



Numerical Investigation of Parallel-Flow Characteristics in Double-Pipe Heat Exchangers Using Different Cross-Sectional Geometries of the Inner Tube

Shani Singh (M.Tech)

Research Scholar, Mechanical Engineering, India

* Corresponding Author: **Shani Singh**; Email: Shanince1@gmail.com

Article Info

ISSN (online): 3049-1215

Impact Factor (RSIF): 8.25

Volume: 03

Issue: 01

Received: 01-12-2025

Accepted: 02-01-2026

Published: 03-02-2026

Page No: 19-25

Abstract

This study investigates the thermal performance of a parallel-flow double-pipe heat exchanger using three different inner-tube geometries—circular, circular-cut, and rectangular-cut—to evaluate how geometric modifications enhance heat transfer in systems with naturally limited thermal gradients. Computational Fluid Dynamics (CFD) simulations were carried out in ANSYS Fluent under steady-state, laminar flow conditions using water as the working fluid, with a 250 mm exchanger modeled to capture conjugate heat transfer and temperature distribution for each geometry. High-quality meshing, boundary-layer refinement, and accurate inlet temperature settings enabled detailed analysis of thermal behaviour across the inner and outer flow regions. The results show that modified geometries significantly improve fluid mixing and boundary-layer disruption, leading to enhanced convective heat transfer compared to the standard circular tube. The circular tube produced cold- and hot-outlet temperatures of 306.30 K and 338.46 K, while the circular-cut tube improved performance with outlet values of 306.19 K (cold) and 331.20 K (hot). The rectangular-cut tube achieved the highest thermal enhancement, raising the cold-fluid outlet temperature to 307.41 K and cooling the hot fluid to 336.81 K, demonstrating that geometric optimization—especially the rectangular-cut design—substantially strengthens heat-transfer effectiveness in parallel-flow heat exchangers.

DOI: <https://doi.org/10.54660/IJFEI.2026.3.1.19-25>

Keywords: Thermal Performance, Parallel-Flow, Double-Pipe, Heat Exchanger, Inner-Tube Circular, Computational Fluid Dynamics, Steady-State and Laminar Flow

1. Introduction

Double-pipe heat exchangers are widely used in industrial applications due to their compact design, structural simplicity, and effective heat transfer capability. They are essential components in systems such as chemical processing, refrigeration, and power generation. In these devices, two fluids exchange heat through concentric pipes — one flowing through the inner tube and the other through the outer annulus. The performance of these exchangers depends significantly on flow arrangement and geometry. In parallel-flow configurations, both fluids move in the same direction, resulting in a gradual temperature equalization along the tube length, which can limit thermal efficiency (Bhattacharjee, 2020) ^[1].

To address this limitation, several passive enhancement techniques have been proposed. Among them, modifying the inner-tube geometry—such as introducing circular-cut or rectangular-cut designs—has proven to be an effective approach to disturb boundary layers and promote better mixing. This modification enhances convective heat transfer by generating secondary flow structures that improve fluid interaction near the walls (Hasgül & Çakmak, 2021) ^[2]. Computational Fluid Dynamics (CFD) allows accurate simulation of such configurations under various operating conditions, helping researchers identify the most efficient tube designs for improved heat transfer (Reddy, 2017) ^[3].

1.1. Background of Double-Pipe Heat Exchangers

Double-pipe heat exchangers are among the simplest and most commonly used configurations for heat transfer between two fluids. They consist of two concentric pipes, where heat is exchanged through the wall separating the inner and outer flow channels. These exchangers are typically used in small- to medium-scale applications where compactness and maintenance accessibility are required. The thermal efficiency largely depends on whether the system operates in counter-flow or parallel-flow mode. Counter-flow systems generally provide higher heat transfer rates due to greater temperature gradients, but parallel-flow designs are often chosen for their ease of installation and lower mechanical stress. However, a limitation of parallel-flow configurations is the reduced driving temperature difference as both fluids progress along the exchanger. This issue has driven researchers to explore ways to enhance heat transfer without adding mechanical complexity. One of the most promising methods involves geometric modifications of the inner tube to increase turbulence and fluid mixing (Vijayaragavan *et al.*, 2023) [4]. Such modifications can substantially improve energy exchange efficiency even under laminar flow conditions.

