Computational Fluid Dynamics and Thermal Performance Optimization of a Shell and Tube Heat Exchanger

Muhammad Abubakar[1*], Mohsin Siddique[1], Shummaila Rasheed[1*] and Mahabat Khan[1]

[1]Department of Mechanical Engineering, Capital University of Science and Technology, 45750,
Islamabad, Pakistan
`shummaila@cust.edu.pk`

Abstract. This study provides an in-depth computational fluid dynamics (CFD) analysis and thermal performance optimization of a shell and tube heat exchanger, a critical component in Heating, Ventilation, and Air Conditioning (HVAC) systems. The heat exchanger, procured from McMaster-Carr, was optimized using CFD simulations and AHED software, focusing on key design parameters such as tube configuration, baffle spacing, and material selection. The study identified the tube length and surface area as pivotal factors in enhancing heat transfer efficiency. Through simulations, temperature and pressure distributions were analyzed, leading to design modifications that improved thermal efficiency by approximately 10% and reduced pressure drop by 52% within the exchanger. The optimized heat exchanger demonstrated enhanced performance, contributing to more energy-efficient HVAC systems.

Keywords. Computational Fluid Dynamics, Heating, Ventilation and Air conditioning, heat exchanger, energy efficiency

1 INTRODUCTION

Shell and tube heat exchangers play a vital role in HVAC systems, enabling efficient thermal management by facilitating heat transfer between fluids without mixing them. Optimizing these exchangers is critical for improving overall energy efficiency, especially in industrial applications. This study focuses on the computational fluid dynamics (CFD) analysis and redesign of a shell and tube heat exchanger, selected from the McMaster-Carr catalog, a trusted source of industrial components. To enhance the thermal performance of the existing design, this research integrates CFD simulations with AHED software, a specialized tool known for its precision in simulation and performance prediction. The optimization process involves refining key design parameters, including

I. U. Ahad (ed.), *Proceedings of the 4th International Conference on Key Enabling Technologies (KEYTECH 2024)*, Atlantis Highlights in Engineering 35,
https://doi.org/10.2991/978-94-6463-602-4_43

tube configuration, baffle spacing, and material selection. Passive methods such as extended surfaces, coiled or twisted tubes, and surface treatments, along with active techniques like surface vibration and electrostatic fields, were considered to improve heat transfer efficiency [1]. Finned tubes, which are effective for fluids with low heat transfer coefficients like gases, increase both the film coefficient and surface area but may result in higher pressure drops. Conversely, tube inserts, particularly beneficial for high-viscosity fluids, enhance turbulence and significantly boost heat transfer coefficients, albeit often at the cost of increased pressure drop. This study aims to optimize the thermal performance of the shell and tube heat exchanger by balancing these factors to achieve a design that is lighter and more energy-efficient than existing models [2,3]. The analysis was conducted using ANSYS software to validate the CFD results, while AHED software was employed to verify and fine-tune the design, ensuring that the optimized heat exchanger meets the desired performance criteria.

2 METHODOLOGY

The methodology employed in this research aimed to redesign and perform CFD analysis of the heat transfer of a heat exchanger for its thermal performance optimization. The first step was to find a suitable heat exchanger already made for design purpose. After that the values from the data sheet were to be placed in the AHED software. Which shows all the details of this heat exchanger and what type of flow is best suitable for a heat exchanger, with the primary objective of designing an already made heat exchanger, the next step was to calculate the values such as length, areas of the tubes and surface areas. The initial design of the heat exchanger, including the shell size and tubes, was based on a reference paper [4]. The next step was analysis. In this part the temperature contours, pressure contours and flow were analyzed, this was analyzed on Ansys fluent to evaluate its performance. Subsequently, different graphs were also made in the process that are static temperature vs position and static pressure vs position.

2.1 Design of Heat Exchanger

A heat exchanger is a critical component in heat transfer applications, where its performance is influenced by several factors, including the type of fluid used, tube arrangement, number of passes, and other key parameters. Achieving optimal performance requires the selection of suitable materials, precise dimensions, and an appropriately sized design. The dimensions of the heat exchanger and its tubes in this study are based on guidelines provided in a reference paper [4]. In this research, various dimensions of the heat exchanger were calculated, as some required values were not provided and needed to be derived using established formulas. This included determining the tube length, tube area, and overall surface area of the heat exchanger. The outlet temperature on the hot side was calculated using Equation (1), ensuring accurate thermal performance evaluation.

$$Q = \dot{m}\, C_p\, \Delta T_h \text{ Corresponds to } T_{h_o} = T_{h_i} - \frac{Q}{\dot{m}\, C_p} \tag{1}$$

