CFD Analysis of Heat Transfer Performance in a Parallel-Flow Double-Pipe Heat Exchanger with Interchanged Hot and Cold Fluids Between Inner and Outer Pipes

Shani Singh[1], Dr. Satnam Singh[2]

[1]Research Scholar, Dept. of Mechanical Engineering P.K University, Shivpuri, (M.P.)
[2]Professor, Dept. of Mechanical Engineering, P.K University, Shivpuri, (M.P.)

Abstract:
This study explores the heat-transfer performance of a parallel-flow double-pipe heat exchanger by evaluating how interchanging the hot and cold fluids between the inner and outer pipes affects thermal behaviour. The topic is significant because the hot fluid enters at 355 K and the cold fluid at 298 K, creating a temperature gradient that determines the rate and direction of heat transfer. The methodology involves developing a detailed computational model, generating a refined mesh with boundary-layer inflation, and applying steady-state CFD simulations that capture conjugate heat transfer and turbulence effects while both fluids flow at a uniform velocity of 1.9 m/s. Temperature contours and area-weighted outlet temperatures are extracted to assess how fluid placement influences the overall effectiveness of the exchanger under identical operating conditions. The conclusions indicate that placing the hot fluid in the inner pipe results in stronger radial heat-transfer gradients and a larger temperature reduction, while positioning it in the outer pipe leads to a comparatively smaller drop, demonstrating reduced thermal effectiveness. These results confirm that fluid placement significantly impacts exchanger performance and that assigning the higher-temperature fluid to the inner pipe enhances heat-transfer efficiency. Overall, the study highlights the importance of strategic fluid arrangement and reinforces the value of CFD-based evaluation for optimizing the design and thermal behaviour of parallel-flow double-pipe heat exchangers.

Keywords: Heat-Transfer, Parallel-Flow, Double-Pipe, Heat Exchanger, Hot and Cold Fluids, Temperature Gradient, Steady-State CFD Simulations and Area-Weighted Outlet Temperatures.

1. INTRODUCTION

Heat exchangers are indispensable components of modern thermal systems, facilitating the efficient transfer of heat between two or more fluids that are at different temperatures without allowing them to mix directly. Their role spans across a variety of sectors, including power generation, chemical and petrochemical processing, refrigeration, HVAC systems, and waste heat recovery operations. In these applications, heat exchangers contribute significantly to the enhancement of energy efficiency, operational economy, and environmental sustainability. Among the numerous types of heat exchangers, the double-pipe heat exchanger stands out for its structural simplicity, flexibility, and ease of maintenance. This device typically consists of two concentric pipes—an inner pipe that carries one fluid and an annulus between the inner and outer pipes through which the other fluid flows. The thermal exchange occurs across the common wall separating these two flows, thereby enabling heat transfer without fluid mixing (Apparao, 2019).

This exchanger type can operate under two fundamental flow configurations: counterflow and parallel flow. In a counterflow arrangement, the hot and cold fluids move in opposite directions, allowing for maximum thermal efficiency. In contrast, the parallel-flow configuration, where both fluids enter from the same end and travel in the same direction, exhibits a gradually decreasing temperature difference along the tube's length. Although this configuration is thermally less effective than the counterflow type, it offers smoother temperature gradients, simple construction, and stable mechanical performance, making it suitable for applications requiring controlled temperature profiles and lower stress levels. The optimization of double-pipe exchangers often involves analytical, numerical, and experimental approaches that consider flow rate, material properties, pipe dimensions, and geometric design to improve heat transfer efficiency and durability (Parekh & Chavda, 2014).

1.1. Overview of Heat Exchangers and Their Industrial Importance

Heat exchangers are central to numerous industrial processes, serving as energy recovery and temperature control units that directly impact process efficiency and cost-effectiveness. Their design allows for the conservation of thermal energy by transferring heat from hot to cold streams, thus reducing fuel consumption and minimizing environmental emissions. Among their many configurations, double-pipe heat exchangers are especially advantageous for operations involving moderate pressures, compact designs, and small flow rates. They are often used in process industries such as food manufacturing, chemical production, and petroleum refining, where thermal regulation is essential for maintaining product quality and process stability (Gabir & Alkhafaji, 2021).

The versatility of double-pipe exchangers lies in their adaptability to various working conditions and fluid types. Their modular structure permits easy scaling, while the concentric pipe arrangement provides a straightforward path for heat exchange, minimizing potential leakage or cross-contamination between the two fluids. Additionally, these exchangers are simple to disassemble for cleaning or inspection, which makes them highly desirable in industries where fouling or contamination is a major concern, such as in dairy or pharmaceutical applications.