1.2. Role of Inner-Tube Geometry in Heat Transfer Enhancement

The cross-sectional geometry of the inner tube strongly influences flow structure and heat transfer characteristics. Conventional circular tubes create smooth and uniform velocity profiles, which minimize turbulence and lead to moderate heat transfer performance. In contrast, modified geometries—such as rectangular, lobed, or wavy profiles—introduce flow disturbances that increase local mixing and enhance thermal energy exchange. These geometries cause velocity gradients to redistribute within the flow field, creating vortices and secondary currents that promote greater heat transfer through boundary layer renewal (Hasgül & Çakmak, 2021) [2]. The performance of these configurations can be evaluated using the Nusselt number, which quantifies the convective heat transfer enhancement relative to conduction.

Optimizing inner-tube geometry involves balancing heat transfer gains with potential increases in pressure drop. Studies have shown that rectangular and corrugated tubes can improve the heat transfer rate by up to 250% compared to smooth tubes, albeit with slightly higher flow resistance (Reddy, 2017) [3]. Corrugation height, cut depth, and aspect ratio are critical design variables that influence the degree of enhancement. Advanced CFD optimization studies have identified geometric parameters that yield the highest thermal efficiency while minimizing energy losses (Han *et al.*, 2015) [5]. Therefore, inner-tube geometry modification represents a practical, passive method to improve overall system performance without additional power input.

1.3. Importance of Flow Regime and Parallel-Flow Configuration

The heat transfer efficiency of a double-pipe heat exchanger depends heavily on the flow regime. In laminar flow, fluid layers move smoothly, resulting in minimal mixing and limited convective transport. While turbulent flow offers better mixing and heat transfer, it requires higher pumping power and is not always suitable for compact or low-flow systems. Under parallel-flow conditions, both fluids move in

the same direction, allowing for uniform temperature gradients and simpler mechanical construction. However, the decreasing temperature difference along the flow path restricts the exchanger's effectiveness. To counter this, geometric modifications—such as adding fins, corrugations, or cut sections—can generate secondary flow structures that mimic turbulent behaviour in laminar regimes (Ahmed *et al.*, 2021) [6]. These structures increase local velocity gradients, improve thermal boundary layer mixing, and enhance the overall heat transfer rate. CFD and experimental results confirm that optimized geometry designs can compensate for the inherent limitations of parallel-flow systems.

1.4. Numerical Simulation and CFD as a Research Tool

Computational Fluid Dynamics (CFD) provides a powerful numerical framework for investigating the flow and heat transfer behaviour in complex heat exchanger geometries. By solving the governing conservation equations for mass, momentum, and energy, CFD allows detailed visualization of temperature fields, pressure distributions, and velocity profiles within double-pipe systems. Researchers employ this method to simulate both steady and transient conditions, offering insights into thermal performance without the need for extensive experimental setups (Apparao & Rao, 2019) [7]. Using ANSYS Fluent, simulations can incorporate conjugate heat transfer between solid walls and fluids, allowing accurate evaluation of different inner-tube geometries. Furthermore, CFD enables parametric studies across a wide range of Reynolds numbers and boundary conditions. Through mesh refinement and turbulence modeling, the precision of the results can be validated against experimental data. This approach helps identify flow regions with high thermal resistance and optimize configurations for maximum heat transfer efficiency (Lahari *et al.*, 2021). Hence, CFD serves as an indispensable design and validation tool in modern heat exchanger research.

2. Effect of Inner Tube Cross-Section on Thermal Performance

The cross-sectional geometry of the inner tube is one of the most influential parameters affecting the thermal and flow characteristics of double-pipe heat exchangers. Altering the shape of the inner tube changes how fluid interacts with the surface, which directly influences turbulence intensity, boundary layer thickness, and overall convective heat transfer. Non-circular cross-sections—such as lobed, wavy, rectangular, or corrugated profiles—generate secondary flow patterns that enhance fluid mixing and disrupt thermal boundary layers, leading to higher Nusselt numbers compared to smooth circular tubes. For instance, Omidi *et al.* (2018) [9] showed that lobed spiral tubes can improve heat transfer by over 200% due to the induced vortices formed along the surface. Similarly, Hasgül and Çakmak (2022) [2] demonstrated that wavy tubes increased heat transfer up to 270%, significantly outperforming smooth tubes under laminar and transitional flow regimes.

