



IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

“CFD SIMULATION AND DESIGN OPTIMIZATION OF SHELL-AND-TUBE HEAT EXCHANGERS FOR THERMAL MANAGEMENT IN HEAVY-DUTY AUTOMOTIVE ENGINES”

Hemant Kushwaha¹, Prof. Vikas Bakshi²

¹ PG scholar, Department of Mechanical Engineering, SAGE University, Indore, Madhya Pradesh, India

² Professor, Department of Mechanical Engineering, SAGE University, Indore, Madhya Pradesh, India

ABSTRACT

The present study investigates the effect of different baffle configurations on the thermal-hydraulic performance of shell-and-tube heat exchangers for heavy-duty automotive engine thermal management. Various configurations, including single-, double-, and triple-segmental, helical, and flower-type baffles, are considered for comparative computational fluid dynamics (CFD) analysis. The numerical investigation focuses on heat transfer rate, overall heat transfer coefficient, temperature distribution, and pressure drop under defined operating conditions. Although segmental baffles can provide enhanced heat transfer, they may also produce significant pressure losses and stagnant flow regions. In comparison, helical and flower-type baffles offer the potential to improve shell-side flow distribution and reduce stagnation. The study aims to evaluate these competing effects and identify a suitable baffle configuration through design optimization. The findings are expected to support the development of efficient shell-and-tube heat exchangers by balancing heat transfer enhancement, pressure drop, and pumping power requirements. This approach can provide useful design guidance for improving thermal management in heavy-duty automotive engines.

Key Words: Shell-and-Tube Heat Exchanger, Computational Fluid Dynamics, Baffle Configuration, Heavy-Duty Automotive Engines, Heat Transfer Coefficient, Pressure Drop, Design Optimization.

I. INTRODUCTION

Thermal management is an essential aspect of heavy-duty automotive engine design because engine performance, fuel efficiency, durability, and operational reliability depend significantly on effective heat dissipation. During engine operation, a considerable amount of heat is generated through fuel combustion, friction, and other thermal processes. If this heat is not adequately controlled, it may lead to overheating, increased component wear, lubricant degradation, and reduced engine efficiency. Therefore, efficient cooling systems are required to maintain suitable operating temperatures under varying engine loads and demanding working conditions.

Shell-and-tube heat exchangers (STHXs) are widely used in industrial thermal systems because of their robust construction, operational reliability, and ability to transfer heat between two fluids. Depending on the system requirements, they can be used for cooling engine lubricating oil, coolant, and other working fluids. Their thermal performance depends on several design parameters, including tube diameter, tube arrangement, shell dimensions, fluid flow rate, inlet temperature, and baffle configuration. Among these parameters, baffle geometry plays a significant role in determining shell-side flow distribution, heat transfer rate, and pressure drop.

hydraulic performance. Furthermore, the performance of different baffle designs depends on operating conditions, exchanger geometry, and fluid properties.

Experimental investigations alone may require considerable time and cost to evaluate multiple design alternatives.

<https://www.ijrtsm.com> © International Journal of Recent Technology Science & Management

Computational fluid dynamics (CFD) provides an effective approach for examining flow patterns, temperature distribution, heat transfer rate, and pressure drop. Nevertheless, a systematic comparison of different baffle configurations under consistent operating conditions is necessary to identify suitable designs for heavy-duty automotive engine applications.

Therefore, the primary problem addressed in this study is to determine how different baffle configurations influence the thermal-hydraulic performance of shell-and-tube heat exchangers and to identify a suitable design through CFD-based comparative analysis and optimization. The study considers the competing requirements of enhanced heat transfer, reduced pressure drop, and acceptable pumping power to support efficient and reliable automotive engine thermal management.

