

Multi-Objective Thermal Performance Enhancement of a Shell-and-Tube Heat Exchanger Using CFD-Based Optimization of Baffle Geometry, Working Fluid, and Material

Abdul Sannan Saboor¹, Prof. Yohan Revanoor²

¹Student, Dept. Of Mechanical Engineering (Thermal Power Engineering), PDA college of engineering, Kalaburagi, India.

²Professor, Dept. Of Mechanical Engineering (Thermal Power Engineering), PDA college of engineering, Kalaburagi, India.

ABSTRACT

This study presents a detailed Computational Fluid Dynamics (CFD) analysis of a shell-and-tube heat exchanger with multiple baffle configurations and working fluids to enhance thermal performance. A total of six cases were investigated under identical inlet conditions, with a hot fluid entering the tube side at 353 K and 2.75 m/s, and a cold fluid entering the shell side at 300 K and 2.75 m/s. The baseline design consisted of a conventional 0°–180° and 180°–360° baffle arrangement with offsets of 90 mm and 40 mm from the center plane. Subsequent iterations focused on optimizing baffle orientation and positioning to improve heat transfer effectiveness. The second and third designs introduced modified angular distributions and offset variations, resulting in enhanced mixing and flow redistribution. Additionally, the influence of different working fluids, including water, air, and ethylene glycol, was analyzed, along with a material change from aluminum to copper in the final iteration. Results indicate that the optimized baffle configurations significantly affect thermal performance and fluid flow characteristics. The second optimized design with water achieved improved tube-side outlet temperature (315.72 K), indicating better heat transfer compared to the baseline. The use of air resulted in lower pressure drops but slightly reduced heat transfer efficiency due to its lower thermal properties. The ethylene glycol case combined with copper tubing showed improved heat retention and thermal conductivity effects. Overall, baffle arrangement, fluid selection, and material properties play a crucial role in enhancing the efficiency of shell-and-tube heat exchangers.

Keywords: CFD Analysis, Shell-and-Tube Heat Exchanger, Baffle Optimization, Heat Transfer Enhancement, Thermal Performance, Ethylene Glycol, Copper Tubes, Fluid Flow Analysis, ANSYS Fluent, Pressure Drop

1. INTRODUCTION

Introduction

The heat exchanger is one of the indispensable devices in modern engineering that transfers heat between fluids without direct mixing. One of the most widely used types of the heat exchanger is the shell and tube version because of its high reliability, easy maintenance, and high performance efficiency. The efficiency of a heat exchanger depends on several factors, such as the characteristics of the fluids, their velocity, materials used, and geometry. In particular, baffles configuration, working media, and material used for tubes play a decisive role in heat exchange process.

Today, increasing energy efficiency becomes crucial because of ever-growing demand for thermal energy and growing environmental requirements. Optimization of heat exchange process is especially critical to achieve this goal. The present paper aims to optimize a shell and tube heat exchanger by using Computational Fluid Dynamics techniques in order to increase its performance by varying baffles configuration, working media, and materials.

Rising energy consumption requires increasing efficiency of thermal engineering devices. Shell and tube heat exchangers are highly efficient tools in terms of energy exchange because they provide efficient cooling and heating at high thermal energy efficiency level. They are characterized by specific internal flow

pattern and thermodynamic resistances depending on certain design parameters, such as baffles configuration and material properties.

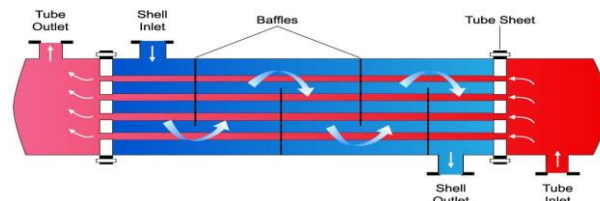


Figure1. Representation of shell and tube heat exchanger with baffle arrangement

In the conventional approach to the problem, correlations of an empirical nature have been used, sometimes missing out on the complex interplay of flows on the shell side. Currently, owing to the advent of CFD modeling, more intricate studies are possible. The current research seeks to achieve improved thermal and hydraulic performance of the shell-and-tube heat exchanger using a multi-objective optimization approach that includes baffle optimization, selection of different fluids, and tube material.

1.1 Overview

This work presents a detailed CFD analysis of a shell-and-tube heat exchanger under different design configurations and operating conditions. A total of six cases were evaluated by varying baffle arrangements, working fluids (water, air, and ethylene glycol), and tube materials (aluminum and copper). The objective is to analyze how these

parameters affect thermal performance, fluid flow behavior, and pressure drop.

The baseline model consists of a conventional baffle arrangement, which is then optimized through iterative modifications in angular positioning and offset distances. The study further extends to analyzing the impact of different fluids with varying thermophysical properties and enhancing thermal conduction by replacing aluminum tubes with copper. The results provide a comparative understanding of performance improvement and help identify the most efficient configuration.

