iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL
FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 13 Issue: IV Month of publication: April 2025

DOI: https://doi.org/10.22214/ijraset.2025.68398

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

A CFD Analysis of Shell and Tube Heat Exchanger with Segmental Baffles

Musaddiq Mohammad[1], K. V. Sharma[2], M.T. Naik[3]
[1] Student, [2, 3]Professor, Center for Energy Studies, NTUH College of Engineering, Hyderabad, Telangana, India

Abstract: A single-pass shell-and-tube heat exchanger with segmental baffles has been analyzed numerically. The study focuses on the crucial aspects of heat transfer and flow patterns, which are numerically investigated by varying the number of baffles. The standard k-ε model is employed to solve the problem, and numerical simulations are conducted for three cases of baffle spacing with a square bundle of tubes. Using a commercial computational fluid dynamics (CFD) solver, the study obtains numerical solutions by solving the three-dimensional continuity, momentum, energy, and turbulence (k-ε) equations. The number of baffles is variable, while other parameters such as velocity, temperature, pressure, and baffle cut are kept constant. The results obtained from the computational analysis are analyzed to understand the effect of several baffles on the heat transfer rate and pressure drop on the shell side.
Keywords: Shell and tube heat exchanger; segmental baffle; k-e model

I. INTRODUCTION

In industries, heat exchangers are essential for cooling, heating, condensation, boiling, and evaporating fluids. Their applications are critical and fundamental to nearly all industrial procedures involving temperature control and fluid phase changes. This study emphasises the practical significance of heat exchangers across a broad range of sectors. Typically, fluids are heated or cooled before undergoing a phase change. The performance and efficiency of heat exchangers are generally assessed using parameters such as heat transfer and pressure drop. The heat transfer coefficient and pressure drop in the heat exchanger design are essential.

In the Literature, You et al. [3] reported the performance of shell-and-tube heat exchangers by implementing trefoil hole baffles to convert the design into a crossflow type. This modification increased the heat transfer area, thus improving heat transfer efficiency. Their research included the practical validation of numerical results using experimental data, employing stainless steel tubes with an internal diameter of 12.8 mm and staggered low-carbon steel baffles. The study concluded that shell-side pressure loss increases with Reynolds numbers. The heat transfer coefficient initially decreased with an increase in the shell-side Reynolds number and then increased after reaching a certain threshold. These findings have a direct impact on the design and operation of heat exchangers in various industrial applications.

Jadhav et al. [8] conducted a numerical study examining the relationship between the heat transfer coefficient and pressure drop in relation to baffle spacing and cut. They compared their results with those obtained using the Bell-Delaware method through simulations of varying flow rates and two different baffle cuts.

Patil et al. [9] analysed the numerical effect of baffle cut on heat transfer coefficient and pressure drop, maintaining a constant baffle spacing. They found that a 30% baffle cut resulted in a lower pressure drop, while the heat transfer coefficients were similar for both 30% and 25% baffle cuts.

Chit et al. [10] investigated the effect of baffle spacing on heat transfer and pressure drop. They proposed that as baffle spacing decreases, the pressure drop increases more rapidly than the heat transfer coefficient. Their findings indicated that the optimal baffle spacing should be between 0.4 and 0.6 times the diameter of the shell.

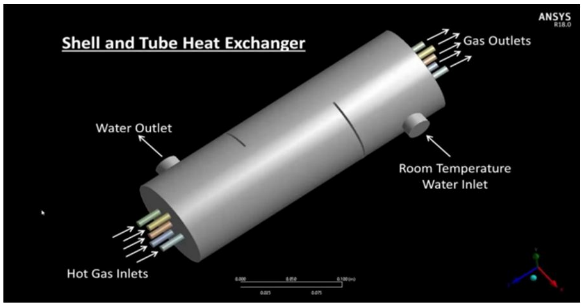
Li et al. [13] utilised mass transfer measurements to derive the local heat transfer coefficient, drawing an analogy between heat and mass transfer to transform the heat transfer coefficient effectively.

