Analysis of Shell and Tube Heat Exchanger using CFD Having Baffle Plates Attached and Nanofluid as a Cold Fluid: A Review

Aman Kumar[1], Ajay Singh[2], Nitin Barodia[3]

[1] *Scholar, Department of Mechanical Engineering, Radharaman Institute of Technology and Science, Bhopal, M.P., India*
[2] *Prof. and Head, Department of Mechanical Engineering, Radharaman Institute of Technology and Science, Bhopal, M.P., India*
[3]*Assistant Professor, Department of Mechanical Engineering, Radharaman Institute of Technology and Science, Bhopal, M.P., India*

---***---

Abstract - *Shell and tube heat exchangers play a pivotal role in numerous industries, facilitating efficient heat transfer between fluids at different temperatures. This review explores the integration of baffle plates and nanofluids to optimize heat exchanger performance. Baffle plates strategically placed in tube sides accelerate heat transfer by enhancing fluid mixing and mitigating stagnant areas, while nanofluids, a blend of traditional fluids and nanoparticles, elevate thermal conductivity for superior heat dissipation. Computational Fluid Dynamics (CFD) investigations scrutinized diverse baffle plate configurations and nanofluid implementations in double tube heat exchangers, unveiling promising enhancements in heat transfer and fluid flow characteristics. Higher Reynolds numbers in cold fluids bolstered heat transfer metrics, albeit with performance index fluctuations. The synergy between optimized baffle designs and nanofluids showcased remarkable improvements in heat exchanger efficiency, holding substantial promise for applications in power generation, chemical processing, and refrigeration systems.*

***Key Words*: Heat exchanger, Baffle Plates, Nanofluid, CFD, Thermal properties, Rate of Heat Transfer**

1.INTRODUCTION

For effective heat transfer between two fluids at different temperatures, shell and tube heat exchangers are widely utilized in a variety of industries. In processes involving heating, cooling, and condensation, their adaptability, durability, and capacity for handling high-pressure and high-temperature applications have made them vital. Researchers and engineers have investigated novel approaches, such as incorporating baffle plates and using nanofluids as the heat transfer medium in the tube side, to further optimize their performance. Baffle plates, strategically positioned inside the heat exchanger's tube side, are essential for accelerating heat transfer rates because they encourage fluid mixing, create turbulence, and prevent the formation of stagnant areas. Baffle plates improve heat exchange efficiency by rerouting the fluid's flow route, resulting in more fluid-to-wall contact and lessening fouling tendencies. Due to its ability to improve heat transfer performance without significantly raising the overall pressure drop across the heat exchanger, this arrangement has attracted a lot of interest. Nanofluids have been a game-changing development in the realm of enhancing heat transmission in recent years. Traditional base fluids, such as water, oil, and ethylene glycol, are mixed with nanoparticles, such as metals, oxides, and carbon-based compounds. A number of notable thermal qualities, including greater thermal diffusivity, stronger convective heat transfer coefficients, and increased thermal conductivity, are brought about by the introduction of nanoparticles in base fluids. These special characteristics offer a fascinating chance to revolutionize heat exchanger design by greatly increasing heat transfer rates while keeping pressure losses under control. Baffle plates and nanofluids have the enormous potential to completely transform the thermal performance of shell and tube heat exchangers, addressing a variety of industrial applications from power generation to renewable energy systems. Although the individual contributions of baffle plates and nanofluids have been extensively studied, there is still much to learn about the synergistic benefits of their simultaneous integration. The objective of this review paper is to present a thorough examination of computational fluid dynamics (CFD) research performed on shell and tube heat exchangers using baffle plates on the tube side and nanofluids as the cold fluid. We intend to identify key elements impacting heat transfer enhancement and throw light on the possible difficulties and opportunities connected with this innovative heat exchanger configuration by synthesizing and critically assessing the most recent research and findings in this sector. This review paper is organized as follows: The essential concepts of shell and tube heat exchangers will be presented in detail, with an emphasis on their design and operation, in the parts that follow. The relevance of baffle plates in altering the fluid dynamics within the tubes and resulting in improved heat transmission characteristics will next be discussed in detail. In addition, a thorough description of nanofluids will be provided, emphasizing the thermal and dispersion characteristics that are crucial for understanding how they affect heat exchanger performance. The various baffle plate layouts used in conjunction with nanofluids will then be examined, with an analysis of their impact on pressure drop, fluid flow patterns, and heat transfer efficiency. We'll also go over the difficulties and restrictions that come with using nanofluids in heat exchangers and look into any potential solutions that have been suggested in the literature. This review's ultimate goal is to provide researchers, engineers, and industrial practitioners with insightful analysis and practical advice on how to integrate baffle plates and use nanofluids in the tube side of shell and tube heat exchangers to improve their thermal performance. These inventions have the potential to improve heat exchange procedures in a variety of industries by improving energy efficiency and sustainability.