Temperatures in the shell and tubes were calculated by

$$T_{shell} = \frac{T_{h_o}+T_{h_i}}{2} + 273 \quad ; T_{Tubes} = \frac{T_{C_o}+T_{C_i}}{2} + 273 \tag{2}$$

$$A_{Tubes} = \frac{Q}{U\Delta T} \tag{3}$$

Pressure drop in the shell was calculated by

$$\Delta P_{shell} = \frac{FGs\,(Nb+1)De}{2\rho De\phi} \tag{4}$$

Length of the tubes was calculated by

$$L_{Tubes} = \frac{A_{Tubes}}{n\pi D_o} \tag{5}$$

To ensure efficient heat transfer and optimal performance of the heat exchanger, the tube length plays a crucial role in regulating these parameters. After conducting detailed calculations, the tube length was optimized to 2.5 meters, with a surface area of approximately 3.77 m². These dimensions were determined to be optimal for achieving the desired operational efficiency of the heat exchanger. The layout of a tubeplate for a heat exchanger, featuring a squared (90°) tube arrangement with a tube pitch of 34 mm. The setup includes 24 tubes distributed in a (12, 12) pattern, with a tube-to-tube spacing of 14 mm. The tubeplate thickness is 40 mm, and the design supports 2 passes with a horizontal division plate layout. The shell cross-sectional area is 0.03 m², while the tubes' cross-sectional area per pass is 2.79E-3 m², and the perimeter is 699 mm.

2.2 CFD Analysis of Tubes within Heat Exchanger

A finite volume method was carried out to examine the heat transfer properties of the heat exchanger. This investigation sought to determine the exact arrangement of the tubes that were subject to greater heat transfer and pressure durability, allowing for further analysis.

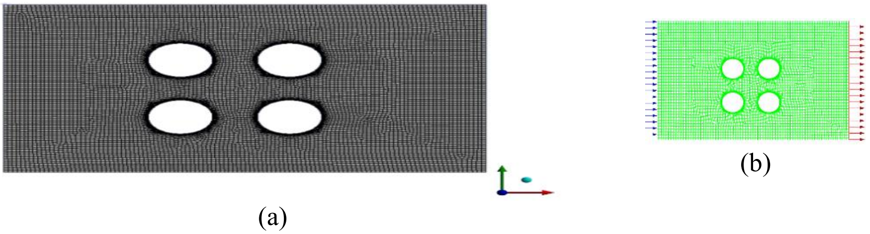


Fig. 1. (a) Mesh of Tubes in a square alignment (b) Boundary condition and CFD of the tubes

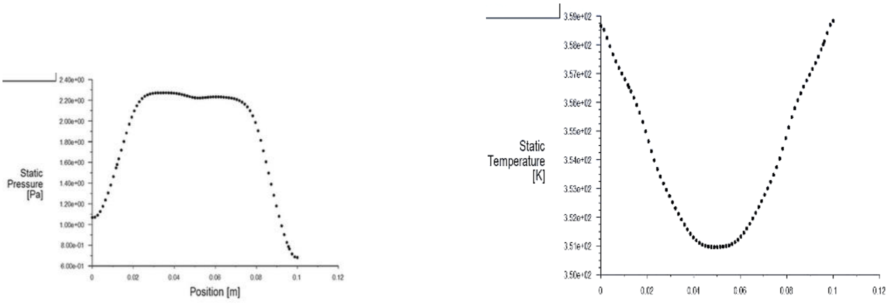
The mesh was generated with a size of 5 mm, ensuring a detailed representation of the heat exchanger geometry. Boundary conditions were applied at the inlet, tube arrangement, and outlet, with various fluids introduced at specific mass flow rates to simulate realistic operational conditions, as depicted in Figure 1. A mesh convergence study was conducted to verify the accuracy of the computational results, leading to a final mesh consisting of 278,145 elements and 368,905 nodes. The analysis produced a temperature value of 357 K and a pressure drop of 6.537 Pa within the shell, as illustrated in Figure 2. Table 1 details the parameters utilized in the CFD analysis of the heat exchanger, which was constructed from steel [5]. These parameters were essential in accurately predicting the thermal and fluid dynamic behavior of the heat exchanger.

Table 1. Known Parameters Of Heat Exchanger

Description	Value
Duty (Q)	$3.95 \times 10^5\ W$
Hot side inlet temperature (T_{h_i})	$82.2°C$
Cold side inlet temperature (T_{c_i})	$29.44°C$
Shell Diameters (D_i & D_o)	222.5 mm & 262.5 mm
Tube Diameters (D_i & D_o)	17.5 mm & 20 mm
Number of Tubes and passes	24 tubes and 2 passes
Mass Flow Rate (kg/hr)	40000 on both Shell side and Tube side
Flow Type	Turbulent

2.3 Thermal Performance Optimization of Shell and Tube Heat Exchanger

To validate the findings, a detailed CFD analysis was conducted. The primary objective of this analysis was to generate optimized graphs that illustrate the thermal and pressure behavior of the entire heat exchanger. Specifically, temperature and pressure distribution graphs were produced. Boundary conditions were meticulously defined on a 2D plane to simulate the crossflow arrangement of fluids with differing temperatures passing through the shell and tubes. These conditions were applied to the tubes and the designated region representing the shell, as depicted in Figures 2(a) and 2(b).



