INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 07, July 2025, pp : 1679-1684
e-ISSN : 2583-1062
Impact Factor : 7.001
www.ijprems.com
editor@ijprems.com

CFD ANALYSIS OF HEAT TRANSFER ENHANCEMENT IN SHELL AND TUBE HEAT EXCHANGER USING FLOWER-SHAPED BAFFLE PLATES

Kamlesh Bramhe[1], Chandra Pratap Patel[2]

[1]Scholar, School Of Mechanical Engineering Sam Global University, Raisen (M.P.), India.

[2]Prof., School Of Mechanical Engineering Sam Global University, Raisen (M.P.), India.

ABSTRACT

Shell and tube heat exchangers (STHEs) play a vital role in thermal energy management across various industrial applications. However, traditional baffle configurations often result in flow-induced vibration, increased pressure drop, and inefficient heat transfer. This study presents a computational fluid dynamics (CFD) analysis of a novel baffle configuration—flower-shaped baffle plates—to investigate their impact on heat transfer and flow characteristics in a shell and tube heat exchanger. A 3D steady-state CFD model was developed and simulated using ANSYS Fluent under turbulent flow conditions. The performance of flower baffles was compared with that of conventional segmental baffles in terms of Nusselt number, pressure drop, velocity profile, and temperature distribution. The results indicate a significant enhancement in heat transfer performance using flower baffles, with an acceptable increase in pressure drop. This research demonstrates the effectiveness of baffle geometry modification as a viable approach for improving thermal performance in STHEs and offers insight for future industrial-scale applications.

Keywords: Shell And Tube Heat Exchanger, Flower Baffle, Heat Transfer Enhancement, CFD Simulation, Pressure Drop, Nusselt Number, Thermal Performance, ANSYS Fluent.

1. INTRODUCTION

Shell and tube heat exchangers (STHEs) are widely employed in industries such as power generation, petrochemicals, refrigeration, and process engineering due to their robust design, large surface area, and operational flexibility. Despite their advantages, the thermal efficiency of STHEs is often constrained by flow maldistribution, dead zones, and excessive pressure drops associated with conventional segmental baffle arrangements. These limitations adversely affect heat transfer rates and overall system performance. Hence, the design and optimization of internal components such as baffles have become a critical area of research to improve the thermal-hydraulic performance of these systems. Among various design modifications, baffle geometry plays a significant role in dictating the fluid flow path, turbulence intensity, and heat transfer rate. Traditional segmental baffles, while simple to manufacture, can lead to vibration, fouling, and high-pressure losses. Therefore, unconventional baffle shapes such as helical, flower-shaped, and doughnut baffles have been introduced to address these issues. Flower-shaped baffles, in particular, offer multiple flow passages and induce localized turbulence, which enhances heat transfer by disrupting boundary layers and promoting better fluid mixing. The use of Computational Fluid Dynamics (CFD) has enabled researchers to simulate and visualize flow and temperature fields within heat exchangers with high accuracy. CFD-based investigations allow parametric studies without the need for costly experimental setups, thus accelerating the design optimization process. In this research, a 3D CFD model of a shell and tube heat exchanger equipped with flower-shaped baffles is developed to evaluate its thermal-hydraulic performance. The numerical results are analyzed in terms of Nusselt number, temperature distribution, pressure drop, and velocity contours, and are compared with a baseline model using segmental baffles. The study aims to assess the feasibility of flower baffles in practical applications and provide insights into the benefits of geometric modifications in enhancing heat exchanger performance.