2. LITERATURE SURVEY

2.1. Aydin et al., 2020

Title: Optimization and CFD Analysis of a Shell-and- Tube Heat Exchanger with Multi-Segmental Baffle

This study presents a detailed CFD-based optimization of a shell-and-tube heat exchanger using a multi-segmental baffle design. The authors focused on improving thermal efficiency while minimizing pressure drop, which is a key challenge in industrial heat exchangers. The simulation results showed that modified baffle geometry significantly enhances turbulence and fluid mixing, leading to improved heat transfer performance compared to conventional segmental baffles. The study concludes that optimized baffle arrangements can achieve better thermo-hydraulic performance without increasing system complexity. The research highlights the importance of geometric modification as a primary parameter in heat exchanger enhancement.

- **Material:** Steel shell, copper tubes
- **Type:** Shell-and-tube heat exchanger
- **Boundary Conditions:** Counterflow, constant inlet temperature and velocity, steady-state CFD

[Download Paper](#)

2.2. Sahoo et al., 2020

Title: Design and Analysis of Heat Exchanger for Determined Heat Transfer Rate

This paper focuses on a multi-model optimization approach for designing a heat exchanger to achieve a specific heat transfer rate. The study combines analytical methods with CFD validation to improve design accuracy. Results indicate that proper selection of flow parameters and geometry significantly enhances performance. The authors conclude that integrating CFD with traditional design methods improves prediction accuracy and reduces design errors. The work emphasizes optimization of flow parameters and thermal characteristics for efficient design.

- **Material:** Aluminum and steel
- **Type:** Shell-and-tube heat exchanger
- **Boundary Conditions:** Steady-state, fixed inlet temperature and flow rate

[Download Paper](#)

2.3. Shiv Kumar et al., 2020

Title: Review on Design and Optimization of Shell and Tube Heat Exchanger

This review paper summarizes various methods used for enhancing heat transfer in shell-and-tube heat exchangers. The study discusses parameters such as baffle spacing, tube diameter, and fluid selection. It concludes that performance improvement can be achieved through combined optimization rather than focusing on a single parameter. The authors highlight that energy savings and cost reduction are major benefits of optimized heat exchanger design.

- **Material:** General (steel, copper, alloys)
- **Type:** Shell-and-tube heat exchanger
- **Boundary Conditions:** Review-based (various studies) [Download Paper](#)

2.4. Shukla et al., 2022

Title: Investigation on Shell and Tube Heat Exchanger by Using CFD

This study uses CFD simulation to analyze the effect of baffle number and material on heat exchanger performance. The results show that increasing the number of baffles improves heat transfer due to enhanced turbulence. Additionally, copper material significantly outperforms aluminum due to higher thermal conductivity. The study concludes that optimal baffle design combined with high-conductivity materials yields maximum performance.

- **Material:** Copper and aluminum
- **Type:** Shell-and-tube heat exchanger
- **Boundary Conditions:** Counterflow, varying mass flow rates, steady-state [Download Paper](#)

2.5. Abdelhamid et al., 2022

Title: Comparative Performance of Shell-and-Tube Heat Exchangers

This paper reviews different configurations of shell-and-tube heat exchangers and compares their performance. The study highlights that shell-and-tube exchangers are widely used due to their flexibility and robustness. It concludes that design modifications such as baffle configuration and tube arrangement significantly affect efficiency. The paper also emphasizes the importance of CFD in modern design processes.

- **Material:** Various industrial materials
- **Type:** Shell-and-tube, plate, and double pipe
- **Boundary Conditions:** Comparative study [Download Paper](#)

2.6 Objectives

- To design a shell-and-tube heat exchanger model for analysis
- To perform CFD simulation under steady-state conditions
- To analyze the effect of different baffle arrangements on heat transfer
- To compare performance using different working fluids (water, air, ethylene glycol)
- To study the influence of tube materials (aluminum and copper)
- To evaluate thermal performance based on outlet temperatures
- To analyze pressure drop and flow characteristics
- To identify the best configuration for maximum efficiency
- To achieve improved heat transfer with minimum energy loss

2.7 Important Observations for CFD

From the analysis of material and fluid properties, it is observed that the choice of working fluid and solid material has a significant impact on the thermal and hydraulic performance of the heat exchanger. Water is the best working fluid due to its high specific heat and good thermal conductivity. The combination enables it to absorb and conduct great heat loads. On the other hand, air has the lowest pressure drop due to its low density and low viscosity; however, poor thermal characteristics limit its ability to transfer heat. Ethylene glycol is better than water in terms of heat absorption; however, it has very high viscosity, making its pressure drop considerable.

When it comes to solids, copper is one of the materials with the highest thermal conductivity that enhances

heat transfer between the tube and the fluid. Aluminium is an inexpensive and lighter alternative that is not inferior to copper in terms of thermal conductivity. Consequently, material choice depends on a number of criteria such as heat transfer capability and pressure drop.

2.8 Assumptions Used in CFD

For the purpose of simplifying the numerical analysis and reducing computational complexity, several assumptions are made in this study. The thermophysical properties of both fluids and solid materials are considered constant and independent of temperature, which is a common approach for moderate temperature ranges. The flow is assumed to be steady-state, meaning that all flow and thermal parameters do not change with time. Additionally, the fluid is treated as incompressible, as density variations are negligible under the given operating conditions.