The use of cut geometries, such as circular-cut and rectangular-cut inner tubes, introduces sharp corners or notches that accelerate local fluid motion, intensifying mixing and energy exchange between the wall and fluid core. However, these geometric alterations must be optimized carefully, as excessive disturbance can lead to pressure penalties and flow instability. Reddy *et al.* (2017) [3] noted that rectangular fins enhance thermal performance but may

increase frictional losses if improperly designed. Overall, cross-sectional modification offers a simple yet powerful passive technique for improving heat exchanger efficiency, making it a critical design consideration in modern CFD-based optimization and industrial applications.

3. Problem Statement

The performance of double-pipe heat exchangers depends greatly on the geometry of the inner tube, particularly under laminar and parallel-flow conditions where fluid mixing is naturally limited. Conventional circular tubes often provide only moderate heat-transfer efficiency due to smooth flow behaviour. Introducing modified inner-tube shapes, such as circular-cut and rectangular-cut geometries, may enhance thermal interaction by altering flow patterns, but their effectiveness has not been clearly established. Understanding how these geometries influence heat transfer and outlet conditions is essential for improving system efficiency. This study investigates the impact of different inner-tube cross-sections using CFD to identify the most effective design.

4. Objectives

1. To compare the heat-transfer performance of circular, circular-cut, and rectangular-cut inner tubes under parallel-flow conditions.
2. To identify which tube geometry provides the highest temperature change and overall thermal efficiency in the 250 mm double-pipe heat exchanger.

5. Literature Review

Omidi *et al.* (2018)^[9] conducted a numerical study on spiral tubes with lobed cross-sections and found that geometric alterations in double-pipe heat exchangers significantly enhance heat transfer performance. Their research demonstrated that lobed geometries increase turbulence intensity and improve the Nusselt number by over 200% compared to circular tubes. Bhattacharjee (2020)^[11] further supported these findings, noting that parallel-flow configurations benefit from passive enhancements due to reduced thermal gradients. Similarly, Bayareh (2019)^[10] investigated corrugated tubes and reported that convex-concave surface designs increased the thermal effectiveness by improving flow mixing and reducing thermal resistance. Gangwar *et al.* (2019)^[11] added that introducing helical tape inserts enhanced hydrothermal performance by increasing secondary flow formation. These studies establish that geometric optimization—through lobed, corrugated, or helical designs—can greatly improve convective heat transfer without altering flow arrangements. Such innovations make double-pipe exchangers more adaptable for chemical, power, and refrigeration systems requiring compact and efficient designs.

Hasgül and Çakmak (2022)^[2] examined wavy inner tubes and observed up to a 270% increase in heat transfer compared to smooth tubes. Their CFD analysis showed that the wavy structure enhanced turbulence and disrupted boundary layers, creating better thermal interaction. Reddy *et al.* (2017)^[3] investigated rectangular fins, demonstrating that optimized fin geometry balances heat transfer and pressure loss. Al-Zahrani (2023)^[12] extended this by testing oval and wavy tube shapes, finding up to a 28% improvement in Nusselt number with minimal pressure penalties. Han *et al.* (2015)^[5] applied Response Surface Methodology to optimize corrugated tubes and achieved ideal heat transfer at reduced

flow resistance. Azizi *et al.* (2023)^[13] used a non-uniform magnetic field with nanofluids in triangular tubes, achieving an additional 15% performance boost. Together, these works confirm that modifying the inner-tube profile and flow environment significantly enhances thermal efficiency.

Vijayaragavan *et al.* (2023)^[4] analyzed externally enhanced labyrinth-shaped inner tubes and observed remarkable heat transfer improvements with limited pressure penalties. Their results showed that increased surface area and secondary vortex formation contributed to higher convective heat transfer rates. Validated these results experimentally, showing strong agreement between CFD predictions and measured outlet temperatures. Ahmed *et al.* (2021)^[6] compared parallel and counter-flow exchangers, finding that optimized geometries in parallel configurations mitigate thermal inefficiency. Alias *et al.* (2020) showed that elliptical tubes combined with ZnO–water nanofluids improved Nusselt numbers by 30%. Incorporated porous metal foams into double-pipe exchangers, improving temperature uniformity and reducing pressure variation. These studies suggest that the integration of enhanced geometries and nanofluids represents a key advancement in sustainable heat exchanger design.