2 LITERATURE SURVEY

This review delves into improving heat transfer in applications like air conditioning and chemical processing by reducing thermal resistance. It focuses on using surface roughness, like ribs and corrugations, to enhance thermo-hydraulic performance. These surface alterations boost surface area and disrupt flow, reducing resistance. The article surveys experimental, analytical, and numerical studies on different fluids flowing through roughened channels, exploring conventional fluids, nanofluids, and hybrid nanofluids. By examining the impact on heat transfer, it aims to lay the groundwork for advanced research in using nanofluids and hybrid nanofluids in ribbed and corrugated channels, potentially enhancing overall system performance [1]. This paper conducts a numerical analysis of a tubular heat exchanger's air-side performance with flower baffles (FBs). It explores in-line and staggered arrangements for different pitch ratios (PR) and Reynolds numbers (Re). Findings reveal that FBs disrupt stagnation zones and enhance air mixing near tube walls, improving thermal transfer coefficients significantly. Higher PR values with a fixed blockage ratio lead to increased Nusselt number (Nu) and friction factor (f). The best thermal performance (η of 1.77) occurs with in-line baffles at PR=2.5, BR=0.5, and Re=3000. Additionally, the study offers correlations to predict Nu and f, beneficial for practical applications [2]. This study employs numerical modeling to analyze convective heat transfer in a rectangular channel carrying a viscous, non-Newtonian fluid. It focuses on the impact of different corrugation shapes (rectangular, triangular, and semi-circular) on fluid dynamics and thermal characteristics. Results show significant alterations in heat exchange performance due to corrugations. The heat transfer superiority ranking follows: rectangular > triangular > semi-circular, while the reverse order applies for low pressure drop. Additionally, the triangular and semi-circular shapes exhibit comparable effects on heat transfer [3]. The study examined an axial flow tubular heat exchanger using innovative circular baffle plates with trapezoidal air deflectors at different inclination angles. These deflectors created swirl flow in the air, enhancing turbulence and heat transfer from heated tubes. By varying the baffle plate pitch ratio and maintaining Reynolds numbers between 16000–28000, a 7.4% average performance enhancement was observed with a baffle plate featuring 30° deflectors under counter flow, compared to a parallel flow setup under similar conditions [4]. This study utilizes 3D computational fluid dynamics (CFD) to assess heat transfer and fluid flow characteristics using a helical screw tape insert in pipe flow. The inserted helical coil, with a 1.92 twist ratio, enhances a tube's geometry. Examining single-phase laminar flow in an annular channel (200 < Re < 2300), the simulation conducted through ANSYS FLUENT undergoes sensitivity analysis and experimental validation. Results indicate a significant increase in heat transfer rate (Nusselt number) by 1.34-2.6 times and friction factor by 3.5-8 times for the wire-wrapped-tube compared to a plain tube. The thermal performance factor peaks at 3.79 under constant pumping power, while pressure drop rises due to increased flow restriction caused by the inserted coils [5]. The experimental analysis evaluates the thermal-hydraulic performance of fluid flow in tubes with helical screw inserts, studying various strip numbers and twist ratios at transition flow regimes. Comparisons between single and double strip inserts, alongside CFD analysis using the k-kl-w model, visualize thermohydraulic characteristics. Double strip helical screw inserts exhibit enhanced Nusselt numbers at higher Reynolds numbers and lower twist ratios compared to single strips. Friction factors increase with double strips at reduced Reynolds numbers and twist ratios [6]. This study conducts a numerical investigation to assess power dissipation and heat transfer enhancement in standard circular-orifice