Abdur Rahim et al. investigated the impact of baffle inclination on shell-side fluid flow and heat transfer while maintaining a constant baffle cut. Their results revealed a 4% reduction in pressure drop with a 10° baffle inclination and a 6% reduction with a 20° inclination.

Kiran et al. [14] simulated a shell-and-tube heat exchanger with baffles to assess the influence of baffle spacing on heat transfer, pressure drop, and outlet temperature on the shell side. They noted that while changes in baffle spacing did not significantly affect outlet temperature, variations in mass flow rate had considerable impacts. The study also found that both pressure drop and baffle spacing corresponded appropriately with changes in mass flow rate.

II. THE STUDY'S METHOD

The ical framework utilizes a shell and tube heat exchanger (STHE) model with segmental baffles to evaluate thermal performance through computational fluid dynamics (CFD).



Shell and tube heat exchanger

The working fluid employed in the simulations is liquid Helium, selected for its superior thermophysical properties, particularly in cryogenic applications. The heat exchanger is designed with multiple baffle configurations (3, 4, and 5 baffles) to investigate the impact of baffle count on heat transfer efficiency and pressure drop.

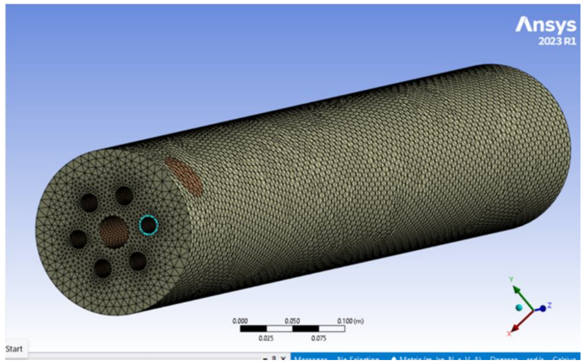
The primary materials used for the heat exchanger components are:
Shell: Stainless steel (SS 304) for enhanced durability and corrosion resistance.
Tubes: Copper due to its excellent thermal conductivity.
Baffles: Aluminum, offering a balance of strength and lightweight properties.

Baffles

The thermophysical properties of liquid Helium were sourced from standard reference databases to ensure accuracy in the CFD simulations.

III. METHODOLOGY

The study utilizes ANSYS Fluent for computational fluid dynamics (CFD) simulations of the STHE. The methodology involves detailed steps: Geometry Creation, Meshing, Boundary Conditions, and Physical Setup. Each step is carefully explained to give the reader a clear understanding of the study's process.
Geometry Creation: The shell and tube heat exchanger geometry is designed using ANSYS Design Modeler. The model features a cylindrical shell, multiple parallel tubes, and segmental baffles positioned at optimally spaced intervals to direct the flow. The dimensions are carefully chosen to replicate realistic industrial heat exchanger setups.
Meshing: A structured mesh is generated using ANSYS Meshing to ensure accurate numerical solutions. The mesh refinement process includes:



Meshing

A finer mesh should be applied to the baffles and tube walls where significant thermal gradients are expected.
A mesh independence study will be performed to verify that further refinement does not significantly alter the results.
Utilizing inflation layers near the walls to resolve boundary layer effects efficiently.

S.No.	Materials	Density (kg/m^3)	Specific heat (J/kg k)	Thermal conductivity (W/m k)
1	Air	1.225	1006.43	0.0242
2	Aluminum	2700	900	235
3	Titanium	4850	544.25	7.44

Properties of fluids and materials

1	Nodes	61874
2	Elements	156107

Table: Mesh Details

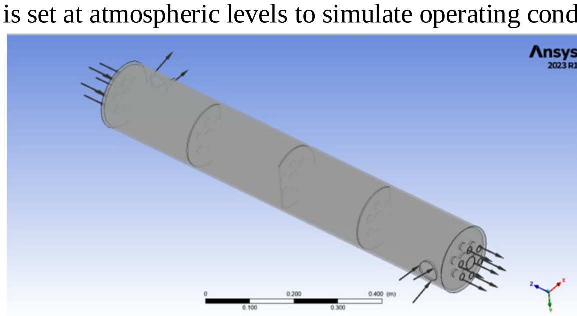
A. Boundary Conditions

Inlet Conditions: The inlet temperature and velocity of liquid Helium are assigned based on experimental data and cryogenic standards.