Chen et al. [1] focused investigation into trefoil-baffle heat exchangers reveals insights into the presence and spatial distribution of dead zones in the shell side. Through residence time distribution analysis, the dead zone volume fraction is quantified at 28%, which is 13% higher than that found in segmental baffle heat exchangers. Velocity contour analysis indicates slower fluid flow near the shell wall and baffles, leading to dead zone development. An optimized tube layout is proposed to reduce the dead zone volume fraction by approximately 13% across various flow conditions, ultimately enhancing system performance. Wang et al. [2] literature review emphasizes improvements in heat transfer efficiency within concentric pipe heat exchangers, which are essential for fulfilling modern industrial demands. The review discusses the advantages of double and triple concentric pipe designs due to their structural simplicity and manufacturing ease. It thoroughly analyzes previous experimental and numerical work related to heat transfer enhancement and flow patterns, offering deep insights into the fundamental mechanisms governing these systems. This knowledge aids in optimizing design and implementation across diverse industrial sectors that utilize concentric pipe exchangers. Li et al. [3] review article outlines various industrial techniques for effective fluid heating

and cooling, especially in solar thermal systems. Methods such as corrugated absorber plates, extended fins, roughened surfaces, and jet impingement are evaluated. Although jet impingement is noted for its high convective heat transfer capability, the review points out a research gap regarding comprehensive assessments of different jet configurations. By systematically analyzing prior studies, the review identifies optimal geometric and operational parameters for roughened surfaces in solar air heaters, providing valuable data for researchers and engineers to minimize experimentation time and cost. Kumar et al. [4] separate experimental investigation examines heat transfer in half-cylindrical shell sections of power plant feedwater heaters and U-tube heat exchangers. The study compares multiple baffle designs—including ladder helical (single, dual inclined, and folded), orifice (fillet triangle, trefoil, and quatrefoil), and segmental baffles—across two pitch lengths. Ladder helical baffles demonstrate the highest shell-side heat transfer coefficients, while orifice baffles result in significantly lower pressure drops. This suggests that while ladder helical designs improve heat transfer, orifice baffles may be better suited where pressure loss is a critical concern. Su et al. [5] innovative design involving alternating V-row triangular tube layouts is introduced for twisted elliptical tube heat exchangers. This study tests eleven configurations, incorporating varying tube layouts, twisted pitch values, and aspect ratios. The setups include five coupling-vortex and five parallel-vortex schemes, along with a baseline smooth round tube system (R10.8). Simulations reveal that coupling-vortex configurations—especially those with larger aspect ratios and shorter twist pitches—generate greater turbulence and secondary flows, thereby boosting heat transfer. These designs show promise for improving thermal efficiency between adjacent tubes. Gu et al. [6] review collates various heat transfer enhancement methods for smooth air channels (SACs). It considers numerous studies that employed numerical and experimental approaches to improve SAC performance through obstacle configurations. These configurations differ by attachment method, orientation, shape, and spacing. The obstacles alter flow direction and local turbulence, which intensifies heat transfer from channel walls. The review provides a valuable summary of the geometrical and flow parameters that significantly affect thermal performance in SACs. Menni et al. [7] numerical simulation investigates the thermal and flow characteristics of turbulent fluid passing through a 2D horizontal rectangular duct. The duct has a top wall at constant temperature and an insulated bottom. To improve fluid mixing and heat transfer, two staggered transverse obstacles—flat rectangular and V-shaped—are added. The periodic arrangement promotes vortex formation, enhancing convective heat transfer in the system. Menni et al. [8] a related study, computational fluid dynamics is used to evaluate turbulent incompressible Newtonian fluid flow in a rectangular heat transfer channel. The channel maintains a constant temperature on the top surface and an adiabatic bottom. Two obstacles—V-shaped and flat rectangular—are placed on opposing surfaces in a staggered layout to generate vortex-induced mixing. The analysis shows that increasing the A/B ratio and reducing spacing (S) leads to stronger secondary flows, increasing the Nusselt number and friction factor. The comprehensive evaluation metric (Nu·f^−1/3) also improves. Compared to a smooth round tube reference (R10.8), all twisted elliptical tube schemes show higher heat transfer and friction. Coupling-vortex layouts yield more uniform temperature profiles than parallel-vortex schemes. For coupling-vortex schemes C12.3S50 and C12.3S100, the average Nu increases by 12.8% and 9.9%, the f increases by 15.2% and 10.5%, and the Nu·f^−1/3 rises by 7.6% and 6.3%, respectively. Lower S values and A/B ratios near unity enhance the thermal performance of twisted elliptical tube heat exchangers with coupling vortices (TETHXs-CV). Gu et al. [9], a detailed review of fin-and-tube heat exchangers evaluates their thermal performance across a broad spectrum of industrial applications, including air conditioning, refrigeration, automotive systems, and electronics. This work examines various enhancement strategies, drawing from both experimental and numerical studies. It focuses on how design geometries and operational conditions influence both heat transfer and pressure drop.