baffled tubes under combined net and oscillatory flows. It examines unsteady pressure drop across these tubes with various operating conditions and fluids to gauge device power consumption, aligning well with existing experimental data. Modeling hydrodynamic and thermal developing flow with uniform heat flux at the tube wall, water and thermal oil are used as working fluids. Spatial and time periodicity are analyzed before computing the Nusselt number [7]. This paper investigates the heat transfer and flow performance of a double shell-pass rod baffle heat exchanger (DS-RBHX) compared to a single shell-pass counterpart (SS-RBHX) using water as the working fluid. Experimental findings reveal that the DS-RBHX consistently outperforms the SS-RBHX, exhibiting higher overall heat transfer coefficients across all measurements. With varying shell-side volume flow rates, the DS-RBHX demonstrates a substantial increase in shell-side heat transfer coefficient (33.5–54.0%) and pressure drop (34.0–74.3%) compared to the SS-RBHX. The DS-RBHX's comprehensive performance surpasses the SS-RBHX by 14.4–24.3% in shell-side heat transfer coefficient under identical shell-side pressure drops [8]. This study compares various baffled longitudinal flow shell-and-tube heat exchangers (STHX) with a segmental baffle shell-and-tube heat exchanger (SG-STHX) using experimental data with municipal water as the working fluid. It analyzes the shell-side pressure drop components across different flow patterns, noting that the rod baffle STHX (RB-STHX) displays a significantly lower proportion of pressure drop in the tube bundle section compared to SG-STHX and large-and-small hole baffle STHX (LSHB-STHX). The findings underscore the superior thermal-hydraulic performance and energy efficiency of longitudinal flow STHX, supporting their continued application and design enhancements for improved performance [9]. The study explores the impact of different baffle patterns on heat transfer and fluid flow in shell-and-tube heat exchangers through experimental investigation involving tri-flower baffle, pore plate baffle, rod baffle, segmental & pore baffle, and segmental baffle configurations. A water-water heat transfer experiment system is employed for comparison. Results show that, at identical flow rates, the heat transfer coefficients (hs) of tri-flower baffle and pore plate baffle are 33.8% and 17.7% higher, respectively, compared to segmental baffle. Additionally, the pressure drop of rod baffle, tri-flower baffle, and segmental & pore baffle is significantly lower—decreasing by 70.1%, 19.5%, and 31.1%, respectively, in comparison to segmental baffle. The study formulates experimental correlations for these heat exchangers and proposes an optimal design scheme for enhancing shell-side heat transfer based on the findings [10]. The study investigates dead zones in the shell side of a trefoil-baffle heat exchanger, analyzing their distribution and volume fraction to propose structure optimizations for better flow conditions. Using residence time distribution, it quantitatively assesses dead zones, revealing a 28% volume fraction, notably larger (by 13%) than in segmental baffle heat exchangers. Velocity contours show higher fluid velocity in the main flow region than near the wall, where fluid moves slower, contributing to dead zone formation near shell walls and baffles. An optimized tube layout is proposed, reducing the volume fraction of dead zones by around 13% across different flow rates. These findings offer insights into dead zone analysis and structural improvements for enhanced heat exchanger performance [11]. The literature review centers on enhancing heat transfer efficiency in concentric pipe heat exchangers, crucial for modern industry's high heat transfer demands. These exchangers, known for their simple design and ease of manufacturing, have seen extensive investigation and technological advancements over the years. Specifically focusing on double and triple concentric pipe heat exchangers, the review systematically analyzes related studies. It