II. PROBLEM IDENTIFICATION

The thermal management of heavy-duty automotive engines requires efficient heat exchangers capable of maintaining suitable operating temperatures under varying engine loads and demanding operating conditions. Shell-and-tube heat exchangers offer robust construction and operational flexibility; however, their thermal-hydraulic performance is strongly influenced by shell-side flow distribution, tube-bundle geometry, baffle configuration, and operating parameters. Conventional designs may experience flow maldistribution, localized recirculation, stagnant regions, and non-uniform temperature distribution, which can limit effective heat transfer and reduce the overall thermal performance of the system. Therefore, improving heat transfer while maintaining acceptable hydraulic performance remains an important design challenge. A major problem in shell-and-tube heat exchanger design is the trade-off between heat transfer enhancement and pressure drop. Increasing turbulence and cross-flow mixing through the tube bundle can improve convective heat transfer, but may also increase flow resistance and the pumping power required to circulate the fluid. Conversely, configurations designed to reduce pressure drop may provide insufficient mixing or lower heat transfer performance. This conflict makes it difficult to select a baffle geometry that simultaneously achieves high thermal effectiveness, low pumping requirements, and reliable operation. The identification of an appropriate thermal-hydraulic balance is particularly important for heavy-duty automotive applications, where energy consumption, packaging constraints, durability, and operating reliability must be considered together.

Conventional segmental baffles are widely used because of their simple construction and established design practices. Nevertheless, their flow patterns may produce significant flow separation, leakage, bypassing, and pressure losses, depending on the design and operating conditions. Alternative configurations, such as double- and triple-segmental baffles, helical baffles, and other modified geometries, have been investigated to improve shell-side flow distribution and reduce hydraulic losses. However, their relative performance cannot be generalized without considering the specific exchanger geometry, fluid properties, flow rate, and thermal boundary conditions. A systematic comparison of these configurations using consistent evaluation criteria is therefore required to understand their suitability for automotive thermal management. Another important challenge is the limitation of conventional heat exchanger design methods in describing complex three-dimensional flow and temperature fields. Empirical and semi-empirical methods, including Kern's method and the Bell–Delaware method, are useful for preliminary design and engineering calculations, but their predictions depend on correlations and modelling assumptions. Computational fluid dynamics provides a more detailed means of investigating velocity distribution, pressure variation, temperature gradients, turbulence, and local heat transfer characteristics. However, CFD results may be affected by mesh quality, turbulence model selection, boundary conditions, numerical convergence, and material-property assumptions. In the absence of mesh-independence studies and experimental validation, numerical predictions may not reliably represent actual heat exchanger performance.

Furthermore, existing investigations may employ different geometric dimensions, fluid combinations, operating conditions, numerical approaches, and performance indicators. Such variations make direct comparisons between reported results difficult and limit the transferability of conclusions to heavy-duty automotive systems. In particular, water-to-water simulations or industrial operating conditions may not fully represent the thermal behavior of automotive coolant and lubricating-oil systems. Automotive-oriented assessment must therefore distinguish between general heat exchanger performance and performance demonstrated under representative vehicle operating conditions. Design optimization introduces an additional challenge because the relevant performance objectives are often

conflicting. Maximizing the overall heat transfer coefficient alone may lead to excessive pressure drop, while minimizing pressure drop alone may reduce heat transfer capability. An effective design assessment should consider multiple indicators, including overall heat transfer coefficient, heat duty, outlet temperatures, pressure drop, pumping power, thermal effectiveness, and manufacturing feasibility. Multi-objective optimization can help identify suitable trade-offs; however, the selected solution must also satisfy practical constraints related to size, durability, maintainability, and operating conditions.

III. RESEARCH OBJECTIVE

The main objectives of the present study are as follows:

1. To investigate the effect of different baffle configurations, including single-, double-, and triple-segmental, helical, and flower-type baffles, on the performance of shell-and-tube heat exchangers.
2. To evaluate the heat transfer rate and overall heat transfer coefficient for different baffle configurations using computational fluid dynamics (CFD).
3. To analyze the influence of baffle geometry on shell-side fluid flow, temperature distribution, and pressure drop.
4. To compare the thermal-hydraulic performance of different baffle configurations under defined operating conditions.
5. To examine the trade-off between heat transfer enhancement, pressure drop, and pumping power requirements.
6. To identify a suitable baffle configuration through CFD-based comparative analysis and design optimization for effective thermal management in heavy-duty automotive engines.

IV. MODELLING AND SIMULATION

The shell and tube device were measured using Kern's method, ASPEN simulation software, and Solid Works flow simulation software and overall heat transfer coefficient values using the identical input parameters using 782, 790.2 and 852.6 W / m²K, respectively. Was designed with. CFD simulation studies on shell and tube heat exchangers are performed with single, double, triple, helical, flower A 'type, and flower' B 'type baffle configurations. Study the effects of various configurations of shells and tube device (STHX) on heat transfer coefficient and pressure drop. Shell and tube heat exchangers improve heat transfer and end in increased pressure. Shell and tube heat exchangers are single, double, triple split baffle, with helical baffle.