Apparao and Rao (2019)^[7] emphasized the importance of CFD in simulating fluid-thermal interactions within modified heat exchangers. Their use of Fe₃O₄ nanofluids demonstrated a significant rise in convective coefficients. Bee and Devaraj (2021)^[8] confirmed these findings experimentally, showing close correlation between CFD results and Cu-based nanofluid data. Patel and Mishra (2019) compared geothermal heat exchangers and noted that horizontal double-pipe systems outperform vertical arrangements due to better natural convection. Wang *et al.* (2021) found that increasing velocity in refrigerant-based exchangers boosts heat transfer while reducing exergy loss. Gomaa *et al.* (2016) studied star-shaped inserts and observed a 25% enhancement in heat transfer. Joshi (2021) verified that Al₂O₃–water nanofluids improve overall efficiency when optimized at low concentrations. Collectively, these findings show that CFD-integrated research, combined with nanofluid use and innovative internal geometries, has transformed the predictive and practical capabilities of double-pipe heat exchangers.

6. Methodology

CFD simulations were performed in ANSYS Fluent to analyze heat transfer and fluid-flow behaviour inside a parallel-flow double-pipe heat exchanger. Two fluids were used in the simulation: a hot fluid at 350 K and a cold fluid at 298 K, both flowing in the same direction to achieve parallel-flow operation. Three different inner-tube geometries—circular, circular-cut, and rectangular-cut—were evaluated to study their effect on temperature distribution and thermal performance. The methodology consisted of geometry creation, mesh generation, boundary-condition assignment, solver setup, and post-processing of simulation results.

6.1. Geometry Creation and Preparation

The geometry was created based on the following specifications:

- **Inner pipe diameter:** 20 mm
- **Outer pipe diameter:** 28 mm
- **Wall thickness:** 1 mm
- **Tube length:** 250 mm

Two separate fluid domains were created: one for the inner pipe and one for the outer annulus. Solid wall regions were also included to enable conjugate heat-transfer modeling. The geometry was imported into ANSYS Fluent Meshing using

the Watertight Geometry workflow. Share Topology was enabled so that all contacting surfaces share common nodes for accurate thermal interaction between solid and fluid regions.

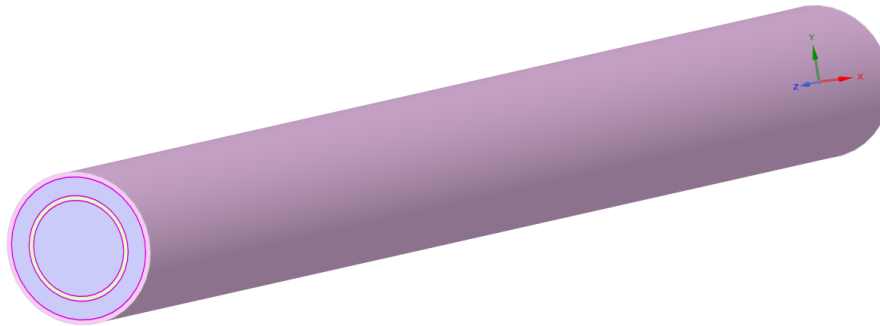


Fig 1: Geometry

6.2. Design configures

The design configuration consists of three inner-tube geometries used in a double-pipe heat exchanger: (a) a circular tube, (b) a circular-cut tube, and (c) a rectangular-cut tube. The circular tube provides a smooth flow with minimal turbulence, serving as the baseline geometry. The circular-cut tube includes symmetric curved indentations along its

perimeter, promoting localized mixing and moderate turbulence. The rectangular-cut tube features deeper angular grooves that generate stronger vortices and enhanced fluid interaction. These geometric variations aim to improve heat transfer by altering boundary layer behaviour and increasing convective surface area within the parallel-flow heat exchanger.