No phase change is considered during the heat transfer process, ensuring that both fluids remain in a single phase throughout the simulation. The flow regime is assumed to be turbulent, and an appropriate turbulence model, specifically the standard $k-\epsilon$ model, is used to capture the effects of turbulence on heat transfer and fluid flow behavior. These assumptions help in achieving a balance between computational efficiency and acceptable accuracy.

2.9 Ready-to-Use ANSYS Fluent Inputs

In the CFD simulation setup using ANSYS Fluent, the material and fluid properties are defined as constant values for simplicity and consistency. The density of each fluid is specified as a constant value, which is suitable for incompressible flow analysis. Similarly, the specific heat capacity (C_p) is defined as a constant to represent the energy storage capacity of the fluid.

Thermal conductivity is also assigned as a constant value for both fluids and solid materials to model heat conduction accurately. The dynamic viscosity of the fluids is specified as a constant, which governs the flow resistance and shear behavior within the domain. These simplified inputs allow for stable and efficient simulations while still providing reliable results for comparative analysis of different configurations.

2.1 Description of Heat Exchanger Design

The current project focuses on the analysis of shell and tube heat exchanger, which is considered to be very reliable and effective equipment, used in industries. The overall length of the heat exchanger equals 500 mm, with 320 mm length of the tube section. In addition, inside the shell there are 9 tubes with diameters of 30 mm, located in the shell with diameter equal to 180 mm. There are several baffles located in the shell area, directing flow inside the heat exchanger, increasing turbulence and thus improving the process of heat exchange in hot and cold streams.

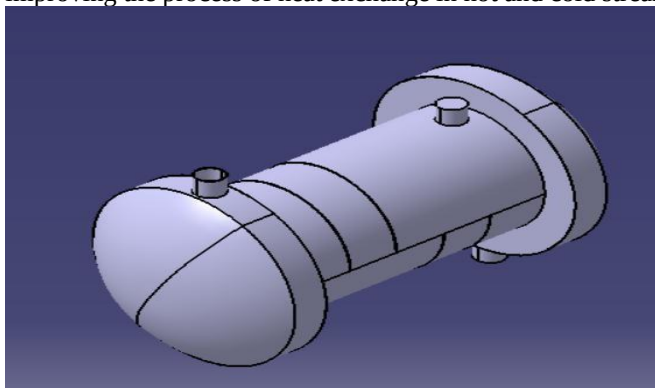


Figure 2. ISO Design of base line shell and tube heat exchanger

Figure 3. 2d front and side wire frame design of shell and tube heat exchanger

2.2 1st Iteration – Baseline Design

The first iteration represents the standard baseline configuration of the heat exchanger, which serves as a reference for performance comparison. In this design, a total of four baffles are arranged

symmetrically along the shell length. Two baffles are positioned on the top side between 0° – 180° , while the remaining two are placed on the bottom side between 180° – 360° . Each pair of baffles is offset from the center plane by 90 mm and 40 mm, creating a staggered flow path.

This configuration forces the shell-side fluid to follow a zig-zag flow pattern, increasing the contact time between the fluid and the tube surfaces. The alternating top and bottom arrangement ensure moderate turbulence and mixing. However, due to the relatively conventional design, some flow maldistribution and dead zones may still exist, which can limit heat transfer efficiency. This baseline case provides a benchmark to evaluate the effectiveness of further optimized designs.

2.3 2nd Iteration – Optimized Baffle Arrangement (Enhanced Efficiency Design)

In the second iteration, the baffle arrangement is modified to improve flow distribution and enhance heat transfer. The first set of two baffles is positioned between 0° – 90° , with offsets of 90 mm and 40 mm from the center plane. The second set of two baffles is placed between 180° – 270° on the opposite side with the same offsets.

This design introduces a more directed and controlled flow path, reducing the possibility of stagnant regions within the shell. The angular positioning ensures that the fluid is guided more effectively across the tube bundle, leading to improved turbulence and better heat exchange. Compared to the baseline design, this configuration minimizes flow bypassing and enhances the interaction between the fluid and tube surfaces. As a result, this iteration is expected to provide higher heat transfer efficiency with a relatively lower pressure drop, making it one of the most effective configurations.

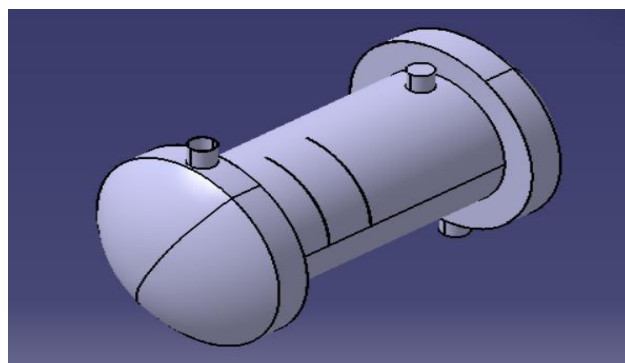


Figure 4 iso view of optimized design 2

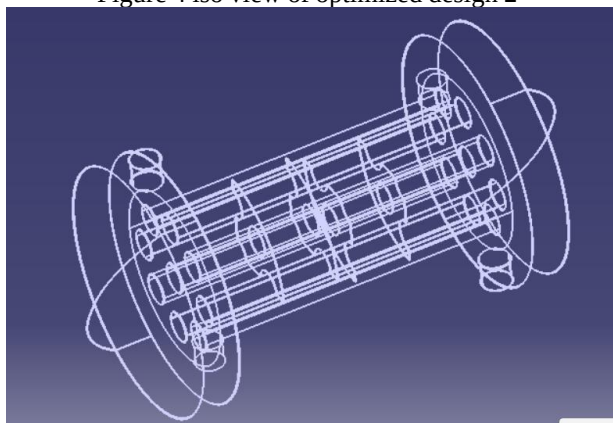


Figure 5 iso wireframe design of optimized design 2

2.4 3rd Iteration – Advanced Optimized Baffle Configuration

The third iteration represents a more advanced optimization of the baffle arrangement, designed to further enhance thermal performance. In this case, four baffles are distributed across different angular regions to create a more complex flow pattern. The first baffle is placed between 0° – 90° with a 90 mm offset, while the second is positioned between 90° – 180° with a 40 mm offset. The third baffle is arranged between 270° – 360° with a 40 mm offset, and the fourth is located between 180° – 270° with a 90 mm offset on the opposite side.

This configuration creates a multi-directional flow path, forcing the fluid to continuously change direction as it passes through the shell. Such an arrangement significantly increases turbulence intensity and promotes better mixing of the fluid. The alternating offsets (90 mm and 40 mm) ensure that the fluid covers a larger surface area of the tubes, thereby enhancing heat transfer. However, the increased flow complexity may also lead to a slightly higher pressure drop compared to simpler designs.

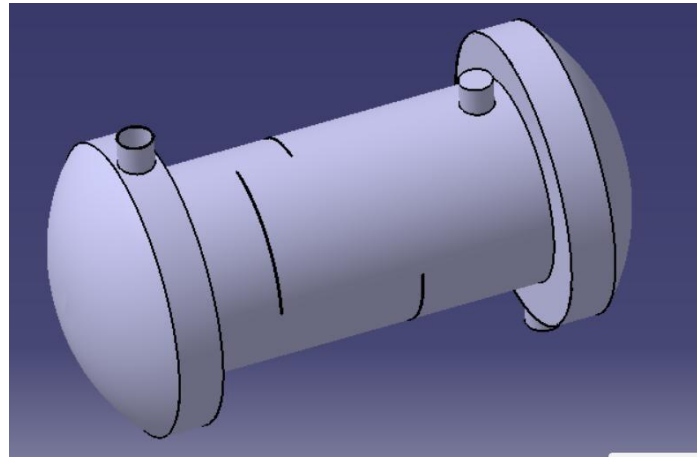


Figure 6 iso view of optimized design 3

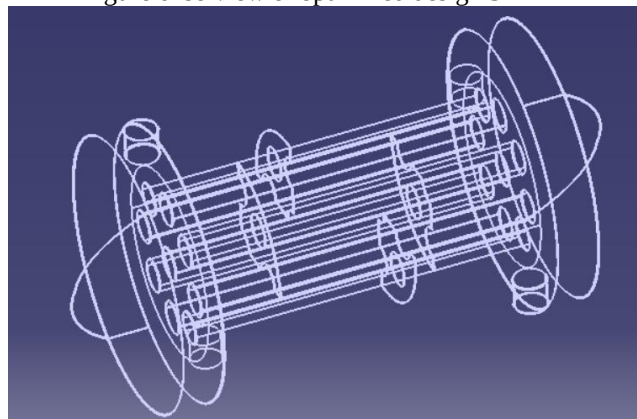


Figure 7 iso wireframe design of optimized design 3 heat exchanger

2.5 Overall Heat Exchanger Design

The present work involves the design and analysis of a shell-and-tube heat exchanger developed to study the effect of baffle arrangement, working fluid, and material on thermal performance. The heat exchanger is designed with an overall length of 500 mm and an effective tube length of 320 mm, ensuring sufficient contact time between the hot and cold fluids for efficient heat transfer. The shell has a diameter of 180 mm, enclosing a bundle of 9 tubes, each with a diameter of 30 mm, arranged to provide adequate heat transfer surface area while maintaining compactness. It consists of two fluids; the hot fluid passes through the tubes while the cold fluid passes through the shell. With this arrangement, heat transfer takes place through the walls of the tubes without mixing the flows. The flow mainly consists of counter-flow, which has a tendency to give high temperature gradients and hence good heat transfer compared to the parallel flow.

One of the features that make this particular heat exchanger unique is the baffles that exist on the shell side of the exchanger. Baffles cause fluid flowing through the shell to take up a zig-zag path thus causing turbulence of the fluid thus enhancing heat transfer. Different arrangements of baffles are examined for their effect in this experiment.