(a)

(b)

FIGURE 2. (a) Static Pressure Vs Position (b) Static Temperature Vs Position

Following the generation of the initial graphs, a comprehensive analysis was performed using AHED software. This tool provided critical input parameters, such as temperature settings, baffle quantity and configuration, pitch, and tube alignment (whether staggered or squared). Upon entering these values, AHED software evaluated the performance of the design and generated a corresponding CAD model. The software's output included not only temperature-related graphs but also additional graphs depicting the relationship between Nusselt and Reynolds numbers for both the tube and shell sides, as illustrated in Figures 4(c) and 4(d).

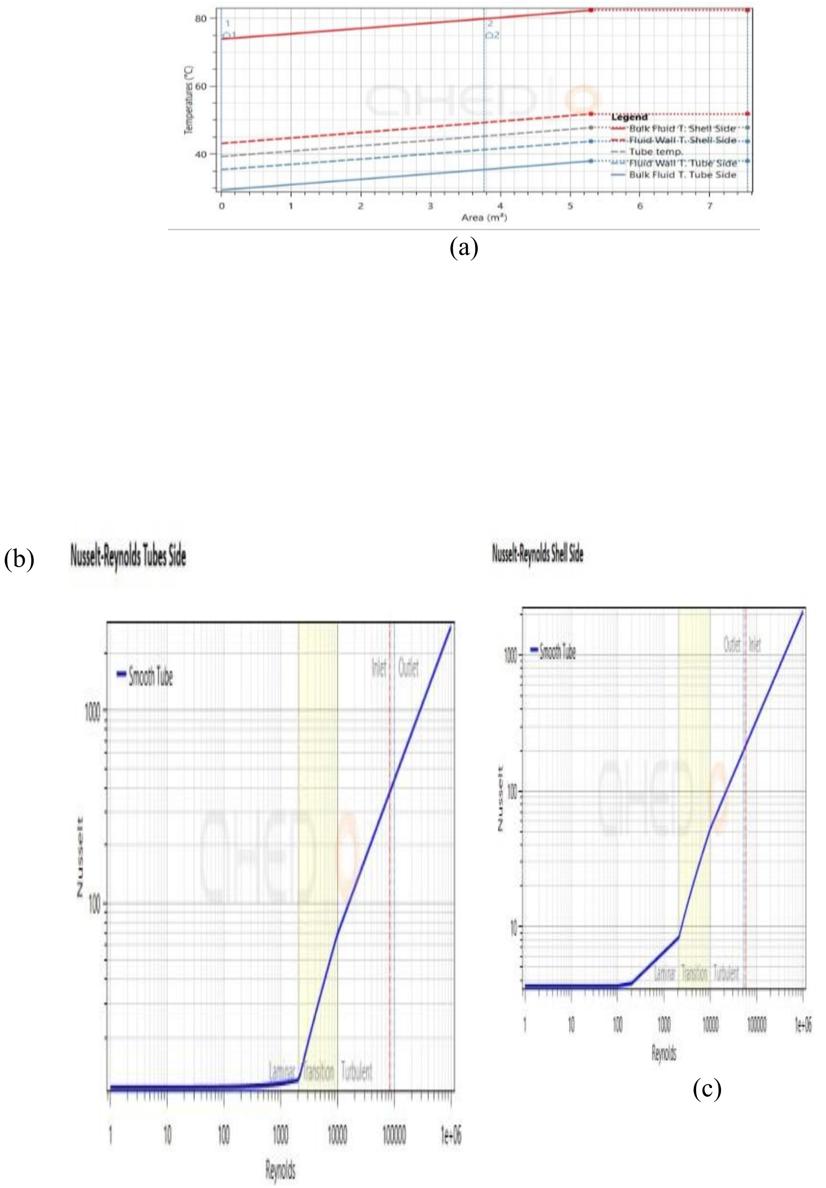
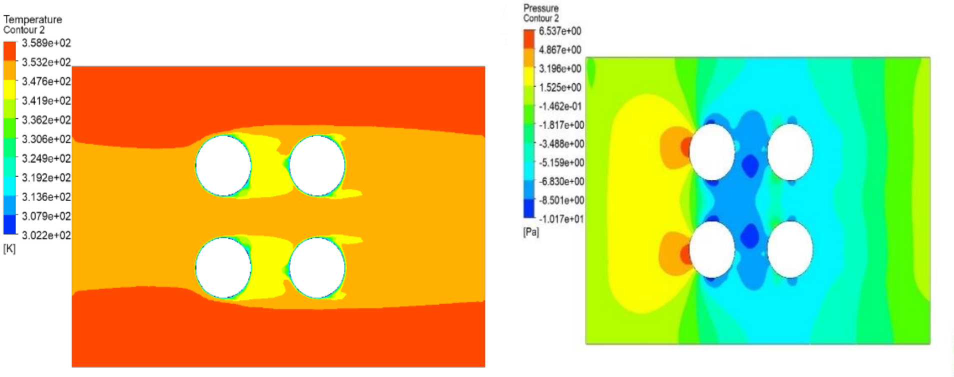