delves into heat transfer enhancement and flow characteristics, providing insights into the heat transfer mechanisms and flow behaviors within these heat exchangers. This comprehensive understanding aims to guide further research and applications in industries reliant on concentric pipe heat exchangers, offering valuable references for their optimization and improved performance [12]. The article focuses on various techniques used in industries for efficient fluid heating and cooling, especially in harnessing solar energy for thermal applications. It highlights methods like corrugated absorber plates, extended surfaces, roughened surfaces, and jet impingement for improved heat transfer. While jet impingement enhances convective heat transfer, the study emphasizes the lack of comprehensive research on diverse jet configurations despite its potential for various engineering applications. By rigorously analyzing previous works, the review provides essential insights into optimal roughened geometries and parameters for enhanced heat transfer in solar air heaters. This detailed information benefits researchers and technical professionals, offering crucial data on flow parameters and heat transfer for future studies, potentially saving time and experimental costs [13]. The study explores heat transfer in half-cylindrical shell spaces of power plant feedwater heaters and countercurrent U-tube heat exchangers. It experimentally investigates overall and shell-side heat transfer coefficients, pressure drop, and comprehensive performance of various baffle structures. These include ladder helical baffles (single inclined, dual inclined, and folded), orifice baffles (fillet triangle, trefoil, and quatrefoil), and segmental baffles, all with two baffle pitches. Results indicate that ladder helical baffles exhibit higher shell-side heat transfer coefficients compared to orifice baffles and segmental ones. However, orifice baffle schemes demonstrate significantly lower shell-side pressure drops compared to ladder helical and segmental schemes [14]. The study introduces an innovative alternating V-rows triangular tube layout in twisted elliptical tube heat exchangers aimed at enhancing heat transfer efficiency between adjacent tubes. Eleven heat exchanger configurations, including two tube layouts, varying aspect ratios of twisted elliptical tubes, and different twisted pitches, were constructed and simulated. These configurations encompassed five coupling-vortex schemes, five parallel-vortex schemes, and a smooth round tube scheme (R10.8), using water as the working fluid with constant properties and shell-side Reynolds numbers ranging from 2000 to 10,000. Results demonstrate that coupling-vortex schemes with concurrent flow and irregular channels, larger aspect ratios, and smaller twisted pitches in the tubes intensify turbulence and secondary flow, enhancing heat transfer [15]. This review consolidates diverse heat transfer strategies employed to enhance the performance of smooth air channels (SACs). Numerous research endeavors, both numerical and experimental, have targeted SACs to improve their efficiency. The review encompasses various obstacle models and configurations, exploring attached, semiattached, or detached obstacles, along with different orientations, shapes, sizes, perforations, and arrangements. These studies primarily focus on altering flow direction, modifying local heat transfer coefficients, and increasing turbulence levels. Such alterations aim to augment heat transfer between the fluid and heated walls, contributing to the overall improvement in system performance [16]. The study aimed to numerically analyze the dynamic and thermal behavior of a turbulent fluid flowing through a two-dimensional horizontal rectangular channel with constant property. In this setup, the upper surface maintained a constant temperature, while the lower surface was thermally insulated. To enhance mixing and subsequently improve heat transfer, two transverse solid obstacles of different shapes—flat rectangular and V-shaped—were strategically inserted into the channel. These obstacles were fixed in a periodically staggered manner, compelling the formation of vortices [17]. The study employed computational fluid dynamics (CFD) to analyze the aerodynamic and thermal characteristics of a turbulent flow of an incompressible Newtonian fluid within a two-dimensional horizontal high-performance heat transfer channel with a rectangular cross-section. The channel's top surface maintained a constant temperature, while the bottom surface was kept adiabatic. To enhance mixing and subsequently improve heat transfer, two obstacles—flat rectangular and V-shaped—were strategically inserted into the channel, fixed to the top and bottom surfaces in a staggered manner to induce vortices [18]. Results revealed that as the A/B ratio increased and S decreased, secondary flow intensified, leading to increased Nusselt number (Nu) and friction factor (f). Additionally, a rise in A/B and S led to higher comprehensive evaluation index Nu·f−1/3. Compared to smooth round tube schemes (R10.8), all twisted tube schemes exhibited higher Nu, f, and Nu·f−1/3 values. Specifically, coupling-vortex schemes demonstrated more uniform temperature fields than corresponding parallel-vortex schemes. The average Nu, f, and Nu·f−1/3 of coupling-vortex schemes C12.3S50/C12.3S100 increased by 12.8%/9.9%, 15.2%/10.5%, and 7.6%/6.3%, respectively, compared to parallel-vortex schemes. Smaller S or an A/B closer to 1 were found to be advantageous for TETHXs-CV, highlighting their potential for improved heat transfer efficiency [19]. The paper offers a comprehensive review of fin-and-tube heat exchangers, extensively exploring their applications in thermal energy conversion across various industries like air conditioning, refrigeration, automotive, and electronics. It delves into the ongoing quest for more efficient cooling systems through compact heat exchangers, prompting extensive research in this domain. The review covers experimental and numerical studies investigating diverse mechanisms for enhancing heat transfer in these heat exchangers. It meticulously examines the influences of operating conditions and the impacts of different geometrical parameters on heat transfer and pressure drop within each mechanism. Additionally, the paper discusses comparative analyses between various heat transfer enhancement mechanisms and explores innovative compound designs for fin-and-tube heat exchangers [20]. The Lead-cooled Fast Reactors (LFRs) stand out as promising candidates for the next generation of nuclear reactors, utilizing molten lead or lead alloys as coolants. These reactors demonstrate exceptional performance in sustainability, thermal-hydraulics, and safety features. The main heat exchanger within LFRs serves as a critical component linking the primary and secondary circuits, significantly influencing the reactor's operational efficiency. In this study, the focus lies on investigating the flow and heat transfer characteristics of fluids within twisted elliptical tubes. This exploration is crucial as it is anticipated that a specific spiral crossflow pattern occurs both inside and outside the tubes. The research begins with theoretical calculations based on a periodical unit model of the twisted tube heat exchanger. Subsequently, numerical simulations are conducted to delve into fluid behavior and heat transfer in both the shell and tube sides [21]. Work done by Ashraf Mimi Elsaid et.al. on shell and tube heat exchanger with helical coil with different inclination angles and use of nanofluid as a heat transfer medium. The study investigates the impact of the inclination angle (ɵ) in a Shell and Helically Coiled Tube Heat Exchanger (SHCT-HE) using water, Al2O3/water, and SiO2/water nanofluids. The experiment explores various volume concentrations (φ) ranging from 0.1 vol% to 0.3 vol% and coil Reynolds numbers (Rec) spanning from 6000 to 15000. Results indicate that increasing the inclination angle enhances the coil Nusselt number (Nuc) and the SHCT-HE effectiveness (ε) while reducing the coil pressure drop (ΔPc). [22].

3. METHODOLOGY

Computational Fluid Dynamics (CFD) is a numerical technique used to simulate the behavior of fluid flows by solving the governing equations that describe fluid motion. Fluent, a commercial CFD software developed by Ansys, offers a wide range of features and capabilities for conducting such simulations. The fundamental governing equations used in Fluent are the Navier-Stokes equations, which describe the conservation of mass, momentum, and energy in fluid flows. These equations are typically solved using numerical methods to obtain approximate solutions.

3.1 Governing Equations:

The Navier-Stokes equations, in their general form, are partial differential equations that represent the conservation of momentum for an incompressible fluid:

Continuity Equation:

$$\nabla \cdot V = 0$$

Momentum Equations:

$$\partial(\rho V)/\partial t + \nabla \cdot (\rho V \otimes V) = -\nabla P + \nabla \cdot \tau + \rho g$$

Energy Equation:

$$\partial(\rho e)/\partial t + \nabla \cdot (\rho e V) = \nabla \cdot (k\nabla T) + Q$$

where V is the velocity vector, ρ is the fluid density, P is the pressure, τ is the stress tensor, g is the gravitational acceleration vector, e is the total energy per unit mass, k is the thermal conductivity, T is the temperature, and Q represents heat sources/sinks.

The governing equations solved by Fluent are the Navier-Stokes equations, which describe the motion of fluid particles and are derived from the principles of conservation of mass, momentum, and energy. These equations are used to numerically simulate fluid flow in a domain.

The 3D time-dependent Navier-Stokes equations for an incompressible fluid flow are as follows:

Conservation of Mass (Continuity equation):

$$\nabla \cdot (\rho * U) = 0$$

where:

ρ is the fluid density,

U is the fluid velocity vector,

∇ is the gradient operator (del operator).

Conservation of Momentum (Navier-Stokes equations):

$$\rho * (\partial U/\partial t + U \cdot \nabla U) = -\nabla P + \mu * \nabla^2 U + \rho * g$$

where:

∂U/∂t is the time rate of change of velocity,

∇U is the velocity gradient tensor,

P is the pressure,

μ is the dynamic viscosity of the fluid,

g is the acceleration due to gravity.

Conservation of Energy (Energy equation):

$$\partial(\rho * E)/\partial t + \nabla \cdot (\rho * E * U) = \nabla \cdot (k * \nabla T) + Q$$

where:

E is the total energy per unit mass (sum of internal energy and kinetic energy),

k is the thermal conductivity of the fluid,

T is the fluid temperature;

Q represents the volumetric heat sources/sinks.

These equations are solved iteratively on a discretized grid within the computational domain using numerical methods to obtain an approximate solution for the fluid flow behaviour. Boundary conditions, initial conditions, and turbulence models (if applicable) are also specified to complete the CFD simulation setup.

3.2 Numerical Methods:

Fluent employs various numerical methods to solve the Navier-Stokes equations efficiently. The Finite Volume Method (FVM) is widely used, where the computational domain is discretized into a grid of control volumes. The integral form of the governing equations is then solved on these control volumes, and the spatial derivatives are approximated using interpolation schemes.

3.3 Turbulence Models:

For simulating turbulent flows, Fluent offers several turbulence models, such as the k-ε model, k-ω SST model, and Reynolds Stress Model (RSM). These models are used to account for the effects of turbulence on the fluid flow and are selected based on the flow characteristics and level of accuracy required.

3.4 Grid Generation:

Accurate grid generation is crucial for obtaining reliable results in CFD simulations. Fluent supports structured, unstructured, and hybrid grid types. The quality of the grid, including cell size, aspect ratio, and smoothness, can significantly influence the accuracy and convergence of the simulation.

3.5 Boundary Conditions:

Appropriate boundary conditions are essential for representing the flow conditions at the domain boundaries. Fluent allows users to specify various boundary conditions, including velocity inlet, pressure outlet, wall, symmetry, and periodic boundaries, among others.

3.6 Post-processing:

Once the simulation is complete, post-processing tools in Fluent enable visualization and analysis of the results. Users can generate contour plots, velocity vectors, streamlines, and other visual representations to gain insights into the fluid flow behavior.

4. CONCLUSIONS

In this review paper, we have extensively investigated the Computational Fluid Dynamics (CFD) analysis of a double tube heat exchanger with various baffle plate configurations, considering the implementation of nanofluid as the cold fluid. The study aimed to explore the potential enhancements in heat transfer performance and fluid flow characteristics through the utilization of nanofluids and different baffle plate designs

In this review, Computational Fluid Dynamics (CFD) analyzed a double tube heat exchanger with varied baffle plate setups, using nanofluids as the cold fluid. Higher Reynolds numbers in the cold fluid boosted heat transfer rate, thermal coefficient, and effectiveness but affected the performance index inversely. Nanofluid integration showed promise, enhancing thermal conductivity for better heat dissipation, notably improving overall heat transfer compared to traditional fluids. Baffle plate designs like helical, segmental, and rod baffles influenced fluid flow and heat transfer differently. Optimized designs and nanofluid use synergistically enhanced heat exchanger performance, offering potential for power generation, chemical processing, and refrigeration applications.

REFERENCES

[1] Sohal, K. Kumar, R. Kumar, Heat transfer enhancement with channel surface roughness: a comprehensive review, Proc. Inst. Mech. Eng., Part C: J. Mech. Eng. Sci. Vol. 236 (11) (2022) 6308–6334 https://doi.org/10.1177/095440622110656.

[2] Sahel Djamel, Thermal performance assessment of a tubular heat exchanger fitted with flower baffles, J. Thermophys. Heat. Transf. 35 (4) (2021) 726–734 https:// doi.org/10.2514/1.T6208.

[3] H. Ameur, D. Sahel, Effect of some parameter on the thermal-hydraulic characteristics of a heat exchanger with corrugated walls, J. Mech. Energy Eng. 3 (1) (2019) 53–60 https://doi.org/10.30464/jmee.2019.3.1.53.

[4] Md Atiqur Rehman, Performance evaluation of turbulent circular heat exchanger with a novel flow deflector-type baffle plate, Journal of Engineering research, https://doi.org/10.1016/j.jer.2023.100105

[5] M. Tusar, K. Ahmed, M. Bhuiya, CFD study of heat transfer enhancement and fluid flow characteristics of laminar flow through tube with helical screw tape insert, Energy Procedia vol.160, (2019) 699–706, https://doi.org/10.1016/j.egypro.2019. 02.190

[6] S.R. Chaurasia, R.M. Sarviya, Comparative thermal hydraulic performance analysis on helical screw insert in tube with different number of strips in transition flow regime, Heat. Mass Transf. vol.57, (2021) 77–91, https://doi.org/10.1007/s00231- 020-02934-6

[7] D. Gonz´ alez-Ju´ arez, R. Herrero-Martín, J.P. Solano, Enhanced heat transfer and power dissipation in oscillatory-flow tubes with circular-orifice baffles: a numerical study, Appl. Therm. Eng. 141 (2018) 494–502, https://doi.org/10.1016/j.applthermaleng.2018.05.115.

[8] X. Wang, Y. Liang, Y. Sun, Z.C. Liu, W. Liu, Experimental and numerical investigation on shell-side performance of a double shell-pass rod baffle heat exchanger, Int. J. Heat Mass Tran. 132 (4) (2019) 631–642, https://doi.org/10.1016/j.ijheatmasstransfer.2018.12.046.

[9] N. Li, J. Chen, T. Cheng, J.J. Kleme, M. Zeng, Analysing thermal-hydraulic performance and energy efficiency of shell-and-tube heat exchangers with longitudinal flow based on experiment and numerical simulation, Energy 202 (2020), 117757, https://doi.org/10.1016/j.energy.2020.117757.

[10] J. Chen, N. Li, Y. Ding, J.J. Kleme, P.S. Varbanov, Q. Wang, M. Zeng, Experimental thermal-hydraulic performances of heat exchangers with different baffle patterns, Energy 205 (2020), 118066, https://doi.org/10.1016/j.energy.2020.118066.

[11] K. Wang, C.P. Bai, Y.Q. Wang, M.S. Liu, Flow dead zone analysis and structure optimization for the trefoil-baffle heat exchanger, Int. J. Therm. Sci. 140 (2019) 127–134, https://doi.org/10.1016/j.ijthermalsci.2019.02.044.

[12] H. Li, Y. Wang, Y. Han, W. Li, L. Yang, J. Guo, Y. Liu, J. Zhang, M. Zhang, F. Jiang, A comprehensive review of heat transfer enhancement and flow characteristics in the concentric pipe heat exchanger, Powder Technol. 397 (2022), 117037, https://doi.org/10.1016/j.powtec.2021.117037.

[13] R. Kumar, R. Nadda, S. Kumar, S. Saboor, C.A. Saleel, M. Abbas, A. Afzal, E. Linul, Convective heat transfer enhancement using impingement jets in channels and tubes: a comprehensive review, Alex. Eng. J. 70 (2023) 349–376, https://doi.org/10.1016/j.aej.2023.02.013.

[14] J. Su, Y. Chen, J. Wu, F. Fei, S. Yang, H. Gu, Experimental investigation on heat transfer performances in half-cylindrical shell space of different heat exchangers, Int. J. Heat Mass Tran. 189 (2022), 122684, https://doi.org/10.1016/j.ijheatmasstransfer.2022.122684.

[15] H. Gu, Y. Chne, B. Sund´ en, J. Wu, N. Song, J. Su, Influence of alternating V-rows tube layout on thermal-hydraulic characteristics of twisted elliptical tube heat exchangers, Int. J. Heat Mass Tran. 159 (2020), 120070, https://doi.org/10.1016/j.ijheatmasstransfer.2020.120070.

[16] Y. Menni, A. Azzi, A. Chamkha, Enhancement of convective heat transfer in smooth air channels with wall-mounted obstacles in the flow path, J. Therm. Anal. Calorimetry 135 (2019) 1951–1976, https://doi.org/10.1007/s10973-018-7268-x.

[17] Y. Menni, A. Azzi, A.J. Chamkha, S. Harmand, Effect of wall-mounted V-baffle position in a turbulent flow through a channel Analysis of best configuration for optimal heat transfer, Int. J. Numer. Methods Heat Fluid Flow 29 (10) (2019) 3908–3937, https://doi.org/10.1108/HFF-06-2018-0270.

[18] Y. Menni, A. Chamkha, C. Zidani, B. Benyoucef, Baffle orientation and geometry effects on turbulent heat

transfer of a constant property incompressible fluid flow inside a rectangular channel, Int. J. Numer. Methods Heat Fluid Flow 30 (6) (2020) 3027–3052, https://doi.org/10.1108/HFF-12-2018-0718.

[19] H. Gu, Y. Chne, J. Wu, B. Sunden, Performance investigation on twisted elliptical tube heat exchangers with coupling-vortex square tube layout, Int. J. Heat Mass Tran. 151 (2020), 119473, https://doi.org/10.1016/j.ijheatmasstransfer.2020.119473.

[20] A. Sadeghianjahromi, C.C. Wang, Heat transfer enhancement in fin-and-tube heat exchangers - a review on different mechanisms, Renew. Sustain. Energy Rev. 137 (2021), 110470, https://doi.org/10.1016/j.rser.2020.110470.

[21] J. Zhao, L. Li, W. Xie, H. Zhao, Flow and heat transfer characteristics of liquid metal and supercritical CO2 in a twisted tube heat exchanger, Int. J. Therm. Sci. 174 (2022), 107453, https://doi.org/10.1016/j.ijthermalsci.2021.107453.

[22] M. Elsaid, M. Ammar, A. Lashin, G. M. R. Assasa, Performance characteristics of shell and helically coiled tube heat exchanger under different tube cross-sections, inclination angles and nanofluids, Case Studies in Thermal Engineering 49 (2023) 103239.