Numerical CFD Study of Heat Transfer Efficiency in a Counter-Flow Double-Pipe Heat Exchanger

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 presents a Computational Fluid Dynamics (CFD) analysis of a double-pipe heat exchanger (DPHE) to evaluate the influence of flow configuration on thermal performance. Using ANSYS Fluent, simulations were carried out for both parallel-flow and counter-flow arrangements with water as the working fluid, where the hot and cold streams entered at 355 K and 298 K, respectively, at a uniform velocity of 1.9 m/s. The model was analyzed using the k–ε turbulence model, providing detailed visualization of temperature and velocity distributions within the exchanger. Results showed that the counter-flow configuration exhibited superior performance, with the hot fluid cooling from 355 K to 327.77 K and the cold fluid heating to 324.11 K, compared to 337.06 K and 321.25 K in the parallel-flow case. These findings indicate that counter-flow operation maintains a more favourable temperature gradient along the exchanger, leading to greater thermal effectiveness and improved heat recovery. Overall, the study demonstrates that CFD is a reliable and precise tool for analyzing and optimizing heat exchanger design, providing valuable insights for enhancing energy efficiency in industrial thermal systems.

Keywords: Computational Fluid Dynamics, Double-Pipe, Heat Exchanger, Parallel-Flow, Counter-Flow, Temperature Gradient and Turbulence Model.

1. INTRODUCTION

Heat exchangers are vital thermal systems used across a range of industrial applications, from power plants to refrigeration systems, due to their role in transferring heat between two fluids at different temperatures efficiently. The design and performance of heat exchangers have evolved significantly to enhance thermal efficiency and sustainability. Among various configurations, the double-pipe heat exchanger (DPHE) has gained prominence for its simplicity, compactness, and adaptability in both laboratory and industrial scales. DPHEs are often used in applications involving moderate heat transfer rates, such as oil cooling, gas heating, and small-scale energy recovery systems (Bartecki, 2015). Their performance is primarily influenced by the fluid flow configuration—parallel or counter-flow—and the associated temperature gradients.
In recent years, significant attention has been directed toward improving the thermal performance of heat exchangers using computational fluid dynamics (CFD), nanofluids, and modified geometries. CFD has become a preferred method for modeling and optimizing these systems due to its ability to simulate complex thermal-fluid interactions under varying operating conditions (Salunke, 2015). This paper provides a detailed overview of double-pipe heat exchangers, their efficiency considerations, and the importance of temperature gradients, with a focus on the advantages of using CFD for heat-transfer analysis.

1.1. Background on Double-Pipe Heat Exchangers

A double-pipe heat exchanger typically consists of two concentric pipes where one fluid flows through the inner pipe while another flows through the annular space between the inner and outer pipes. The configuration is flexible, allowing either counter-flow or parallel-flow operation depending on the thermal requirements of the application (Bhattacherjee, 2020). Counter-flow configurations are generally more efficient due to higher temperature differentials along the length of the exchanger. DPHEs are widely used in small-scale systems and laboratory experiments because they provide a straightforward way to study fundamental heat-transfer principles (Ebieto et al., 2020). Recent research focuses on optimizing these exchangers by modifying pipe

geometries, adding fins, and introducing nanofluids to enhance convective heat transfer. For instance, rectangular fins attached to the inner pipe were found to increase the surface area and overall heat transfer coefficient significantly compared to plain designs (Reddy, 2017).
Additionally, novel configurations, such as oval or wavy tubes, have been introduced to further enhance the heat-transfer characteristics. A recent study demonstrated that an oval wavy DPHE can improve the Nusselt number by up to 28% compared to conventional circular designs. These advancements highlight how geometrical innovations directly impact heat exchanger efficiency and compactness.