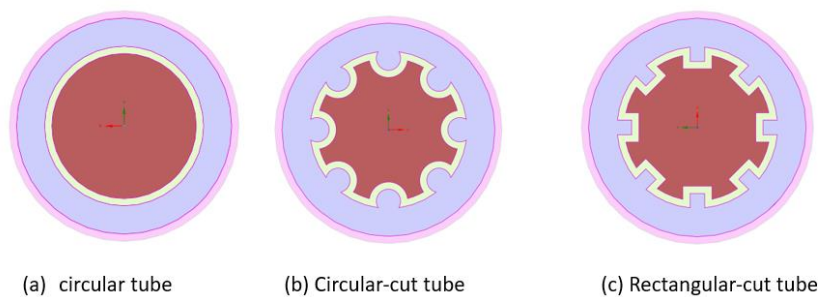


Fig 2: Design configures

6.3. Meshing and Boundary Layer Setup

A high-quality mesh was generated using curvature and proximity-based controls to accurately capture all geometric features. Local sizing was applied to refine the mesh near the pipe walls. Three inflation (boundary layer) layers were

added on all fluid-wall surfaces to resolve temperature and velocity gradients. A polyhedral volume mesh was created to ensure good convergence and accurate prediction of heat transfer under laminar flow conditions.

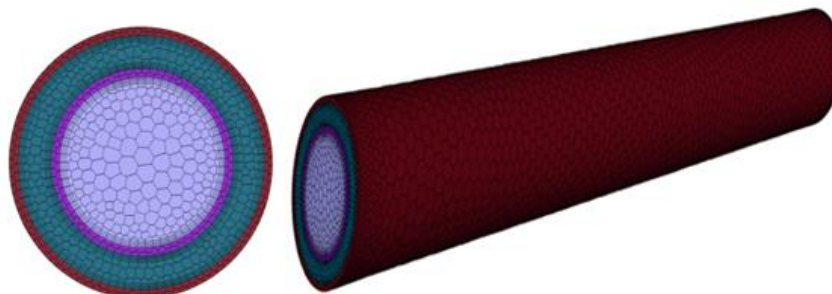


Fig 3: Meshing

6.4. 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.

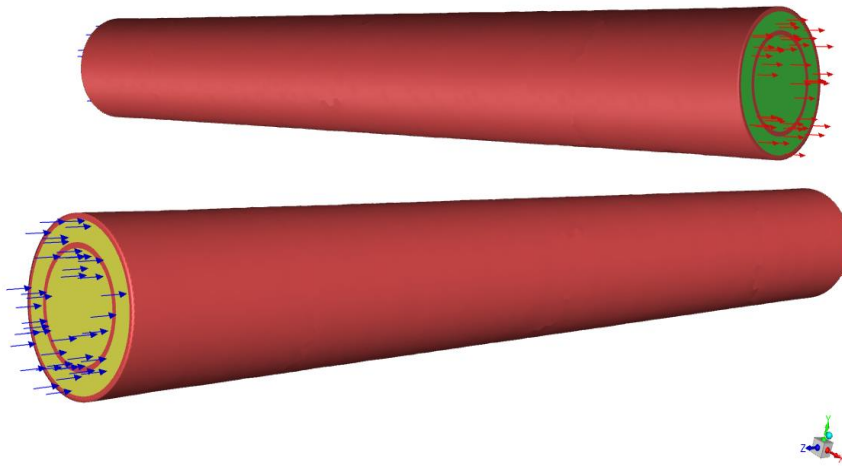


Fig 4: Inlet and outlet

6.5. Boundary Conditions and Region Setup

In ANSYS Fluent, all regions were assigned as either fluid or solid. The inner pipe fluid zone and the outer annulus fluid zone were defined separately to allow accurate modeling of heat transfer between them. Both inlets were specified as velocity inlets, with temperatures set to 350 K for the hot fluid and 298 K for the cold fluid, depending on the inner-tube geometry being tested. Since the heat exchanger operates under parallel-flow conditions, both fluids entered from the same direction. The outlets for both fluid domains were defined as pressure outlets with zero-gauge pressure. All solid surfaces were assigned no-slip boundary conditions. This setup ensured correct simulation of fluid-wall interaction and conjugate heat transfer throughout the double-pipe system.

