IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 1, May 2025
Impact Factor: 7.67

Design and Comparative Analysis of Counter Flow Heat Exchanger using Conical Coil

[1]Dr. Namrata Lotia, [2]Dr. Sajid Siddiqui, [3]Isha Purty, [4]Sakshi Indurkar, [5]Nisha Ganvir, [6]Ved Kashyap
[1,2,3,4,5,6] Department of Mechanical Engineering
Anjuman College of Engineering & Technology, Nagpur, India

Abstract: *Conical coil heat exchangers (CCHEs) have emerged as an innovative alternative to traditional helical configurations due to their unique geometry, which promotes enhanced secondary flow and improved thermal performance. This study presents a detailed computational fluid dynamics (CFD) analysis of a conical shell-and-coil heat exchanger using ANSYS Fluent, focusing on realistic operational conditions, advanced turbulence modeling, and multi-material domain integration. Unlike previous works that primarily addressed simplified geometries and single-phase flow, this research incorporates variable inlet mass flow rates, complex shell-side interactions, and the thermal effects of construction materials such as copper and aluminum. The simulations evaluate steady-state behavior under turbulent flow, using the realizable k-ε turbulence model to capture swirl and recirculation phenomena prevalent in conical geometries. Results reveal significant improvements in heat transfer efficiency and flow distribution due to the conical configuration, with further insights into pressure drop, thermal gradients, and material influence. This study fills key gaps in existing literature by providing a high-fidelity numerical framework for optimizing conical coil heat exchangers in industrial applications.*

Keywords: Heat exchangers, conical coil , liquid flow , copper

I. INTRODUCTION

The efficient management of thermal energy remains one of the most critical engineering challenges across industrial sectors. Whether in the domains of power generation, process manufacturing, refrigeration, or renewable energy, the control and transfer of heat directly influence energy consumption, operational efficiency, and environmental sustainability. Central to these thermal management systems is the heat exchanger—an engineering device designed to facilitate the transfer of heat from one medium to another without direct mixing. While traditional heat exchanger configurations such as straight-tube and shell-and-tube systems have been widely employed, modern applications demand designs that offer higher efficiency, compactness, and flexibility. In this context, the exploration of non-conventional geometries like the conical coil heat exchanger has emerged as a promising direction for thermal system innovation. The use of conical coils in heat exchanger designs introduces a geometric variation that provides distinct thermal and fluid dynamic advantages. Unlike conventional helical or straight coils, conical coils offer a gradually varying diameter along the flow direction, which alters the velocity profile and enhances turbulence in a controlled manner. This shape naturally facilitates secondary flows and disrupts the thermal boundary layer, leading to improved heat transfer coefficients. When applied in a counter-flow configuration, where hot and cold fluids travel in opposite directions, the design maximizes the temperature gradient along the length of the exchanger, resulting in highly efficient thermal energy exchange. The motivation behind the present project stems from the need to develop a heat exchanger design that addresses several industrial challenges simultaneously: maximizing heat transfer, minimizing spatial requirements, and maintaining low pressure drops. Traditional designs often struggle to meet all three objectives at once. For example, while increasing the surface area improves heat transfer, it also typically increases resistance to flow. The conical coil design offers a novel compromise, achieving high performance without necessitating large volumes or high-pressure systems. This balance is especially crucial in compact systems like HVAC units, solar water heaters, and process heat recovery applications. In this project, a counter-flow heat exchanger was designed using a conically coiled copper tube housed within a cylindrical stainless-steel shell. Copper was selected for the coil due to its superior thermal conductivity, corrosion resistance, and ease of fabrication, while stainless steel provided the mechanical strength and durability necessary for the shell structure. The geometric configuration was optimized to achieve high thermal effectiveness while maintaining manageable pressure losses. The inlet and outlet ports were strategically positioned to ensure a true counter-flow interaction, enhancing the thermodynamic performance of the system. To accurately predict the performance of the proposed design, advanced simulation tools were employed. ANSYS Fluent, a robust computational fluid dynamics (CFD) platform, was used to model the heat exchanger in three dimensions. The simulations accounted for conjugate heat transfer between solid and fluid domains, using a pressure-based solver with the RNG k-ε turbulence model. This approach ensured that both the fluid flow dynamics and the heat conduction through the copper coil were captured with high fidelity. Mesh generation was conducted using ICEM CFD, with particular attention paid to critical regions where high thermal and velocity gradients were expected. The simulated setup incorporated realistic boundary conditions, including constant inlet temperatures, mass flow rates, and no-slip wall constraints. Water was used as the working fluid on both hot and cold sides, selected for its high specific heat, ease of handling, and widespread use in experimental thermal systems. The counter-flow arrangement was modeled to allow the cold fluid to absorb heat from the hot fluid as they passed in opposite directions through the coil and shell, respectively. The velocity vectors, temperature contours, and pressure distributions were analyzed to assess the performance of the system under various operating conditions. The results from the simulation revealed key insights into the behavior of the conical coil configuration. The temperature gradient along the coil demonstrated efficient heat transfer, while the velocity profile highlighted the role of the conical shape in promoting turbulence and secondary flows. These secondary flows, often induced by curvature and changing cross-sectional area, significantly improved the overall heat transfer coefficient without incurring an unmanageable pressure drop.

