipe as shown in figure 15. The flow rate of the cold inlet is high therefore cold fluid is dominant (fluid at the main pipe is laminar while fluid at the branch pipe is turbulent) at the centerline along which temperature distribution is investigated.

Case 3:

In Case 3, the flow rates in both the main and branch pipes were identical, each maintained at 0.3 kg/s. The effect of this balanced flow condition on temperature distribution along the length of the main pipe is illustrated in Figure 17 and Figure 18.

Statistical Validation of Results:

To ensure the reliability of the CFD predictions, statistical error analysis was conducted using Root Mean Square Error (RMSE) and Mean Absolute Percentage Error (MAPE). For the simulated temperature profiles, the RMSE was calculated as ±4.2 K, while the MAPE remained below 7.5% when compared with experimental data reported [23] [3]. These metrics indicate a strong correlation between the numerical results and experimental benchmarks. Furthermore, Grid Convergence Index (GCI) analysis was employed to verify mesh independence, with GCI values remaining below 5%, thereby confirming the adequacy of the grid resolution and the numerical accuracy of the simulations.

Discussion:

The findings of the present study reinforce and extend previous investigations into thermal mixing behavior within T-junction configurations, particularly in high-Reynolds-number flow regimes relevant to nuclear thermal-hydraulics and industrial heat transfer systems.

The predicted temperature distributions and mixing patterns show strong agreement with the computational and experimental results ofauthors. [5] [7], particularly with the influence of geometric and flow parameters on mitigating thermal gradients. Like their observations, the present study confirms that optimized inlet conditions can significantly reduce temperature stratification, thereby enhancing thermal homogenization.

The velocity fields and turbulence structures predicted using the SST k–ω turbulence model are consistent with the computational results reported [2], as well as with the experimental validations presented. [3]. This alignment supports the continued application of the SST model in capturing flow separation, recirculation zones, and anisotropic turbulence effects, findings that are further substantiated by author[18].

Moreover, the present findings are consistent with those reported by the authors [11], demonstrating that higher inlet velocities in the main pipe effectively suppress reverse flow and enhance mixing stability. The impact of momentum ratios on the extent and efficiency of thermal mixing is also in agreement [12], where author emphasized the importance of momentum interplay in driving convective transport and improving heat transfer uniformity.

The study also reproduces thermal fluctuation behaviors associated with fatigue risks [13], confirming that large-scale turbulence plays a critical role in generating temperature instabilities, an effect previously highlighted by [22]. This validates concerns about thermal fatigue in real-world applications such as nuclear T-junctions and pipeline systems.

Finally, while this study primarily utilizes RANS-based turbulence models, the observed discrepancies in small-scale fluctuation capture reinforce the superiority of Large Eddy Simulation (LES) in resolving detailed thermal and velocity fields, as demonstrated [23]. This highlights a potential direction for future research aiming at higher-fidelity turbulence resolution.

The comparative alignment of this study's findings with existing literature not only validates the numerical approach but also contributes to a broader understanding of thermal mixing dynamics. These insights offer practical guidance for mitigating thermal fatigue and optimizing heat transfer in critical engineering systems.

Conclusion:

The research investigates strategies for enhancing thermal interaction and temperature uniformity in T-junctions, with a focus on the mixing of hot and cold fluid streams. The main challenge lies in understanding how variations in inlet velocities, pipe diameters, turbulence models, and flow rates impact heat transfer efficiency and temperature uniformity. To overcome this, the research utilizes detailed Computational Fluid Dynamics (CFD) simulations under controlled boundary conditions. A 40 m/s hot inlet speed, a 30 m/s cold inlet speed, and a 15 K temperature differential between the main and branch pipes were the parameters used in the simulation.