In addition to their structural advantages, the energy-saving capability of double-pipe exchangers has made them integral to modern sustainable engineering. They are widely implemented in systems designed for waste heat recovery, where excess thermal energy from one process is repurposed to preheat incoming fluids in another process, thereby improving overall system efficiency. Recent research and industrial innovations have focused on enhancing these exchangers' performance through improved materials, enhanced surface geometries, and flow optimization techniques, all of which contribute to maximizing heat transfer while reducing energy losses (Tongkratoke et al., 2019).

1.2. Principles of Heat Transfer in Double-Pipe Exchanger Systems

The fundamental mechanism governing the operation of a double-pipe heat exchanger is based on conduction through the separating wall and convection within both fluid streams. Heat is transferred from the hot fluid flowing through one section (typically the inner pipe) to the colder fluid flowing through the other section (the annulus). The rate of this heat transfer depends on several variables, including the temperature difference between the fluids, the thermal conductivity of the wall material, the flow velocity, and the turbulent or laminar nature of the flow. According to the first law of thermodynamics, the heat lost by the hot fluid equals the heat gained by the cold fluid, ensuring an energy balance throughout the exchanger's length (Bartecki, 2015).

In a parallel-flow arrangement, both fluids travel in the same direction. This causes the largest temperature difference at the inlet, which gradually decreases along the flow path. The diminishing temperature gradient limits the maximum heat transfer rate compared to a counterflow system, but it also creates a smoother and more uniform temperature distribution across the wall. Such characteristics are particularly advantageous when temperature uniformity is critical, or when the materials involved are sensitive to sudden temperature changes that could cause thermal stress or fatigue.

The flow regime within the exchanger—laminar or turbulent—plays a critical role in determining the overall heat transfer performance. Turbulent flow enhances convective mixing, thereby increasing the heat transfer coefficient, while laminar flow provides stable yet lower heat transfer efficiency. Computational and experimental investigations have expanded the understanding of these dynamics, enabling accurate prediction and optimization of heat transfer coefficients. Recent advances in Computational Fluid Dynamics (CFD) and transfer function modeling have made it possible to visualize fluid behavior, assess flow patterns, and identify temperature distribution trends under a wide range of operating conditions (Bartecki, 2020).

1.3. Parallel-Flow Configuration: Characteristics and Limitations

In a parallel-flow double-pipe heat exchanger, both the hot and cold fluids enter the exchanger at the same end and flow in the same direction, exiting together at the opposite end. This configuration is widely used because it ensures a uniform wall temperature throughout the exchanger, reducing the risk of thermal stresses that could otherwise lead to material fatigue or damage. The even temperature distribution is especially beneficial for systems that must operate under steady thermal loads or where delicate materials require gradual heating or cooling (Ebieto et al., 2020).

However, one of the principal drawbacks of the parallel-flow configuration is its lower thermal efficiency compared to counterflow designs. Since both fluids move in the same direction, the temperature difference between them continuously decreases along the length of the exchanger. This reduced driving force results in a lower average temperature difference and, consequently, a lower rate of heat transfer. Nonetheless, this disadvantage is often offset by the configuration's simplicity, predictable temperature profiles, and mechanical stability.

Parallel-flow exchangers are frequently employed in applications where moderate heat recovery, ease of fabrication, and maintenance accessibility are prioritized. For instance, industries such as food processing or pharmaceuticals favor this configuration because it provides smoother temperature changes, reducing the risk of thermal degradation of heat-sensitive substances. Additionally, the lower efficiency is often acceptable in processes where high-temperature approaches are unnecessary, and compactness or cost reduction is desired (Martin, 2019).

1.4. Significance of Fluid Swapping in the Inner Tube

The placement of fluids—specifically which one occupies the inner or outer tube—has a substantial impact on the overall thermal and hydraulic performance of a double-pipe heat exchanger. This concept, known as fluid swapping, involves alternating the hot and cold fluids between the inner and outer pipes to optimize thermal efficiency, minimize pressure losses, or control fouling tendencies. The selection depends largely on the thermophysical properties of the working fluids, such as viscosity, density, and specific heat capacity.

When the hot fluid is placed in the inner tube, the heat transfer rate generally increases due to higher wall temperatures, resulting in enhanced convective effects. However, this setup also increases the risk of fouling and scaling, especially when the hot fluid contains dissolved salts or particulates that may precipitate on the hot surfaces. Conversely, when the cold fluid occupies the inner tube, fouling risk diminishes, and temperature distribution tends to be more uniform. This configuration is often preferred in long-term industrial operations where ease of maintenance and system longevity are critical (Sharma et al., 2020).

Empirical studies have demonstrated that fluid swapping can significantly affect outlet temperatures, temperature gradients, and even flow-induced vibrations within the system. The choice of which fluid to assign to the inner or outer pipe must consider not only thermal performance but also the pressure drop and pumping power requirements, as the smaller cross-sectional area of the inner tube can increase flow resistance for high-viscosity fluids. Consequently, fluid swapping serves as a design optimization technique that allows engineers to tailor heat exchanger performance for specific industrial applications (Sunu et al., 2016).