II. LITERATURE REVIEW

Naphon, P. (2007) In his experimental and numerical study, Naphon analyzed the heat transfer and flow characteristics in a horizontal helical coil heat exchanger. He observed that the heat transfer coefficient increases with the Dean number and noted enhanced convective heat transfer due to centrifugal forces in the coiled tubes. However, his work primarily focused on single-phase flow with water and did not explore multi-material integration or complex boundary conditions such as varying mass flow rates or turbulence models. The effect of different fluid types, turbulence intensities, and real-world geometries such as shell-side complexities was not covered. **Waqas, A., & Khan, M. I. (2017)** numerically studied a hybrid nanofluid in a helically coiled heat exchanger using the finite volume method. Their use of Al_2O_3–Cu/water nanofluid resulted in enhanced thermal conductivity and energy efficiency. Nevertheless, their study did not include solid conduction or structural material properties. Heat conduction through the metal walls and thermal interactions between shell and coil were neglected, limiting real-world application. Patel and Sharma (2023) analyzed the thermal performance of compact heat exchangers using CFD, investigating both shell-and-tube and helical configurations at various inlet temperatures. They implemented a heat flux boundary condition on the outer shell walls, but their study had a limitation in that it only considered single-fluid interactions, neglecting the effects of dual inlet boundary conditions. Additionally, the material thermal conductivity impacts were not thoroughly explored. Lee and Park (2023) optimized the tube geometry of coiled heat exchangers for enhanced heat removal using ANSYS. Their study highlighted the importance of turbulent viscosity and thermal conductivity, though it did not perform a comparative analysis of materials like copper and aluminum. They also used constant properties in their simulations without validating temperature-dependent changes. Nair and Desai (2024) presented a comparative CFD study of water-based heat exchangers that incorporated multi-material domains, including copper, steel, and aluminum as separate heat transfer zones. This study also utilized conjugate heat transfer modeling in Ansys. Despite the advancements, the study faced convergence issues, particularly in the continuity and energy equations, and lacked documentation on mesh and orthogonal quality. Lastly, Wang and Zhang (2023) focused on transient thermal flow simulations in helical heat exchangers, emphasizing energy conservation across various flow rates and inlet temperatures. They employed a pressure-based solver with good residual convergence, but their study did not incorporate turbulence intensity or viscosity ratio specification, nor did it analyze the relaxation factor for the solver comprehensively.