- **Hot Inlet Velocity's Effect:** The results demonstrated that increasing the hot inlet velocity from 20 m/s to 40 m/s led to a significant rise in the outlet temperature, from 223.382 K to 325.975 K. The higher inlet velocity improved thermal mixing, as faster fluid flow enhances the interaction between hot and cold fluids, thereby improving heat transfer.
- **Impact of hot inlet diameter:** The study found that increasing the hot inlet diameter from 2 cm to 4 cm resulted in an increase in the outlet temperature from 325.95 K to 329.797 K. The larger diameter allows more fluid to flow through, which contributes to better thermal mixing by enhancing the heat transfer between the fluids.
- **Dual hot inlets:** Introducing dual hot inlets further improved the thermal mixing, resulting in an outlet temperature of 329.797 K. This configuration helped distribute the hot fluid more uniformly across the junction, leading to enhanced heat transfer and better temperature distribution.
- **Turbulence models:** The comparison between two turbulence models, k-ω and k-ε, revealed that the k-ω model provided better results for temperature distribution. The simulations using the k-ω model showed more uniform temperature profiles and better prediction of heat transfer behavior. As a result, the k-ω model was identified as more suitable for simulating turbulent flow in T-junctions, as it effectively captured the complex flow dynamics.
- **Flow rate variations:** The impact of flow rates on temperature dispersion was investigated. The exit temperature increased to 312 K as the main pipe's flow rates increased. Conversely, a lower exit temperature of 305 K was the consequence of higher flow rates in the branch pipe. These findings emphasize the significance of maximizing flow rates in the main and branch pipes to improve heat transfer efficiency and attain efficient temperature distribution.
- **Circular vs. rectangular T-junctions:** A comparative analysis of circular and rectangular T-junctions showed that the rectangular configuration provided superior thermal mixing. The larger surface area and more abrupt changes in flow direction within the rectangular T-junction resulted in greater turbulence, which in turn enhanced heat transfer. As a result, the outlet temperature in the rectangular T-junction was higher, showcasing its advantage for improving thermal performance.

This research provides significant insights into the factors influencing thermal mixing and temperature distribution in T-junctions. The findings underscore the importance of optimizing parameters such as inlet velocity, inlet diameter, turbulence model, and flow rate to enhance heat transfer and ensure more uniform temperature distribution. The use of the k-ω turbulence model, the introduction of dual hot inlets, and the optimization of the T-junction geometry (particularly the rectangular design) all contribute to improved thermal performance. The results also highlight the importance of balancing flow rates in both the main and branch pipes for optimal temperature distribution. By bridging the gap between theoretical analysis and practical implementation, this work offers valuable recommendations for the design and optimization of T-junctions in industrial applications. The study paves the way for further research aimed at refining these configurations to achieve even more efficient thermal mixing and heat transfer.

References:

[1] A. M. S. Chapuliot, C. Gourdin, T. Payen, J.P. Magnaud, "Hydro-thermal-mechanical analysis of thermal fatigue in a mixing tee," *Nucl. Eng. Des.*, vol. 235, no. 5, pp. 575–596, 2005, doi: https://doi.org/10.1016/j.nucengdes.2004.09.011.

[2] J. T. Adeosun and A. Lawal, "Numerical and experimental studies of mixing characteristics in a T-junction microchannel using residence-time distribution," *Chem. Eng. Sci.*, vol. 64, no. 10, pp. 2422–2432, 2009, doi: https://doi.org/10.1016/j.ces.2009.02.013.

[3] H. D. Zughbi, Z. H. Khokhar, and R. N. Sharma, "Mixing in pipelines with side and opposed tees," *Ind. Eng. Chem. Res.*, vol. 42, no. 21, pp. 5333–5344, Oct. 2003, doi: 10.1021/IE0209935;PAGEGROUP:STRING:PUBLICATION.

[4] K. O. A. Ait Msaad, M. Mahdaoui, T. Kousksou, A. Allouhi, T. El Rhafiki, A. Jamil, "Numerical simulation of thermal chaotic mixing in multiple rods rotating mixer," *Case Stud. Therm. Eng.*, vol. 10, pp. 388–398, 2017, doi: https://doi.org/10.1016/j.csite.2017.09.005.