Fig. 4. **(a)** Temperature graph, **(b)** Tubes side Nusselt vs Reynolds and **(c)** Shell side Nusselt vs Reynolds from Ahed Software

3 RESULTS AND DISCUSSIONS

Following the calculation that was done and the CFD analysis of the model. Subsequently, a mesh convergence study was conducted to ensure accurate and reliable CFD analysis [5]. The initial design of the heat exchanger exhibited a moderate temperature gradient across the tubes and a discernible pressure drop across the shell, indicating inefficiencies. Optimization efforts led to a more uniform temperature distribution and a reduced pressure drop, significantly enhancing heat transfer efficiency and reducing energy losses. AHED (Advanced Heat Exchanger Design) software, which specializes in designing shell and tube heat exchangers and includes a Fluids and Mixing Assistant, was instrumental in this process. By utilizing its client-server architecture and cloud-based engine, AHED provided detailed thermodynamic analysis and allowed for precise adjustments to design parameters. This comprehensive analysis, combined with the data from McMaster-Carr, validated the improvements observed in temperature distribution and pressure drop through ANSYS analysis, resulting in a heat exchanger with greater reliability and effectiveness for industrial applications.



(b)

(a)

Fig. 5. (a) Temperature Contour and **(b)** Pressure Contour

Table 2. Optimal Design Parameter

Description	Calculated	Analyzed
Tube Length (m)	2.5	-
Tube Area (m^2)	3.77	-

Temperature in Shell (K)	350	358
Pressure in Shell (Pa)	13.612	6.537

4 Conclusion

The computational fluid dynamics (CFD) and thermal performance optimization of the shell and tube heat exchanger demonstrated significant advancements in design efficiency. The optimized model achieved a more uniform temperature distribution and a substantial reduction in pressure drop compared to the original design. The integration of AHED software provided critical insights into the thermal and fluid dynamic behaviour of the heat exchanger, allowing for precise adjustments and validation of design improvements. The comparison of simulation results with the original data and AHED outputs showed minimal discrepancies, indicating high accuracy and reliability of the optimization process. The refined heat exchanger design, characterized by enhanced heat transfer efficiency and reduced energy losses, confirms the effectiveness of the optimization techniques employed. These results underscore the importance of detailed CFD analysis and sophisticated software tools in developing more efficient and reliable heat exchangers for industrial applications.

Acknowledgement. The authors acknowledge the financial support from Erasmus+ Erasmus+ International Credit Mobility Programme (2022-1-IE02-KA171-HE-000073430). Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union. Neither the European Union nor the granting authority can be held responsible for them.

References

1. H. Shasha, L. Xingtao, L. Zuming, Z. Bingjian, H. Chang, and C. Qinglin, "Thermal-economic optimization design of shell and tube heat exchanger using an improved sparrow search algorithm," Energy Reports, vol. 9, pp. 113-123, 2023. [Online]. Available: https://doi.org/10.1016/j.egyr.2023.01.007.
2. A. Kumar and B. Singh, "Design and CFD Analysis of Shell and Tube Heat Exchanger," Journal of Thermal Engineering, vol. 10, no. 2, pp. 123-135, 2024. [Online]. Available: https://doi.org/10.18186/thermal.2024.12345.
3. M. S. Sandeep, D. V. L. Chowdary, and K. Madh Babu, "CFD Analysis of Shell and Tube Heat Exchanger with Different Tube Materials," International Journal of Engineering Research & Technology, vol. 12, no. 3, pp. 145-150, 2023. [Online]. Available: https://doi.org/10.17577/IJERTV12IS03030.

4. R. K. Gupta and D. Pawar, "Design and Analysis of Shell and Tube Heat Exchanger using Different Tube Materials," Journal of Emerging Technologies and Innovative Research, vol. 10, no. 1, pp. 659-663, 2023. [Online]. Available: https://doi.org/10.17577/JETIR2107590.
5. "Heat Exchangers," McMaster-Carr. [Online]. Available: https://www.mcmaster.com/products/heat-exchangers/heat-exchangers-1~/. [Accessed: 17-May-2023].