S.No.	Parameters	Materials
1	Hot fluid	Helium
2	Cold fluid	Helium

Table: Boundary condition details 1

Outlet Conditions: The outlet pressure is set at atmospheric levels to simulate operating conditions.



Physical setup

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

B. Wall Conditions

No-slip conditions are applied at solid-fluid interfaces.

Heat flux or constant temperature boundary conditions are assigned to analyze heat transfer behavior.

Turbulence Modeling: The k-ε turbulence model is chosen because it is robust in predicting recirculation and turbulence effects in industrial flows.

C. Solver Setup

Discretization Scheme: Second-order upwind schemes enhance the accuracy of momentum, energy, and turbulence equations.

Pressure-Velocity Coupling: The SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm ensures stable convergence.

Residual Criteria: The solution is iterated until the residual values for the continuity, momentum, and energy equations drop below 10 ⁵, ensuring numerical accuracy.

D. Simulation and Analysis

The CFD simulations are performed for different baffle configurations (3, 4, and 5 baffles) to compare heat transfer rates and pressure drops.

E. Post-processing in ANSYS Fluent

Temperature contours and velocity profiles are analyzed to identify areas of high and low temperatures.

Streamline visualization is used to assess the distribution of fluid flow and the effects of turbulence.

A pressure drop across the exchanger is evaluated to ensure minimal energy losses.

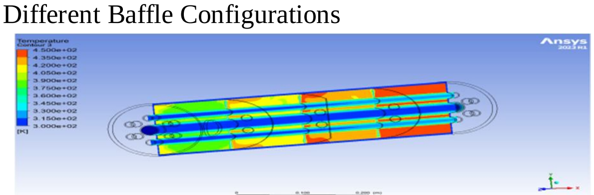
Validation: The numerical results are validated against theoretical correlations and available experimental data.

IV. RESULTS & DISCUSSIONS

The numerical analysis of the shell-and-tube heat exchanger revealed significant variations in temperature distribution for different baffle configurations. The inlet temperature of liquid Helium remained relatively stable across all simulations, while the outlet temperature exhibited a decreasing trend with an increasing number of baffles. The observed outlet temperatures for the 3, 4, and 5 baffles were 80 K, 78 K, and 75 K, respectively. This reduction in outlet temperature indicates improved heat transfer performance with higher baffle counts.

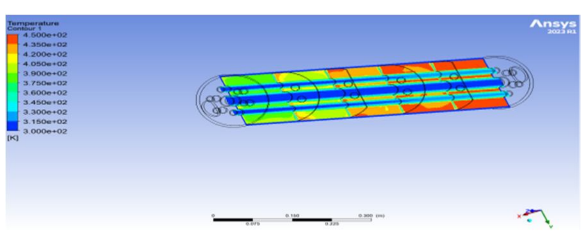
A. Case 1

Figure 1: Temperature Variation with Different Baffle Configurations



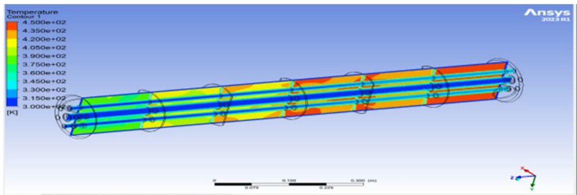
1) Temperature Contour with 3 baffles

In this heat exchanger, the hot fluid in the shell cools from 450 K to 350 K, while the cold fluid in the tubes warms up from 300 K to 357 K. Since there is little flow obstruction, the hot zones stay larger near the inlet, but the air in the tubes does not heat up much because it moves through quickly.



2) Temperature Contour with 4 baffles

The shell-side hot fluid cools from left to right, reaching ~330 K. In contrast, the tube-side cold fluid heats up to ~362 K. Improved flow redirection enhances water-air interaction, ensuring uniform heat transfer with minimal stagnation and moderate pressure drop.