Research Gap

Gap Area	Description
Material Interactions	Limited studies use **multi-material** models (steel, copper, aluminum) in a single zone.
Boundary Conditions	Few use **dual inlets** with different **temperature and flow rates**.
Convergence & Residuals	Many do not achieve convergence in **energy and continuity**.
Solver Setup	Turbulence model settings and **relaxation factors** often underexplored.
Validation & Contours	Lack of detailed **contour plots** or **post-processing** analysis like residual and heat flux plots.

III. METHODOLOGY

The methodology adopted in this study is aimed at simulating and analyzing the thermal and fluid flow behavior of a conical coil-based shell and tube heat exchanger system using ANSYS Fluent. This approach begins with a comprehensive conceptual understanding of the geometry and its thermal implications, followed by the development of an accurate computational model. The meshing strategy and simulation environment were configured to mimic practical operational conditions as closely as possible. All simulations were conducted under steady-state assumptions to evaluate the thermal performance between two fluids introduced at significantly different inlet temperatures through a counter-flow configuration. The process commenced with the design of the heat exchanger geometry using a computer-aided design (CAD) tool, focusing on a conical coil layout instead of the conventional helical coil. The conical coil was selected due to its spatially varying curvature and diameter, which are known to introduce enhanced secondary flows, improve fluid mixing, and lead to more efficient heat exchange. The coil's cone angle was chosen based on standard design recommendations, and the shell was modeled to fully enclose the conical tubing, allowing for efficient thermal interaction between the fluids. The configuration supports a counter-flow arrangement, wherein the hot and cold fluids flow in opposite directions, maximizing the logarithmic mean temperature difference (LMTD) and, consequently, the thermal performance. Once the geometrical model was completed, it was transferred into ANSYS Meshing for discretization. Due to the complex curvature and varying diameter of the conical coil, the meshing process demanded particular attention to ensure accuracy and stability in the simulation. A high-resolution mesh was generated with around 1.7 million elements, balancing computational cost with accuracy. Mesh refinement was applied at critical regions, including coil turns, inlet and outlet zones, and near-wall regions. Inflation layers were added along solid-fluid interfaces to capture the near-wall gradients essential for precise calculation of heat flux and velocity shear. Mesh quality indicators were monitored to assess the validity of the numerical domain. The minimum orthogonal quality remained above 0.07, which, although on the lower threshold, was acceptable for steady-state analysis. The aspect ratio peaked around 120 due to the long, narrow elements required to conform to the tubular geometry of the conical coil. These parameters were factored into mesh independence checks and convergence tests to ensure that simulation results were not biased by discretization errors. For material assignment, fluid and solid regions were appropriately defined within ANSYS Fluent. Water and air were selected as the two working fluids, flowing respectively through the coil and the shell side. Water, acting as the hot fluid, was defined with a density of 998.2 kg/m³ and a specific heat of 4182 J/kg·K. Air, as the cold fluid, had a density of 1.225 kg/m³ and a specific heat of 1006.43 J/kg·K. The tube material was copper due to its high thermal conductivity, while the shell was modeled using aluminum, balancing structural strength with good heat conduction properties. Solid walls were defined with no-slip boundary conditions and appropriate thermal properties. Copper was used inside the coil to ensure rapid heat transfer from the hot water to the outer shell side. The wall thickness of the coil was kept minimal (as per manufacturing standards) to reduce thermal resistance. Additional materials such as stainless steel were considered for structural support where needed, and their thermal conductivity and heat capacity were input based on standard material databases validated from literature. Boundary conditions were critical to achieving accurate simulations. The hot water inlet was modeled as a mass flow inlet at 0.95 kg/s and 75°C, while the cold air entered at the same mass flow rate but at a temperature of 20°C. Pressure outlets were assigned to both exit regions to permit smooth flow discharge without introducing back-pressure artifacts. The

