



IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

“ADVANCES IN SHELL-AND-TUBE HEAT EXCHANGER DESIGN FOR HEAVY-DUTY AUTOMOTIVE THERMAL MANAGEMENT: A REVIEW OF CFD METHODS, BAFFLE GEOMETRIES, AND OPTIMIZATION STRATEGIES”

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

Shell-and-tube heat exchangers play an important role in thermal management systems by facilitating heat transfer between fluids in automotive applications. In heavy-duty vehicles, effective thermal management is essential for maintaining engine operating temperatures, improving energy efficiency, reducing thermal losses, and ensuring reliable performance under varying operating conditions. However, conventional shell-and-tube heat exchangers may exhibit limitations associated with non-uniform shell-side flow distribution, flow recirculation, excessive pressure drop, and limited heat transfer performance. Recent advances in computational fluid dynamics (CFD), innovative baffle geometries, and design optimization techniques have provided opportunities to address these challenges. This review paper presents a comprehensive assessment of developments in shell-and-tube heat exchanger design for heavy-duty automotive thermal management, with particular emphasis on CFD modelling approaches, baffle configurations, thermal-hydraulic performance, and optimization strategies. Conventional segmental baffles, including single-, double-, and triple-segmental designs, are examined alongside alternative configurations such as helical and other advanced baffle geometries. The review discusses the influence of geometric parameters, fluid properties, operating conditions, turbulence models, and numerical discretization on heat transfer characteristics and pressure drop. Furthermore, the application of multi-objective optimization techniques for balancing the overall heat transfer coefficient, pumping power, pressure drop, and thermal effectiveness is considered. The importance of mesh independence studies, numerical validation, and experimental verification in establishing the reliability of CFD predictions is also highlighted. The review identifies key research gaps concerning automotive-specific operating conditions, transient thermal loads, realistic coolant and lubricant properties, and the integration of advanced heat exchanger designs with vehicle thermal management systems. The findings are intended to support the development of efficient, compact, and hydraulically optimized shell-and-tube heat exchangers for heavy-duty automotive applications.

Key Words: *Shell-and-tube heat exchanger; computational fluid dynamics; heavy-duty automotive engines; thermal management; baffle geometry; heat transfer enhancement; pressure drop; design optimization.*

I. INTRODUCTION

Thermal management is a critical aspect of heavy-duty automotive engineering because engine performance, fuel efficiency, component durability, and operational reliability are strongly influenced by the effective control of operating temperatures. Heavy-duty vehicles, including trucks, buses, construction equipment, and commercial transport vehicles, operate under varying load conditions and often experience prolonged periods of high thermal demand. During engine operation, a considerable amount of energy is released as heat, which must be managed

effectively to maintain suitable operating temperatures and prevent overheating. Inadequate heat removal can contribute to thermal stresses, lubricant degradation, reduced component life, and deterioration in engine performance. Consequently, the development of efficient heat transfer systems is an important area of research in automotive thermal management. Heat exchangers are widely used in automotive and industrial thermal management systems to transfer thermal energy between fluids at different temperatures. Depending on the application, they support engine coolant temperature control, lubricating oil cooling, transmission fluid cooling, and the thermal management of other vehicle subsystems. Among the different types of heat exchangers, shell-and-tube heat exchangers are recognized for their robust construction, operational reliability, adaptability to different operating conditions, and suitability for applications involving relatively high thermal loads. Their design consists of a bundle of tubes enclosed within a shell, allowing heat to transfer between fluids flowing through the tubes and the shell-side passage. Although compact plate-type and other specialized heat exchangers are also widely used in modern vehicles, shell-and-tube configurations remain relevant to selected heavy-duty and auxiliary thermal management applications where durability, maintainability, and operating flexibility are important considerations.