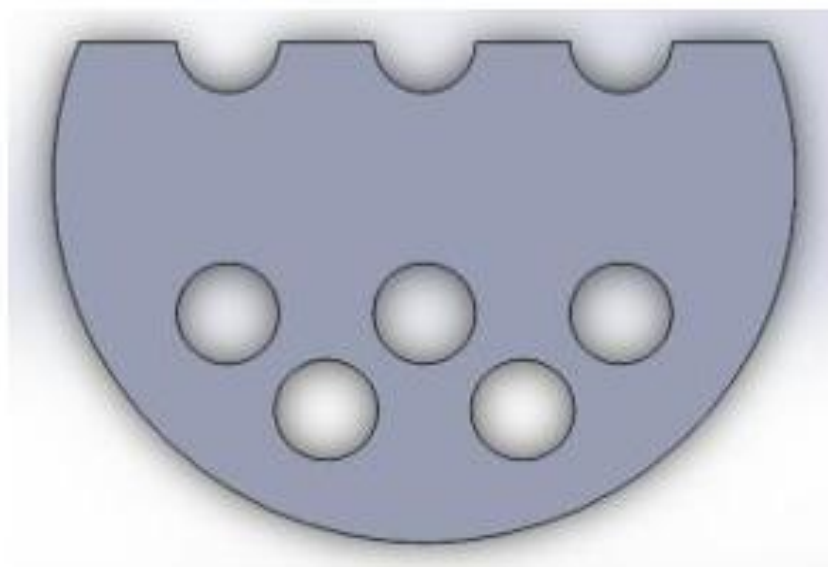


Figure 6.1 Single Segmental Baffles

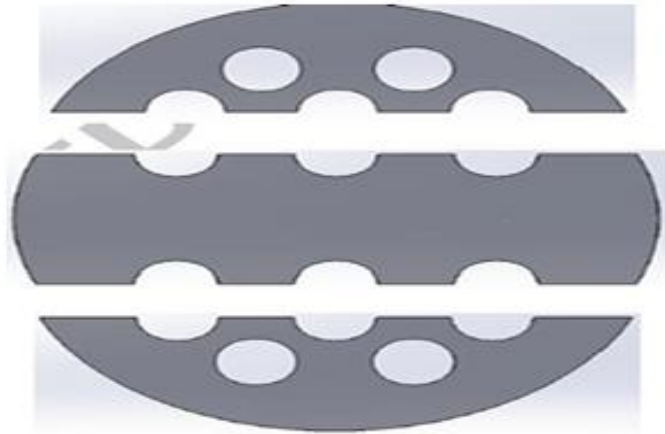


Figure 6.2 Double Segmental Baffles



Figure 6.3 Triple Segmental Baffles

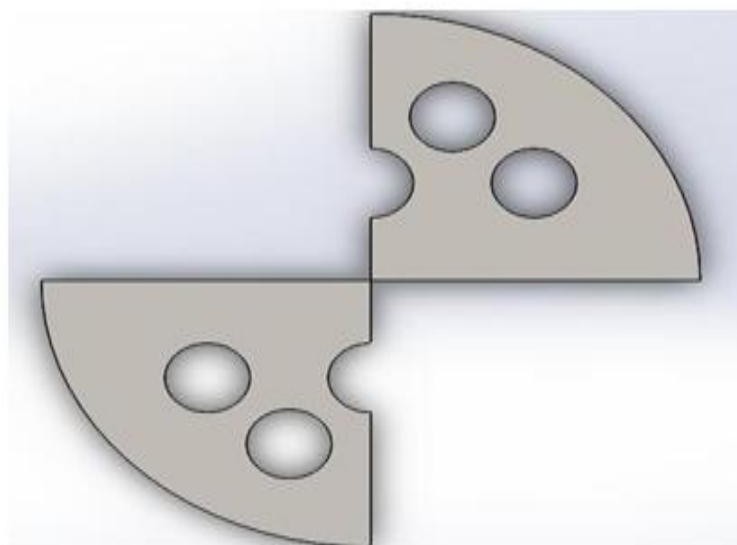


Figure 6.4 Flower "A" Baffles

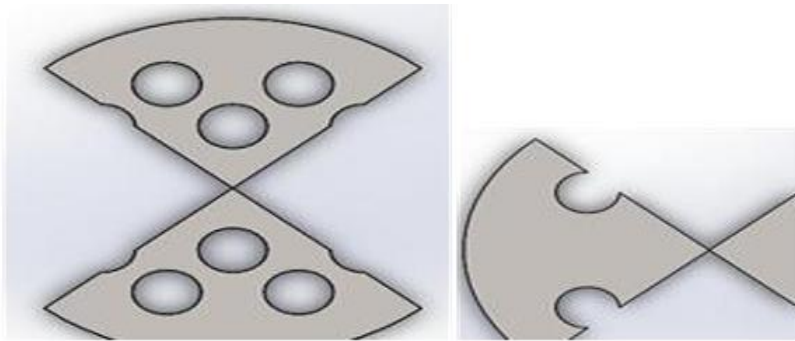


Figure 6.5 Flower “B” Baffles

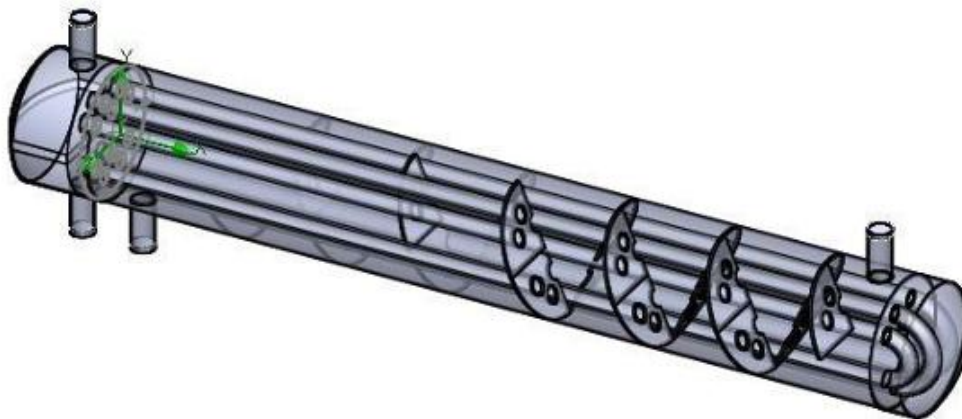


Figure 6.6 3D view of Helical baffled U-tube Heat Exchanger

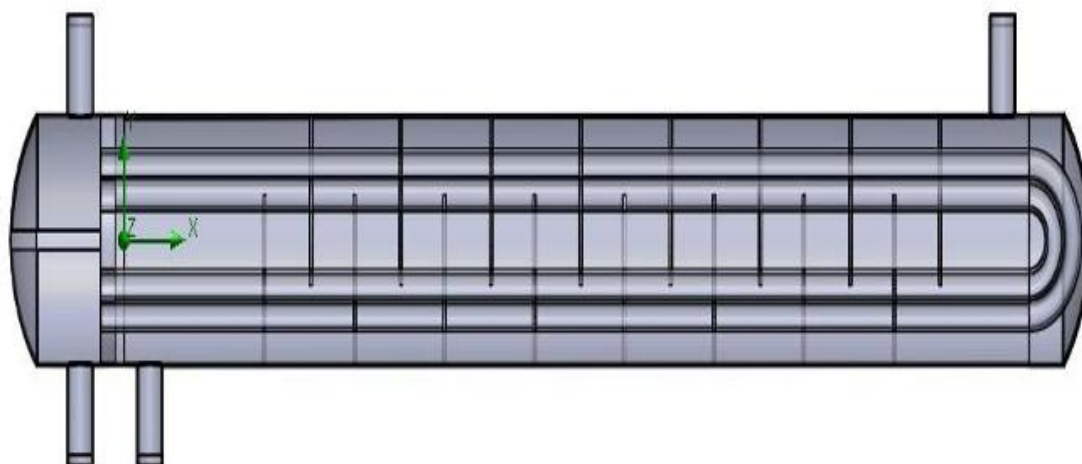


Figure 6.7 2D view of the Shell and Tube Heat Exchanger designed

Baffles are essential components of shell-and-tube heat exchangers because they support tubes, guide shell-side fluid flow, and enhance heat transfer by increasing turbulence. However, conventional single-segmental baffles may cause flow stagnation, bypassing, leakage, and high pressure drop, resulting in increased pumping power and flow-induced tube vibration. These limitations highlight the need for improved baffle designs to enhance thermal-hydraulic performance.