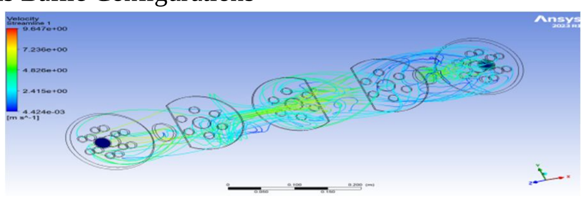
3) Temperature Contour with 5 baffles

The shell-side fluid cools to ~330 K - 350 K, while the tube-side fluid heats to ~350 K - 375 K, indicating efficient heat exchange. Baffles enhance turbulence and mixing, ensuring a uniform temperature distribution, a longer flow path for improved heat transfer, and minimizing hot spots.

4) Velocity Profile Analysis

Velocity contour analysis showed enhanced fluid mixing with an increased number of baffles. The turbulence created by the baffles resulted in improved heat transfer rates but also introduced a higher pressure drop. The velocity distribution demonstrated that regions near the tube walls experienced enhanced convective effects, improving thermal performance.

Figure 2: Velocity Contours for Various Baffle Configurations



5) Velocity contour with 3 baffles

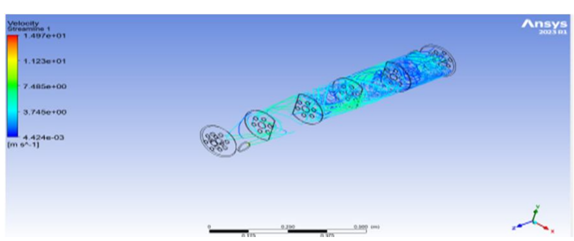
The baffles cause the shell-side fluid to travel zigzag, thus augmenting turbulence and heat transfer. There are recirculation areas where the flow slows and forms eddies that enhance mixing.

Inlet and Outlet Velocities:

Inlet Velocity: 4.42

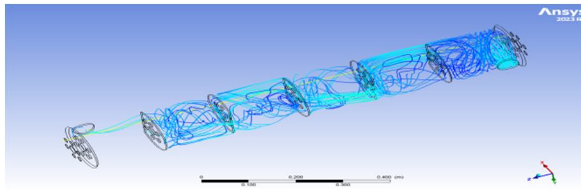
Outlet Velocity: The section on the right-hand side, close to the exit, has a moderate velocity of approximately 4.5-5 m/s (green to yellow zone).

6) Velocity contour with 4 baffles



Baffles create a zigzag flow, increasing turbulence and heat transfer while forming recirculation zones that enhance mixing. The inlet velocity is 4.42 m/s, and the outlet velocity ranges from 4.5 to 5 m/s, indicating moderate flow acceleration.

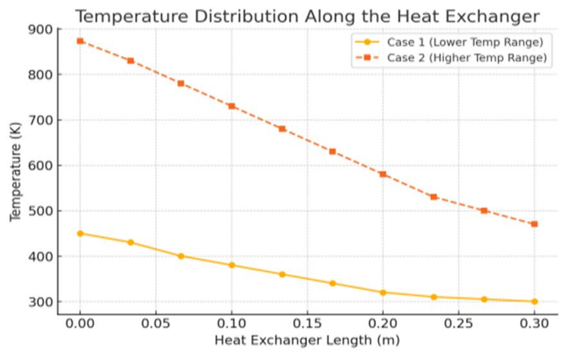
7) Velocity contour with 5 baffles



The fluid enters from the left at a high to moderate velocity and moves through multiple baffles before exiting on the right. While the inlet shows a yellow green (higher) velocity, the outlet velocity decreases (blue) due to resistance from the baffles.

8) Thermal Performance Comparison

Heat transfer effectiveness improved with the addition of baffles, due to increased flow redirection and turbulence. However, beyond a certain threshold, the added resistance led to diminishing returns in heat exchange efficiency. The comparative study indicates that an optimal balance between heat transfer enhancement and pressure drop limitations must be maintained.