The review offers comparative assessments of different enhancement mechanisms and investigates novel compound geometries that can optimize the thermal efficiency of fin-and-tube configurations. Menni et al. [10] 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.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 07, July 2025, pp : 1679-1684
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

2. METHODOLOGY

ANSYS Workbench 2022 is used to design a shell and tube heat exchanger with and without a flower baffle. With 44 baffle plates placed in a helical spiral, create the geometry for flower baffles in Solid Works. The geometry of the shell and tube heat exchanger can then be finished by importing these baffle plates into ANSYS 2022. The heat exchanger's dimensions are displayed in Table 1 and its model may be seen in Fig. 1.

Table 1: Dimensions Of The Heat Exchanger

S. No	Parameter	Value
1	Length of test section	500 mm
2	Shell diameter	80 mm
3	Tube outer diameter	43 mm
4	Tube inner diameter	40 mm
5	Inlet and outlet diameter of cold and hot fluid	40 mm
6	Thickness of baffle plates	4.5 mm
7	Number of baffle plates	44

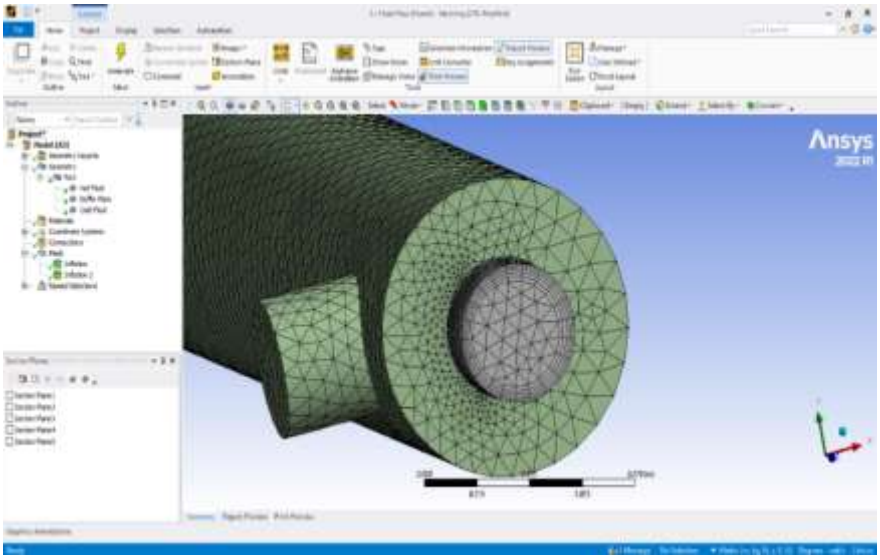

Fig 1: Model of the Heat Exchanger

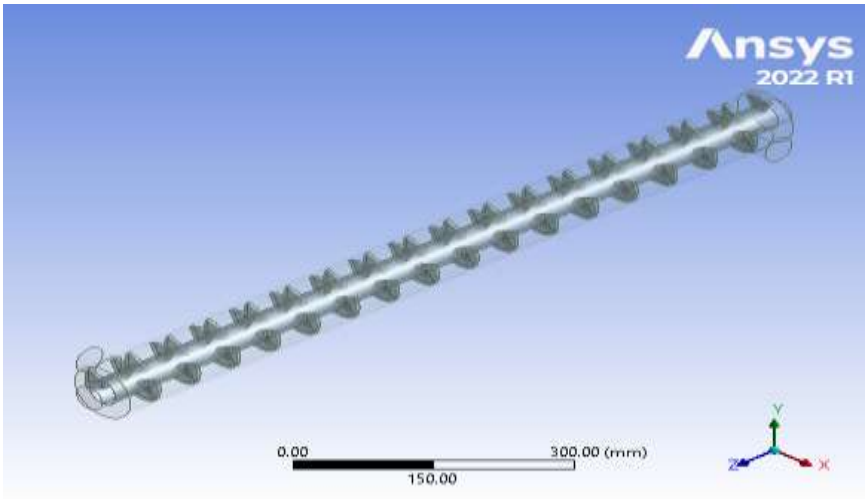

Fig 2: Inflation on the Tube Side

The computational fluid dynamics (CFD) analysis of shell and tube heat exchangers with a novel flower baffle design requires careful consideration of mesh generation and inflation techniques to accurately capture the intricate geometry

and complex flow patterns. Meshing plays a pivotal role in determining the reliability and efficiency of CFD simulations, as it directly impacts the solution quality and computational cost. In this study, we address the challenges involved in meshing the flower baffle geometry and the surrounding fluid domain. A hybrid approach combining structured and unstructured meshing methods is adopted to adequately resolve the intricate features of the flower baffle. The mesh resolution is optimized to strike a balance