6.6. Solver Setup and Simulation

The simulation was performed in steady-state mode using a pressure-based solver, suitable for incompressible internal flow. The energy equation was activated to capture heat transfer between the two fluids and through the pipe walls. Since the flow velocities were low and the geometry supported smooth flow, the laminar flow model was selected. Material properties were assigned as follows: water for both fluid zones and standard solid material for the pipe walls. Second-order upwind discretization was used for the momentum and energy equations to enhance accuracy, while first-order upwind was applied only where needed for stability. The simulation was initialized using Hybrid Initialization and iterated until convergence was achieved.

6.7. Post-Processing

Post-processing was carried out in ANSYS Fluent and included extraction of temperature contours, velocity distributions, and area-weighted outlet temperatures for both fluids. Temperature readings were collected at the inner pipe inlet/outlet and outer annulus inlet/outlet to evaluate the change in thermal energy as the fluids moved along the 250 mm heat exchanger. The contour plots and numerical data were used to compare the thermal performance of the three

geometries and determine how each inner-tube shape influenced heat transfer efficiency under parallel-flow conditions.

7. Results

The temperature-based analysis provides a clear comparison of how the three inner-tube geometries—circular, circular-cut, and rectangular-cut—affect the heat transfer performance of a parallel-flow double-pipe heat exchanger. By focusing on the change in temperature from inlet to outlet for both hot and cold fluids, the effectiveness of each configuration becomes evident. Higher cold outlet temperatures indicate stronger heat absorption, while lower hot outlet temperatures reflect improved heat rejection and more effective thermal interaction between the streams. The circular tube serves as the baseline, showing moderate temperature changes due to its smooth geometry and limited flow disturbance. In contrast, the circular-cut and rectangular-cut tubes demonstrate enhanced thermal performance as the modified shapes promote increased mixing and convection. These enhancements are directly visible through greater temperature rises on the cold side and more pronounced cooling on the hot side. Overall, temperature behaviour across the three cases highlights how geometric modification strengthens heat transfer efficiency.

7.1. Case 1: Circular Tube

In the circular tube configuration, the heat transfer performance is moderate, reflecting the smooth and symmetric geometry. The cold fluid temperature increases from 298 K to 306.30 K, showing a steady but limited heat absorption. Meanwhile, the hot fluid temperature decreases from 350 K to 338.46 K, indicating a modest heat release. The net temperature of 323.02 K represents balanced thermal mixing without significant enhancement. Due to minimal flow disruption, convection remains relatively low, resulting in stable but less efficient heat transfer. This geometry offers predictable thermal behaviour but does not promote strong thermal interaction between hot and cold streams.

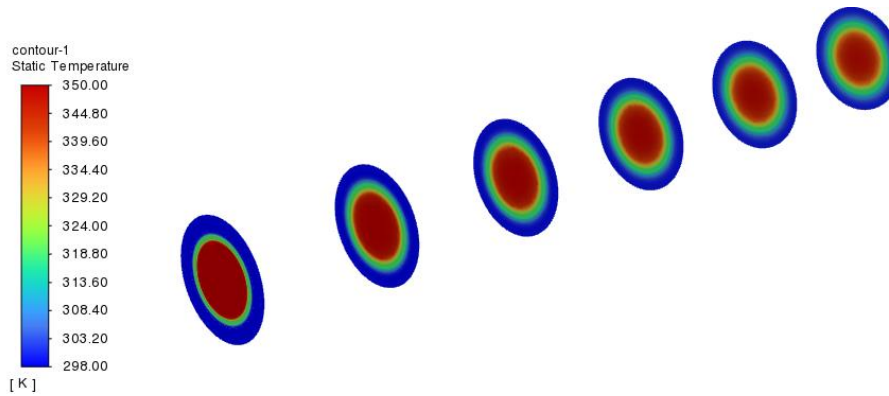


Fig 5: Circular Tube

7.2. Case 2: Circular-Cut Tube

The circular-cut tube demonstrates improved heat transfer compared to the circular tube due to the modified geometry. The cold outlet temperature rises to 306.19 K, indicating better heat absorption, while the hot outlet temperature drops more significantly to 331.20 K. This reduction reflects stronger cooling and enhanced energy exchange. The net

temperature of 316.70 K suggests more effective mixing and improved thermal interaction. The circular cut promotes localized turbulence, allowing heat to transfer more efficiently across the thermal boundary. Overall, this configuration enhances thermal performance noticeably, although the increased heat transfer corresponds to greater internal flow disturbance.