The thermal-hydraulic performance of a shell-and-tube heat exchanger depends on several interacting design and operating parameters, including tube diameter, tube arrangement, shell diameter, baffle spacing, baffle cut, fluid flow rate, inlet temperature, and thermophysical properties of the working fluids. Among these parameters, baffle geometry has a particularly significant influence on shell-side flow distribution and heat transfer. Baffles direct the shell-side fluid across the tube bundle, promote flow mixing, and interrupt the development of thermal boundary layers. These effects can enhance convective heat transfer; however, they may also increase flow resistance and pressure drop. Excessive pressure drop increases the pumping power required to circulate the fluid, potentially offsetting the benefits obtained from improved heat transfer. Therefore, the design of an effective shell-and-tube heat exchanger requires a suitable balance between thermal performance and hydraulic efficiency rather than maximizing the heat transfer coefficient alone. Conventional segmental baffles are widely employed because of their relatively simple construction, ease of manufacturing, and established design practices. Single-segmental baffles can provide substantial cross-flow mixing and heat transfer, but may also generate pronounced flow separation, recirculation zones, and comparatively high pressure losses. Double- and triple-segmental baffles modify the shell-side flow path and can reduce flow resistance under appropriate geometric and operating conditions. Alternative arrangements, including helical baffles and other modified baffle geometries, have also attracted research interest because they can promote more continuous flow paths and improve shell-side flow distribution. Nevertheless, the performance of these configurations depends on their detailed geometry, the operating regime, and the design constraints. A baffle configuration that reduces pressure drop may also reduce heat transfer, while a design that improves heat transfer may require additional pumping power. Accordingly, comparative assessment is necessary to determine which configuration provides the most appropriate thermal-hydraulic performance for a particular application.

Computational fluid dynamics (CFD) has become an important tool for investigating the complex flow and heat transfer phenomena occurring inside shell-and-tube heat exchangers. Unlike conventional analytical and empirical design methods, CFD can provide detailed information about velocity fields, temperature distributions, pressure variations, turbulence characteristics, and local heat transfer behavior. These capabilities allow researchers to examine flow maldistribution, recirculation regions, thermal boundary layers, and the influence of geometric modifications. CFD also enables the evaluation of alternative designs before physical prototypes are manufactured, potentially reducing development time and experimental costs. Commercial simulation platforms and numerical solvers have made it increasingly practical to investigate three-dimensional heat exchanger geometries under specified thermal and hydraulic boundary conditions. Traditional methods, such as Kern's method, remain useful for preliminary heat exchanger design and engineering estimation. More detailed approaches, including the Bell–Delaware method, account for several shell-side leakage and bypass effects and can provide improved estimates of heat transfer and pressure drop when the required geometric information is available. Process simulation and specialized heat exchanger design software can further support design evaluation. However, empirical and semi-empirical methods generally do not provide the same spatially resolved flow and temperature fields as three-dimensional CFD simulations. At the same time, CFD predictions are not inherently accurate simply because a detailed numerical model is used. Their reliability depends on the quality of the computational mesh, appropriate turbulence and heat transfer models, boundary conditions, numerical convergence, and validation against experimental measurements or credible benchmark data. Grid-independence assessment and systematic uncertainty evaluation are therefore important components of reliable

CFD-based heat exchanger research. In addition to CFD modelling, design optimization has emerged as a promising approach for improving shell-and-tube heat exchanger performance. Optimization may involve changes to baffle spacing, baffle cut, tube layout, shell dimensions, and other geometric variables to achieve specified thermal and hydraulic objectives. Single-objective optimization may focus on maximizing the overall heat transfer coefficient or minimizing pressure drop. However, these objectives can conflict with one another, making multi-objective optimization particularly relevant. Approaches based on response surface methodology, genetic algorithms, and other numerical optimization techniques can be used to explore design alternatives and identify trade-offs between heat transfer, pumping power, pressure drop, equipment size, and manufacturing requirements. For heavy-duty automotive applications, optimization should also consider variations in engine load, coolant flow rate, inlet temperature, packaging constraints, reliability, and maintenance requirements.