between accuracy and computational feasibility. Furthermore, proper inflation is implemented to accurately represent the boundary layer near the solid surfaces. The generation of thin prism layers of mesh cells ensures precise resolution of the velocity and thermal gradients, essential for capturing the flow behaviour close to the surface. The meshing and inflation techniques are validated against experimental data and numerical convergence studies. The resulting mesh quality is assessed to ensure its suitability for reliable CFD predictions. By employing these meshing and inflation strategies, the CFD simulations provide an in-depth understanding of the heat transfer characteristics and convective heat transfer coefficients within the shell and tube heat exchanger with a flower baffle. The findings offer valuable insights into the heat exchanger's performance and the potential enhancement brought about by the flower baffle design, paving the way for improved heat exchanger designs in

Thermal properties used used for the nanofluid

At volume fraction of 0.6

Nano fluid density (ρnf):

ρnf = 2727.28 kg/m3

Nano fluid specific heat (Cp nf):

Cp nf = 1389.02 J/kg-K

Nano fluid viscosity (μnf):

μnf = 0.0025075 kg/m-s

3. RESULTS AND DISCUSSION

3.1 Heat Flux in Shell and Tube Side

In the below mentioned fig. 3, the heat flux value is higher for without flower baffle in comparison with the other case, due to the higher thermal conductivity of nanoparticles present in the nanofluids. They acquire more heat from the hot fluid and increase the wall temperature as well as the heat flux due to the temperature difference. For the heat flux value in the case of without flower baffle has higher heat flux in comparison with the other case but for similar mass flow rate on both side (shell and tube side) with flower baffle has slightly high heat flux value. In the case of baffle plates with similar mass flow rate the heat flux is more for hot fluid. As the difference in mass flow rate increases, there is an increase in heat flux.