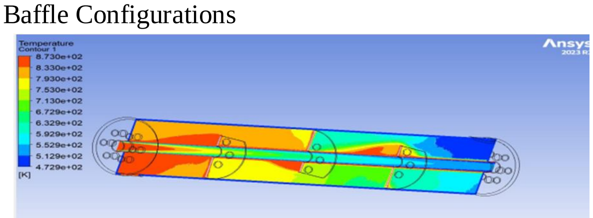


Graph representing the Temperature distribution among the heat exchangers

B. Case - 2

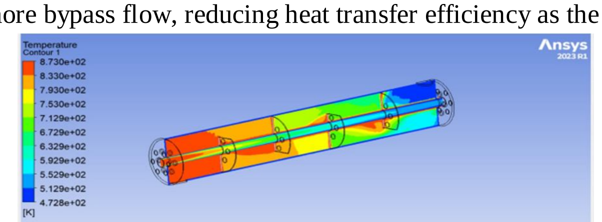
Fluid - HELIUM

Temperature Variation with Different Baffle Configurations



Temperature contour with 3 baffles

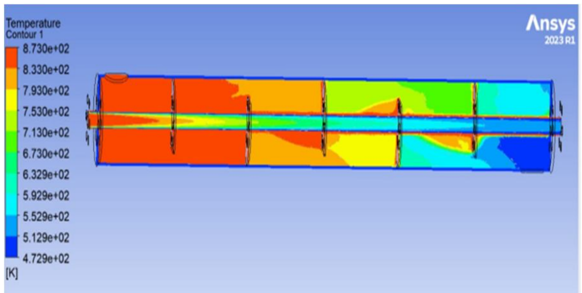
The inlet shows a hot region (red), while the outlet cools to a temperature range of approximately 472.9 K to 530 K (green/blue). More significant baffles gaps allow more bypass flow, reducing heat transfer efficiency as the shell-side flow is not fully utilised.



Temperature contour with 4 baffles

The inlet has a high-temperature zone in the middle tube, while the outlet cools to a temperature range of approximately 472.8 K to 530 K (blue green). A structured flow path enhances mixing and heat transfer, leading to a more uniform temperature distribution with minimal bypass regions.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

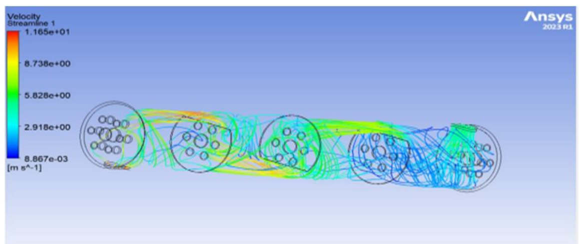


Temperature contour with 5 baffles

The center tube's high-temperature zone indicates the entry of hot fluid, while the rightmost blue-green region shows cooling, with an outlet temperature ranging from ~472.9 K to 530 K. The extended flow path maximizes heat exchange, ensuring a uniform temperature gradient and the highest heat transfer efficiency.

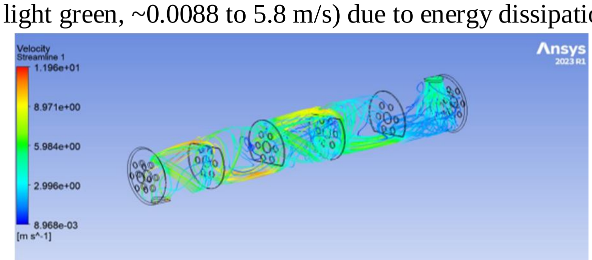
1) Velocity contour variations

The velocity streamline contours obtained from ANSYS simulations for different baffle configurations in a shell and tube heat exchanger.



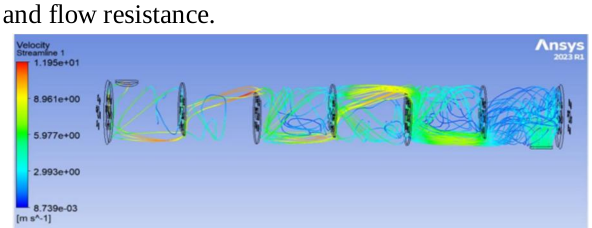
2) Velocity contour with 3 baffles

The velocity streamlines show flow entering from the left, moving through baffles, and exiting on the right. Initially smooth, the flow slows down at the outlet (blue to light green, ~0.0088 to 5.8 m/s) due to energy dissipation and baffle interactions**.**



3) Velocity contour with 4 baffles

Some regions show velocity peaks near 11.96 m/s (red zones), while the outlet velocity decreases to around 0.009 to 5.98 m/s (green to blue), indicating energy dissipation and flow resistance.