V. DESIGN OF SHELL AND TUBE HEAT EXCHANGER

A shell-and-tube heat exchanger (STHX) with different baffle geometries is designed to investigate the effects of baffle configuration on its thermal and hydraulic performance. Water-to-water, 1–2 pass shell-and-tube heat exchanger is designed using the specifications provided in Table 1. Figures 1–3 illustrate the single-, double-, and triple-segmental baffle configurations, respectively. Figures 4 and 5 present the Flower-A and Flower-B baffle configurations, respectively, while Figure 6 illustrates the helical baffle arrangement. Figure 7 shows the overall configuration of the designed shell-and-tube heat exchanger.

Table 5.1:-Data for design of Shell and tube heat exchanger

Shell Side Fluid-Hot Water		
Property	Unit	Value
T_{HI}		90
T_{HO}		70
Density	kg/m ³	971.8
Specific Heat Capacity	kJ/kgK	4.1963
Viscosity	mPas	0.354
Conductivity	W/mK	0.67
Fouling Factor	m ² K/W	0.0002
Flow Rate	kg/s	0.3
Tube Side Fluid-Cold Water		
T_{CI}		30
T_{CO}		38
Density	kg/m ³	984
Specific Heat Capacity	kJ/kgK	4.178
Viscosity	mPas	0.725
Conductivity	W/mK	0.623
Fouling Factor	m ² K/W	0.0002
Flow Rate	kg/s	0.7533