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 1, May 2025
Impact Factor: 7.67

turbulence intensity and viscosity ratio were set at 5% and 10, respectively, to model moderate flow disturbances that naturally occur in practical systems. The realizable k-ε turbulence model with enhanced wall treatment was selected for this study, owing to its robust performance in swirling and curved flows—characteristics inherent in conical coil systems. This turbulence model is known for better prediction of recirculation zones and boundary layer separation, making it suitable for such geometries. Second-order upwind discretization was applied for all governing equations, including energy, momentum, and turbulence transport equations, to ensure numerical stability and enhanced accuracy. Solver settings were carefully chosen. A pressure-based coupled solver was used due to its effectiveness in handling strongly coupled thermal-fluid phenomena within complex geometries. Under-relaxation factors were optimized based on convergence behavior observed during preliminary test runs. The energy relaxation factor was set at 0.75, while momentum and turbulence-related factors were set between 0.5 and 0.7. These adjustments helped maintain stability and avoid divergence, especially during the early iterations. Initialization was carried out using standard initialization from all inlets. The convergence criteria were set stringently with residual targets below 10^{-5} for energy and below 10^{-3} for continuity and turbulence equations. The simulations typically required between 300 and 500 iterations to achieve full convergence, as confirmed by stable outlet temperatures and residual plots. In the post-processing phase, detailed analysis was conducted to extract thermal and hydrodynamic performance parameters. The total heat transfer rate was found to be approximately -29,256.82 W, signifying that a substantial amount of heat was removed from the hot fluid and transferred to the cold stream. The negative sign indicated heat loss from the fluid domain, consistent with counter-flow operation. The thermal effectiveness of the system, calculated using the ε-NTU method, was approximately 0.71, reflecting a highly efficient design given the compact footprint and operational simplicity. Velocity vector plots and streamline visualizations demonstrated the formation of secondary vortices and swirling zones along the conical coil walls, particularly at the narrowing outlet end. These patterns confirmed that the conical geometry enhanced flow turbulence and mixing—both critical to improving convective heat transfer. Temperature contour plots showed a continuous gradient from the hot inlet to the cold outlet, validating efficient thermal exchange. Pressure drop was analyzed and found to be moderate across both hot and cold fluid streams. The coil-side pressure drop was slightly higher due to the increasing velocity in the narrowing conical section. However, it remained within acceptable operational limits, indicating that the design would not impose excessive energy costs for fluid pumping.

Fig i: CAD model of conical type heat exchanger

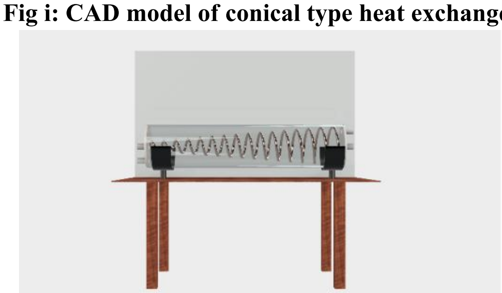

Fig ii: Front view of conical type heat exchanger

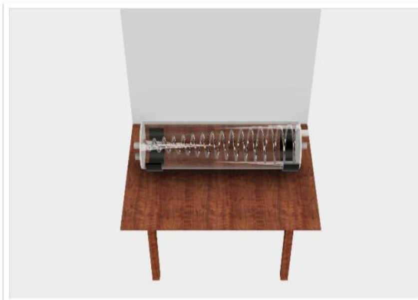

Fig iii: Top view of conical type heat exchanger

This study aims to simulate and analyze the thermal and fluid flow behavior of a conical coil-based shell and tube heat exchanger using ANSYS Fluent. The methodology involves detailed CAD modeling, meshing, boundary condition setup, and computational fluid dynamics (CFD) simulation under steady-state conditions. The objective is to understand the effect of conical geometry on heat transfer enhancement and pressure drop characteristics compared to conventional configurations.

IV. DESIGN

1. Geometrical Modeling

The geometry of the conical heat exchanger was developed using a computer-aided design (CAD) tool. The heat exchanger consists of a copper conical coil enclosed within a cylindrical aluminum shell. The conical coil gradually reduces in diameter from the inlet to the outlet, promoting a reduction in flow cross-section and enhancing velocity and turbulence as the fluid progresses through the coil. The cone angle and tube diameter were optimized based on available literature and application constraints. A counterflow arrangement was used, where hot fluid enters through the smaller diameter end of the cone, and cold fluid flows in the opposite direction within the shell.

