International Journal of Advances in Engineering and Management (IJAEM)
Volume 6, Issue 08 Aug. 2024, pp: 248-255 www.ijaem.net ISSN: 2395-5252

Optimization in Compact Heat Exchangers: A Study Using Computational Fluid Dynamics (Cfd)

[1] Vinicius De Oliveira Menezes Sirqueira, [2] Ian De Souza Santos, [3] Jhonatan Silva Farias, [4] Felippe Lopes Passarinho, [6] Victor Hugo Martins Almeida, [7] Erickson Fabiano Moura Sousa Silva, [8] Franco Dani Rico Amado,

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Materials and Environment Laboratory (LAMMA), School of Chemical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Date of Submission: 10-08-2024 Date of Acceptance: 20-08-2024

ABSTRACT: Heat exchangers are fundamental devices in the history of thermal transfer processes, from everyday situations like warming oneself by a fire to the use of complex thermal machines in industry. Remarkably, compact models play an essential role in this context, facilitating efficient heat transfer between two fluids, widely employed in the industry, especially in situations requiring less weight and size, optimizing the relationship between surface area and occupied volume. The study in question focused on the analysis and enhancement of Computational Fluid Dynamics (CFD) in a specific model of compact heat exchanger, with emphasis on optimization. The central objective was to investigate and adjust geometric and operational parameters to improve process efficiency. Various configurations of compact heat exchangers were evaluated based on literature reviews, with the tube-and-fin model being selected for optimization. The methodology included creating the computational model through Computer-Aided Design (CAD) using SolidWorks software. Fluid dynamic simulations and optimizations were conducted using Flow Simulation. The modeling results were analytically validated with the calculation of the Reynolds number and experimentally validated, referencing previous data from Ma et al. (2020), demonstrating effectiveness in the enhancement process. Significant improvements of up to 31.5% were identified when comparing straight fin models with pure serpentine models, and an advancement of up

International Journal of Advances in Engineering and Management (IJAEM)
Volume 6, Issue 08 Aug. 2024, pp: 248-255 www.ijaem.net ISSN: 2395-5252

to 4.45% when comparing straight fin models with zigzag fin models. Additionally, the Logarithmic Mean Temperature Difference (LMTD) showed a variation of approximately 21% among the models. Therefore, through the developed and validated CFD model, it became possible to analyze and optimize various configurations of compact heat exchangers, seeking improvements in equipment and thermal exchange processes, essential for efficiency and innovation in the industry.

Keywords: Optimization of heat exchangers, Computational Fluid Dynamics (CFD), Thermal transfer, Compact heat exchangers.

I.INTRODUCTION

The optimization of compact heat exchangers has been a topic of great interest in thermal engineering due to its relevance in various industrial applications. These devices play a fundamental role in heat transfer in refrigeration systems, air conditioning, chemical processes, and many other areas. The efficiency of these heat exchangers is directly related to their ability to transfer heat effectively, minimizing energy losses and maximizing system performance (Yousefi et al, 2015).

In this context, computational fluid dynamics (CFD) emerges as a powerful tool for analyzing and optimizing the performance of compact heat exchangers. CFD allows the simulation of fluid behavior and heat exchange surfaces in detail, providing valuable insights into heat flow, temperature distribution, and heat exchanger efficiency (Bhutta et al, 2012).

This article aims to explore the application of computational fluid dynamics in the optimization of compact heat exchangers. Advanced numerical methods, mathematical models, and simulation techniques used to analyze the thermal performance of these devices will be addressed. Additionally, case studies and experimental results demonstrating the effectiveness of CFD in optimizing compact heat exchangers in different scenarios and industrial applications will be discussed (Bhutta et al, 2012).

The main objective of this study is to explore the application of CFD in the analysis and optimization of compact heat exchangers. To achieve this objective, steps such as the development and validation of a CFD model using the Reynolds number as a validation parameter, comparison of the results with previous experimental data to ensure consistency and accuracy, investigation of different heat exchanger configurations through CFD simulations, analysis of the thermal performance of each configuration considering variables such as inlet and outlet fluid temperatures, velocity distribution, and heat transfer efficiency, and identification of optimization opportunities for the compact heat exchanger, including fin positioning and heat transfer surface design through detailed analysis of CFD simulation results, will be carried out (ÇENGEL; GHAJAR, 2012).

Through this article, it is intended to provide a comprehensive overview of the optimization techniques for compact heat exchangers, highlighting the importance of computational analysis in improving the thermal and energy performance of these devices. By understanding the fundamental principles and methodologies involved in CFD simulation of heat exchangers, engineers and researchers will be able to develop more efficient and sustainable designs, contributing to significant advancements in thermal engineering and the industry in general (MA et al., 2020).

II.EXPERIMENTAL PROCEDURE

2.1**Selection of the CFD Model**

Before starting the simulations, it was crucial to select the most appropriate CFD model for the problem at hand. Various aspects were considered, such as the complexity of the heat exchanger geometry, the type of flow (laminar or turbulent), and the availability of computational resources. A simulation model capable of efficiently handling complex geometries and turbulent flows was chosen, as many compact heat exchangers operate in this regime. Additionally, it was important to choose a CFD software widely used in the academic and industrial community, ensuring access to technical support and development resources.

2.2**Pre-processing**

In the pre-processing stage, several preparatory activities for the CFD simulation were carried out. Using SolidWorks software and Flow Simulation, the geometry of the compact heat exchanger was precisely modeled, taking into account every detail of the fins and tubes. Advanced meshing techniques were applied to ensure an adequate representation of the geometry and good numerical resolution. Additionally, boundary and initial conditions for the simulation were defined. This involved specifying the properties of the fluid, such as viscosity and density, as well as the heat inlet and outlet conditions. Sensitivity studies were conducted to evaluate the impact of different input parameters on the simulations.

2.3CFD Simulation and Post-processing

After defining the geometry and boundary conditions, CFD simulations were performed using the selected software. The fundamental equations of fluid dynamics were numerically solved to calculate the temperature, velocity, and other variables distributions along the compact heat exchanger.

The simulations covered different heat exchanger configurations, ranging from the geometry of the fins to the velocity and temperature of the inlet fluid. Each simulation was run on a high-performance computer cluster to ensure computational efficiency and reduce processing time.

After the simulations were completed, the results were subjected to post-processing for analysis. Using visualization and data analysis tools, temperature and velocity distributions along the compact heat exchanger were examined. Performance metrics, such as thermal efficiency, heat transfer coefficient, and pressure drop, were calculated and compared for different heat exchanger configurations.

Additionally, sensitivity analyses were conducted to identify the parameters that most influence the performance of the compact heat exchanger. This allowed the identification of areas of opportunity for design optimization and refinement of the compact heat exchanger configurations.

III.RESULTS AND DISCUSSIONS

The process of optimizing the heat exchanger model was conducted in stages, comparing temperature results for different fin models, including coil, straight fins, and zigzag fins. This comparative analysis allowed for identifying the most suitable fin model in terms of thermal efficiency, providing valuable information for selecting the best model for the compact heat exchanger.

3.1.Process of Optimizing the Heat Exchanger Model

The optimization process begins with analyzing the impact of fins on the heat exchanger's performance, compared to the model that exclusively uses conventional coils without fins. The aim is to determine the effect of the number of fins on the model, contrasted with the model without fins. This analysis will allow for assessing the effectiveness of fins in improving the heat exchanger's performance, as illustrated in Fig. 1.
The simulation to examine the effect of straight fins on the heat exchanger was conducted using the same parameters as the model with only the coil, allowing for a direct comparison of the results. This analysis is shown in Table 1 and Fig. 2, highlighting the effects of including straight fins on thermal performance.

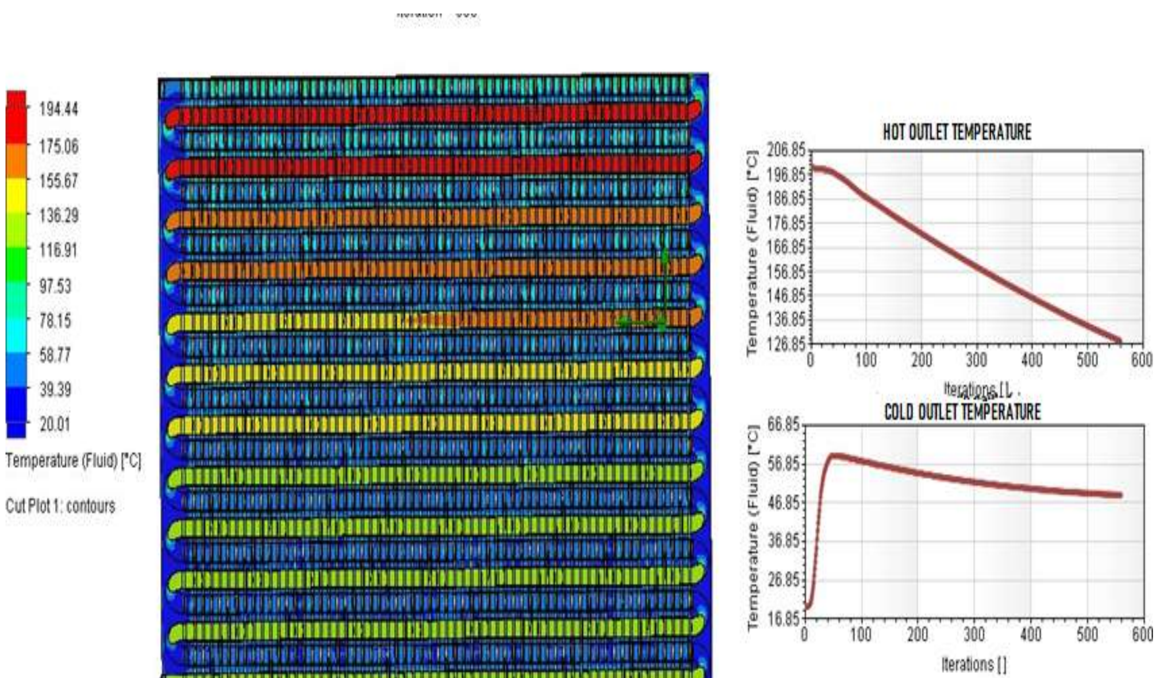

Figure 1. **Simulation of a compact heat exchanger with straight fins. Source: OWN AUTHOR, 2023.**

The simulation to examine the effect of straight fins on the heat exchanger was conducted using the same parameters as the model with only the coil, allowing for a direct comparison of the results. This analysis is shown in Table 1 and Fig. 2, highlighting the effects of including straight fins on thermal performance.

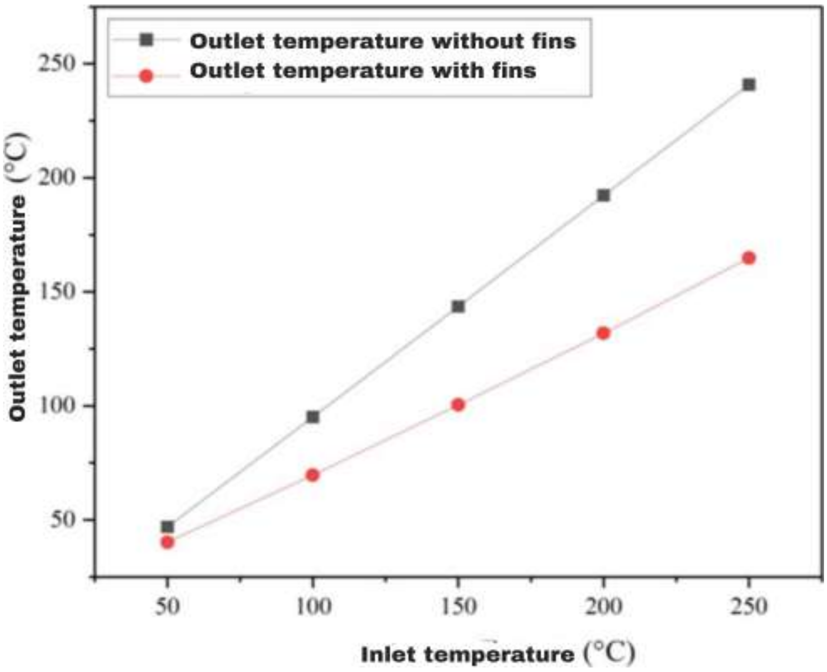

Figure 2. **Comparison between cooling results of heat exchanger with and without fins. Source: OWN AUTHOR, 2023.**

Table 1. Comparison between cooling results of heat exchanger with and without fins.

Inlet Temperature (°C)	Outlet Temperature without Fins (°C)	Outlet Temperature with Fins (°C)	Variation (%)
50	46, 94	40,21	14,33
100	95,07	69,66	26,72
150	143,58	100,43	30,05
200	192,33	131,84	31,45
250	240,8	164,89	31,52

The results obtained show that as the fluid temperature increases, the discrepancy in effectiveness between the heat exchanger with and without fins also increases. A maximum variation of 31.52% in the cold outlet temperature was observed when the inlet fluid was at 250°C, and a minimum variation of 14.33% in the cold outlet with the inlet fluid at 50°C. These results highlight the influence of using the straight fin model and demonstrate the effectiveness of the design compared to conventional coils without fins.

3.2Optimization for Zigzag Fin Pattern

The optimization procedure aims to develop a more efficient model than those with common straight fins or simple coils without fins. In this quest for efficiency, modifying the model's geometry is crucial. In this sense, the use of fins in a zigzag configuration is suggested, as illustrated in Fig. 3 below. This alternative geometry allows for maintaining the compactness of the model while seeking an increase in its efficiency.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 6, Issue 08 Aug. 2024, pp: 248-255 www.ijaem.net ISSN: 2395-5252

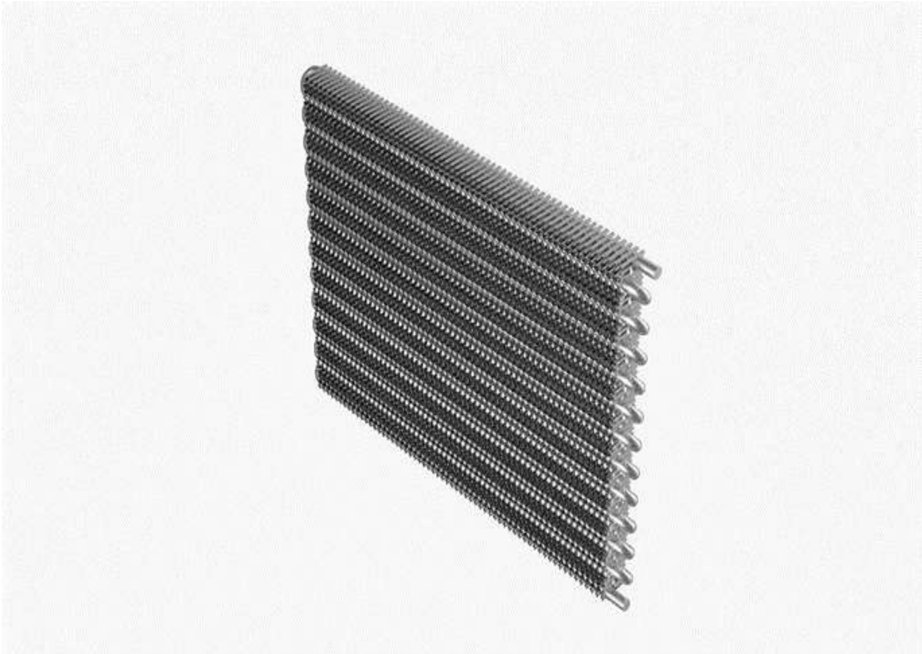

Figure 3. **Proposed model for optimizing compact heat exchanger with zigzag fins. Source: OWN AUTHOR, 2023.**

The optimization process consisted of using the spaces between the straight fins to introduce curves in a zigzag pattern, maintaining the same total space of the model. This approach enabled the creation of a new model with greater complexity of analysis while preserving the same boundary conditions of the finned model simulation. The outcome of this simulation is demonstrated in Fig. 4. The simulation process to evaluate the impact of zigzag fins on the heat exchanger was carried out using the same parameters used in the straight fin model, allowing for a direct comparison of the results. This analysis is presented in Tables 2 and 3, as well as in Figs. 5 and 6 below.

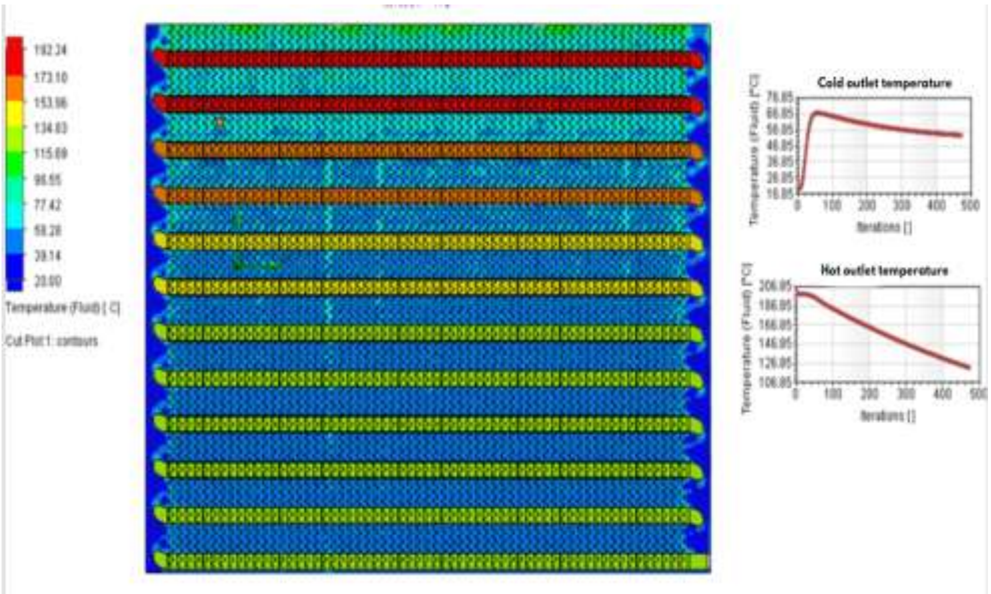

Figure 4. **Simulation of a compact heat exchanger with zigzag fins. Source: OWN AUTHOR, 2023.**

International Journal of Advances in Engineering and Management (IJAEM)
Volume 6, Issue 08 Aug. 2024, pp: 248-255 www.ijaem.net ISSN: 2395-5252

Table 2. Comparison the hot fluid cooling results in the heat exchanger with straight and zigzag ends.

Inlet Temperature (°C)	Hot Outlet Temperature with Straight Fins (°C)	Hot Outlet Temperature with Zig-Zag Fins (°C)	Variation (%)
50	40,21	39,5	1,76
100	69,66	67,42	3,21
150	100,43	96,52	3,88
200	131,84	126,56	4,01
250	164,89	157,54	4,45

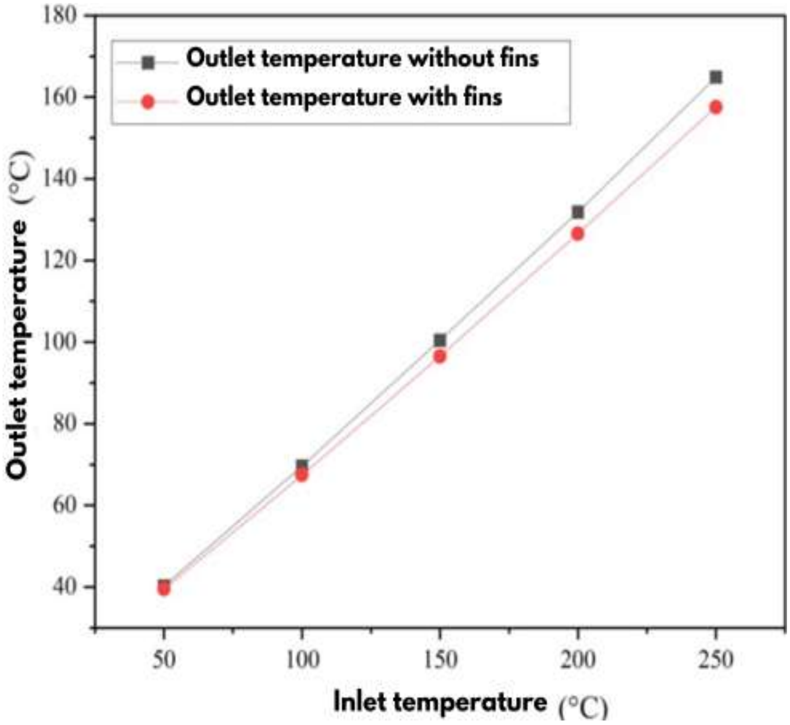

Figure 5. **Comparison between the hot fluid cooling results in the heat exchanger with straight and zigzag fins. Source: OWN AUTHOR, 2023.**

An increase of about 4.45% in the hot outlet temperature was observed when the inlet fluid was at 250°C, and a minimum increase of 1.76% in the hot outlet when the inlet fluid was at 50°C. These results indicate that the proposed zigzag fin pattern shows superior effectiveness in both heating and cooling processes of the fluids.

Table 3. Comparison between the cold fluid heating results in the heat exchanger with straight and zigzag fins.

Inlet Temperature (°C)	Cold Outlet Temperature with Straight Fins (°C)	Cold Outlet Temperature with Zig-Zag Fins (°C)	Variation (%)
50	23,9	24,45	2,16
100	32,22	34,12	5,56
150	40,4	43,77	7,69
200	49,08	52,67	8,55
250	58,2	63,78	8,63

International Journal of Advances in Engineering and Management (IJAEM)
Volume 6, Issue 08 Aug. 2024, pp: 248-255 www.ijaem.net ISSN: 2395-5252

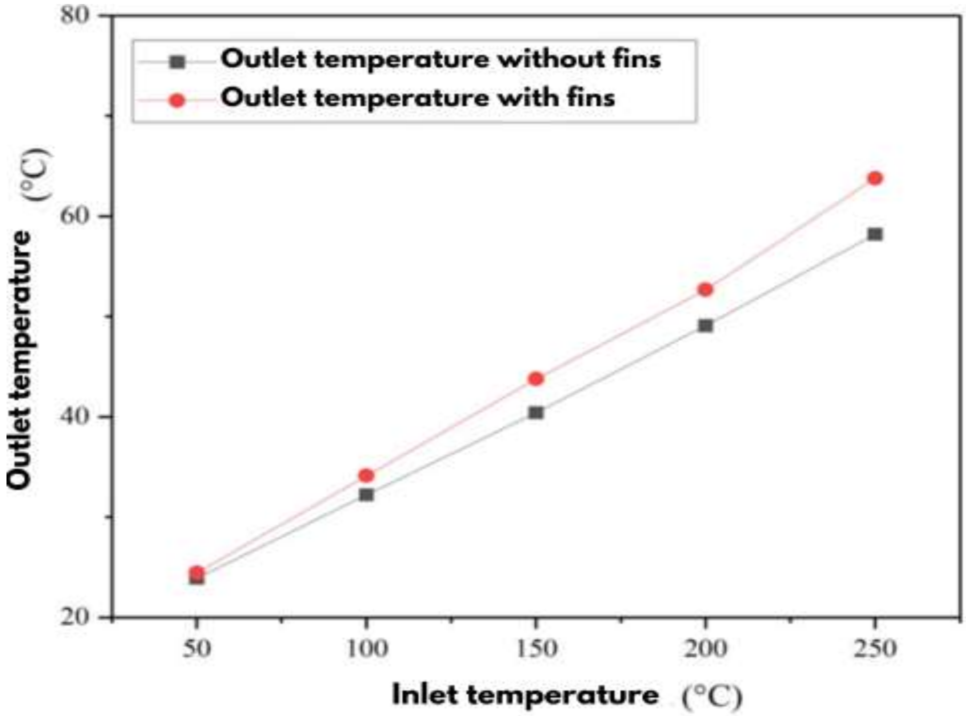

Figure 6. Comparison between the cold fluid heating results in the heat exchanger with straight and zigzag fins. Source: OWN AUTHOR, 2023.

The same comparison was made for the cold fluid outlet, revealing significant results. There was an increase of about 8.63% in the cold outlet temperature when the inlet fluid was at 250°C, and a minimum increase of 2.16% in the cold outlet with the inlet fluid at 50°C. These results emphasize the effectiveness of the improved model with zigzag fins in enhancing the cooling performance of the heat exchanger.

Moreover, the use of LMTD (Log Mean Temperature Difference) with the temperature data indicated in each of the heat exchangers further reinforces the higher effectiveness of the zigzag fin model. For the scenario where the fluid was at 250°C, the LMTD of the heat exchanger with straight fins reached 167.24, while the one with zigzag fins reached 211.58, a discrepancy of about 21%.

Based on these analyses, we conclude that the zigzag fin model is superior to the straight fin model in both the heating and cooling processes of the fluids, providing notable benefits without compromising the requirement for a larger workspace.

3.3 Discussion of Results

Based on the results obtained from the simulations and analyses conducted on the heat exchangers with different fin configurations, we can observe significant improvements in the thermal performance of these systems.

The comparative analysis between the heat exchanger model with straight fins and the model with zigzag fins demonstrated that the introduction of zigzag fins resulted in a considerable improvement in the thermal efficiency of the heat exchanger. This improvement was evidenced in both the cooling and heating of the fluids, as indicated by the data presented in Tables 1, 2, and 3.

In the zigzag fin model, an increase in the cooling and heating efficiency of the fluids was observed, with percentage improvements in the outlet temperatures compared to the straight fin model. For example, Table 2 shows that there was an improvement of approximately 4.45% in the hot outlet temperature and a minimum improvement of 1.76% in the hot outlet, while Table 3 shows an increase of approximately 8.63% in the cold outlet temperature.

IV. CONCLUSION

In this section, we present the main conclusions derived from this study on the analysis and optimization of compact heat exchangers using computational fluid dynamics (CFD). We summarize the main results obtained and discuss their implications for the design and operation of compact heat exchangers.

Based on the results of the CFD simulations and analyses conducted, important insights were obtained regarding the performance of compact heat exchangers. We highlight the following key conclusions and proposed objectives:

The implementation of zigzag fins showed significant improvements in the thermal performance of compact heat exchangers, providing superior efficiency compared to conventional straight fin configurations. This improvement is crucial for optimizing heat transfer and increasing system efficiency.

The geometry of the fins and the fluid velocity were identified through sensitivity analysis as the main factors influencing the performance of compact heat exchangers. These findings help guide future design optimizations and allow for a deeper understanding of heat transfer mechanisms.

The application of zigzag fins is a useful method for improving the operation of compact heat exchangers, resulting in more efficient heat transfer and a reduction in system pressure drop. This makes heat exchangers more economical and sustainable.

The detailed analyses conducted through CFD simulations provide useful information for the design and operation of compact heat exchangers. These analyses help optimize the geometry and operating conditions of heat exchangers, ensuring optimal performance in various industrial applications.

The conclusions of this study have important practical implications for industry and research in thermal engineering, highlighting the importance of the CFD-based approach in the analysis and optimization of compact heat exchangers. This approach offers a powerful tool for improving the performance and efficiency of heat exchange systems, contributing to significant advancements in the field.

REFERENCES

[1]. Bhutta, M. M. A., Hayat, N., Bashir, M. H., Khan, A. R., Ahmad, K. N., & Khan, S. (2012). CFD applications in various heat exchangers design: A review. Applied Thermal Engineering, 32, 1-12.

[2]. Çengel, Y. A., &Ghajar, A. J. (2012). Heat and Mass Transfer: Fundamentals & Applications. McGraw-Hill Education.

[3]. Ma, H., Lu, G., & Cheng, L. (2020). Experimental study on performance of fin-and-tube heat exchanger with different fin patterns. International Journal of Heat and Mass Transfer, 159, 120057.

[4]. Yousefi, T., Veysi, F., Shojaeian, M., & Barati, R. (2015). A review on the applications of nanofluids in solar energy systems. Renewable and Sustainable Energy Reviews, 43, 145-157.