Other parameters that will be varied in this experiment include the working fluids, which are water, air, and

ethylene glycol; other parameters are also varied including the materials used to make the tubes, which are aluminum and copper.

3. CFD ANALYSIS

3.1 Case 1: Baseline Design with Water

In the first case, the standard baseline baffle arrangement is used with water as the working fluid. The baffles are arranged in the 0°–180° (top) and 180°–360° (bottom) regions with offsets of 90 mm and 40 mm, creating a conventional zig-zag flow path. This design ensures moderate turbulence and heat transfer between the shell-side and tube-side fluids.

The primary function of this arrangement is to generate moderate turbulence within the shell-side flow. As the fluid repeatedly changes direction while passing through the baffles, it disrupts the thermal boundary layer formed on the tube surfaces. This improves the rate of heat exchange between the hot fluid inside the tubes and the cold fluid in the shell. However, since this is a conventional design, the flow distribution is not fully optimized. Certain regions inside the shell may experience low velocity zones or recirculation areas, commonly referred to as dead zones, which reduce the effective heat transfer area.

CFD

3.2 1st iteration CFD with base paper design for validation

Geometry Details

- **Heat Exchanger Type:** Shell and Tube
- **Overall Length:** 500 mm
- **Tube Length:** 320 mm
- **Number of Tubes:** 9
- **Tube Diameter:** 30 mm
- **Shell Diameter:** 180 mm
- **Working Fluid:** Water (both hot and cold side)

A computational fluid dynamics (CFD) analysis was performed on a shell-and-tube hydro-thermal heatexchanger to investigate the thermal and flow characteristics of water as the working medium on both the hot and cold sides. The heat exchanger has an overall length of 500 mm, with nine tubes of 30 mm diameter arranged inside a shell of 180 mm diameter. The effective tube length for heat transfer is 320 mm. Both hot and cold fluids enter the exchanger with an inlet velocity of 2.57 m/s. The inlet temperature of the hot fluid is 353 K, while the cold fluid enters at 300 K.

Parameter	Hot Fluid	Cold Fluid
Inlet Velocity	2.75 m/s	2.75 m/s
Inlet Temperature	353 K	299 K
Turbulence Model	RNG k-ε Enhanced Wall Treatment Enabled 50	
Wall Treatment		
Energy Equation		
Iterations		

table 1. tabular of boundaries

The simulation was carried out using a steady-state, pressure- based solver with the RNG k-ε turbulence model and enhanced wall treatment to accurately capture near-wall effects and turbulent behavior. The energy equation was enabled to account for conjugate heat transfer between the hot and cold streams. Water properties were considered for the operating temperature range, and the solution was iterated up to 50 iterations until residual convergence was achieved and outlet temperatures stabilized.

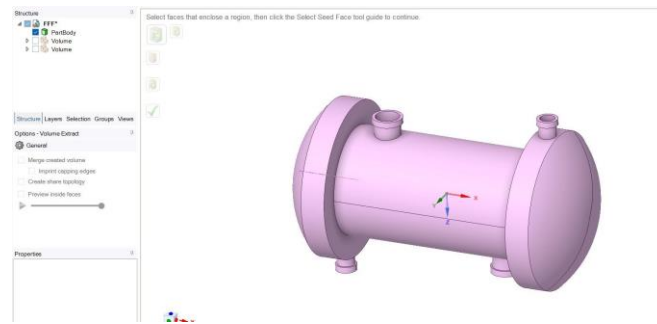


Figure 8 Shell Solid Design

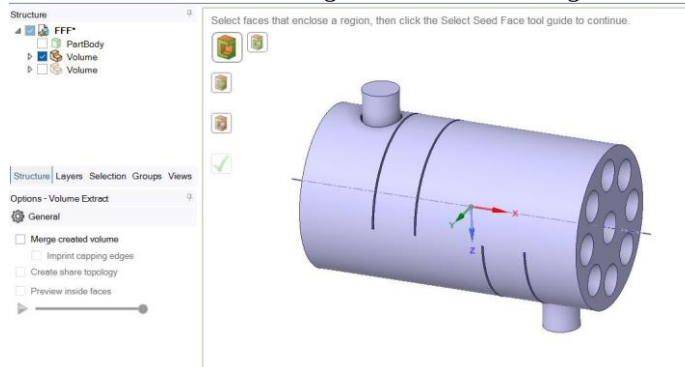


Figure 9 Cold Water Volume Extraction from Solid Shell Using Space claim

Based on the inlet velocity and tube diameter, the Reynolds number is in the order of 10^4 – 10^5 , indicating fully turbulent flow in both the tube and shell sides. The use of the RNG k- ϵ model is therefore appropriate for predicting turbulence effects and improving heat transfer estimation. The velocity distribution shows higher magnitudes at the inlet regions, with gradual reduction along the flow direction due to frictional effects. Minor flow interactions and mixing are observed within the shell region because of the presence of multiple tubes, enhancing convective heat transfer.

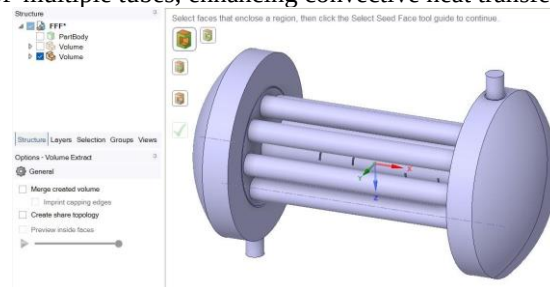


Figure 10 Hot Water Volume Extraction from Solid Shell Using Space Claim

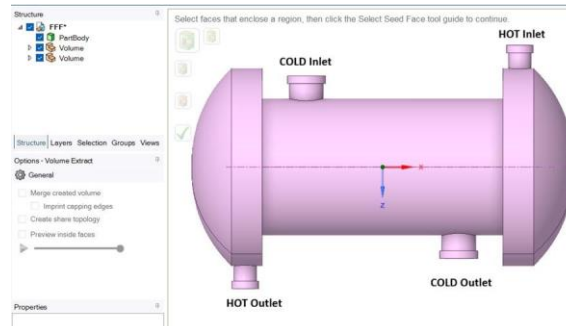


Figure 11 Heat Exchanger Fluid Parameters

The temperature contours indicate a progressive decrease in hot fluid temperature along the tube length, while the cold fluid temperature increases correspondingly. The maximum temperature gradient occurs near the inlet section where the temperature difference between the two fluids is highest. As the fluids move downstream, the temperature difference reduces, indicating effective heat exchange. The uniform arrangement of nine tubes promotes distributed heat transfer across the shell domain. Pressure drop

is observed along both flow paths due to viscous and turbulent losses. However, because the velocity is moderate (2.75 m/s), the pressure drop remains within acceptable limits while still maintaining strong convective heat transfer. Overall, the CFD analysis confirms that the heat exchanger operates in a fully turbulent regime, providing effective thermal energy transfer between the hot and cold water streams under the specified boundary conditions.

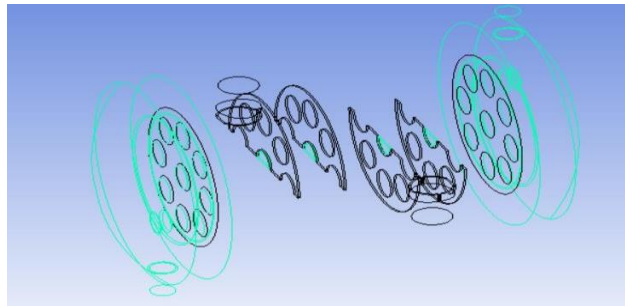


Figure 12 Wireframe Diagram of Complete assembly of Hydro Thermal Shell Type Heat Exchanger

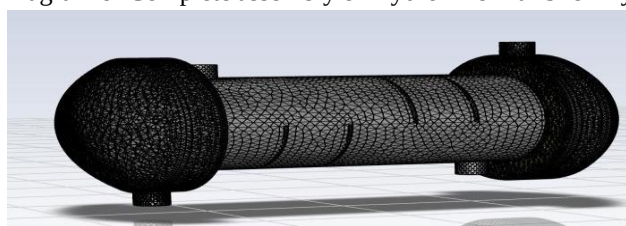


Figure 13 Mesh Configuration on Cold Fluid Domain

Elements 444032

Table Mesh Statistics

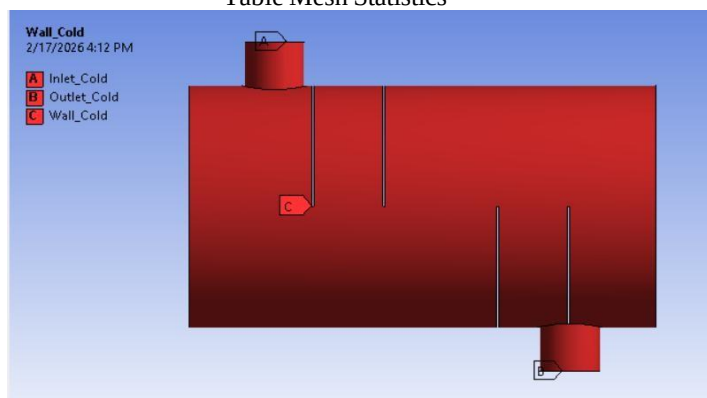


Figure Named Selection For cold Domain

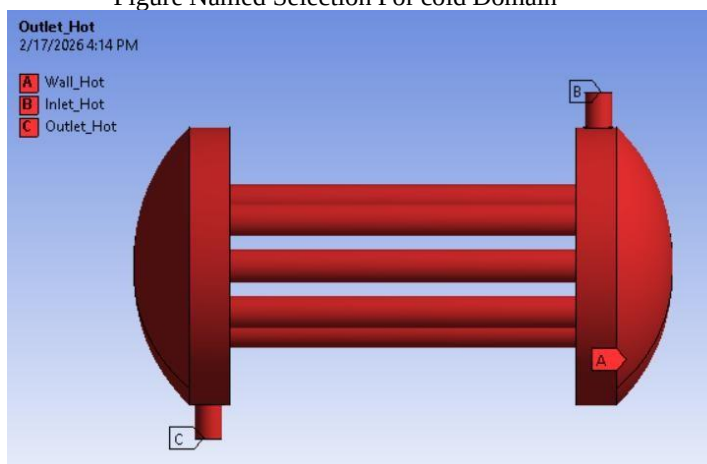


Figure 14 Named Selection For Hot Domain

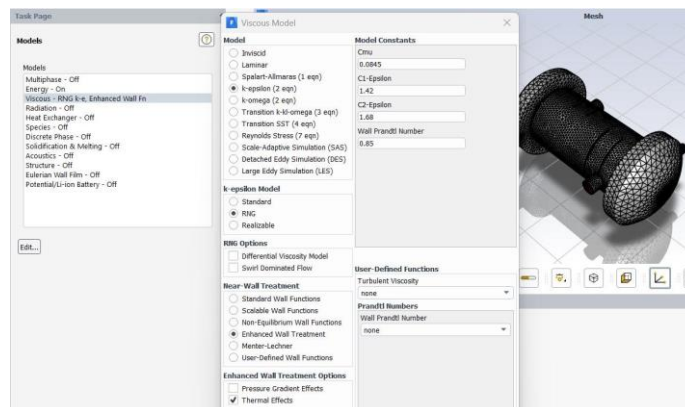


Figure 15 Setup Viscous Flow to the modal

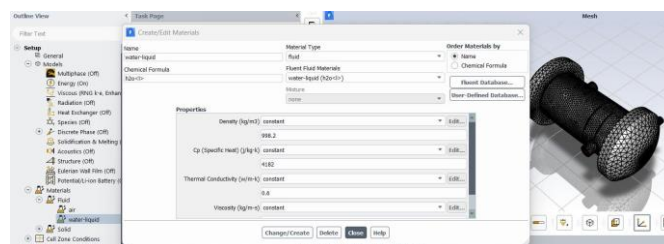


Figure 16 Material Assignment for the both Fluid Domain

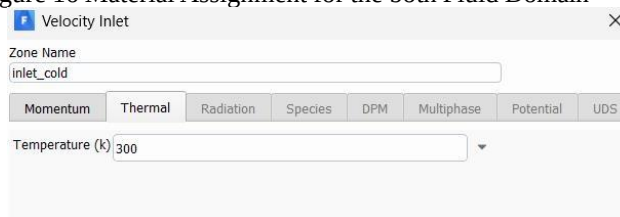


Figure 17 Cold Domain Boundary temperature assignment

Statistics	
Nodes	94848

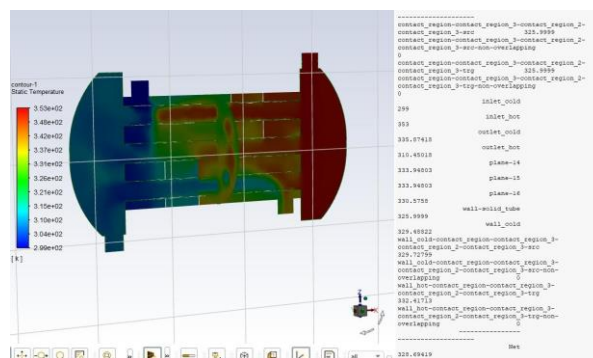


Figure 18 temperature difference contour plot

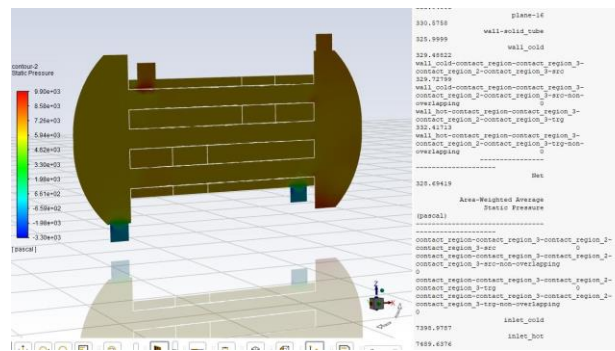


Figure 19 Pressure distribution across the heat exchanger

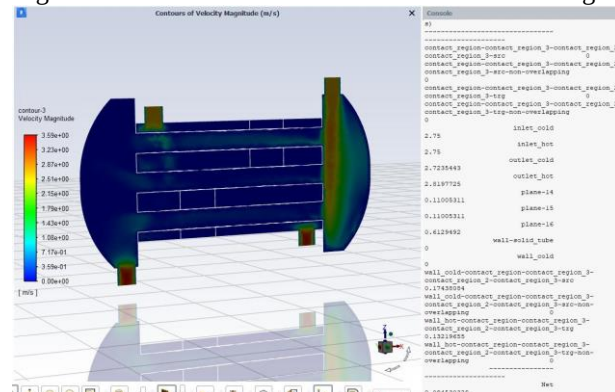


Figure 20 change in velocity in m/sec at inlets and outlets

3.3 Case 2: Optimized Baffle Arrangement (2nd Iteration) – Detailed Explanation

From the simulation results, the shell-side outlet temperature is 335.87 K, indicating that the cold fluid has absorbed heat from the hot fluid. The tube-side outlet temperature is 310.45 K, showing a corresponding decrease in the hot fluid temperature. This temperature difference confirms that heat transfer is occurring effectively, but not at the maximum possible level. The pressure at the shell inlet is 9.90×10^3 Pa, which reflects a moderate resistance to flow due to the presence of baffles. The velocity of 3.59 m/s indicates a reasonably turbulent flow regime, supporting convective heat transfer.

Despite achieving stable operation, the baseline design has certain limitations. The symmetric placement of baffles can lead to uneven flow distribution, where some regions experience higher flow while others receive less fluid interaction. This reduces the overall thermal efficiency of the system. Additionally, the presence of dead zones and bypass streams allows some portion of the fluid to pass through the shell without effectively participating in heat exchange.

Overall, this case provides a balanced but non-optimized performance, making it an essential reference point for evaluating improvements in subsequent optimized designs. It clearly demonstrates the need for better baffle arrangement to enhance flow uniformity, reduce thermalIn the second iteration, the baffle configuration is modified to improve the overall thermal efficiency of the heat exchanger by enhancing flow distribution and minimizing stagnant zones. In this design, two baffles are positioned in the 0° – 90° region with offsets of 90 mm and 40 mm from the center plane, while the remaining two baffles are arranged in the 180° – 270° region on the opposite side with the same offsets. This configuration creates a more asymmetric and directed flow path compared to the baseline design, allowing better control over the movement of the shell-side fluid.