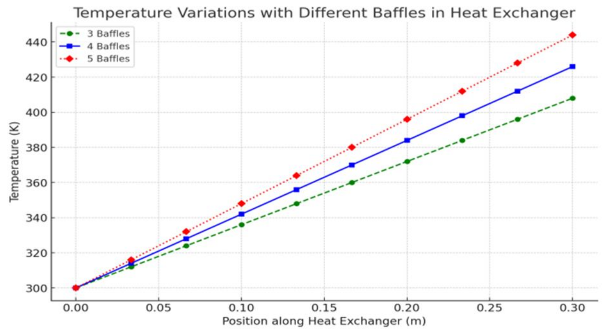


4) Velocity contour with 5 baffles

Some regions show velocity peaks near 11.95 m/s (red zones), while the outlet velocity decreases to around 0.0087 to 5.97 m/s (green to blue) due to baffle-induced flow resistance and energy dissipation.

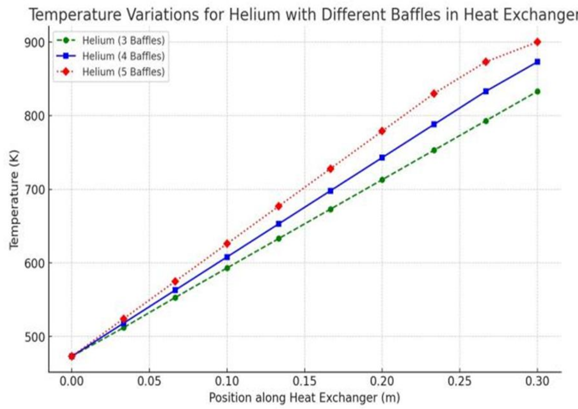
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

Graphs



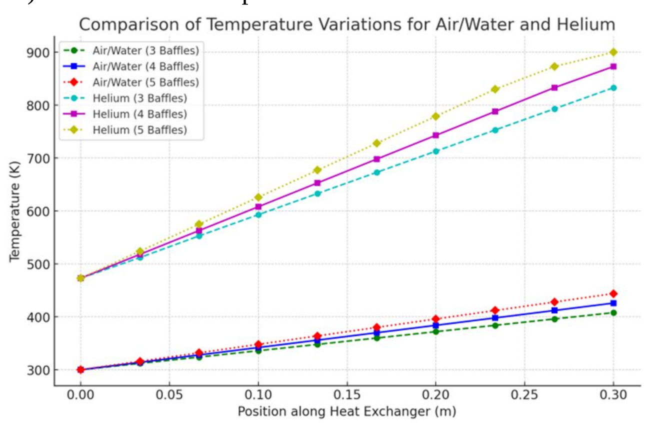
Graph showing temperature variations for different numbers of baffles (3, 4, and 5)

The chart illustrates the decline in the exchanger's temperature. In Case 1 (yellow line), the temperature drops from 450 K to 300 K, while in Case 2 (orange dashed line), it starts at 873 K and decreases to 470 K at the exit.



Graph showing temperature variations of Liquid Helium for different numbers of baffles (3, 4, and 5)

More baffles raise the heat exchanger's temperature. The red line (5 baffles) shows the highest increase, the blue line (4 baffles) is moderate, and the green line (3 baffles) has the lowest temperature rise.



Comparison of temperature variations for Air/Water and Helium

The chart compares temperature variations along a heat exchanger for air/water and Helium with different numbers of baffles (3, 4, and 5). Key observations: More Baffles Lead to Higher Temperature Gains: For both air/water and Helium, the 5-baffle case (red for air/water, yellow for Helium) shows the highest temperature increase along the exchanger. The 3-baffle case (green for air/water, cyan for Helium) has the lowest temperature increase. Helium Experiences a Greater Temperature Rise than Air/Water: Helium has a significantly higher temperature at every position than air/water.