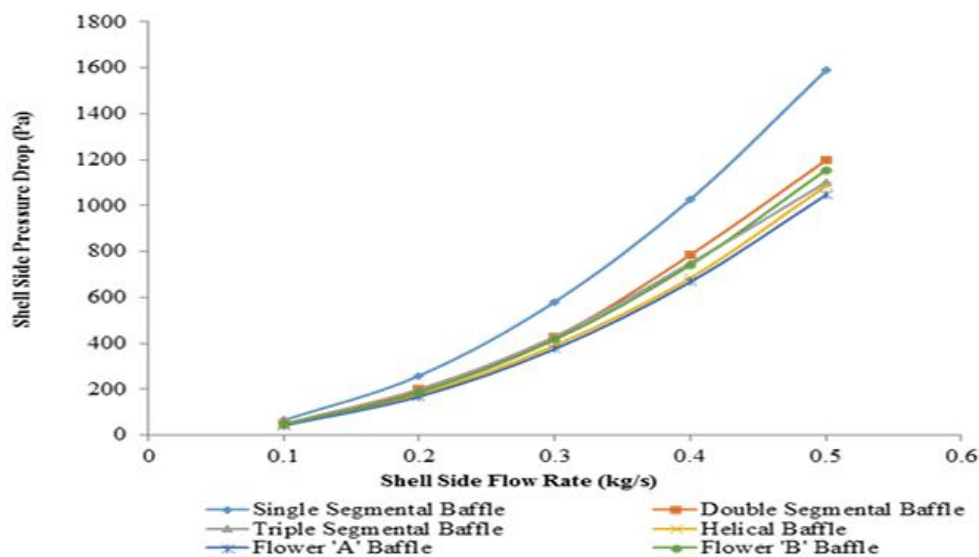


Figure 5.1 Shell Side Pressure Drop vs. Shell Side Flow Rate

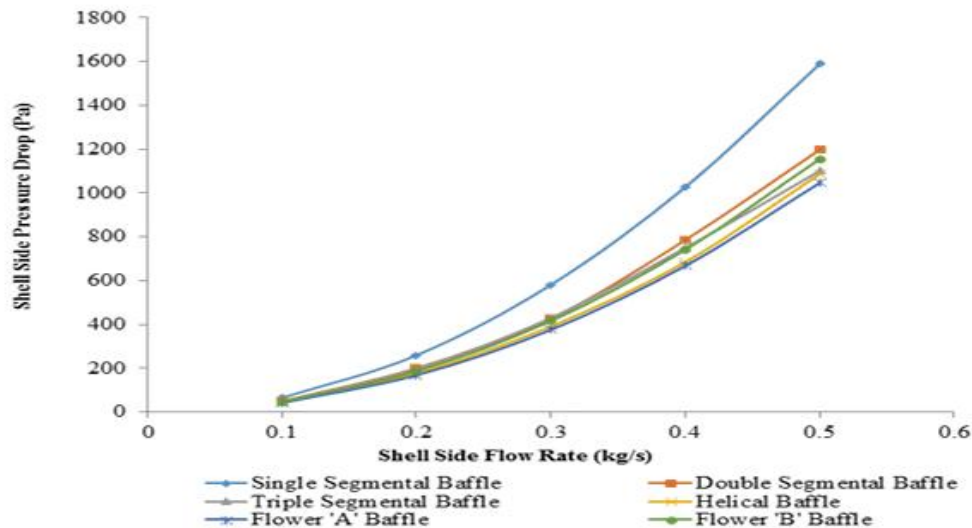


Figure 5.2 Shell Side Pressure Drop vs. Shell Side Flow Rate

VI. RESULT AND DISCUSSION

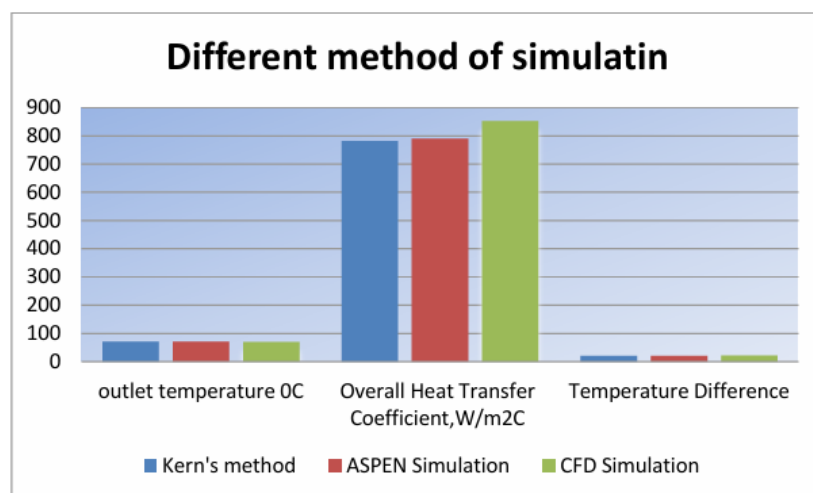


Fig 6.1 Different method of simulation

Table 6.1 Comparison of Overall Heat Transfer Coefficient, Shell side

Heat Exchanger Design Method	Outlet temperature 0C	Overall Heat Transfer Coefficient, W/m2C	Temperature Difference
Kern's method	70	782	20
ASPEN Simulation	70.08	790.2	19.92
CFD Simulation	68.79	852.46	21.21

Kern's method and ASPEN simulations produced similar overall heat transfer coefficients, while SOLIDWORKS CFD predicted a value approximately 9% higher than Kern's method. Similarly, the shell-side temperature difference obtained from CFD was about 6% higher, whereas the shell-side outlet temperature was approximately 1.7% lower. These variations may be attributed to differences between empirical correlations and numerical flow-field calculations.

VII. CONCLUSION

A shell-and-tube heat exchanger was analyzed using Kern's method, ASPEN, HTRI, and SOLIDWORKS Flow Simulation, yielding overall heat transfer coefficients of 782, 790.2, 781.9, and 852.6 W/m²K, respectively. CFD studies comparing different baffle configurations indicated that single-segmental baffles provide higher heat transfer but cause greater pressure drop and pumping power. Double-segmental and flower-type baffles reduce pressure drop by approximately 25–35%, while helical baffles reduce it by 30–35% but may require additional heat transfer area. Triple-segmental baffles were unsuitable for the investigated configuration. SOLIDWORKS CFD predicted an overall heat transfer coefficient approximately 9% higher than Kern's method and a shell-side temperature difference about 6% higher.