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 objective of this review paper is to critically examine recent advances in shell-and-tube heat exchanger design for heavy-duty automotive thermal management, with particular emphasis on computational fluid dynamics (CFD), baffle geometry, thermal-hydraulic performance, and design optimization. The specific research objectives are as follows:

1. To review CFD methodologies: Examine the numerical approaches, turbulence models, boundary conditions, and computational techniques used to investigate fluid flow and heat transfer in shell-and-tube heat exchangers.
2. To evaluate different baffle geometries: Review the thermal-hydraulic performance of conventional segmental baffles, including single-, double-, and triple-segmental designs, and alternative configurations such as helical and other modified baffles.
3. To analyze heat transfer characteristics: Assess the influence of baffle configuration, tube arrangement, baffle spacing, fluid flow rate, and operating temperature on the overall heat transfer coefficient, heat duty, and temperature distribution.
4. To investigate pressure drop and pumping power: Examine the relationship between heat transfer enhancement, shell-side pressure drop, and pumping power to identify the trade-offs associated with different heat exchanger designs.
5. To compare numerical and conventional design methods: Evaluate the capabilities and limitations of CFD simulations in comparison with established approaches such as Kern's method, the Bell–Delaware method, and specialized heat exchanger design software.
6. To review design optimization strategies: Examine single-objective and multi-objective optimization techniques used to improve thermal performance, reduce hydraulic losses, and identify suitable design configurations under practical constraints.
7. To assess CFD reliability and validation: Investigate the role of mesh-independence studies, numerical convergence, turbulence model selection, sensitivity analysis, and experimental validation in establishing confidence in simulation results.
8. To identify research gaps in automotive applications: Identify limitations in existing studies concerning representative heavy-duty vehicle operating conditions, variable thermal loads, automotive coolant and lubricant properties, packaging constraints, and system-level thermal management.

IV. LITERATURE REVIEW

The design and optimization of shell-and-tube heat exchangers have received considerable attention because of their importance in industrial heat transfer, energy conversion, and thermal management systems. Their performance depends on baffle geometry, tube arrangement, shell-side flow distribution, operating temperature, fluid properties, and mass flow rate. Computational fluid dynamics (CFD) has become an important tool for investigating the complex interaction between fluid flow and heat transfer. Researchers have also explored advanced baffle configurations, geometric modifications, and optimization algorithms to improve the overall heat transfer coefficient while reducing pressure drop and pumping power. The following review summarizes relevant research published between 2026 and 2020, highlighting its major findings and significance for future automotive thermal management applications.

Bhardwaj et al. (2026) investigated the effect of baffle geometry and perforation shape on the thermal-hydraulic performance of shell-and-tube heat exchangers using CFD. The study compared segmental, double-segmental, flower-type, and disc-and-doughnut baffles with different perforation geometries at varying shell-side mass flow rates. The

results indicated that circular perforations in double-segmental baffles provided a favorable balance between heat transfer and pressure drop under the investigated conditions. The authors also observed improved flow mixing and reduced stagnation regions for selected perforated configurations. This study demonstrates the potential of combining baffle geometry modification with perforation design to improve heat exchanger performance. However, further evaluation is required under operating conditions representative of heavy-duty automotive cooling systems.

Yadav et al. (2026) investigated the influence of helical-baffle pitch on the thermal and hydraulic behavior of shell-and-tube heat exchangers. The study evaluated how changes in baffle pitch affect shell-side fluid flow, heat transfer, and pressure drop. The investigation emphasized the importance of selecting suitable geometric parameters to achieve a balance between heat transfer enhancement and hydraulic resistance. The study is relevant to the present review because helical baffles offer an alternative to conventional segmental arrangements and may help control shell-side flow distribution. Their suitability for automotive thermal management, however, depends on the exchanger dimensions, operating conditions, and practical installation requirements.

Ni (2026) examined the influence of shell-side baffle number on the thermal-hydraulic performance of a shell-and-tube heat exchanger intended for marine cooling applications. The study investigated the relationship between baffle number, cooling performance, shell-side flow distribution, and pumping-power requirements. The research highlights the importance of evaluating baffle quantity as a design parameter because increasing the number of baffles may improve flow guidance but can also increase hydraulic resistance. Although the investigation concerns marine rather than automotive cooling, its findings are relevant to heavy-duty thermal management systems where continuous heat rejection and energy-efficient fluid circulation are important.