1.5. Thermal Performance Parameters and Analytical Approaches

The evaluation of a double-pipe heat exchanger's performance relies heavily on analyzing temperature distribution and heat transfer behavior along its length. In a parallel-flow system, the temperature difference between the hot and cold fluids is greatest at the inlet and gradually decreases toward the outlet, leading to an exponential decline in heat transfer rate along the tube. The performance of such exchangers is typically characterized by measuring the temperature at several points, from which the overall heat transfer coefficient and heat duty can be derived. Analytical models developed from energy balance equations are employed to predict these temperature variations and assess exchanger performance (Okafor et al., 2016).

Experimental and simulation-based investigations complement these analytical models by visualizing fluid dynamics and temperature fields under various conditions. For example, Computational Fluid Dynamics (CFD) simulations enable researchers to model flow structures, evaluate turbulence intensity, and identify zones of high or low heat transfer activity. Such studies have shown that small variations in flow rate or inlet temperature can significantly alter thermal profiles, confirming the sensitivity of double-pipe exchangers to operating parameters.

To enhance thermal efficiency, recent research has explored the introduction of internal surface modifications, such as ribs, fins, or spiral grooves, which promote turbulence and increase the effective heat transfer area. These enhancements provide a practical means of boosting performance without increasing the exchanger's size or energy consumption (Kumar et al., 2018).

2. PROBLEM STATEMENT

Efficient thermal management remains a crucial challenge in industrial heat exchanger systems, particularly in compact configurations where space and material limitations constrain performance. The parallel-flow double-pipe heat exchanger, though structurally simple and easy to fabricate, often experiences reduced heat-transfer efficiency due to the decreasing temperature difference between fluids along its length. Moreover, the influence of fluid placement—whether the hot or cold fluid occupies the inner pipe—can alter convective behavior, wall temperature distribution, and fouling tendencies, thereby affecting the exchanger's overall thermal performance. Despite its widespread use in laboratories and process industries, limited research exists on quantifying how fluid swapping within the inner tube influences temperature distribution, outlet temperatures, and overall heat-transfer effectiveness under identical flow and material conditions. This study addresses this gap using Computational Fluid Dynamics (CFD) in ANSYS Fluent to provide a comparative analysis of both configurations and to optimize exchanger performance through simulation-based insights.

3. OBJECTIVES

1. To analyze the heat transfer performance of a parallel-flow double-pipe heat exchanger when hot and cold fluids are interchanged between the inner and outer pipes.
2. To compare temperature distribution, outlet temperatures, and overall heat transfer efficiency for both fluid arrangements.

4. LITERATURE REVIEW

Apparao (2019) conducted a computational fluid dynamics (CFD) analysis of a double-pipe heat exchanger to investigate fluid temperature profiles and overall heat transfer performance. The study revealed that CFD models can closely replicate experimental data, making them effective tools for analyzing the influence of velocity, inlet temperature, and flow direction on exchanger performance. Similarly, Parekh and Chavda (2014) performed an experimental and exergy analysis, demonstrating that although parallel-flow exchangers exhibit slightly lower efficiency compared to counterflow ones, they offer smoother thermal gradients and reduced material stress. Bartecki (2015) introduced a transfer function-based model to represent the exchanger's thermal dynamics, emphasizing that linear system theory can accurately simulate the transient response of symmetric heat exchangers. Later, Bartecki (2020) expanded on this by developing a rational transfer function model for parallel-flow designs, enhancing the accuracy of dynamic heat transfer predictions. Joshi (2021) further validated CFD simulation reliability by demonstrating that flow symmetry and geometry strongly influence temperature fields. Collectively, these studies highlight that CFD modeling and mathematical simulation are indispensable for understanding the symmetrical behavior of double-pipe exchangers and optimizing their geometry for balanced and efficient heat transfer.

Gabir and Alkhafaji (2021) presented a comprehensive review of double-pipe heat exchanger techniques and emphasized that material selection, geometry, and flow structure play crucial roles in achieving optimal performance. Their findings revealed that aluminum and copper are highly efficient due to their excellent thermal conductivity and low fouling tendencies. Martin (2019) experimentally studied turbulent water flow in double-pipe exchangers, reporting that optimizing Reynolds numbers significantly improves outlet temperature uniformity and overall heat transfer. Similarly, Gangwar et al. (2019) investigated hydrothermal performance with helical tape inserts, demonstrating a 20% increase in heat transfer efficiency due to enhanced mixing. Kumar et al. (2018) complemented these results by performing a numerical analysis that showed internal surface grooves promote turbulence and increase Nusselt numbers, which leads to improved thermal effectiveness without a large rise in pressure drop. Together, these studies emphasize that the integration of computational optimization with experimental validation ensures symmetry in temperature distribution, energy conservation, and mechanical stability across double-pipe exchanger systems, enhancing their practical industrial performance.

Ebieto et al. (2020) designed and constructed a laboratory-scale double-pipe heat exchanger to experimentally validate the relationship between geometry and thermal performance, finding that dimensional precision enhances symmetry in temperature distribution. Sharma et al. (2020) analyzed heat transfer in a double-pipe exchanger using DOT 4 brake fluid and showed that viscosity and flow rate have significant effects on outlet temperatures and thermal balance. Expanded on this concept by comparing circular and oval cross-sections, concluding that oval designs improved heat transfer by up to 28% while maintaining lower pressure drops. In a similar direction, Anantha et al. (2021) used CFD to evaluate helical baffle designs, observing that baffles increased turbulence intensity, thereby boosting heat transfer by 5–8% with manageable pressure losses. Supporting evidence from Tongkratoke et al. (2019) demonstrated that optimized tube geometry, including grooved and finned surfaces, contributes to improved temperature uniformity. Collectively, these structural innovations highlight that symmetry in design—achieved through geometric refinement, internal baffles, or surface modifications—significantly enhances performance in parallel-flow exchangers without compromising operational efficiency.

Okafor and Tagbo (2016) explored the influence of material selection on double-pipe heat exchangers and found that aluminum and copper tubes exhibit superior thermal conductivity and uniform wall temperatures, which support symmetrical heat distribution in parallel-flow systems. Nwokolo et al. (2020) extended this by integrating a biomass energy source with a double-pipe exchanger and reported that symmetrical geometry between the inner and outer tubes ensured balanced temperature profiles and minimized heat loss. Hamzah and Nima (2019) investigated the use of metal foam fins and demonstrated that they enhance convective heat transfer while preserving temperature uniformity across the pipe length. Deb et al. (2022) used CFD to analyze secondary coolant behavior and verified that symmetric configurations yield consistent conjugate heat transfer and steady temperature gradients. Bartecki (2020) confirmed that maintaining symmetry in physical and computational models reduces numerical instability and improves predictive reliability. Overall, these studies collectively affirm that material selection, geometric alignment, and symmetric design are fundamental to achieving high thermal efficiency, stability, and long-term reliability in double-pipe parallel-flow exchangers.

International Journal on Science and Technology (IJSAT)

E-ISSN: 2229-7677 ● Website: www.ijsat.org ● Email: editor@ijsat.org

5. METHODOLOGY

CFD simulations were performed in ANSYS Fluent to analyze heat transfer inside a parallel-flow double-pipe heat exchanger. Two working fluids were used: a hot fluid at 355 K and a cold fluid at 298 K. The inner and outer pipes were modeled using thermally conductive material to ensure efficient heat transfer between the two flowing streams. Two operating cases were simulated by swapping the hot and cold fluids between the inner pipe and the outer pipe. The overall process included geometry creation, mesh generation, boundary-condition assignment, solver setup, and post-processing of the CFD results.

5.1. Geometry Creation and Preparation

The geometry was created based on the specified dimensions:

- Inner pipe: **20 mm inner diameter**, **1 mm wall thickness**
- Outer pipe: **28 mm inner diameter**, **1 mm wall thickness**
- Total heat-exchanger length: **300 mm**

The model included two fluid domains (inner pipe and outer pipe regions) and two solid domains representing the pipe walls. The complete model was imported into ANSYS Fluent Meshing using the Watertight Geometry workflow. Share Topology was enabled to ensure that all contacting surfaces shared common nodes, which is necessary for accurate conjugate heat-transfer modeling between the fluids and the solid walls.