REFERENCES

- [1] A. Bhardwaj, A. K. Mishra, S. Rao, G. Kumar, and R. K. Singh, "A comparative CFD study of baffle geometry and perforation shape for performance optimization in shell-and-tube heat exchangers," *International Communications in Heat and Mass Transfer*, vol. 178, 2026, Art. no. 111565, doi: 10.1016/j.icheatmasstransfer.2026.111565.
- [2] A. M. Mohammadzadeh, B. Jafari, K. Hosseinzadeh, et al., "Numerical investigation of segmental baffle design in shell and tube heat exchangers with varying inclination angles and spacing," *Scientific Reports*, vol. 15, 2025, Art. no. 4683, doi: 10.1038/s41598-025-87652-x.
- [3] H. M. K. Hoang, H. L. Cao, P. M. Q. Pham, A. Hajjar, and N. V. Linh, "A novel shell-and-tube heat exchanger design with alternative inclined baffles," *Case Studies in Thermal Engineering*, vol. 65, 2025, Art. no. 105542, doi: 10.1016/j.csite.2024.105542.
- [4] M. Waleed, S. M. A. Naqvi, H. Mustafa, M. K. Al Mesfer, M. Danish, K. Irshad, and H. Shahzad, "Numerical analysis of shell and tube heat exchanger with combination of different baffles," *Case Studies in Thermal Engineering*, vol. 65, 2025, Art. no. 105658, doi: 10.1016/j.csite.2024.105658.
- [5] A. M. Mohammadzadeh, B. Jafari, and K. Hosseinzadeh, "Comprehensive numerical investigation of the effect of various baffle design and baffle spacing on a shell and tube heat exchanger," *Applied Thermal Engineering*, vol. 249, 2024, Art. no. 123305, doi: 10.1016/j.applthermaleng.2024.123305.
- [6] S. Saha and N. Hasan, "Numerical evaluation of thermohydraulic parameters for diverse configurations of shell-and-tube heat exchanger," *Results in Engineering*, vol. 23, 2024, Art. no. 102509, doi: 10.1016/j.rineng.2024.102509.
- [7] "Developing an algorithm to evaluate the tube and baffle choice influences on enhancing the hydrothermal performances of shell and tube heat exchangers," *International Journal of Heat and Mass Transfer*, vol. 222, 2024, Art. no. 125178, doi: 10.1016/j.ijheatmasstransfer.2024.125178.
- [8] "Numerical study on hydrodynamics, mixing and heat transfer of highly viscous fluid in a novel shell-and-tube heat exchanger with X-type baffles," *Chemical Engineering Research and Design*, vol. 205, pp. 556–568, 2024, doi: 10.1016/j.cherd.2024.04.031.
- [9] S. A. Marzouk, M. M. Abou Al-Sood, E. M. S. El-Said, M. M. Younes, and M. K. El-Fakharany, "A comprehensive review of methods of heat transfer enhancement in shell and tube heat exchangers," *Journal of Thermal Analysis and Calorimetry*, vol. 148, pp. 7539–7578, 2023, doi: 10.1007/s10973-023-12265-3.
- [10] D. Arunachalam, A. K. Shukla, A. Dewan, R. Salhotra, and D. K. Singh, "Computation of thermal properties of CMSP shell and tube heat exchanger using different types of helical baffles," *Heat Transfer*, vol. 52, pp. 4401–4421, 2023, doi: 10.1002/htj.22882.
- [11] "CFD-based optimization of a shell-and-tube heat exchanger," *Fluid Dynamics and Materials Processing*, vol. 19, no. 11, pp. 2761–2775, 2023, doi: 10.32604/fdmp.2023.021175.
- [12] "CFD simulation of helical shell and tube heat exchanger using optimization techniques," *Journal of Engineering Research*, vol. 11, no. 1, pt. B, pp. 253–265, 2023, doi: 10.36909/jer.11985.
- [13] S. A. Marzouk, M. M. Abou Al-Sood, M. K. El-Fakharany, and E. M. S. El-Said, "A comparative numerical study of shell and multi-tube heat exchanger performance with different baffles configurations," *International Journal of Thermal Sciences*, vol. 179, 2022, Art. no. 107655, doi: 10.1016/j.ijthermalsci.2022.107655.

- [14] “Multi-objective optimization of helical baffles in the shell-and-tube heat exchanger by computational fluid dynamics and genetic algorithm,” *Energy Reports*, vol. 8, pp. 11064–11077, 2022, doi: 10.1016/j.egyr.2022.08.249.
- [15] “Numerical investigation of the flow dynamics and heat transfer in a rectangular shell-and-tube heat exchanger,” *Case Studies in Thermal Engineering*, vol. 32, 2022, Art. no. 101873, doi: 10.1016/j.csite.2022.101873.