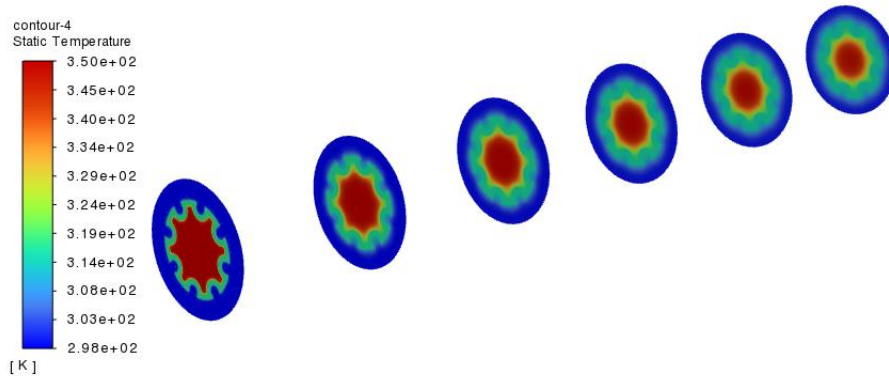


Fig 6: Circular-Cut Tube

7.3. Case 3: Rectangular-Cut Tube

The rectangular-cut tube provides the strongest thermal interaction among all configurations. The cold outlet temperature rises to 307.41 K, the highest value recorded, indicating superior heat absorption. Meanwhile, the hot fluid temperature decreases from 355 K to 336.81 K, showing a substantial cooling effect. The net temperature of 320.84 K

reflects enhanced energy exchange and more effective parallel-flow heat transfer. The rectangular cut generates broader flow disturbances, increasing convection and improving thermal contact between the fluids. This configuration balances improved heat transfer with moderate geometric modification, making it the most thermally efficient option in the study.

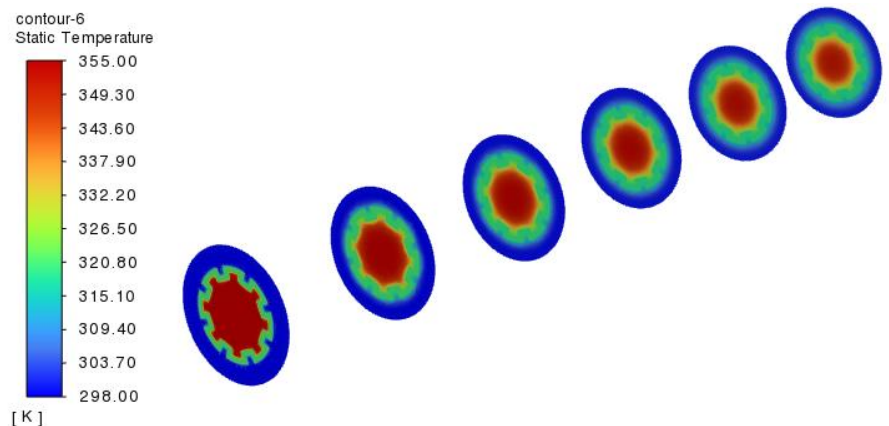


Fig 7: Rectangular-Cut Tube

8. Conclusion

The numerical investigation confirms that the geometry of the inner tube plays a decisive role in determining the thermal effectiveness of a parallel-flow double-pipe heat exchanger. The temperature-based results show clear differences in performance among the circular, circular-cut, and rectangular-cut tubes. The circular tube, used as the reference configuration, exhibits smooth flow with minimal disturbance, resulting in moderate heat transfer. The cold fluid temperature increases from 298 K to 306.30 K, while the hot fluid decreases from 350 K to 338.46 K. These values indicate stable but limited thermal interaction, as the circular geometry does not significantly enhance turbulence or secondary flows. The circular-cut tube introduces asymmetry that increases mixing and improves thermal contact between the fluids. This results in a cold outlet temperature of 306.19 K and a hot outlet temperature of 331.20 K, demonstrating stronger cooling and improved heat absorption compared to the circular tube. The increased temperature difference confirms that geometric modification enhances convective heat transfer mechanisms.