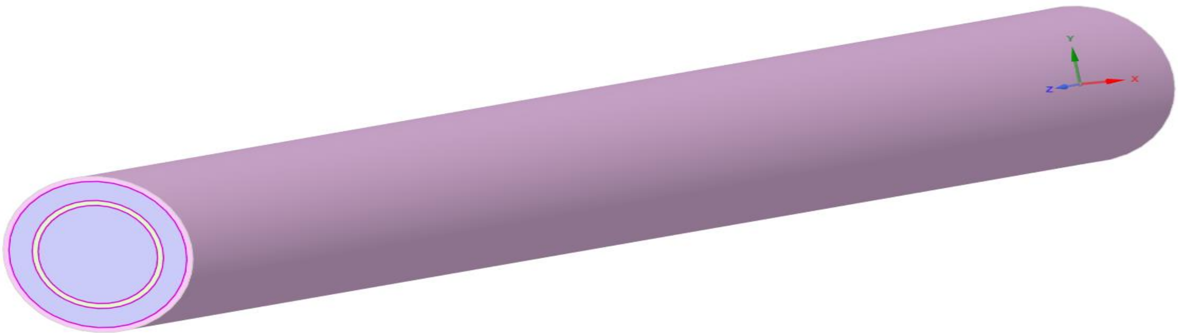

Figure 1: Geometry

5.2. Meshing and Boundary Layer Setup

A high-quality mesh was generated using local sizing controls to refine areas near the walls. The mesh size was set to around 4 mm with a growth rate of 1.2 to keep smooth transitions. A detailed surface mesh was created first using curvature and proximity functions so that both circular pipes and the narrow annulus were accurately captured. Three layers of inflation (boundary layers) were added on all fluid-wall surfaces to resolve velocity and temperature gradients close to the pipe walls. A polyhedral volume mesh was finally generated, providing good convergence and accurate heat-transfer prediction for internal flow.

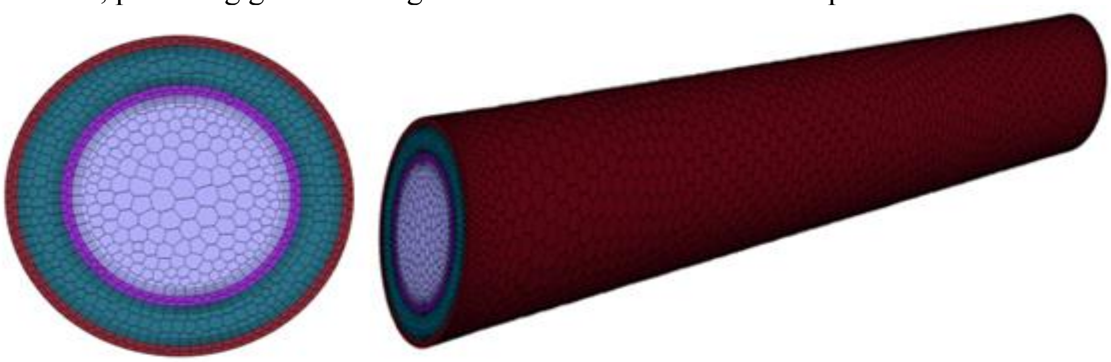

Figure 2: Meshing

5.3. Inlet and outlet

In parallel flow, both the hot fluid at 350 K and the cold fluid at 298 K enter the heat exchanger from the same side and flow in the same direction. Each fluid exits at the opposite end, allowing heat to transfer gradually along the tube length.

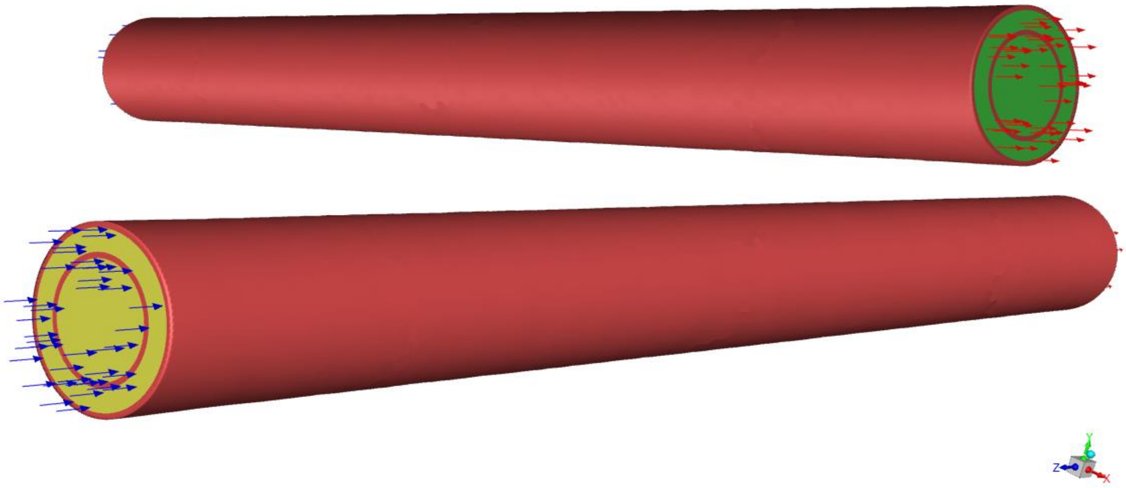

Figure 3: Inlet and outlet

5.4. Boundary Conditions and Region Setup

All regions were identified in Fluent as either fluid or solid. The inner pipe fluid and outer annulus fluid were assigned as fluid zones, while the pipe walls were assigned as solid regions. The inlets of both pipes were defined as velocity inlets, where the temperature was set to either 298 K or 355 K depending on the case being simulated. The outlets of the pipes were set as pressure outlets with zero gauge pressure. No-slip boundary conditions were applied on all solid walls. This setup allowed Fluent to properly simulate conjugate heat transfer between the flowing fluids and the solid pipe walls.

5.5. Solver Setup and Simulation

The simulation was run in **steady-state mode** using the following solver settings:

- **Pressure-based solver**, suitable for incompressible internal flow
- **Energy equation** enabled for heat-transfer prediction
- **Standard k–ε turbulence model** for modeling turbulence inside the pipes
- **Material properties:**
- Water for fluid regions
- Solid pipe walls
- **Discretization schemes:**
- Second-order upwind for most equations
- Turbulent kinetic energy automatically switched to *first-order upwind* (as noted in transcript)
- **Hybrid initialization** was used to start the solution

5.6. Post-Processing

After solving, temperature contours, velocity distributions, and area-weighted outlet temperatures were extracted from Fluent. The results were recorded at the inner pipe inlet and outlet, as well as at the outer annulus inlet and outlet. These values were used to compare the thermal performance of the heat exchanger under both fluid-swapping cases. The contour plots clearly show how heat transfers along the pipe and how outlet temperatures differ between the two cases.

6. RESULTS

The numerical results obtained from the CFD simulations provide insight into the thermal performance of the parallel-flow double-pipe heat exchanger under two different operating conditions. The analysis focuses on:

1. The configuration in which the hot fluid flows through the inner pipe while the cold fluid flows through the outer pipe.
2. The configuration in which the hot and cold fluids are interchanged between the inner and outer pipes.

Area-weighted inlet and outlet temperatures are extracted from the CFD solver for both cases to evaluate the temperature variation along the flow, the direction of heat transfer, and the influence of fluid swapping on overall heat-exchange effectiveness.

6.1. Case 1: Hot Fluid in the Inner Pipe (355 K) and Cold Fluid in the Outer Pipe (298 K)

The temperature contour for Case 1 (Figure 4) clearly illustrates the heat-transfer process from the hot inner fluid toward the colder fluid flowing through the outer pipe. As the fluid progresses along the length of the heat exchanger, the inner hot stream gradually cools, while the outer cold stream undergoes a corresponding rise in temperature due to continuous heat absorption from the inner pipe.

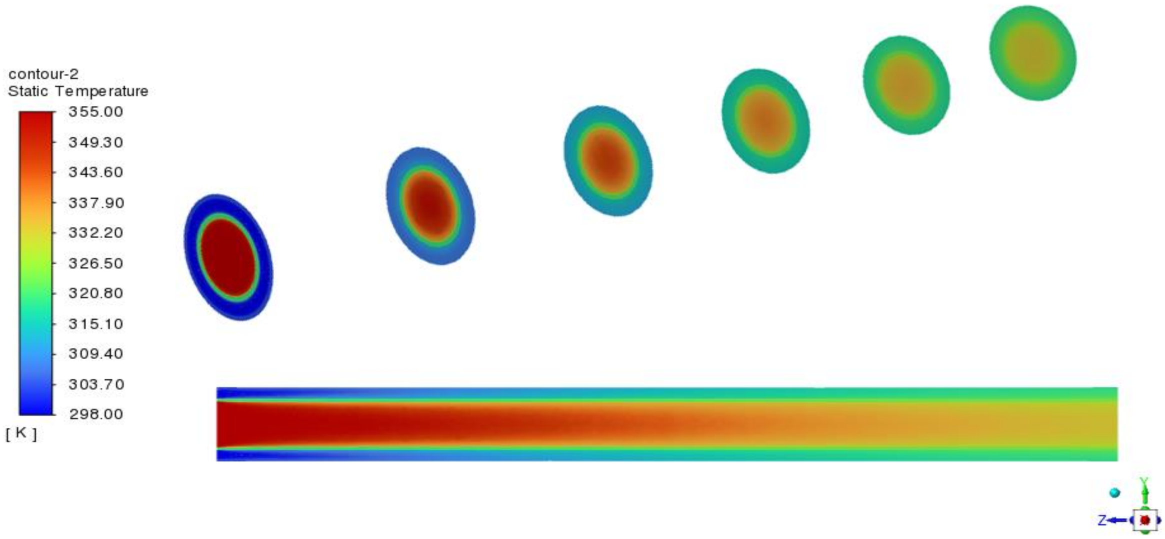

Figure 4: Hot Fluid in the Inner Pipe and Cold Fluid in the Outer Pipe contour

6.2. Case 2: Cold Fluid in the Inner Pipe (298 K) and Hot Fluid in the Outer Pipe (355 K)

The temperature contour for Case 2 (Figure 5) clearly shows the reverse heat-transfer process compared to Case 1. In this configuration, the hot fluid enters through the outer pipe, and heat is transferred inward toward the colder fluid flowing through the inner pipe. As the fluids move along the length of the heat exchanger, the cold inner stream gradually heats up, while the hot outer stream cools down due to continuous energy exchange through the pipe wall.

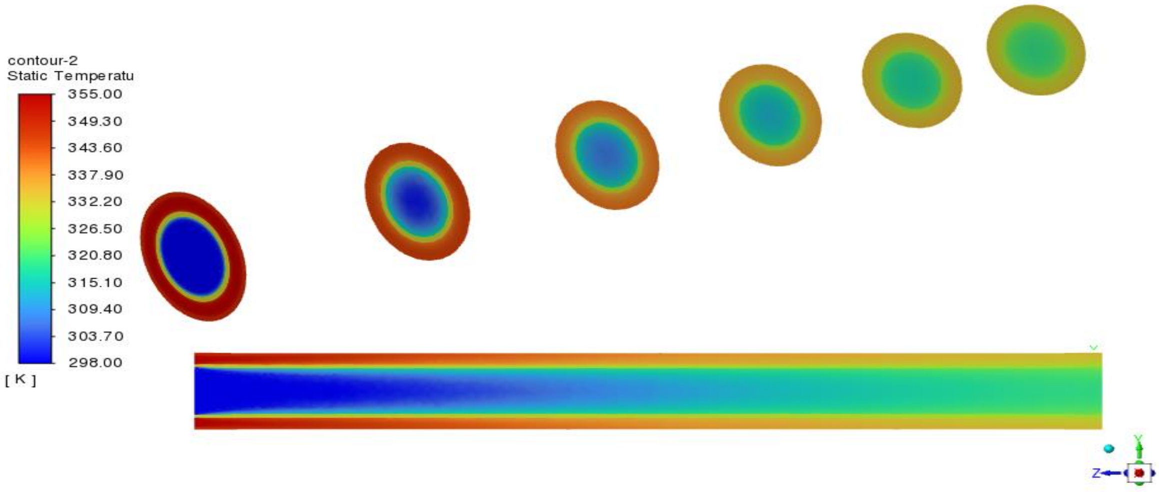

Figure 5: Cold Fluid in the Inner Pipe and Hot Fluid in the Outer Pipe contour

7. COMPARISON BETWEEN THE TWO CASES

The two CFD cases evaluate how the location of the hot fluid inside the double-pipe heat exchanger affects overall heat-transfer performance. In Case 1, the hot fluid flows through the inner pipe, cooling from 355 K to 330 K, resulting in a temperature drop of 25 K. Heat is transferred outward to the surrounding cold fluid, and the temperature contour shows a strong radial gradient moving from the inner region toward the outer pipe.

In Case 2, the hot fluid is positioned in the outer pipe, cooling from 355 K to 337 K, giving a smaller temperature drop of 18 K. Heat moves inward toward the colder inner stream, but the reduced temperature difference indicates slightly lower heat-transfer effectiveness compared to Case 1. Overall, the results suggest that placing the hot fluid in the inner pipe enhances thermal performance due to stronger radial heat flow and greater temperature reduction.

Summary

Case	Hot-fluid location	Inlet (K)	Outlet (K)	ΔT (K)	Effectiveness*
1	Hot in inner pipe	355	330	25	Higher
2	Hot in outer pipe	355	337	18	Lower

8. CONCLUSION

The CFD investigation of the parallel-flow double-pipe heat exchanger demonstrates the significant influence of fluid placement on thermal performance, temperature distribution, and overall heat-transfer effectiveness. When the hot fluid flows through the inner pipe, the system exhibits a stronger radial temperature gradient and a larger temperature drop of 25 K, indicating more effective heat transfer between the two streams. This enhancement is attributed to the higher wall temperatures and increased convective activity in the inner pipe, which promote more vigorous heat exchange with the surrounding colder annulus. Conversely, when the hot fluid is positioned in the outer pipe, the observed temperature drop decreases to 18 K, and the temperature contours reveal a less pronounced gradient. This comparative reduction in thermal performance confirms that the inner pipe location is thermally more favourable for high-temperature fluids due to intensified heat flux and improved interaction with the surrounding pipe walls. These findings align with previous literature highlighting how flow arrangement, material conductivity, and geometric symmetry directly influence exchanger performance and temperature uniformity.

Moreover, the CFD analysis validates the effectiveness of numerical simulation as a tool for predicting thermal behaviour in compact heat-exchange systems, supporting earlier studies that emphasize the accuracy of CFD in modeling internal flows and conjugate heat transfer. The results highlight that parallel-flow configurations, although inherently limited by declining temperature differences along their length, can be optimized through strategic fluid placement that enhances thermal gradients and minimizes inefficiencies. The insights gained from fluid swapping also underscore its value as a design optimization technique, particularly in systems where fouling tendencies, pressure drop, and mechanical stability must be balanced against heat-transfer requirements. Overall, this study provides a comprehensive comparison of two operating modes and demonstrates that placing the hot fluid in the inner pipe yields superior heat-transfer performance, offering practical guidance for improving the design and operation of parallel-flow double-pipe heat exchangers in industrial applications.

REFERENCES:

1. Apparao, G. V., & Rao, K. S. (2019). CFD analysis of a double pipe heat exchanger by using fluid based nanomaterials. Int J Trend Scientific Res Dev, 3(2).
2. Parekh, P. P., & Chavda, N. K. (2014). Experimental and exergy analysis of a double pipe heat exchanger for parallel flow arrangement. Mh, 4, 0c.
3. Bartecki, K. (2015). Transfer function-based analysis of the frequency-domain properties of a double pipe heat exchanger. Heat and Mass Transfer, 51(2), 277-287.
4. Bartecki, K. (2020). Rational transfer function model for a double-pipe parallel-flow heat exchanger. Symmetry, 12(8), 1212.
5. Gabir, M. M., & Alkhafaji, D. (2021, August). Comprehensive review on double pipe heat exchanger techniques. In Journal of Physics: Conference Series (Vol. 1973, No. 1, p. 012013). IOP Publishing.
6. Ebieto, C. E., Ana, R. R., Nyong, O. E., & Saturday, E. G. (2020). Design and construction of a double pipe heat exchanger for laboratory application. European Journal of Engineering and Technology Research, 5(11), 1301-1306.
7. Martin, K. (2019, October). Water-to-water turbulent flow heat exchanger performance analysis. The World Conference on Engineering and Technology.
8. Sharma, C., Siddiqui, M., Imran, M., & Manikandaraja, G. (2020, August). Heat transfer analysis of the DOT 4 brake fluid using a double pipe heat exchanger. In IOP Conference Series: Materials Science and Engineering (Vol. 912, No. 4, p. 042011). IOP Publishing.
9. Sunu, P. W., Anakottapary, D. S., & Santika, W. G. (2016). Temperature approach optimization in the double pipe heat exchanger with groove. In MATEC Web of Conferences (Vol. 58, p. 04006). EDP Sciences.
10. Kumar, A., Sureka, U., & Kumar, S. (2018). Numerical Analysis of Heat transfer Enhancement in a double pipe heat exchanger with a holed twisted tape. In MATEC Web of Conferences (Vol. 144, p. 04012). EDP Sciences.
11. Tongkratoke, A., Pramuanjaroenkij, A., Phankhoksoong, S., & Kakaç, S. (2019, March). The experimental investigation of double pipe heat exchangers prepared from two techniques. In IOP Conference Series: Materials Science and Engineering (Vol. 501, No. 1, p. 012064). IOP Publishing.
12. Okafor, C. E., Tagbo, A., Obiafudo, O., & Nwadike, E. (2016). Material Selection and Fluid Flow Analysis of Parallel Flow Heat Exchanger. Archives of Current Research International, 6(3), 1-14.
13. Nwokolo, N., Mukumba, P., & Obileke, K. (2020). Thermal performance evaluation of a double pipe heat exchanger installed in a biomass gasification system. Journal of Engineering, 2020(1), 6762489.

14. Hamzah, J. A., & Nima, M. A. (2019). Numerical investigation of heat transfer enhancement of double pipe heat exchanger using metal foam fins. Journal of Engineering, 25(6), 1-18.
15. Gangwar, A., Pathariya, A., & Kumar, H. (2019). Thermal analysis on hydrothermal performance of double pipe heat exchanger with single and double helical tape. International Journal of Innovative Technology and Exploring Engineering, 2(9), 1965–1972. https://doi.org/10.35940/ijitee.B7861.129219
16. Anantha, S., Gnanamani, S., Mahendran, V., Rathinavelu, V., Rajagopal, R., Tafesse, D., & Nadarajan, P. (2021). A CFD investigation and heat transfer augmentation of double pipe heat exchanger by employing helical baffles on shell and tube side. Thermal Science, 26(2 Part A), 991-998.
17. Deb, J. S., Prodhan, M. M. H., Hossain, N., & Rakib, G. S. (2022). Preliminary CFD investigation for secondary coolant in molten salt reactor with alumina nanofluid considering FLiBe and FLiNaK as base fluid through a double pipe heat exchanger. Journal of Bangladesh Academy of Sciences, 46(1), 45-55.
18. Joshi, N. (2021). Simulation and Analysis of Double Pipe Heat Exchanger. International Journal for Research in Applied Science and Engineering Technology, 9(VI), 2388-2394.