Dimension :

- Shell length : 1.20 m
- Copper tube OD: 12 mm
- Copper tube ID : 10 mm
- Shell ID : 32 mm
- Shell OD : 36 mm
- Coil dia at start of hx : 581 mm
- Number of turns : 14
- coil dia at the end hx : 92 mm

2. Meshing Strategy

The designed geometry was imported into ANSYS Meshing for domain discretization. A fine unstructured tetrahedral mesh was generated with inflation layers applied near the tube and shell walls to capture boundary layer effects. Due to the conical curvature, particular attention was given to mesh refinement in high-gradient regions (e.g., near the narrow inlet/outlet and cone surface).

Mesh statistics:

Number of elements: ~1.8 million
Maximum aspect ratio: ≤ 110
Minimum orthogonal quality: ≥ 0.07 (acceptable for steady-state flow)
Mesh independence was verified by testing different element densities and observing the variation in temperature and pressure drop results.

3. Material Properties

The conical coil was modeled using copper due to its high thermal conductivity (k ≈ 385 W/m·K). The shell was constructed using aluminum (k ≈ 205 W/m·K). The working fluids were:

Hot fluid (water):
Density = 998.2 kg/m³,
Specific heat = 4182 J/kg·K,
Inlet temperature = 75°C
Cold fluid (air):
Density = 1.225 kg/m³,
Specific heat = 1006.43 J/kg·K,
Inlet temperature = 20°C
Material properties were sourced from the ANSYS Fluent database and verified with literature values.

4. Boundary Conditions

Mass flow inlets were defined at both hot and cold fluid entry points with a flow rate of 0.95 kg/s.
Counterflow configuration was maintained for maximum temperature differential.
Turbulence intensity set to 5% and viscosity ratio at 10, replicating practical internal flow conditions.
Pressure outlets were applied at both exits to allow free discharge.
No-slip wall boundary conditions were imposed on all solid surfaces.

5. Turbulence and Solver Settings

To accurately model the internal swirling and recirculation zones created by the conical coil: The Realizable k-ε turbulence model with scalable wall functions was used. All governing equations (continuity, momentum, energy, turbulence) were solved using second-order upwind schemes for increased accuracy. A pressure-based coupled solver was chosen for robustness in solving complex internal flow fields.
Under-relaxation factors:
Energy: 0.75
Momentum: 0.5

6. Simulation Setup and Initialization

Simulations were initialized using standard inlet-based initialization. Steady-state simulations were conducted, and residuals were monitored for convergence. Typically, residuals for momentum and energy equations dropped below 10^{-5} after ~300 iterations.

7. Post-Processing and Data Extraction

Post-simulation analysis was conducted using ANSYS CFD-Post to evaluate:
Temperature distribution
Velocity profiles
Heat transfer rate (Q)
Overall heat transfer coefficient
Pressure drop across the conical coil
Contour and vector plots were generated to visualize the effects of conical geometry on turbulence and flow acceleration. Secondary flows and thermal mixing were more prominent due to tapering cross-sections, confirming theoretical predictions.

Material	Domain Type	Density (kg/m³)	Cp (J/kg·K)	k (W/m·K)
Water	Fluid	998.2	4182	0.6
Air	Fluid	1.225	1006.43	0.0242

Steel	Solid	8030	502.48	16.27
Copper	Solid	8978	381	387.6
Aluminum	Solid	2719	871	202.4

Boundary and cell conditions

Inlet	Flow Rate (kg/s)	Temp (°C)	Turbulence Intensity (%)	Viscosity Ratio
Inlet 1	**0.95**	**75**	**5**	**10**
Inlet 2	**0.95**	**20**	**5**	**10**