It suggests that Helium, having better thermal conductivity, absorbs and transfers heat more efficiently.

V. CONCLUSION

The research has successfully evaluated the transient thermal behavior of a heat exchanger-type shell-and-tube (STHE) with segmental baffles using CFD simulations in ANSYS.

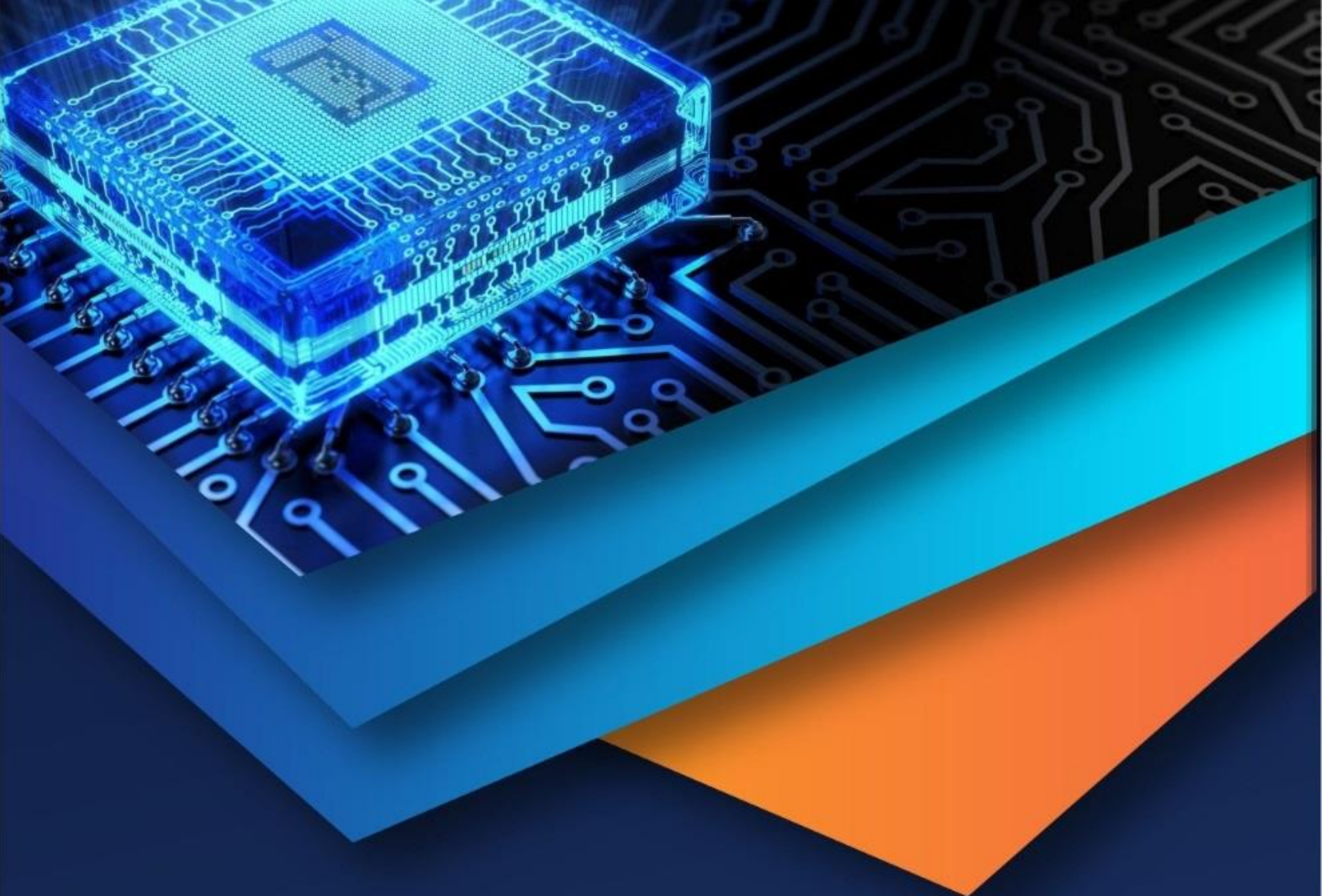
The simulations demonstrated that the heat exchanger's performance was heavily dependent on the number and positioning of baffles; specifically, one arrangement was recommended that achieved a tradeoff between enhancing thermal performance and improving flow distribution.