1.2. Importance of Thermal Efficiency in Heat Exchanger Applications

Thermal efficiency is a crucial performance indicator in heat exchangers as it directly affects the overall energy utilization of industrial systems. Higher thermal efficiency translates to reduced energy losses, improved sustainability, and cost-effectiveness. The effectiveness (ε) of a heat exchanger represents the ratio between the actual and maximum possible heat transfer, depending on the temperature difference and flow arrangement (Fernández-Torrijos et al., 2016).
In biomass gasification systems, for example, heat recovery through DPHEs helps minimize energy losses from syngas cooling, leading to a more sustainable process. An experimental study showed that a counter-flow DPHE improved effectiveness by approximately 14% over a parallel-flow configuration (Nwokolo et al., 2020). Likewise, in low-temperature applications such as textile dryers, optimized heat recovery systems using parallel manifolds and staggered fins achieved an increase in energy recuperation of up to 180% compared to conventional configurations (Fiaschi et al., 2017). The choice of materials and working fluids also influences efficiency. Copper-based exchangers and the use of nanofluids such as Al_2O_3/water mixtures significantly enhance heat-transfer rates due to higher thermal conductivities (Joshi, 2021). These developments underline the critical role of thermal efficiency in both design and operational optimization.

1.3. Role of Temperature Gradient in Heat-Transfer Performance

The temperature gradient between the two fluids is the driving force for heat exchange in a DPHE. A higher temperature difference results in a greater rate of heat transfer according to Fourier’s law of heat conduction. The configuration of the flow—whether parallel or counter—determines how this gradient changes along the exchanger length. In counter-flow systems, the gradient remains more uniform, leading to a higher mean temperature difference and, consequently, better heat-transfer performance (**Siavashi** et al., 2019).
Temperature gradients can be manipulated by varying mass flow rates, using fins, or introducing nanofluids to alter the thermal boundary layers. For instance, the use of alumina-based nanofluids increased the overall heat-transfer coefficient by up to 17.62%, with a corresponding 10.8% rise in exchanger effectiveness (Bendaraa et al., 2021). Similarly, temperature distribution analysis in three-fluid helical exchangers showed that higher flow rates lead to enhanced temperature uniformity and greater overall heat-transfer coefficients (Mohapatra et al., 2017). The use of advanced analytical techniques such as the ε–NTU method provides accurate determination of the thermal performance under different gradient conditions. This allows engineers to predict exchanger effectiveness without assuming outlet temperatures, optimizing systems for specific applications (Fernández-Torrijos et al., 2016).

1.4. Parallel vs. Counter-Flow Configurations in Heat Exchangers

The performance of a heat exchanger depends heavily on the direction of fluid flow. In parallel-flow configurations, both fluids move in the same direction, while in counter-flow arrangements, they flow in opposite directions. The counter-flow configuration generally provides a higher temperature difference along the length of the exchanger, resulting in superior heat-transfer performance (Kishan, R, 2020).
Empirical studies confirm this advantage. A CFD and experimental comparison showed that counter-flow DPHEs achieved higher overall heat-transfer coefficients and lower outlet temperatures for hot fluids compared to parallel-flow designs (Apparao & Rao, 2019). However, in some two-phase flow systems, parallel flow may outperform counter flow due to early vapor generation and higher latent heat utilization (Abishek et al., 2017).
Furthermore, in micro-scale applications, CFD-based simulations reveal that the choice of flow direction significantly impacts pressure drops and temperature profiles. A hybrid CFD-porous media approach for

microchannel exchangers showed consistent results across both configurations with counter-flow yielding slightly higher effectiveness (Rehman et al., 2020).

1.5. Advantages of Using CFD for Heat-Transfer Analysis

Computational Fluid Dynamics (CFD) has become an indispensable tool in heat exchanger design, analysis, and optimization. CFD simulations enable researchers to predict velocity fields, temperature distributions, and pressure drops without the need for extensive physical experimentation. This not only reduces development costs but also allows detailed investigation of complex flow geometries and multi-phase interactions (Gabir & Alkhafaji, 2021).

For example, CFD modeling of DPHEs using nanofluids revealed significant improvements in convective heat transfer and friction factors, enabling optimization of flow parameters for better performance (Apparao & Rao, 2019). Moreover, in systems with complex fin geometries, CFD provides accurate visualization of thermal boundary layers and local heat fluxes, aiding in fin design for improved heat dissipation (**Siavashi** et al., 2019).

CFD has also proven valuable in analyzing novel geometrical configurations. In one study, oval and wavy pipe geometries were simulated to identify optimal designs that balance pressure drop and Nusselt number enhancements. Similarly, CFD-aided redesign of industrial heat exchangers led to a 97% improvement in heat recovery through optimized fin arrangements (Fiaschi et al., 2017).

In addition to steady-state analysis, CFD allows transient simulations to study the effects of fluctuating inlet conditions and variable flow rates. This capability supports the development of adaptive control systems that maintain optimal performance under dynamic thermal loads (Shukla & Kishan, 2020).

2. STATEMENT OF PROBLEM

The performance of double-pipe heat exchangers is strongly influenced by the temperature difference maintained between the hot and cold fluids, yet this behavior can vary significantly depending on whether the system operates in parallel-flow or counter-flow mode. Traditional analytical methods often simplify flow behavior and wall interactions, making it difficult to capture detailed thermal patterns within the exchanger. Therefore, a numerical CFD investigation is necessary to provide accurate insight into fluid temperature variations and heat-transfer efficiency along the exchanger length. This study focuses on evaluating the inlet and outlet temperature differences of hot and cold water streams under both parallel- and counter-flow arrangements using ANSYS Fluent. By comparing these temperature differences, the research aims to determine how flow configuration influences thermal performance and overall heat-transfer effectiveness. Understanding these variations is essential for improving heat-exchanger design, enhancing energy efficiency, and identifying the configuration that delivers superior thermal output in practical applications.

3. OBJECTIVE

- To determine inlet and outlet temperature differences between hot and cold fluids for parallel- and counter-flow conditions.
- To compare hot and cold fluid temperature differences in both configurations to evaluate overall thermal performance effectiveness.

4. LITERATURE REVIEW

Apparao and Rao (2019) conducted a CFD analysis of a double-pipe heat exchanger using fluid-based nanomaterials to enhance convective heat transfer and found that nanofluid inclusion significantly improved overall efficiency and temperature uniformity across the exchanger. Their study demonstrated that nanoparticles increase the effective thermal conductivity of the working fluid, leading to better heat dispersion. Similarly, Bhattacherjee (2020) performed a detailed CFD study on counter-flow configurations and revealed that variations in inlet velocity and temperature substantially influence thermal gradients, Nusselt number distribution, and overall exchanger performance. Complementing these findings, Bartecki (2015) developed a transfer function-based model to represent the frequency-domain thermal dynamics of a double-pipe exchanger, showing that linear system theory can accurately simulate transient heat responses. Fernández-Torrijos et al. (2016) expanded on these numerical insights by examining ε–NTU relationships for various flow configurations, confirming that counter-flow arrangements consistently exhibit superior thermal effectiveness compared to parallel flows. Together, these studies illustrate how CFD and mathematical models play a vital role in understanding and optimizing the thermal performance of double-pipe heat exchangers, particularly under counter-flow conditions, by predicting temperature distributions and enhancing geometric design precision.

Apparao, G. V., & Rao, K. S. (2019) conducted a computational analysis of a counter-flow double-pipe heat exchanger fitted with fins on the inner pipe surface, demonstrating that the inclusion of extended surfaces significantly increases the convective area and improves overall heat transfer rates. Abishek, King, and Narayanaswamy (2017) further investigated two-phase counter-flow evaporators using CFD simulations and observed that counter-flow orientation yielded a more uniform temperature distribution and higher local heat transfer coefficients than parallel configurations. Likewise, Kishan, R (2020) compared various exchanger geometries through CFD modeling and concluded that counter-flow systems provide the best compromise between heat transfer efficiency and pressure drop, highlighting their superior thermal management characteristics. Murthy, K (2015) experimentally validated CFD simulations for double-pipe exchangers and confirmed that numerical predictions closely matched measured temperature gradients, reinforcing CFD's reliability for real-world heat transfer applications. Collectively, these findings confirm that computational modeling not only aligns with experimental results but also offers predictive capabilities for performance evaluation, providing valuable design insights for improving efficiency in counter-flow double-pipe heat exchangers.

More recently, Bendaraa, Charafi, and Hasnaoui (2021) examined alumina-based nanofluids in double-pipe exchangers, finding that nanoparticle concentration enhances heat transfer efficiency by increasing thermal conductivity and reducing boundary layer resistance. Nwokolo, Mukumba, and Obileke (2020) evaluated the performance of a double-pipe exchanger integrated into a biomass gasification system and reported that optimal counter-flow velocity substantially improves thermal recovery and system performance. Rehman et al. (2020) introduced a hybrid CFD–porous medium model that accurately predicted the thermal performance of gas-to-gas micro heat exchangers, demonstrating strong correlation with experimental observations. Explored innovative counter-flow exchanger configurations and showed that optimized geometrical modifications can significantly enhance efficiency without introducing major pressure losses. These studies collectively emphasize that advanced CFD modeling techniques, combined with nanofluid and geometric optimization strategies, can substantially improve the thermal performance of counter-flow double-pipe exchangers. The findings reinforce that CFD remains a powerful design and analysis tool, providing predictive insight for industrial-scale and renewable energy heat transfer systems.

Reddy (2017) carried out a numerical study on a double-tube heat exchanger equipped with rectangular fins on the annular side, concluding that fin geometry and placement significantly influence the Nusselt number and overall heat transfer performance without causing excessive pressure losses. Mohapatra, Padhi, and Sahoo (2017) explored convective heat transfer within an inserted coiled tube exchanger, demonstrating that secondary flow generation leads to improved fluid mixing and enhanced convective coefficients. Fiaschi, Manfrida, Russo, and Talluri (2017) applied CFD simulations to redesign industrial heat exchangers, showing that optimized counter-flow configurations significantly boost waste heat recovery potential and overall system efficiency. In a related study, Shukla and Kishan (2020) analyzed latent heat storage systems with different flow conditions, finding that counter-flow orientations provide superior temperature uniformity and faster thermal charging and discharging rates. Together, these findings highlight the versatility of CFD in analyzing different configurations and underline the significant role of geometric optimization, fin design, and flow dynamics in improving heat transfer rates and thermal performance within counter-flow double-pipe heat exchangers.

Gabir and Alkhafaji (2021) conducted a comprehensive review of double-pipe heat exchanger design and optimization techniques, emphasizing CFD's growing importance in identifying critical design variables that influence performance under fluctuating thermal loads. Rehman et al. (2020) complemented this perspective by integrating CFD and porous-medium simulations to model microscale heat exchangers with high predictive accuracy, effectively bridging the gap between theory and experimental validation. Nwokolo, Mukumba, and

Obileke (2020) also employed numerical methods to optimize counter-flow exchangers for biomass gasification systems, demonstrating substantial improvements in thermal efficiency through CFD-based design tuning. Finally, Proposed innovative geometric configurations that enhanced heat transfer rates and minimized material stress, reinforcing CFD's value in validating design accuracy and optimizing heat exchanger performance. Collectively, these works affirm that CFD-based numerical studies are indispensable in advancing the understanding of counter-flow double-pipe exchangers, providing engineers with a powerful framework for improving heat transfer efficiency, reducing operational costs, and enabling energy-efficient system designs.

5. METHODOLOGY

CFD simulations were carried out in ANSYS Fluent to investigate the heat-transfer performance of a counter-flow double-pipe heat exchanger operating with water as the working fluid. The hot stream entered at 355 K, while the cold stream entered at 298 K. The flow velocity for both streams was fixed at 1.9 m/s, ensuring comparable flow conditions along the exchanger length. The numerical procedure included geometry creation, mesh generation, specification of boundary conditions, solver configuration, and post-processing of temperature and velocity fields.

5.1. Geometry Creation and Preparation

The model was constructed according to the following specifications:

- Fluid passage (inner region): 20 mm diameter, 1 mm wall thickness
- Surrounding annular passage: 28 mm diameter, 1 mm wall thickness
- Total heat-exchanger length: 300 mm

The model consisted of two fluid domains, representing the inner and outer pipe flow regions, along with two solid wall domains corresponding to the pipe boundaries. The geometry was imported into ANSYS Fluent Meshing using the Watertight Geometry workflow, ensuring a clean and continuous surface for meshing. Share topology was activated so that all adjacent surfaces shared common nodes, allowing proper interaction between the solid and fluid regions. This setup ensured accurate conjugate heat-transfer modeling, enabling realistic simulation of heat conduction through the pipe walls and convection between the hot and cold fluid streams.

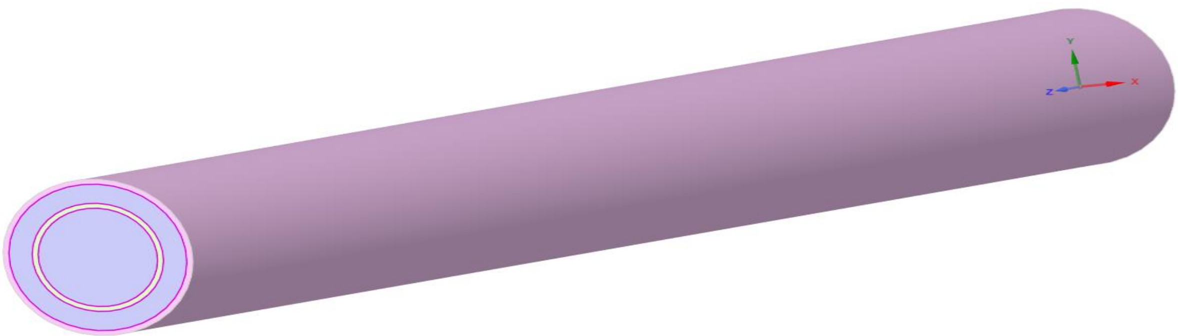

Figure 1: Geometry

5.2. Meshing and Boundary-Layer Setup

A high-quality mesh was generated with refinement applied to regions near the walls and within the narrow annular gap. The primary settings included:

- Element size: approximately 4 mm
- Growth rate: 1.2, ensuring smooth transitions
- Curvature and proximity-based refinement for circular and tight-gap regions
- Inflation layers: three layers on all fluid–wall boundaries to resolve steep gradients in velocity and temperature

A polyhedral volume mesh was created to improve solution stability and reduce numerical diffusion, providing accurate prediction of convective heat transfer along the exchanger.

5.3. 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.

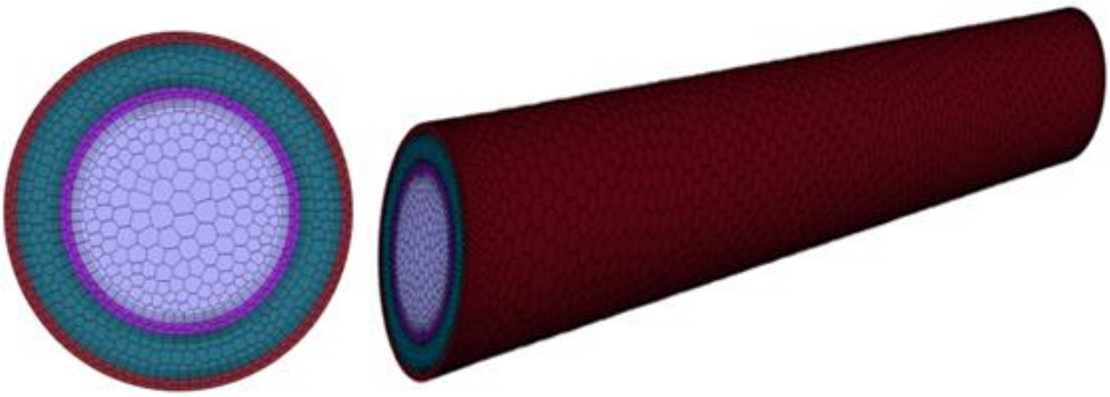

Figure 2: Meshing

5.4. Inlet and outlet

In a parallel-flow arrangement, the hot fluid (350 K) and cold fluid (298 K) enter from the same side and move in the same direction. In a counter-flow arrangement, the fluids enter from opposite ends and flow against each other, creating a larger temperature gradient and improving heat-transfer effectiveness.

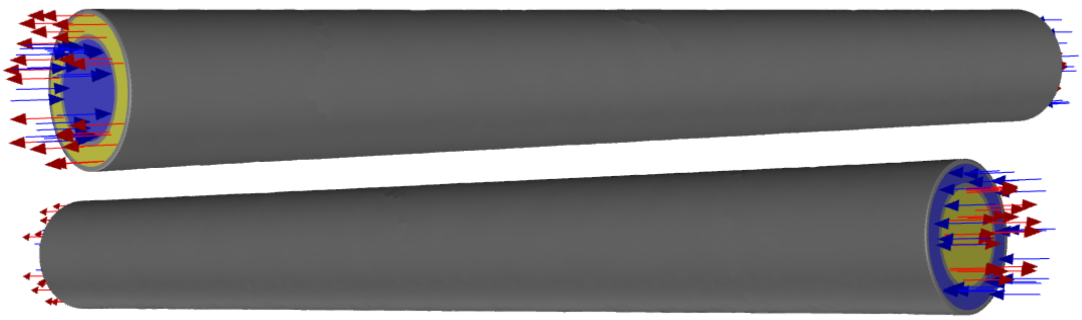

Figure 3: Inlet and outlet

5.5. Boundary Conditions and Region Setup

The fluid passages were defined as liquid water zones, and the surrounding walls were assigned as solid to enable accurate conjugate heat-transfer modeling. Counter-flow operation was established by applying velocity inlets of 1.9 m/s to both fluid streams, with one stream entering at 355 K while the opposing stream entered at 298 K. The downstream ends of both passages were set as pressure outlets with zero gauge pressure, and no-slip boundary conditions were applied to all internal solid surfaces. This configuration ensured realistic representation of the opposing flow directions and allowed efficient simulation of both convective heat exchange between the fluids and conductive heat transfer through the metal walls.

5.6. 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

5.7. 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 performance of the double-pipe heat exchanger was evaluated under two operating modes: parallel flow and counter flow. For both cases, the hot fluid entered at 355 K while the cold fluid entered at 298 K, with the flow rate kept constant. Outlet temperatures were obtained using area-weighted averaging to accurately assess the heat-transfer effectiveness of each configuration. The results for both cases are presented below.

6.1. Case 1: Parallel-Flow Condition

In the parallel-flow arrangement, the hot and cold streams moved in the same direction along the heat exchanger. The hot fluid temperature decreased from 355 K to 337.06 K, resulting in a temperature drop of 17.94 K. Meanwhile, the cold fluid temperature increased from 298 K to 321.25 K, gaining 23.25 K as it absorbed heat from the hot stream.

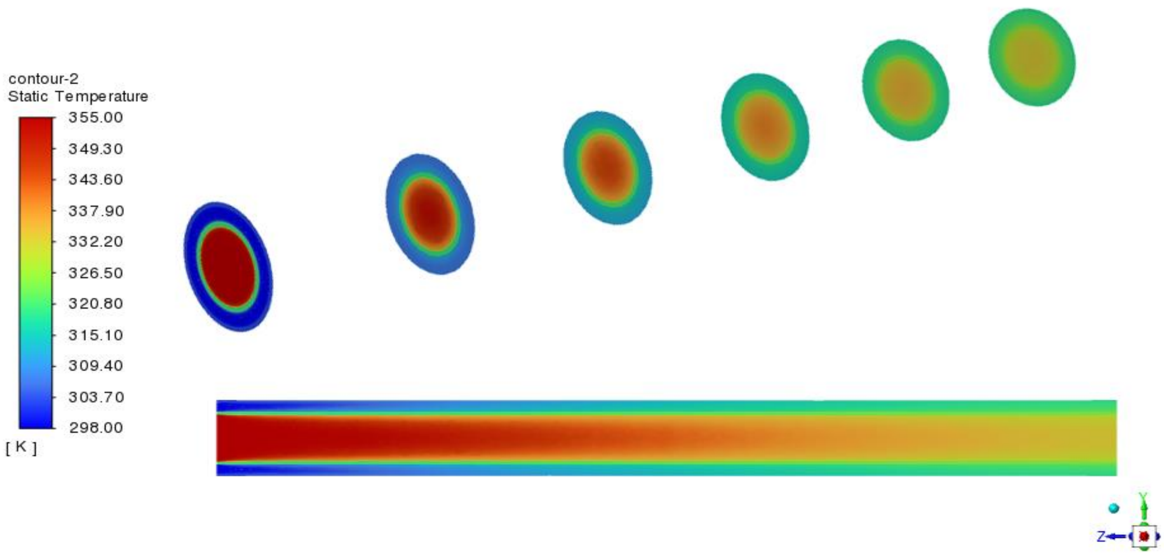

Figure 4: Parallel-Flow Condition

6.2. Case 2: Counter-Flow Condition

In the counter-flow configuration, the two fluids entered from opposite ends, maintaining a higher temperature gradient throughout the exchanger. The hot fluid cooled from 355 K to 327.77 K, achieving a larger temperature drop of 27.23 K compared to the parallel-flow case. The cold fluid warmed from 298 K to 324.11 K, which is higher than the outlet temperature obtained in parallel flow.

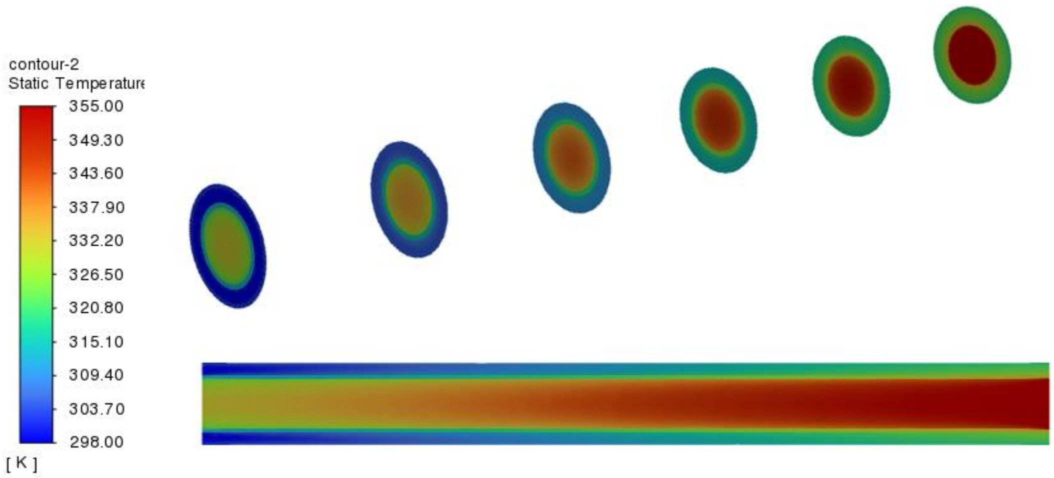

Figure 5: Counter-Flow Condition

6.3. Comparison Between the Two Cases

A comparison between the parallel-flow and counter-flow configurations highlights the differences in thermal performance achieved under each operating mode. Although both cases were operated with identical inlet temperatures and flow conditions, the resulting outlet temperatures show that counter flow provides superior heat-transfer effectiveness.

In the parallel-flow arrangement, the hot fluid exited at 337.06 K, while the cold fluid reached 321.25 K. The temperature difference between the outlets was relatively small, indicating that the thermal driving force diminished along the length of the exchanger, as expected in parallel-flow systems.

In contrast, the counter-flow configuration produced a significantly larger temperature drop for the hot fluid, cooling it to 327.77 K, while the cold fluid was heated to 324.11 K. The higher cold-fluid outlet temperature and greater cooling of the hot stream demonstrate the improved thermal gradient maintained throughout the exchanger when the flows move in opposite directions.

Overall, the counter-flow heat exchanger shows greater thermal efficiency, better heat recovery, and more effective cooling compared to the parallel-flow system.

Table 1: Comparison of Parallel-Flow and Counter-Flow Results

Condition	Hot Fluid Inlet (K)	Hot Fluid Outlet (K)	Hot Fluid ΔT (K)	Cold Fluid Inlet (K)	Cold Fluid Outlet (K)	Cold Fluid ΔT (K)
Parallel Flow	355	337.06	17.94	298	321.25	23.25
Counter Flow	355	327.77	27.23	298	324.11	26.11

7. CONCLUSION

The CFD-based numerical analysis of the double-pipe heat exchanger (DPHE) clearly demonstrates that counter-flow configurations exhibit superior heat-transfer performance compared to parallel-flow systems. Under identical inlet conditions of 355 K for the hot stream and 298 K for the cold stream, the counter-flow arrangement achieved a greater temperature drop of 27.23 K for the hot fluid and a higher outlet temperature of 324.11 K for the cold fluid. In contrast, the parallel-flow case resulted in a smaller hot-fluid temperature drop of 17.94 K and a cold-fluid outlet of 321.25 K, indicating reduced thermal effectiveness. These results confirm that the counter-flow setup maintains a more favorable and uniform temperature gradient along the exchanger length, leading to better utilization of the available temperature difference and enhanced overall thermal efficiency. The simulation outcomes align closely with experimental and analytical results from previous studies such as those by Apparao and Rao (2019), Bhattacherjee (2020), and Fernández-Torrijos et al. (2016), who also emphasized the effectiveness of counter-flow heat exchangers in maximizing heat recovery and energy utilization.

Furthermore, the results validate the accuracy and reliability of Computational Fluid Dynamics (CFD) as an analytical tool for predicting temperature distributions and optimizing heat exchanger designs. The application of CFD allowed detailed visualization of thermal and velocity fields, enabling a deeper understanding of flow interactions and wall heat transfer processes that traditional analytical methods often oversimplify. The counter-flow configuration demonstrated more efficient heat recovery with minimal pressure loss, confirming its advantage in maintaining strong thermal gradients throughout the exchanger. This study reinforces that CFD-driven investigations can effectively guide the design and optimization of energy systems by identifying parameters that maximize exchanger effectiveness, reduce thermal losses, and improve sustainability in industrial applications. Overall, counter-flow DPHEs represent the most efficient configuration for compact and high-performance heat transfer systems.

REFERENCES:

1. 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.
2. Kumar, S., Karanth, K. V., & Murthy, K. (2015). Numerical study of heat transfer in a finned double pipe heat exchanger. *World Journal of Modelling and Simulation*, *11*(1), 43-54.
3. Bhattacherjee, D. CFD Analysis of Double Pipe Counter Flow Heat Exchanger. *International Journal of Engineering Research & Technology (IJERT)(Oct, 2020) ISSN*, 2278-0181.
4. 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.
5. Reddy, N. S., Rajagopal, K., & Veena, P. H. (2017). Numerical investigation of heat transfer enhancement and pressure drop of a double tube heat exchanger with rectangular fins in the annulus side. *International Journal of Dynamics of Fluids*, *13*(2), 295-308.
6. Fernández-Torrijos, M., Almendros-Ibáñez, J. A., Sobrino, C., & Santana, D. (2016). ε–NTU relationships in parallel–series arrangements: Application to plate and tubular heat exchangers. *Applied Thermal Engineering*, *99*, 1119-1132.
7. 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.
8. Fiaschi, D., Manfrida, G., Russo, L., & Talluri, L. (2017). Improvement of waste heat recuperation on an industrial textile dryer: Redesign of heat exchangers network and components. *Energy Conversion and Management*, *150*, 924-940.
9. 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.
10. Siavashi, M., & Miri Joibary, S. M. (2019). Numerical performance analysis of a counter-flow double-pipe heat exchanger with using nanofluid and both sides partly filled with porous media. *Journal of Thermal Analysis and Calorimetry*, *135*(2), 1595-1610.
11. Bendaraa, A., Charafi, M. M., & Hasnaoui, A. (2021). Numerical and experimental investigation of alumina-based nanofluid effects on double-pipe heat exchanger thermal performances. *SN Applied Sciences*, *3*(2), 172.
12. Mohapatra, T., Padhi, B. N., & Sahoo, S. S. (2017). Experimental investigation of convective heat transfer in an inserted coiled tube type three fluid heat exchanger. *Applied Thermal Engineering*, *117*, 297-307.
13. Kishan, R., Singh, D., & Sharma, A. K. (2020). CFD Analysis of heat exchanger models design using ansys fluent. *International Journal of Mechanical Engineering and Technology*, *11*(2).
14. 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).
15. Abishek, S., King, A. J., & Narayanaswamy, R. (2017). Computational analysis of two-phase flow and heat transfer in parallel and counter flow double-pipe evaporators. *International Journal of Heat and Mass Transfer*, *104*, 615-626.

16. Rehman, D., Joseph, J., Morini, G. L., Delanaye, M., & Brandner, J. (2020). A hybrid numerical methodology based on CFD and porous medium for thermal performance evaluation of gas to gas micro heat exchanger. *Micromachines*, *11*(2), 218.
17. 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.
18. Shukla, P. K., & Kishan, P. A. (2019). CFD analysis of latent heat energy storage system with different geometric configurations and flow conditions. In *Proceedings of the 25th National and 3rd International ISHMT-ASTFE Heat and Mass Transfer Conference (IHMTC-2019)*. Begel House Inc..