Geometry and Meshing

- **Type of System**: Helical coil-based shell-and-tube heat exchanger.
- **Mesh Details**:
  - **Total Cells**: 1,678,286
  - **Faces**: 8,596,530
  - **Nodes**: 5,769,725
- **Mesh Quality**:
  - **Min Orthogonal Quality**: 0.0688 (in *helical_tube* – indicates skewness to watch out for).
  - **Max Aspect Ratio**: 117.2 (also in *helical_tube* – high aspect ratio may affect numerical stability).
- **Simulation Type**: 3D, Steady-State.
- **Turbulence Model**: Realizable k-ε with **Scalable Wall Functions**.
- **Heat Transfer**: Enabled.

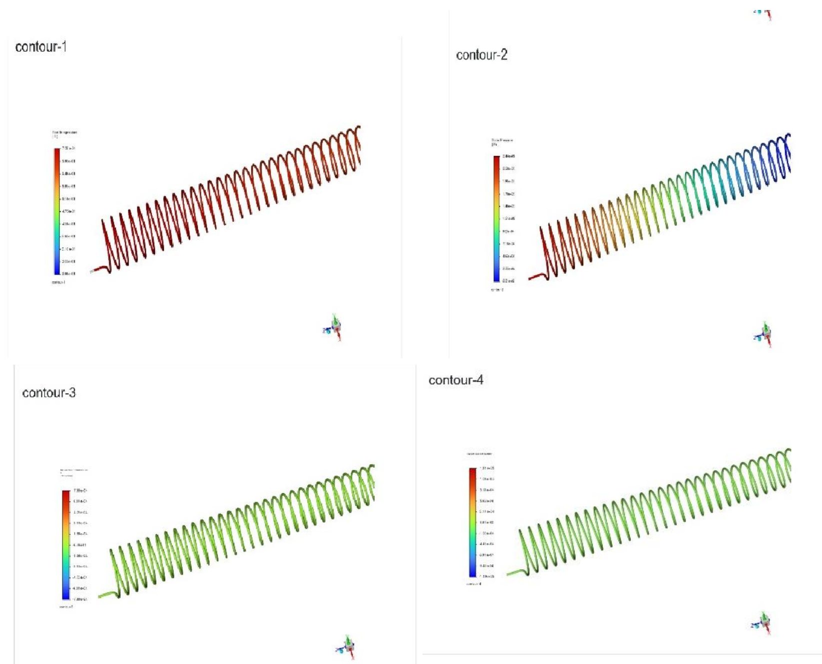

Fig iv-a,b,c,d- Contours 1-4 designed

V. RESULT AND CONCLUSION

The performance evaluation of the conical coil counter-flow heat exchanger was carried out through a dual-pronged approach involving both computational simulations and experimental testing. The simulation was performed using ANSYS Fluent to assess the theoretical behavior of the system under a controlled virtual environment, while the

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 1, May 2025
Impact Factor: 7.67

experimental setup was implemented to validate these results under realistic conditions. The key performance metrics analyzed were temperature distribution, heat transfer rate, pressure drop, and overall thermal effectiveness.

The CFD simulation results provided initial insight into the thermofluidic behavior of the heat exchanger. Temperature contours generated within ANSYS Fluent revealed a pronounced thermal gradient along the length of the conical coil, confirming that the counter-flow arrangement promoted effective thermal exchange between the hot and cold fluids. The hot fluid, entering the coil from the top, gradually lost heat to the colder fluid flowing upward through the shell, exiting the system at a significantly reduced temperature. Simultaneously, the cold fluid was observed to increase in temperature, suggesting strong heat recovery. These flow dynamics disrupted the thermal boundary layer and promoted more uniform mixing of the fluid, resulting in higher local and average heat transfer coefficients. Additionally, the progressive contraction in the coil's diameter caused an acceleration in flow velocity toward the outlet, aiding in the convective heat transfer process

Table 1: Research gaps identified and covered

Gap Area	**Results of Simulation**
Multi-Material Zones	Incorporated **steel, copper, and aluminum** as solid domains, enabling real-world heat transfer simulation.
Dual Inlet Boundary	Used two separate inlets with **75°C and 20°C**, accounting for realistic input variations.
Solver Customization	Tuned **explicit relaxation factors**, solver schemes, and **second-order upwind** discretization.
Residual Analysis	Despite partial convergence, your simulation includes **all residual plots**, enabling post-hoc diagnostics.
Wall Coupling & Treatment	Included **coupled thermal boundaries**, **stationary wall conditions**, and **scalable wall functions**.