Among all configurations, the rectangular-cut tube delivers the highest overall thermal performance. The cold fluid reaches 307.41 K, the largest temperature increase recorded, showing superior heat absorption. Meanwhile, the hot fluid temperature decreases from 355 K to 336.81 K, indicating a substantial heat release. This performance improvement arises from the rectangular cut generating intensified flow disturbance, larger mixing zones, and more active thermal interaction across the fluid boundary layers. The pronounced cooling of the hot stream and the higher heating of the cold stream confirm that this geometry maximizes thermal energy exchange within the parallel-flow arrangement. Overall, the study concludes that inner-tube modifications meaningfully enhance heat transfer, with the rectangular-cut geometry providing the strongest improvement. These findings highlight the importance of geometric optimization when designing or upgrading double-pipe heat exchangers for improved temperature control and energy efficiency.

9. References

1. Bhattacharjee D. CFD analysis of double pipe counter flow heat exchanger. *Int J Eng Res Technol*. 2020 Oct. ISSN: 2278-0181.
2. Hasgul C, Cakmak G. Heat transfer analysis of double tube heat exchanger with wavy inner tube. *Therm Sci*. 2022;26(4 Pt B):3455-62.
3. Reddy NS, Rajagopal K, Veena PH. Numerical investigation of heat transfer enhancement and pressure drop of a double tube heat exchanger with rectangular fins in the annulus side. *Int J Dyn Fluids*. 2017;13(2):295-308.
4. Vijayaragavan B, Asok SP, Shakthi Ganesh CR. Heat transfer characteristics of double pipe heat exchanger having externally enhanced inner pipe. 2023.
5. Han HZ, Li BX, Wu H, Shao W. Multi-objective shape optimization of double pipe heat exchanger with inner corrugated tube using RSM method. *Int J Therm Sci*. 2015;90:173-86.
6. Ahmed F, Sumon MM, Fuad M, Gugulothu RAV, Mollah AS. Numerical simulation of heat exchanger for analyzing the performance of parallel and counter flow. *WSEAS Trans Heat Mass Transf*. 2021;16:145-52.
7. Apparao GV, Rao KS. CFD analysis of a double pipe

heat exchanger by using fluid based nanomaterials. *Int J Trend Sci Res Dev*. 2019;3(2).

8. Bee H, Devaraj S. Experimental and CFD analysis of GW70 based Cu nanofluids in a parallel flow heat exchanger.
9. Omidi M, Farhadi M, Jafari M. Numerical study on the effect of using spiral tube with lobed cross section in double-pipe heat exchangers. *J Therm Anal Calorim*. 2018;134(3):2397-408.
10. Bayareh M. Numerical simulation and analysis of heat transfer for different geometries of corrugated tubes in a double pipe heat exchanger. *J Therm Eng*. 2019;5(4):293-301.
11. Gangwar A, Pathariya A, Kumar H. Thermal analysis on hydrothermal performance of double pipe heat exchanger with single and double helical tape. *Int J Innov Technol Explor Eng*. 2019;9(2):1965-72.
12. Al-Zahrani S. Heat transfer characteristics of innovative configurations of double pipe heat exchanger. *Heat Mass Transf*. 2023;59(9):1661-75.
13. Azizi Y, Bahramkhoo M, Kazemi A. Influence of non-uniform magnetic field on the thermal efficiency hydrodynamic characteristics of nanofluid in double pipe heat exchanger. *Sci Rep*. 2023;13(1):407.

How to Cite This Article

Singh S, Singh S. Numerical investigation of parallel-flow characteristics in double-pipe heat exchangers using different cross-sectional geometries of the inner tube. *Int J Future Eng Innov*. 2026;3(1):19–25. doi:10.54660/IJFEI.2026.3.1.19-25.

Creative Commons (CC) License

This is an open access journal, and articles are distributed under the terms of the Creative Commons Attribution-NonCommercial-ShareAlike 4.0 International (CC BY-NC-SA 4.0) License, which allows others to remix, tweak, and build upon the work non-commercially, as long as appropriate credit is given and the new creations are licensed under the identical terms.