Helium exhibits a significantly higher temperature increase for all baffle arrangements compared to air/water. The growth is much more pronounced for Helium, while for air and water, it is moderate. Helium reaches 900K with 5 baffles, while air and water remain below 350K, proving that Helium is a better heat transfer fluid.

Validation: The simulation results were validated against experimental and theoretical data with a reasonable degree of accuracy for the measured temperature profiles and heat transfer rates. The simulated numerical results were consistent with the correlations confirmed in the literature, providing accuracy to the model and reliable simulation data for the experimental component.

REFERENCES

[1] G. V. S. Rao, C. J. Rao, and H. N. HariBabu, "Heat transfer analysis on shell and tube heat exchanger," International Journal of Research in Aeronautical and Mechanical Engineering, vol. 2, pp. 11–26, January 2014.
[2] Y. You, Y. Chen, M. Xie, X. Luo, L. Jiao, and S. Huang, "Numerical simulation and performance improvement for a small-size shell-and-tube heat exchanger with trefoil-hole baffles," Applied Thermal Engineering, vol. 89, pp. 220–228, 2015
[3] G.-Y. Zhou, J. Xiao, L. Zhu, J. Wang, and S.-T. Tu, "A numerical study on the shell-side turbulent heat transfer enhancement of shell-and-tube heat exchanger with trefoil-hole baffles," In The 7th International Conference on Applied Energy, Energy Procedia, pp. 3174–3179, 2015
[4] J. Yang and W. Liu, "Numerical investigation on a novel shell-and-tube heat exchanger with plate baffles and experimental validation," Energy Conversion and Management, vol. 101, pp. 689–696, 2015.
[5] J. Wen, H. Yang, S. Wang, Y. Xue, and X. Tong, "Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffles," International Journal of Heat and Mass Transfer, vol. 84, pp. 990–997, 2015.
[6] E. Ozden and I. Tari, “Shell side CFD analysis of a small shell-and-tube heat ex- changer”, Energy Conversion and Management, vol. 51, pp. 1004–1014, 2010.
[7] C. N. Patil and N. S. Bhalkikar, "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop and Heat Transfer Coefficient," International Journal for Scientific Research and Development, vol. 2, pp. 649–654, 2014.
[8] S. P. Chit, N. A. San, and M. M. Soe, "Flow Analysis in Shell Side on the Effect of Baffle Spacing of Shell and Tube Heat Exchanger," International Journal of Science, Technology, and Society, vol. 3(5), pp. 254–259, 2015.
[9] B. Peng, Q. W. Wang, C. Zhang, G. N. Xie, L. Q. Luo, Q. Y. Chen, and M. Zeng, "An experimental study of Sheel-and-Tube Heat Exchangers with continuous helical baffles," Journal of Heat Transfer, vol. 129, pp. 1425–1431, 2007
[10] E. S. Gaddis and V. Gnielinski, "Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles," Chemical Engineering and Processing, vol. 36, pp. 149–159, 1997
[11] H. D. Li and V. Kottke, "Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for in-line tube arrangement by mass transfer measurements," Heat and Mass Transfer, vol. 33, pp. 371–376, 1998.
[12] K. Kiran, A. M. M. Manoj, and C. J. Umesh, "Investigation of baffle spacing effect on shell-side heat transfer characteristics in shell-and-tube heat exchangers using computational fluid dynamics," Thermal Engineering, vol. 73, pp. 26022–26026, 2014
[13] J. Prasanna, H. R. Purushothama, and D. K. Murugeshan, "A numerical analysis of hydrodynamic and heat transfer effects of shell and tube heat exchanger for different baffle space and cut," Mechanica Confab, vol. 2, June-July 2013.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY