Maximizing Heat Exchanger Efficiency through Optimized Perforation Patterns

[1]Ankit Dwivedi, [2]Amit Kumar Asthana
[1]Research Scholar, Department of Mechanical Engineering, Truba Institute of Engineering & Information Technology Bhopal (M.P.) India
[2]Assistant Professor, Department of Mechanical Engineering, Truba Institute of Engineering & Information Technology Bhopal (M.P.) India
dwivediankit45@gmail.com, asthana603@gmail.com

* Corresponding Author: Ankit Dwivedi

Abstract: *Improving the thermal-hydraulic performance of compact heat exchangers is a major step toward improving the energy efficiency of industrial thermal systems. As part of the present study, a thorough computational fluid dynamics (CFD) examination was conducted to look at the convective heat transfer and pressure drop behavior in an annular double-tube heat exchanger that had perforated segmental spiral plate baffles designed with vertical orientation along the inner heated tube surface. Both tube and shell sides had the working fluid being water under turbulent flow conditions. A three-dimensional numerical model was constructed employing ANSYS Fluent to simulate the heat transfer with temperature-dependent heat transfer coefficients and heat source terms. Two-layer k-εRNG is utilized for turbulent modeling, along with the standard wall function to include the offsetting of the near-wall effects. A new and fresh three-dimensional model was created and implanted in the commercial code ANSYS Fluent in order to investigate the behavior of the SSPE and heat transfer mechanisms in its shell. The study presents a systematic comparison of multiple SSPB configurations, including solid baffles and perforated baffles carrying perforations in various shapes mainly- circular, square, elliptical, triangular, and rhombus. Based on numerical simulation outcomes from both the velocity field and temperature distribution, detailed analyses of heat transfer and pressure dropped are conducted. Also, a configuration with solid baffles is reported as the baseline. The results demonstrate that introducing perforated serrated finned tubes appreciably enhanced heat transfer due to the increased flow disturbance, secondary flow generation, and improvement in fluid mixing within the annulus. A triangular-perforated SSPB outperforms comparison configurations in terms of overall thermal-hydraulic performance. It achieves the optimum heat transfer enhancement with an acceptable increase in pressure drop. Besides, a specially designed SSPB model is proposed based on the numerical results and compared to some of the earlier works exposed in the literature. The outcomes of this research provide significant recommendations for the development of high-performance, energy-efficient double-tube heat exchangers in few industrial settings.*

Keywords: *Double-tube heat exchanger, Perforated baffles, Convective heat transfer, Pressure drop, CFD analysis, Flow enhancement, Thermo-hydraulic*

I. INTRODUCTION

The heat exchangers play an essential role in the versatile engineering practice, namely, nuclear energy technology, chemical processing industry, refrigeration, automotive systems, and thermal engineering. Amongst the various configurations, 'annular double tube heat exchanger' became widely popular because of its advantageous merits, including simplicity in construction, ease of maintenance, compact design, and affordable compliance towards operation under moderate pressure or temperature [1]. However, as heat transfer performance is concerned, the conventional double tube heat exchangers lack enough process with comparative poor convective heat transfer, particularly because of the formation of thermal boundary layers and little fluid diffusion. The design and optimization of double tube heat exchangers, therefore, suffer from the biggest challenge in applications because enhancing the convective heat transfer alongside a tolerable pressure drop continues to become impossible task [2].

There have been numerous solid ideas as well as research over the last decades for solving these limitations. These methods could be broadly grouped into three main categories: active, passive, and compound methods. Active methods involve the addition of energy from external sources such as vibration, pulsating flow, or fluid injection [3], while no external energy is supplied system into the passive techniques, i.e., surface modifications or flow-disturbing devices, keeping the basic objective of saving energy consumption. Of the passive methods, the use of baffles, inserts, ribs, twisted tapes, and surface roughness elements are of considerable interest, owing to their simplicity, reliability, and cost-effectiveness. Baffles particularly are effective in redirecting the flow, promoting turbulence, and enhancing fluid mixing, beneficially contributing to higher convective heat transfer coefficients [4].

Segmental and spiral baffles are highly used with tubular heat exchangers and shell-and-tube exchangers primarily due to a practice involving inducing secondary flows and swirl motion, where spiral baffles outperform segmental baffles as far as their fluid momentum is concerned [5]. However, these solid section baffles impart a considerable drag on the flow due to the associated pressure drop, thus creating counterbalancing by the thermal benefits resulting out of the enhanced heat transfer coefficients. The use of a faired baffle type of a permeate or perforated baffle has solved the problem to a large extent [6]. The essential existence of perforations acts as a means for some of the flow to bypass through the disc of baffle thus diminishing flow resistance yet adaquately disrupting the boundary layer. It is believed that perforated spiral plate baffles can deliver a beautiful balance between the heat transfer improvement and pressure drop punishment if these remain in combination with spiral as well as segmental geometry [7].

Today, Computational Fluid Dynamics (CFD) has marked its presence as an established and powerful tool among engineers, who significantly use it for the in-depth study of heat transfer and fluid flow behavior in complex heat exchanger geometries. CFD offers comprehensive visualization of fields like velocity, temperature distribution, turbulence characteristics [8], pressure variations, which in some cases are impossible to achieve with so much uncertainty with experiments. The analysis on heat transfer and fluid flow behavior through CFD paves the way for exploring ventilation manipulation baffle geometry; orientation, pitch, hole diameter, flow condition; and achieves a better optimum design with minimum cost and design time [9].

Although significant though limited research has explored passive enhancement techniques, there are few studies looking primarily into the role of spiral baffle-equipped annular double-tube heat exchangers, with segmental perforated spiral plate annular baffles placed in parallel with the inner heated tube surface [10]. The effect of the spiral flow induction, segmental flow redirection, and such additional effects as the expansion of the surface area or the flattening of the local temperature gradient to the outer surface owing to the perforations on convective heat transfer characteristics remains to be fully understood. Moreover, most of the studies available dwell on purely experimental studies on fairly simplified baffle configurations, which makes a transition to a CFD-based comprehensive diagnostics even more difficult [11].

A complete CFD investigation is planned in this research on heat transfer through convection enhancement in an annular double-tube heat exchanger with perforated segmental spiral plate baffles. The working fluid used is water, and the thermal-hydraulic performance is evaluated in terms of the Nusselt number [12]-[13], pressure drop, friction factor, and enhancement efficiency. The numerical results shall be carefully validated against experimental data to provide changes in the models not yet observed. It also aims to discover the best baffle design from both higher heat transfer and minimum pressure loss perspectives [14]-[15]. The outcomes of this study are anticipated to help the optimal design of heat exchangers in annular configurations in industrial thermal systems based on improved heat transfer performance. Figure 1 shows Annular Double-Tube Heat Exchanger

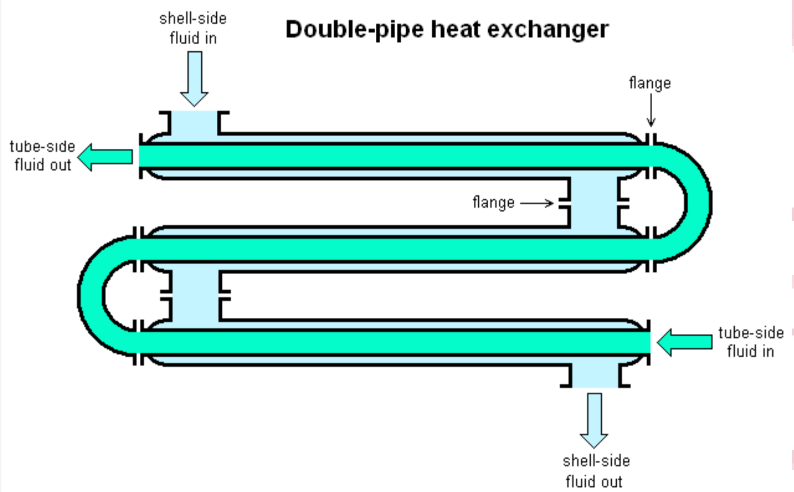

Figure 1: Annular Double-Tube Heat Exchanger

II. RELATED WORK

Various compound passive heat transfer enhancement techniques have been elaborately reviewed with particular emphasis on their applications on numerous heat exchanger systems. An array of enhancement techniques such as baffles, inserts, ribs, and surface modifications have been systematically grouped to bring into focus the advantage of applying multiple passive methods simultaneously. The imbalance between heat transfer enhancement and pressure drop penalties is brought into focus along with the challenges and open issues in optimizing compound passive combinations [1]. Review of passive heat transfer enhancements applicable to different heat exchanger geometries has also been extensively reviewed, along with the twisted tapes, baffles, extended surfaces, and roughened walls. These methodologies are critically shared to be the dominant ones in the industrial sector due to their easy operation, trustworthiness, and low running cost [2].

A combined investigation of passive and active heating and cooling techniques with regard to their potential for effectiveness pertaining to heat transfer, pressure drop, and entropy generation appears in peer-reviewed literatures. Compound methods win in terms of operation over each stand-alone method under several selected operational conditions, although careful optimization is required [3]. Active heat transfer enhancement techniques such as vibration, pulsating flow, electro hydrodynamic forces, and fluid injection showed good potential of improving heat transfer but could be implemented only to a limited extent due to increased complexity and electricity consumption [4]. Nevertheless, passive cooling techniques such as surface roughness, inserts, and flow obstacles proved their effectiveness in terms of energy consumption and system reliability despite the presence of pressure losses [5].

Recently enhanced techniques in heat transfer have been explored due to porous media, microorganisms, and nanofluids, in order to understand their impact on heat transport mechanisms. While these might provide great potentials for enhancement, such factors as stability, fouling, or complexity in modeling pose certain challenges to widespread applications in units of conventional heat exchangers. Studies of theoretical nature or experiments conducted on double-pipe-type heat exchangers have given evidence that active-passive rather than active or passive only techniques can lead to substantial improvements in the heat transfer coefficients, although carefully managing the resulting pressure drop. Reviews every now and then encapsulate the experiments and computational findings about the heat transfer enhancement with an emphasis on the right design strategies.

Experiments and simulations of straight and U-tubes with CFD tools, in general, confirm good flow and thermal prediction compared to lab experiments. Further acceptance can be reported from considering negatively re-target flow into the minimum entropy generation in heat transfer by enhancing the geometry shape of baffles. Studies with twisted tapes or Nano fluids provide insight into how both viscosity and vortex form affect heat transfer. Friction losses will nevertheless occur to maintain a slightly higher heat-transfer rate.

Numerical investigations show that there is an increased level of turbulence and mixing due to winglet configurations, resulting in enhanced Nusselt numbers as compared to steady flow [12]. Jet impingement mechanisms also produce high local heat transfers, although they come with design complexities and practical implementation challenges [13]. With the combination of novel baffle geometries and fluid injection, marked enhancement came about in heat transfer; a good comparison between the experimental and CFD findings was seen [14].

Alternative heat exchanger geometries, such as flat-plate and finned-tube designs, have been experimentally examined and proven to provide enhanced thermal performance compared with conventional designs in terms of heat transfer effectiveness [15–16]. Currently, much attention is focused on perforated baffle plates because they prove to provide an efficient mixing of flow but restrict high pressure drops, thereby resulting in substantial improvement in the convective heat-transfer enhancement [17]. Heat transfer enhancements due to nanofluids and data-based modeling were also demonstrated for annulus heat exchanger geometries, exploring how fluid properties have an impact on heat-transfer performance [18].

The study of heat transfer at microscale and phase change is driven towards understanding the effect of boiling and condensation processes on annular microchannels and augmented tubes. This investigation reported that among the flow regimes, surface properties, heat flux, and input quantities are influencing factors on the heat transfer performance [19–23]. Spirals and resonant flow passages have been carefully examined to enhance mixing and increase the surface area thereby enhancing condensation and two-phase heat transfer characteristics [24,25]. In partial-channels having micro-geometries, surface-modeled with pin-fin and the like, heat transfer under boiling conditions improves somewhat, though the increase in heat drop becomes a factor [26].

A combined study of numerical and experimental approaches was pursued in the exploration of the thermal behavior of double-layer tube and annular systems under extreme conditions; this was aimed at improving understanding of the subsurface dynamics and thermal systems of the ocean's depths. Studies with numerical methods on non-Newtonian and magnetohydrodynamic flows in inclined annular tubes yield insight into the advanced transport phenomena of interest to specialized heat transfer applications [28]. And the latest studies have been on annular plate heat exchangers for thermal energy storage, cooling system for reactor coolant pumps, enhanced microchannel boiling, and oscillatory cooling mechanism in engine applications, reflecting the increasing importance of CFD and experimental validations in modern heat transfer research [29–32].

Table 1: Literature Review on Heat Transfer Enhancement Techniques

Ref.	Technique Used	Key Contributions	Results	Limitations
[11]	Twisted tape + nanofluid	Investigated vortex interactions	Significant Nusselt number enhancement	Increased friction factor
[12]	Pulsating flow with winglets	Analyzed unsteady flow effects numerically	Improved mixing and heat transfer	Complex control requirements
[13]	Jet impingement	Reviewed jet-based enhancement methods	Very high local heat transfer	Non-uniform temperature distribution
[14]	Baffles + air injection	Combined experimental and CFD analysis	Enhanced heat transfer validated numerically	Increased system complexity
[15]	Modified flat plate geometry	Investigated geometric innovation	Improved exchanger effectiveness	Limited scalability

[16]	Corrugated channels	Reviewed hydrothermal performance	Enhanced turbulence and mixing	Pressure drop increases
[17]	Perforated baffles	Evaluated perforation effects experimentally	Improved heat transfer with lower pressure loss	Optimal perforation parameters not generalized
[18]	Nanofluids in annulus	Integrated ANN with experiments	Improved prediction accuracy	Model depends on data quality
[19]	Annular boiling	Studied flow boiling regimes	High heat flux capability	Limited to microscale
[20]	Hydrophobic enhanced tubes	Developed condensation correlations	Improved condensation efficiency	Restricted to specific refrigerants
[21]	Two-phase microchannel flow	Analyzed flow regime impacts	Detailed heat transfer characterization	Complex experimental setup
[22]	Structured boiling surfaces	Investigated nucleation mechanisms	Significant boiling enhancement	Manufacturing complexity
[23]	Structured boiling surfaces	Reinforced findings of [22]	Consistent enhancement trends	Duplicate reference
[24]	Spiral tube condensation	Studied non-azeotropic mixtures	Enhanced condensation heat transfer	Limited mixture types
[25]	Oil–air two-phase flow	Experimental thermal characterization	Relevant aerospace insights	Narrow operating range

III. RESEARCH OBJECTIVES

- To investigate the convective heat transfer and pressure drop in an annulus with perforated SSPBs aligned along the inner heated tube surface, using water as a working fluid by using CFD analysis.
- To develop a proposed model for SSPB with optimised geometrical parameters and compare with existing base experimental results

IV. RESEARCH METODOLOGY

This particular study aimed to explore forced convective heat transfer and pressure drop characteristics in an annular double-tube heat exchanger (DTHE) fitted focused with perforated segmental spiral plate baffles (SSPBs) using a systematic Computational Fluid Dynamics (CFD) study. The investigation was therefore confined only in improving the thermal performance incorporating differing perforation geometries of ordered SSPBs using water as the working fluid on both tube and shell sides. Given the appropriateness of baffle design to computational accuracy, physical realism and cross-comparison, the methodology presented several objectives which included evaluating the effect due to perforated SSPBs on heat transfer and pressure drop, trying to figure out the influence of different perforation figures and angles, and questing an optimal pattern in baffle plug type which allows maximum thermal enhancement and minimum hydraulic loss. An elaborate CFD package is a comprehensive precondition enabling a profound comprehension of flow dynamics and thermal interactions to foster a stronger realization for creating more efficient DTHE systems of this design, that can consume less energy while recovering the overall losses.

Geometry Modeling

Three-dimensional CAD models of the annular double-tube heat exchanger (DTHE) were developed and studied with its geometrical and thermal details. The geometry consists of only one forceful flow copper tube, very slim in diameter, with an outer PVC shell that carries the annular flow channel. For further enhancement of convective heat transfer, segmented spiral plate baffles (SSPBs) were placed along the inner tube surface. A model of different baffle configurations affecting fluid flow and thermal performance were investigated, starting from a no-baffle baseline case, which is considered the most primitive baffle configuration, towards different configurations with solid baffles without perforations, as well as variation on perforated baffles including different hole shapes, such as circular, square, triangular, rhombus, and elliptical, in both horizontal and vertical orientations. This diversity of baffles taken into account aims to enable a rational examination of perforation effects of orientation on heat enhancement ad pressure drop characteristics. In the preprocessing stage, the CAD models were imported into the ANSYS Workbench. Design Modeler was used for preparing the geometry of the model. During geometry cleanup, surface issues like overlapping faces, sharp edges, etc. are rectified in order to ensure smooth mesh quality for consistent numerical reliability. Clarity of model preparation leads to the foundation of a high-quality computational domain, allowing an analysis of flow patterns, vortex formation on the deflected baffle configurations, and thermal boundary layer development. The attention to baffle geometry this way while keeping the per-processing to be rigid and trying out a variety of baffles and baffle designs acts as a baseline for a strong foundation in further CFD simulations that allow accurate prediction of the thermal enhancement and hydraulic performance evaluations in the annular double tube heat exchanger (DTHE) system.

Meshing Strategy

The computational domain is covered through two heterogeneous mesh types of grid, viz., structured and unstructured, which are optimized for perfectly resolving complex flow and heat phenomena through the annular DTHE. The structured mesh is applied through the geometrical-abnormality-free areas of regularity, like straight sections on the inner tube, so that high accuracy and numerical stability are upheld. Such unstructured mesh application on the spirally baffled segmental plates and circular perforations allows the mesh to very much smoothen on the boundary layers, without bothering their parallelism, in a way that the geometricity as the smallness of fluid neglectance. The localized refinement of meshes anywhere required, when steep gradients of velocity and temperature are anticipated, should be done, such as near the inner tube walls, near the baffle walls, and near the perforation edges. The targeted mesh refinement, hence created between boundary layers, flow separation, and vortex formation. with its accomplishment, ensures the latter to be resolved on effect and served with adequate accuracy for any meaningful prediction for heat transfer and pressure drop correlations. As statistical simulations demand extensive CPU limits, mesh independence tests are also put in action in order to determine the exact resolution and fine accuracy for the meshing stage in use. Such mesh independence tests will generally be carried on continually refining meshes progressively, each time one would watch for potential changes in any core parameter like the Nusselt number and pressure drop until changes reduce to a predefined threshold. This is an indication of introduction of insensitivity of the solution to any further increment of mesh refinement. This method helps in ensuring computational results are not only accurate but also time-efficient, building the groundwork for a more detailed CFD analysis on the thermal-hydraulic performance of DTHE with distinct baffle configurations.

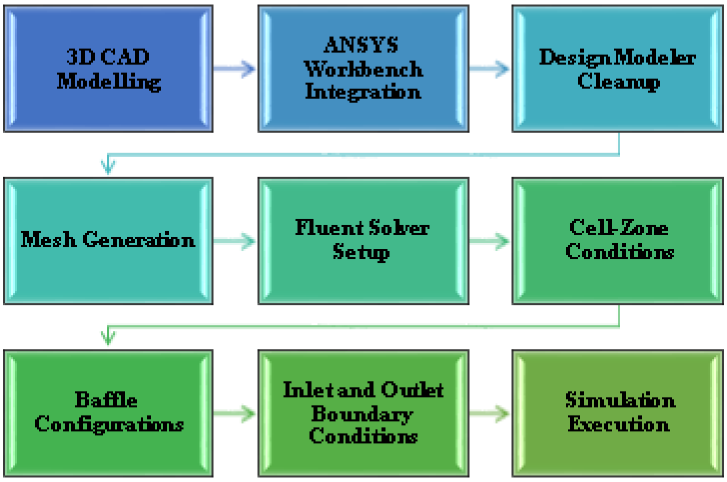

Figure 2: Flow Chart of Adopted Methodology

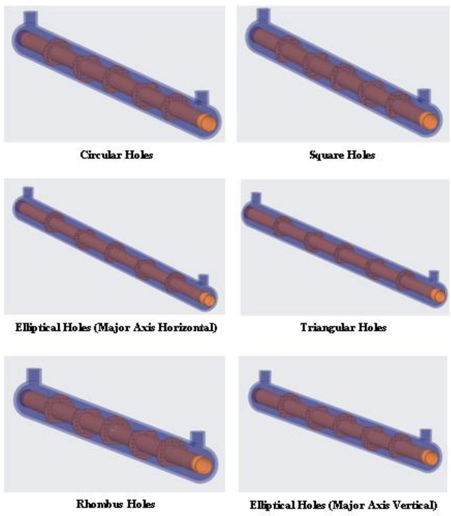

Figure 3: Specific CASES Considered in this Study

Governing Equations and Physical Models

CFD was based on the steady state using the pressure-based solver in ANSYS Fluent to model the flow and thermal fields within the double-tube heat exchanger (DTHE). The flow is 3D and incompressible, which is more appropriate for the low-speed state of the liquid flow in that it occurs in this experiment. The energy equation is implemented to account for convectional heat transfer between the inside of the copper tubes and the working fluid, leading to better prediction capabilities of temperature distribution and thermal performance. To describe turbulence, the realizable k–ε model combined with standard wall treatment models is used for its high robustness and reliability in handling complex internal flow fields with various effects like separations, recirculation, and swirling associated with the spiral segmental plate baffles (SSPBs). This model optimally describes the interaction between turbulent eddies and boundary layers, and hence through that, it ensures a fair resolution of velocity and thermal gradients near walls and baffle surfaces. Appropriate boundary conditions to the specified inlet velocity and temperature are applied, while the outlet pressure is specified to be at ambient conditions to exhibit realistic operating scenarios. As the convergence criteria are strictly monitored to attain numerical stability and accuracy in the forecasts, residuals continue dropping to negligible levels. In sum, this CFD model offers a reliable framework that pertains to investigating the impacts brought forth by various baffle configurations upon improvement of the convective heat transfe-r and pressure drop characteristics in the DTHE mechanism.

Material Properties and Cell Zone

The computational domain was discretized into two different regions corresponding to the fluid and solid, so as to truly represent the configuration of the heat exchanger. Water was chosen as the working fluid, which exhibits the most favorable thermal properties and is widely applied in practical heat transfer devices. The inner tube and the perforated baffles were modeled using copper, belonging to a material category of high thermal conductivities, which is intended to efficiently assist heat transfer along the solid surfaces with the flowing fluid. The external shell is a polyvinyl chloride (PVC), being a material of low thermal conductivity, to minimize heat loss to the environment. Conjugate heat transfer has been accommodated within the framework to solve for two sets of energy equations simultaneously with the fluid and solid domains allowing for the modeling of sensible predictions for temperature gradients, heat flux distributions, and interfacial thermal interactions, hence offering an improved and physically sound representation of the system's total heat transfer response within the annular heat exchanger.

Boundary Conditions

The relatively realistic and practical boundaries defined herein will enable a detailed simulation of the heat exchanger system, particularly focusing on some of the typical operating conditions. On the tube side, we need to prescribe a mass flow rate of 0.1343 kg/s, with the exit temperature at 50 °C, in which the fluid is now relatively at higher temperature. On the shell side, a mass flow rate of 0.2 kg/s is applied as an initial design at an entry temperature of 15 °C, meaning the working fluid is rather cold. These initial conditions encompass a clear thermal driving force, which is the key to effectively implementing the convective heat transfer analysis under consideration. Over the exits, boundary conditions are specified as 'pressure-outlet' on the baffles and at the tube-side ends. This enables the observation and enhancement of a more natural flow, and accordingly, will provide some nudge to the solver in adjusting the outlet velocities as per the flow field inside the tube, so as to bring in numerical stability. Conversely, the application of outlet conditions in the pressure and suction at the outgoing places definitely prevents the deflection of inlets to gradients or even the establishment of fluid informational flows. Some form of 'no-slip' was based on boundary conditions which were also considered on impassable solid boundaries encompassing inner tubes, baffles, and outer shell. For these, the wall velocity is kept at zero velocity, thereby ensuring the norms of boundary-layer performance. These combined boundary conditions for their proximity bear striking resemblance to the actual operating environment involved in industrial double-tube heat exchangers, thus leading to enhancing the replicability and practical relevance of the findings of simulations.

Performance Evaluation Parameters

Various flow discharge and heat transfer parameters are used when checking for several thermohydraulic behaviors of configurations. This is aimed to basically serve as the tool in comparing configurations against the particular basecase. It is found that the temperature descent on the tube side denotes the amount of heat being transferred as the hot fluid passes through the inner tube. Here lies the correct understanding of the given leg of heat transfer augmentation application. Similarly, the temperature decrease on the shell side is used to justify annular absorptivity of the cold fluid. Besides thermophysical, effect of baffle positioning on velocity distribution within circulation is meticulously analyzed so as to inform its importance on flow motion and hence mixing, turbulence rise, and recirculatory zones. All these flow parameters are critically important ingredients in augmentative heat transfer by convection. In the last exercise, the global performance assessment of each configuration was compared against a basic structure by calculating a percentage increase in performance indicators. This comparison thus enables the clear evaluation of the effectiveness of the proposed design modifications in improved heat transfer while maintaining acceptable ways for hydraulic and flow.

V. RESULT AND DISCUSSION

A holistic analysis of thermal and fluid flow behavior within a double-tube heat exchanger (DTHE) in the specified operating conditions was conducted. The results indicated the different cross-sectional visualizations of the temperature distribution through DTHE. These contours firmly adopt the existence of gradients in the temperature and show the character of the thermal development corresponding to the absence of baffling.
In addition to the thermal analysis, a detailed understanding was achieved on the velocity fields within the fluid domain. The velocity contour plots and streamlines have shown how flow velocity is spatially distributed exhibiting flow patterns specific to the parametric design of the unbaffled configuration. The analysis elaborated on how the unrestricted flow would affect matters of velocity distribution and mixing and then made within the heat exchanger.
Important temperature and velocity values were recorded at the inlet and outlet of the tube side and shell side of the DTHE for the purpose of quantitative valuation and comparative evaluation. The data furnish the basic point of reference for gauging the influence of different baffle configurations taken into consideration for subsequent cases.

Case 1: No Baffles

This work is going to be diving into how the presence of baffles affects a double tube heat exchanger (DTHE). Representations were introduced in the literature for the temperature distributions for not having baffles and the variation in temperature presented as graphs. A few more visual representations were introduced as fluid velocity distributions and their corresponding flow picture in an absence of baffles. DTHE behavior in terms of heat and fluid becomes clear with such insights under such circumstances.

Table 3: Temperature and Velocity for Tube and Shell Sides with No Baffles

No baffles		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	48.44
Shell side	15.00	18.54
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.37
Shell side	0.90	1.06

Case 2: Baffles with No Holes

The explored issue is to comprehend the influence that solid baffles, devoid of any openings or apertures, have on the overall performance of the heat exchanger. Solid, rigid baffles were made to change the flow pattern of the fluid and also to facilitate the movement of heat flow in DTHE. This is their original initiation. In fact, they could not have apertures or holes to avoid causing any additional complexities. The study aimed to explain how solid baffles have an influence on such parameters as temperature distribution and fluid velocity profiles whenever unperforated.

Table 4: Temperature and Velocity for Tube and Shell Sides with No hole Baffles

Baffles with No hole		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	44.75
Shell side	15.00	24.14
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.37
Shell side	0.90	0.96

The temperatures read while analyzing the "Baffles with No Holes" condition within the double-tube heat exchanger (DTHE) appear as follows: Figure 6 Temperature distributions within the DTHE with Baffle without holes.
On the tube side of the DTHE, the temperature at the inlet was 50.00°C, and it decreased to 44.75°C at the outlet. As opposed to the tube side, on the shell side of the DTHE, the temperature at the inlet was much lower, only 15.00°C, but it managed to increase with an increase to about 24.14°C.
Some of the measurements of velocity had been in this analysis in the condition "Baffles with No Holes" of double-tube heat exchanger (DTHE). Figure 7 Velocity distributions within the DTHE with Baffle without holes. In the double-pipe heat exchanger, flows at the inlet were at a velocity of 0.26 m/s through the tubes, increasing to 0.37 m/s at the outlet with the exit outlet losses at the outlet. Shell-side flows were at 0.90 m/s near the inlet, accompanied by further increases around the quench-zone region, to 0.96 m/s at the outlet.

Case 3: Baffles with Circular Holes

In this study, the experimental investigation has been carried out to investigate the performance of a double-tube heat exchanger (DTHE) with perforated baffles. The main objective is to examine how the introduction of circular openings influences fluid flow behavior and enhances thermal energy transfer in the DTHE. Hence, the study has comparative and quantitative ideas on how circular perforations may influence core performance factors like temperature distribution and fluid velocity profiles.

Table 5: Temperature and Velocity with Circular hole Baffles

Circular holes		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	44.57
Shell side	15.00	18.90
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	1.12

Case 4: Baffles with Square Holes

The main stress of the present work involves examining whether or not strategically introduced square holes in the baffle attributes to the optimization of heat exchanger performance. Square perforations are introduced, through which alteration in fluid flow patterns is engineered to enhance the heat transfer efficiency of DTHE. Therefore, this study is planned to provide a conceptual foundation to examine how these specific square openings tune a number of cardinal parameters, which can be reduction of temperature distribution profiles and a slightly increased pattern of fluid velocity profiles within heat exchangers as per the state of the problem.

Table 6: Temperature and Velocity with Square Hole Baffles

Square holes		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	44.84
Shell side	15.00	16.93
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	1.01

Case 5: Baffles with Elliptical Holes (Major Axis Horizontally Aligned)

The aim of this study was to examine how elliptical orifices with specific alignments impact the functionality of the heat exchanger. Elliptical holes are ingeniously introduced in a way that changes the fluid flow-paths and improves heat transfer processes within the DTHE. In other words, this study aims to offer a complete understanding of the various impacts of these elliptical openings arranged horizontally as far as critical parameters like temperature distribution and fluid-velocity profiles are required.

Table 7: Temperature and Velocity with Elliptical Holes Horizontally Aligned Baffles

Elliptical holes horizontally aligned		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	43.70
Shell side	15.00	19.61
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	1.08

Case 6: Baffles with Triangular Holes

This case study will provide a detailed description of the impact of incorporating three sides, namely, triangular holes in the baffles on the overall performance of the DTHE. The ultimate aim is to determine how efficiently the DTHE can perform when these carefully placed triangular holes in the baffles are employed. The triangular holes were introduced to induce changes in fluid flow patterns, and hence enhance heat transfer within the DTHE.

Table 8: Temperature and Velocity with Triangular holes

Triangular holes		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	43.57
Shell side	15.00	20.40
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	1.03

Case 7: Baffles with Rhombus Holes

The aim of the case study was to determine the performance characteristics of a double-tube heat exchanger (DTHE) configuration full with rhombus-shaped perforated baffles. It was important to evaluate how baffle rhombus-shaped perforations influence heat transfer performance of the heat exchanger. Rhombus-shaped perforations insert a new dimension to the DTHE flow, improving it with the heat transfer process.

Table 9: Temperature and Velocity with Rhombus Holes

Rhombus holes		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	44.18
Shell side	15.00	19.90
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	1.02

Table10: Temperature and Velocity with Elliptical Holes Vertically Aligned

Elliptical holes vertically aligned		
Temperature (°C)	At Inlet	At Outlet
Tube side	50.00	44.38
Shell side	15.00	19.50
Velocity (m/s)	At Inlet	At Outlet
Tube side	0.26	0.38
Shell side	0.90	0.95

Comparative Results for Temperature

Considering the proposed inlet conditions are 50°C and 15°C for tube and shell side, each thermocouple must be exposed to these temperatures, proceed the desired effect under pressure; assess the calculated inlet conditions, and plot the theoretical temperature using Weibull fit line for each thermocouple.

Table 11: Comparison of Temperature Drop on Tube Side and Temperature Rise on Shell Side for Different Baffle Configurations

C. No.	Baffle Configuration	Tube side TS	Shell side SS	% Reduction TS	% Accession SS
1	W/o Baffle	48.44	18.54	3.12	3.54
2	No Hole	44.75	19.14	10.494	4.14
3	Circular	44.57	19.90	10.856	4.90
4	Square	43.88	20.93	12.24	5.93
5	Elliptical horizontal	44.78	20.51	10.44	5.51
6	Triangle	43.57	24.40	12.852	9.40
7	Rhombus	44.18	20.70	11.646	5.70
8	Elliptical vertical	44.38	19.80	11.244	4.80

Comparative Results for Velocity

The recorded velocity readings at both the tube side outlet and shell side outlet provide crucial insights into how different baffle configurations impact fluid flow within the double-tube heat exchanger (DTHE). With consistent inlet velocities of 0.26 m/s for the tube side and 0.90 m/s for the shell side, variations in velocity profiles are observed as the fluids interact with the heat exchanger's internal components and baffles.

Table 12: Percentage Reduction in Temperature and Velocity for Various Baffle Configurations

C.No.	Baffle Configuration	Tube side TS	Shell side SS	% Reduction TS	% Reduction SS
1	W/o Baffle	0.38	0.950	46.15	5.56
2	No Hole	0.38	1.027	46.15	14.07
3	Circular	0.38	1.030	46.15	14.44
4	Square	0.37	1.080	42.31	20.00
5	Elliptical horizontal	0.38	1.029	46.15	14.31
6	Triangle	0.37	1.120	42.31	24.44
7	Rhombus	0.38	1.050	46.15	16.67
8	Elliptical vertical	0.38	1.040	46.15	15.56

VI. CONCLUSION AND FUTURE WORK

In this study, the research gap was filled by investigating the thermal-hydraulic performance of a double-tube heat exchanger with segmental spiral plate baffles featuring perforations (SSPBs) using a detailed CFD analysis. Numerical simulations showed the effects of baffle geometry, perforation shape, and orientation on convective heat transfer and pressure drop characteristics in the annular flow region, with water as the working fluid. A geometrical arrangement without any baffle served as the benchmark against which various arrangements with baffles were set and later analyzed.

The experimental outcomes signify the improvement of heat transfer intensity with the introduction of SSPBs; thanks to turbulence intensification, the secondary flow formation and fluid mixing at the vicinity of the heated inner tube. Among all configurations investigated, perforated baffles indeed proved to create a fine balance between augmentation of heat transfer and pressure drop. Particularly, triangular perforated type SSPBs displayed significantly prominent performance in heat and thermodynamics. Their absorption charge values suggested a high decrease in tube side outlet temperature and an increased-upon-thermal dissipation at shell-side with tolerable pressure drops. The strength of these SSPBs comes down to strong flow separation and swirling that Taylor-ordered sharp-edge triangular perforations provoke. As clearly seen in comparison, hole shape and alignment have significant influence in determining flow behavior and thermal performance. The optimized configuration of the SSPB identified in this study displays a decent proximity to the experimental results obtained, giving evidence for the reliability of the CFD tools in use. This work provided the opportunity to shed some light upon the enhancement potential of geometrically optimized, perforated SSPBs as an effective passive technique to augment performance of a double-tube heat exchanger. The design will enhance energy efficiency while cutting down operational costs considerably. Future research may focus on experimental validation of the optimized SSPB configuration, investigation of different working fluids including nanofluids, and transient flow analysis. Additionally, multi-objective optimization considering heat transfer, pressure drop, and manufacturing feasibility can further enhance practical applicability.

REFERENCES

[1] Al-Chlaihawi, Kadhim K. Idan, Moayed R. Hasan, and Ali L. Ekaid. "Compound passive heat transfer augmentation techniques: A comprehensive review." *Heat Transfer* 53.2 (2024): 363-421.
[2] Nitturi, Lalitha Krishna, et al. "Augmentation of heat transfer through passive techniques." *Heat Transfer* 52.6 (2023): 4422-4449.
[3] Bhattacharyya, Suvanjan, et al. "Thermal performance enhancement in heat exchangers using active and passive techniques: a detailed review." *Journal of Thermal Analysis and Calorimetry* 147.17 (2022): 9229-9281.
[4] Mraiza, Duaa, and Foad Faraji. "Active Techniques of Heat Transfer Enhancement: A review." *Babylonian Journal of Mechanical Engineering* 2024 (2024): 99-105.
[5] Mebarek-Oudina, F. "Exploring passive heat transfer enhancement techniques: applications, benefits, and challenges." *Babylonian Journal of Mechanical Engineering* 2024 (2024): 122-127.
[6] Habibishandiz, M., and M. Z. Saghir. "A critical review of heat transfer enhancement methods in the presence of porous media, nanofluids, and microorganisms." *Thermal Science and Engineering Progress* 30 (2022): 101267.
[7] Hamed, Hussein H., Awad E. Mohammed, and OMAR ABED HABEEB. "The effect of using compound techniques (passive and active) on the double pipe heat exchanger performance." *Egyptian Journal of Chemistry* 64.6 (2021): 2797-2802.
[8] Guo, Zhixiong, et al. "Heat transfer enhancement− A brief review of literature in 2020 and prospects." *Heat transfer research* 52.10 (2021).
[9] Andrzejczyk, Rafał, Tomasz Muszynski, and Przemysław Kozak. "Experimental and computational fluid dynamics studies on straight and U-bend double tube heat exchangers with active and passive enhancement methods." *Heat Transfer Engineering* 42.3-4 (2021): 167-180.

[10] Jayranaiwachira, Nuthvipa, et al. "Entropy generation and thermal performance of tubular heat exchanger fitted with louvered corner-curved V-baffles." *International Journal of Heat and Mass Transfer* 201 (2023): 123638.
[11] Chaurasiya, Prem Kumar, et al. "Exploring the combined influence of primary and secondary vortex flows on heat transfer enhancement and friction factor in a dimpled configuration twisted tape with double pipe heat exchanger using SiO2 nano fluid." *International Journal of Thermofluids* 22 (2024): 100684.
[12] Akcay, Selma. "Numerical analysis of heat transfer improvement for pulsating flow in a periodic corrugated channel with discrete V-type winglets." *International Communications in Heat and Mass Transfer* 134 (2022): 105991.
[13] Waware, Shital Yashwant, Sandeep Sadashiv Kore, and Suhas Prakashrao Patil. "Heat transfer enhancement in tubular heat exchanger with jet impingement: A review." (2023).
[14] Al-darraji, Ahmed Rahmah, et al. "Enhancement of heat transfer in a vertical shell and tube heat exchanger using air injection and new baffles: Experimental and numerical approach." *Applied Thermal Engineering* 236 (2024): 121493.
[15] Islam, Mohammad S., and Suvash C. Saha. "Heat transfer enhancement investigation in a novel flat plate heat exchanger." *International Journal of Thermal Sciences* 161 (2021): 106763.
[16] Alfellag, Mohanad A., et al. "The hydrothermal performance enhancement techniques of corrugated channels: a review." *Journal of Thermal Analysis and Calorimetry* 147.19 (2022): 10177-10206.
[17] Rahman, Atıqur. "Experimental investigations on single-phase heat transfer enhancement in an air-to-water heat exchanger with rectangular perforated flow deflector baffle plate." *International Journal of Thermodynamics* 26.4 (2023): 31-39.

[18] Borhani, Mohammad Amin, et al. "Heat transfer evaluation of combined nanofluids in vertical annulus: An experimental and artificial neural network approach." *Progress in Nuclear Energy* 175 (2024): 105339.
[19] Zhang, Zheng, et al. "Boiling heat transfer of annular flow in a microchannel." *Physics of Fluids* 37.5 (2025).
[20] Li, Wei, et al. "Correlations of condensation heat transfer in hydrophobic enhanced tubes with using R32." *International Communications in Heat and Mass Transfer* 164 (2025): 108886.
[21] Huang, Guanghan, et al. "The impacts of gas-liquid two-phase flow regimes on heat transfer performance and pressure drop in microchannels-an experimental study." *International Journal of Thermal Sciences* 219 (2026): 110232.
[22] Inanlu, Mohammad Jalal, et al. "Unveiling the fundamentals of flow boiling heat transfer enhancement on structured surfaces." *Science Advances* 10.45 (2024): eadp8632.
[23] Inanlu, Mohammad Jalal, et al. "Unveiling the fundamentals of flow boiling heat transfer enhancement on structured surfaces." *Science Advances* 10.45 (2024): eadp8632.
[24] Tian, Zhongyun, et al. "Experimental analysis on the flow patterns and conversion mechanisms of condensing flow with non-azeotropic mixtures in spiral tube." *Experimental Thermal and Fluid Science* 158 (2024): 111245.
[25] Liu, Yewei, et al. "Experimental study of oil-air two-phase flow and heat transfer characteristics in vertical vent pipes of aviation engine." *Case Studies in Thermal Engineering* (2025): 106687.
[26] Alnaimat, Fadi, Abdul Hanan, and Bobby Mathew. "Boiling heat transfer of water on smooth and square pin fins surfaces in minichannel." *International Communications in Heat and Mass Transfer* 159 (2024): 108186.
[27] Zhang, Zheng, et al. "Research on heat transfer mechanisms in deep-water double-layer tube twin gradient drilling systems." *Geoenergy Science and Engineering* 240 (2024): 213025.
[28] Ahmed, M. M., et al. "The profound effect of heat transfer on magnetic peristaltic flow of a couple stress fluid in an inclined annular tube." *Modern Physics Letters B* 38.25 (2024): 2450233.
[29] Lu, Shixiang, et al. "Operational characteristics of a novel phase change thermal storage device under rotational actuation of the annular plate heat exchanger." *Energy* (2025): 138039.
[30] Tang, Kun, et al. "Research on flow and heat transfer characteristics of built-in cooling system in shaft sealed reactor coolant pump." *Annals of Nuclear Energy* 224 (2025): 111698.
[31] Yin, Liaofei, et al. "Flow boiling heat transfer in multi-stage enhanced open microchannels with micro/nano structures." *Applied Thermal Engineering* 258 (2025): 124695.
[32] Wang, Zhaowen, et al. "Mechanisms of enhanced heat transfer of piston oscillating cooling in a heavy-duty diesel engine." *Applied Thermal Engineering* 245 (2024): 122875.