Mohammadzadeh et al. (2025) investigated the influence of segmental-baffle inclination angles and spacing on shell-and-tube heat exchanger performance using three-dimensional CFD simulations. The study examined the relationship between baffle geometry, turbulent flow behavior, heat transfer, and pressure drop. The findings demonstrated that the thermal-hydraulic performance of a heat exchanger is sensitive to baffle arrangement and spacing. The research supports the use of CFD for examining geometric modifications before physical implementation and emphasizes the need to evaluate heat transfer and pressure drop together. These considerations are important when developing compact and efficient heat exchangers for heavy-duty automotive applications.

Waleed et al. (2025) conducted a numerical investigation of shell-and-tube heat exchangers incorporating combinations of different flower-type baffles. The study compared combinations of double-, triple-, and quadruple-flower baffles with selected reference configurations. CFD simulations were used to examine the effects of baffle arrangements on fluid flow and thermal-hydraulic performance. The findings indicated that certain combinations of flower-type baffles could improve performance compared with the reference designs under the investigated conditions. This research highlights the potential of non-conventional baffle geometries to improve shell-side flow distribution. Further studies are needed to assess their manufacturing feasibility, durability, and performance under automotive operating conditions.

Ajmal et al. (2025) investigated the optimization of shell-and-tube heat exchanger geometry by varying baffle spacing, baffle cut, and sealing arrangements. The researchers combined the Bell–Delaware method for preliminary thermal analysis with CFD simulations to evaluate heat transfer and pressure drop. Their findings indicated that appropriate selection of baffle dimensions and sealing arrangements could substantially influence exchanger performance. The study demonstrates the value of combining conventional design calculations with numerical simulation to evaluate alternative geometries. For heavy-duty automotive applications, similar optimization approaches could be used to identify designs that satisfy thermal requirements while controlling pumping power and space limitations.

Mohammadzadeh et al. (2024) performed a comprehensive numerical investigation of different baffle designs and baffle spacing in shell-and-tube heat exchangers. The study examined conventional segmental and double-segmental baffles alongside helical, disc-and-doughnut, clamping anti-vibration, and flower-type configurations. The researchers evaluated the overall heat transfer coefficient, pressure drop, and performance evaluation criterion (PEC). The results indicated that disc-and-doughnut baffles achieved favorable thermal-hydraulic performance in the investigated configuration, while some alternative arrangements reduced pressure drop at the expense of heat transfer. The findings

emphasize that baffle selection should be based on multiple performance indicators rather than heat transfer coefficient alone.

Wang et al. (2024) investigated shell-side flow paths and the influence of dead zones on the thermal performance of shell-and-tube heat exchangers. The study examined how clearances between baffles, tubes, and the shell influence fluid distribution and heat transfer. The findings highlighted that shell-side dead zones and flow maldistribution can limit thermal performance and that appropriate control of geometric clearances can improve flow distribution. This research is relevant to CFD-based heat exchanger optimization because local flow structures may not be adequately represented by a single overall heat transfer coefficient. Detailed flow-field analysis is therefore useful for identifying regions requiring geometric improvement.

Marzouk et al. (2023) presented a comprehensive review of heat transfer enhancement methods for shell-and-tube heat exchangers. The review examined different passive and active enhancement techniques and discussed their influence on heat transfer, pressure drop, and overall performance. The authors highlighted the importance of improving heat transfer while accounting for the associated hydraulic penalty and practical design requirements. This review provides a broad foundation for understanding the development of heat exchanger enhancement techniques. Nevertheless, the application of these approaches to heavy-duty automotive thermal management requires further consideration of vehicle operating conditions, packaging constraints, and variable thermal loads.

Al-Darraji et al. (2024; published online in 2023) investigated heat transfer enhancement in a vertical shell-and-tube heat exchanger using air injection and modified baffle arrangements. The study combined experimental and numerical approaches to assess the influence of disc-and-ring baffles, including configurations with circular holes, on exchanger performance. The reported findings indicated improvements in thermal effectiveness for selected modified baffle designs, with air injection and baffle geometry influencing the heat transfer process. This study demonstrates the value of combining experimental measurements with CFD analysis when evaluating novel enhancement techniques. However, the practicality of air injection in a particular automotive cooling system would need to be evaluated separately.

Yadav et al. (2023) investigated helical shell-and-tube heat exchanger configurations using CFD and optimization techniques. The researchers evaluated different helical-tube arrangements, numbers of baffles, and baffle spacings to assess their influence on heat exchanger performance. The study reported improved performance for selected configurations incorporating additional baffles and a copper oxide nanofluid compared with the reference models. The findings suggest that baffle number, spacing, and working-fluid characteristics can jointly influence heat transfer. However, the use of nanofluids in practical automotive systems requires further assessment of stability, material compatibility, maintenance, and pumping requirements.

Daneshparvar and Beigzadeh (2022) investigated the thermal-hydraulic performance of helical-baffle shell-and-tube heat exchangers using CFD and a multi-objective genetic algorithm. The study examined the effects of baffle pitch and baffle angle on heat transfer coefficient and pressure drop. The numerical model was validated against experimental data, and the validated results were used to develop predictive correlations and identify suitable geometric configurations. The optimization results demonstrated the trade-off between increasing heat transfer and controlling pressure drop. This study supports the integration of validated CFD models and multi-objective optimization to improve heat exchanger design.

Aydin et al. (2022) investigated the optimization and CFD analysis of a shell-and-tube heat exchanger incorporating multi-segmental baffles. The research considered thermal and economic performance and examined the effects of a modified baffle arrangement on shell-side flow and pressure drop. The study compared CFD predictions with experimental results and reported that the investigated baffle design could reduce shell-side pressure losses without substantially compromising thermal performance. The findings indicate that modified multi-segmental baffles may provide an alternative to conventional designs when reducing operating costs is an important objective. The research also reinforces the importance of experimental comparison when assessing the reliability of numerical results.

Thondiyil and Kodakkattu (2021) investigated the optimization of shell-and-tube heat exchangers with staggered

baffles using the Taguchi method and CFD. The study varied several geometric parameters, including shell diameter, tube diameter, baffle cut, baffle spacing, and baffle orientation. An orthogonal-array experimental design was used to reduce the number of numerical simulations required to explore the design space. Separate parameter combinations were identified for maximizing heat transfer rate and minimizing pressure drop. The findings demonstrate that systematic design-of-experiments techniques can reduce computational effort and help identify promising configurations. The study is relevant to automotive applications where design optimization must consider competing thermal and hydraulic requirements.

Biçer et al. (2020) proposed a novel three-zonal baffle design and optimized its parameters using CFD and the Taguchi method. The study aimed to reduce shell-side pressure loss while maintaining or improving thermal performance. The numerical predictions were compared with experimental data, and the optimized design showed a reduction in shell-side pressure loss of approximately 49%, together with an increase in shell-side temperature difference of up to 7% under the investigated conditions. The study demonstrated the potential of combining innovative baffle geometry, design-of-experiments methods, and CFD validation to improve heat exchanger performance. These results provide a useful basis for investigating alternative baffle configurations for energy-efficient thermal management. [15]

Mohammadi et al. (2020) investigated thermal optimization of a shell-and-tube heat exchanger using porous baffles. The researchers numerically examined heat transfer and pressure drop and evaluated the influence of different porous-baffle parameters. Artificial neural networks were used for sensitivity analysis, while multi-objective optimization was applied to identify promising design parameters. The study illustrates how modified baffle structures and data-driven optimization methods can be combined to improve thermal-hydraulic performance. However, porous-baffle designs also require consideration of manufacturing complexity, fouling, mechanical strength, and long-term operational reliability before they can be recommended for heavy-duty automotive systems.

V. CONCLUSION

This review examined advances in shell-and-tube heat exchanger design for heavy-duty automotive thermal management, focusing on CFD methods, baffle geometries, and optimization strategies. The literature indicates that baffle configuration, spacing, flow rate, and operating temperature significantly influence heat transfer and pressure drop. Conventional segmental baffles are widely used because of their simple design, while helical, perforated, and other modified baffles offer potential improvements in flow distribution and thermal-hydraulic performance. CFD provides valuable insights into temperature distribution, velocity fields, and pressure drop. However, reliable predictions require mesh independence, suitable turbulence models, and validation against experimental data. Conventional methods such as Kern and Bell–Delaware remain useful for preliminary design and comparison. Future research should focus on validated CFD simulations, multi-objective optimization, and automotive-specific operating conditions, including variable loads and transient thermal behavior. Integrating advanced baffle designs with reliable numerical analysis can support the development of efficient, compact, and reliable heat exchangers for heavy-duty automotive applications.

REFERENCES

- [1] G. M. P. Yadav, M. G. M. J. Basha, B. Omprakash, and E. M. Manohar, "Modeling and CFD analysis of shell-and-tube heat exchanger with helical baffles: Performance evaluation through baffle pitch optimization," *Journal of Thermal Engineering*, vol. 12, no. 5, pp. 1915–1928, 2026, doi: 10.47481/jten.0074.
- [2] Z. Ni, "Shell side baffle number governs thermal hydraulic trade offs in a marine shell and tube heat exchanger," *Discover Applied Sciences*, 2026, doi: 10.1007/s42452-026-09476-2.
- [3] B. Tian, N. Wang, L. Liu, P. Peng, Y. Guo, L. Liu, and S. Shao, "A comprehensive investigation of shell and tube heat exchangers based on porous baffle design and multi-objective parameter optimization," *International Journal of Thermal Sciences*, vol. 221, Art. no. 110463, 2026, doi: 10.1016/j.ijthermalsci.2025.110463.
- [4] 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*, 2026, doi: 10.1016/j.icheatmasstransfer.2026.111565.

- [5] “Design optimization of a shell-and-tube heat exchanger based on variable baffle cuts and sizing,” *Journal of Thermal Science and Engineering Applications*, vol. 17, no. 2, Art. no. 021008, 2025, doi: 10.1115/1.4067235.
- [6] 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.
- [7] J. Wang, J. Nan, and Y. Wang, “CFD-based optimization of a shell-and-tube heat exchanger,” *Fluid Dynamics & Materials Processing*, vol. 19, no. 11, pp. 2761–2775, 2023, doi: 10.32604/fdmp.2023.021175.
- [8] “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.
- [9] “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.egy.2022.08.249.
- [10] A. Aydin, H. Yaşar, T. Engin, and E. Büyükkaya, “Optimization and CFD analysis of a shell-and-tube heat exchanger with a multi segmental baffle,” *Thermal Science*, vol. 26, no. 1, pt. A, pp. 1–12, 2022, doi: 10.2298/TSCI200111293A.
- [11] S. M. A. Naqvi and Q. Wang, “Performance enhancement of shell–tube heat exchanger by clamping anti-vibration baffles with porous media involvement,” *Heat Transfer Engineering*, vol. 42, no. 18, pp. 1523–1538, 2021, doi: 10.1080/01457632.2020.1807098.
- [12] N. Biçer, T. Engin, H. Yaşar, E. Büyükkaya, A. Aydın, and A. Topuz, “Design optimization of a shell-and-tube heat exchanger with novel three-zonal baffle by using CFD and Taguchi method,” *International Journal of Thermal Sciences*, vol. 155, Art. no. 106417, 2020, doi: 10.1016/j.ijthermalsci.2020.106417.
- [13] M. H. Mohammadi, H. R. Abbasi, A. Yavarinasab, and H. Pourrahmani, “Thermal optimization of shell and tube heat exchanger using porous baffles,” *Applied Thermal Engineering*, vol. 170, Art. no. 115005, 2020, doi: 10.1016/j.applthermaleng.2020.115005.
- [14] H. R. Abbasi, E. Sharifi Sadeh, H. Pourrahmani, and M. H. Mohammadi, “Shape optimization of segmental porous baffles for enhanced thermo-hydraulic performance of shell-and-tube heat exchanger,” *Applied Thermal Engineering*, vol. 180, Art. no. 115835, 2020, doi: 10.1016/j.applthermaleng.2020.115835.