[5] Q. Cao, D. Lu, and J. Lv, "Numerical investigation on temperature fluctuation of the parallel triple-jet," *Nucl. Eng. Des.*, vol. 249, pp. 82–89, 2012, doi: https://doi.org/10.1016/j.nucengdes.2011.07.018.

[6] S. M. Hosseini, K. Yuki, and H. Hashizume, "Classification of turbulent jets in a T-junction area with a 90-deg bend upstream," *Int. J. Heat Mass Transf.*, vol. 51, no. 9–10, pp. 2444–2454, 2008, doi: https://doi.org/10.1016/j.ijheatmasstransfer.2007.08.024.

[7] H. L. T. Lu, P.X. Jiang, Z.J. Guo, Y.W. Zhang, "Large-eddy simulations (LES) of temperature fluctuations in a mixing tee with/without a porous medium," *Int. J. Heat Mass Transf.*, vol. 53, no. 21–22, pp. 4458–4466, 2010, doi: https://www.sciencedirect.com/science/article/abs/pii/S0017931010003650?via%3Dihub.

[8] R. I. Damora Rhakasywi, Harinaldi, Engkos A. Kosasih, "Computational and Experimental Study of Heat Transfer on the heat sink with an impinging synthetic jet under Various Excitation Wave," *Case Stud. Therm. Eng.*, vol. 26, p. 101106, 2021, doi: https://doi.org/10.1016/j.csite.2021.101106.

[9] L.-W. Hu and M. S. Kazimi, "LES benchmark study of high cycle temperature fluctuations caused by thermal striping in a mixing tee," *Int. J. Heat Fluid Flow*, vol. 27, no. 1, pp. 54–64, 2006, doi: https://doi.org/10.1016/j.ijheatfluidflow.2005.08.001.

[10] Q. W. Kexin Huang, Bo Su, Tong Li, Hanbing Ke, Mei Lin, "Numerical simulation of the mixing behaviour of hot and cold fluids in the rectangular T-junction with/without an impeller," *Appl. Therm. Eng.*, vol. 204, p. 117942, 2022, doi: https://doi.org/10.1016/j.applthermaleng.2021.117942.

[11] Y. M. F. C.H. Lin, "Investigating thermal mixing and reverse flow characteristics in a T-junction using CFD methodology," *Appl. Therm. Eng.*, vol. 102, pp. 733–741, 2016, doi: https://doi.org/10.1016/j.applthermaleng.2016.03.124.

[12] C. S. Yu Shao, Y. Jeffrey Yang, Lijie Jiang, Tingchao Yu, "Experimental testing and modeling analysis of solute mixing at water distribution pipe junctions," *Water Res.*, vol. 56, pp. 133–147, 2014, doi: https://doi.org/10.1016/j.watres.2014.02.053.

[13] Z. Mi, R. Kulenovic, and E. Laurien, "T-junction experiment with high temperature and high pressure to investigate flow rate influence on mixing characteristics," *Int. J. Heat Fluid Flow*, vol. 71, pp. 451–459, 2018, doi: https://doi.org/10.1016/j.ijheatfluidflow.2018.05.004.

[14] A. Sakowitz, M. Mihaescu, and L. Fuchs, "Effects of velocity ratio and inflow pulsations on the flow in a T-junction by Large Eddy Simulation," *Comput. Fluids*, vol. 88, pp. 374–385, 2013, doi: https://doi.org/10.1016/j.compfluid.2013.10.001.

[15] S. J. Wang and A.S. Mujumdar, "Flow and mixing characteristics of multiple and multi-set

opposing jets," *Chem. Eng. Process. Process Intensif.*, vol. 46, no. 8, pp. 703–712, 2007, doi: https://doi.org/10.1016/j.cep.2006.09.006.
[16] C. Evrim and E. Laurien, "Numerical study of thermal mixing mechanisms in T-junctions," *Appl. Therm. Eng.*, vol. 183, no. 1, p. 116155, 2021, doi: https://doi.org/10.1016/j.applthermaleng.2020.116155.
[17] A. Maurya, N. Tiwari, and R. P. Chhabra, "Thermal Mixing of Impinging Laminar Streams of Shear-Thinning Fluids," *Heat Transf. Eng.*, vol. 41, no. 18, pp. 1576–1595, Oct. 2020, doi: 10.1080/01457632.2019.1661667.
[18] F. M. Th. Frank, C. Lifante, H.-M. Prasser, "Simulation of turbulent and thermal mixing in T-junctions using URANS and scale-resolving turbulence models in ANSYS CFX," *Nucl. Eng. Des.*, vol. 240, no. 9, pp. 2313–2328, 2010, doi: https://doi.org/10.1016/j.nucengdes.2009.11.008.
[19] J. Dahlberg, M ; Chapuliot, D ; Bretherton, I ; Faidy, C ; Church, M ; Chapuliot, S ; Wilke, U ; Taylor, N ; Solin, "Development of a European procedure for assessment of high cycle thermal fatigue in light water reactors: Final report of the NESC-thermal fatigue project," *Publ. Off. Eur. Union*, 2007, [Online]. Available: https://op.europa.eu/en/publication-detail/-/publication/f74e08dd-b6f1-45ec-b985-a378067f16af
[20] C. Brücker, "Study of the three-dimensional flow in a T-junction using a dual-scanning method for three-dimensional scanning-particle-image velocimetry (3-D SPIV)," *Exp. Therm. Fluid Sci.*, vol. 14, no. 1, pp. 35–44, 1997, doi: https://doi.org/10.1016/S0894-1777(96)00110-0.
[21] K. H. H. Kamide, M. Igarashi, S. Kawashima, N. Kimura, "Study on mixing behavior in a tee piping and numerical analyses for evaluation of thermal striping," *Nucl. Eng. Des.*, vol. 239, no. 1, pp. 58–67, 2009, doi: https://doi.org/10.1016/j.nucengdes.2008.09.005.
[22] A. E. A. and S. A. K. Aleksandr V. Obabko, Paul F. Fischer, Timothy J. Tautges, Vasily M. Goloviznin, Mikhail A. Zaytsev, Vladimir V. Chudanov, Valeriy A. Pervichko, "Large Eddy Simulation of Thermo-Hydraulic Mixing in a T-Junction," *Nucl. React. Therm. Hydraul. Other Appl.*, 2013, doi: 10.5772/53143.
[23] A. Blahoianu and A. Huerta, "OECD/NEA Activities to Support Long Term Operation," *J. Disaster Res.*, vol. 5, no. 6, pp. 707–711, Dec. 2010, doi: 10.20965/JDR.2010.P0707.
[24] Thomas Höhne, "Scale resolved simulations of the OECD/NEA–Vattenfall T-junction benchmark," *Nucl. Eng. Des.*, vol. 269, pp. 149–154, 2014, doi: https://doi.org/10.1016/j.nucengdes.2013.08.021.
[25] H. Ayhan and C. N. Sökmen, "CFD modeling of thermal mixing in a T-junction geometry using LES model," *Nucl. Eng. Des.*, vol. 253, pp. 183–191, 2012, doi: https://doi.org/10.1016/j.nucengdes.2012.08.010.
[26] K. A. B.L. Smith, J.H. Mahaffy, "A CFD benchmarking exercise based on flow mixing in a T-junction," *Nucl. Eng. Des.*, vol. 264, pp. 80–88, 2013, doi: https://doi.org/10.1016/j.nucengdes.2013.02.030.
[27] R. D. Lazarov, I. D. Mishev, and P. S. Vassilevski, "Finite volume methods for convection-diffusion problems," *SIAM J. Numer. Anal.*, vol. 33, no. 1, pp. 31–55, Jul. 1996, doi: 10.1137/0733003;WGROUP:STRING:PUBLICATION.