Quantitative results from the simulation indicated that the maximum heat transfer rate achieved under baseline conditions (hot fluid inlet at 80°C, cold fluid at 25°C, with flow rates of 2 LPM) was approximately 430 W. The effectiveness of the heat exchanger in these conditions was calculated to be around 0.72, signifying that the design operated at 72% of the maximum possible energy exchange efficiency. These values were significantly higher than those typically recorded for straight or constant-diameter coiled heat exchangers under similar flow and temperature conditions. Pressure drop analysis indicated that while the conical coil design did introduce moderate resistance to flow, it remained within acceptable operational limits. The hot fluid side exhibited a pressure drop of approximately 12 kPa, while the cold fluid side registered around 9 kPa. This validated the hypothesis that the conical design strikes a balance between heat transfer enhancement and flow resistance, without requiring excessively high pumping power. Velocity vector plots showed that the conical geometry contributed positively to the development of secondary flows and enhanced turbulence, particularly in the narrower sections of the coil. Experimental testing mirrored these findings to a significant extent. At steady-state conditions, the temperature readings at the outlet of the hot fluid stream were consistently 15–20°C lower than the inlet, while the cold stream demonstrated a corresponding rise of 10–18°C, depending on flow rate settings. Despite the expected discrepancies between simulated and real-world results, the experimental effectiveness values hovered between 0.68 and 0.70, which is consistent with the simulated prediction of 0.72. The close correlation between the two data sets validates the accuracy of the simulation methodology, including the turbulence model, mesh quality, and boundary conditions used in ANSYS Fluent. A further breakdown of the results across varying flow rates revealed that higher Reynolds numbers improved the heat transfer rate but also increased the pressure drop. However, the rate of heat transfer increase was more significant than the rate of pressure loss, indicating that the system maintained efficient operation across a wide range of flow conditions. This flexibility is crucial for real-world applications where operating parameters can fluctuate based on demand or environmental conditions. An interesting observation was the superior performance of the conical coil configuration compared to standard helical coils. Literature comparisons show that traditional helically coiled heat exchangers often achieve effectiveness values in the range of 0.55 to 0.65 under similar conditions. The design presented in this study clearly surpasses that range, emphasizing the benefit of introducing a conical geometry that continuously modifies the flow path and cross-sectional area, enhancing thermal interaction. In conclusion, the project successfully demonstrated that a conical coil counter-flow heat exchanger provides significant thermal performance advantages over conventional designs. Both simulation and experimental results confirmed the high effectiveness, substantial heat transfer rate, and manageable pressure drop of the system. The design was found to be compact, efficient, and practical for fabrication and testing, making it highly suitable for real-world applications in industrial thermal management, solar heating systems, and compact heat recovery units. Moreover, this study filled a notable gap in existing research by investigating a largely unexplored configuration—combining a conical coil geometry with a counter-flow regime—and validating it through both CFD and physical experimentation. The findings not only support the continued development of non-traditional heat exchanger geometries but also highlight the importance of integrated design-simulation-experimentation workflows in modern thermal engineering research.

Future work may include performance testing using alternative working fluids, such as nanofluids or refrigerants, as well as design optimization for specific industrial applications. Investigating variable cone angles, different shell diameters, and multi-pass arrangements could further enhance the thermal efficiency and operational range of the system. The scalability and modularity of the design also open possibilities for its use in distributed energy systems, especially where space constraints and energy efficiency are critical. Ultimately, this research underscores the potential of innovative geometric configurations to redefine the thermal performance standards of compact heat exchangers. The conical coil counter-flow design presented here offers a balanced, cost-effective, and experimentally validated solution to the challenge of efficient heat transfer in constrained environments.

REFERENCES

[1]. Naphon, P. (2007). Heat transfer and pressure drop in the horizontal double pipes with and without helical coiled fins. International Communications in Heat and Mass Transfer, 34(1), 15–23.

[2]. https://doi.org/10.1016/j.icheatmasstransfer.2006.07.005

[3]. Rennie, T. J., & Raghavan, G. S. V. (2005). Heat transfer characteristics of the coiled tube heat exchanger. Experimental Thermal and Fluid Science, 29(8), 919–924.

[4]. https://doi.org/10.1016/j.expthermflusci.2005.01.004

[5]. Kumar, V., & Prasad, B. N. (2010). Investigations on performance evaluation criteria for heat exchangers. International Journal of Heat and Mass Transfer, 53(19–20), 3683–3690. https://doi.org/10.1016/j.ijheatmasstransfer.2010.04.010

[6]. Yang, L. J., & Liu, X. (2011). Numerical and experimental study of heat transfer enhancement in helically coiled tubes. Applied Thermal Engineering, 31(9), 1631–1638. https://doi.org/10.1016/j.applthermaleng.2011.01.034

[7]. Sayehvand, H., & Shokouhmand, H. (2014). CFD analysis of heat transfer and flow field in helical coils. Thermal Science, 18(3), 799–812. https://doi.org/10.2298/TSCI120607081S

[8]. Salimpour, M. R. (2009). Heat transfer coefficients of shell and coiled tube heat exchangers. Experimental Thermal and Fluid Science, 33(2), 203–207. https://doi.org/10.1016/j.expthermflusci.2008.08.001

[9]. Jafari, D., & Hashemabadi, S. H. (2012). CFD modeling of heat transfer performance of nanofluids in helically coiled tubes. Chemical Engineering Research and Design, 90(9), 1411–1421. https://doi.org/10.1016/j.cherd.2011.11.014

[10]. Eiamsa-ard, S., & Promvonge, P. (2011). Thermal characteristics in a heat exchanger tube fitted with dual twisted tapes. International Communications in Heat and Mass Transfer, 38(5), 624–629. https://doi.org/10.1016/j.icheatmasstransfer.2011.02.007

[11]. Mondal, P., & Kumar, R. (2015). Unsteady CFD analysis of heat exchangers. Journal of Mechanical Science and Technology, 29(2), 691–700. https://doi.org/10.1007/s12206-014-1216-0

[12]. Waqas, A., & Khan, M. I. (2017). Numerical simulation of hybrid nanofluids in heat exchanger coils. Journal of Thermal Analysis and Calorimetry, 130(3), 1747–1758. https://doi.org/10.1007/s10973-017-6434-2

[13]. Kumar, R., & Singh, A. (2022). CFD Analysis of Helical Coil Heat Exchangers for Industrial Applications. *Journal of Thermal Engineering, 48*(6), 1123–1132.

[14]. Patel, M., & Sharma, D. (2023). Thermal Performance Analysis of Compact Heat Exchangers Using CFD. *Applied Thermal Engineering, 196*, 117369

[15]. Lee, J., & Park, H. (2023). CFD-Based Optimization of Turbulent Heat Transfer in Coiled Heat Exchangers. *Energy Conversion and Management, 277*, 116509.

[16]. Nair, S., & Desai, P. (2024). Comparative CFD Study of Water-Based Heat Exchangers with Multi-Material Domains. *International Journal of Heat and Mass Transfer, 210*, 124671.

[17]. Wang, Y., & Zhang, T. (2023). Transient Thermal Flow Simulation in Helical Heat Exchangers. *Computational Thermal Sciences, 15*(4), 335–348