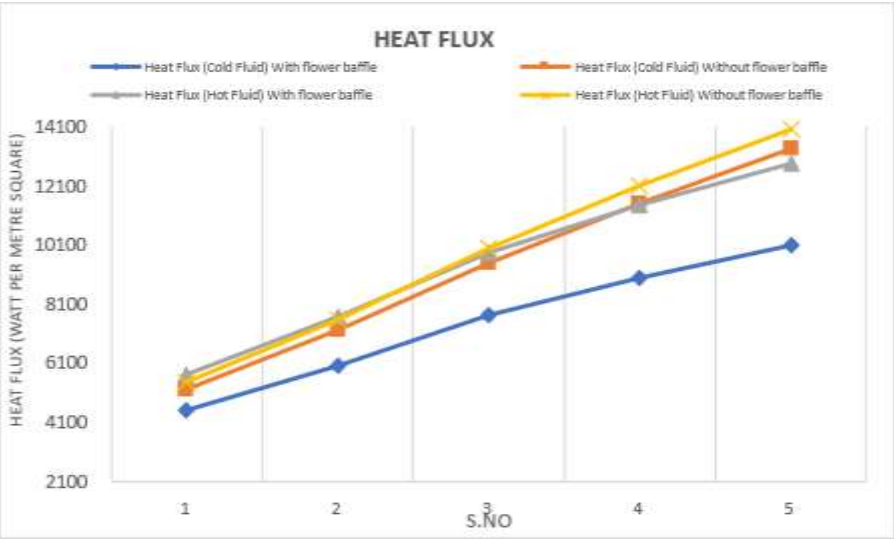

Fig 3: Graph of heat flux in shell and tube heat exchanger

3.2 Convective Heat Transfer Coefficient in Cold Fluid

As shown in the below fig. 4, the convective heat transfer coefficient for cold fluid is higher for STHX with flower baffle and nanofluid in comparison with STHX with nanofluid only. The flower baffle creates turbulence in cold fluid causing to increase the convective heat transfer coefficient of nanofluid.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 05, Issue 07, July 2025, pp : 1679-1684
e-ISSN : 2583-1062
Impact Factor : 7.001
www.ijprems.com
editor@ijprems.com

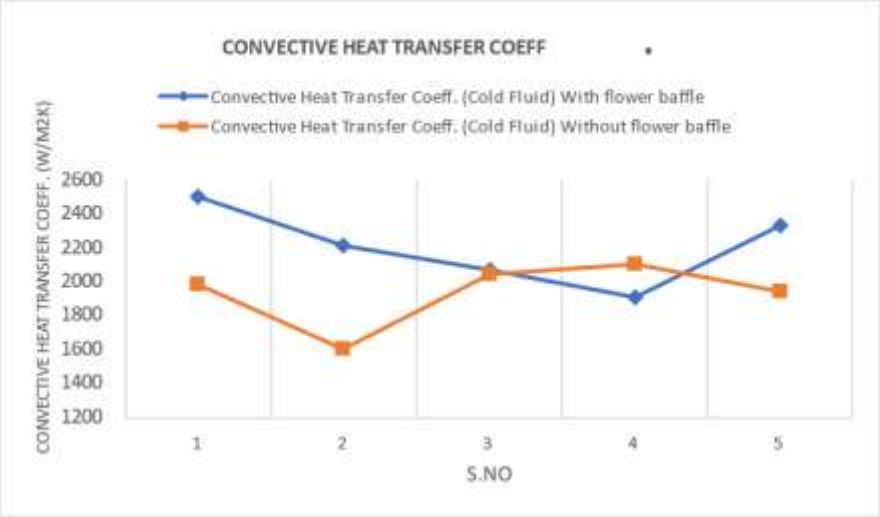

Fig 4: Graph of convective heat transfer coefficient in cold fluid

3.3 Convective Heat Transfer Coefficient in Hot Fluid

Whereas, for hot fluid the convective heat transfer coefficient of hot fluid(water) for STHX with nanofluid is high in comparison with other case as shown in fig. 5.

Fig 5: Graph of convective heat transfer coefficient in hot fluid

4. CONCLUSION

Compared to the individual effects, the combination of flower-shaped baffles and SiO_2 nanofluid significantly enhances the convective heat transfer coefficient on the cold fluid side. The nanofluid contributes to a higher heat flux, particularly when there is a greater difference between the mass flow rates of the hot and cold fluids. Notably, employing nanofluid at flow rates of 2 LPM (cold side) and 10 LPM (hot side) results in effective thermal performance with a cold-side pressure drop of 4.75 Pa. Similarly, using nanofluid at 2 LPM (cold side) and 8 LPM (hot side) leads to satisfactory outcomes, with a tube-side pressure drop of 2.48 Pa. While pressure drop on the hot side increases with higher mass flow rates, the integration of SiO_2 nanofluid and flower baffles helps maintain a lower pressure drop on the cold side under constant flow conditions. Overall, the use of SiO_2 nanofluid on the low-pressure drop tube side proves advantageous for achieving higher heat flux.

5. REFERENCES

[1] 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.

[2] 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.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] 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.

[8] 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.

[9] 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.

[10] 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.

[11] 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.