The primary objective of this arrangement is to reduce flow maldistribution and eliminate dead zones that were present in the baseline configuration. By concentrating the baffles in specific angular regions, the fluid is forced to pass more effectively across the tube bundle, increasing cross-flow interaction between the fluid and the tubes. This leads to improved turbulence and a more uniform velocity distribution throughout the shell. The staggered offsets of 90 mm and 40 mm further ensure that the fluid repeatedly changes direction, which helps in disrupting the thermal boundary layer and enhances convective heat transfer.

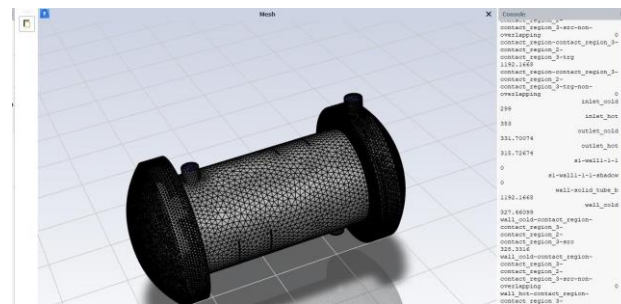


Figure 21 temperature result compute

From the simulation results, this design shows a significant improvement in performance compared to the baseline case. The tube-side outlet temperature increases to 315.72 K, which indicates that more heat is transferred from the hot fluid to the cold fluid. At the same time, the shell-side outlet temperature decreases to 331.70 K, confirming that the cold fluid absorbs a greater amount of heat. The pressure at the shell inlet reduces slightly to 9.60×10^3 Pa, which suggests that the optimized flow path reduces unnecessary resistance while maintaining effective turbulence. The velocity of 3.56 m/s indicates stable and well-distributed flow conditions.

This configuration achieves an optimal balance between heat transfer enhancement and pressure drop control. Unlike the baseline design, where some regions experienced poor flow interaction, this optimized arrangement ensures that most of the shell-side fluid actively participates in heat exchange. The improved flow uniformity and reduced bypassing result in higher thermal efficiency without increasing energy consumption significantly.

Overall, the second iteration demonstrates that strategic repositioning of baffles can greatly enhance heat exchanger performance. It provides the best combination of high heat transfer, controlled pressure drop, and efficient fluid flow, making it the most effective design among all the cases studied.

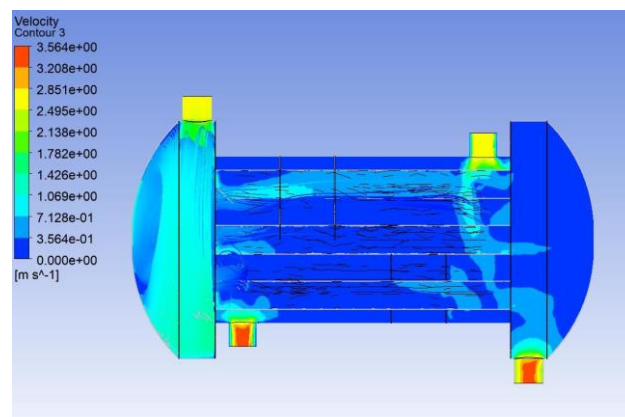


Figure 22 velocity result compute contour

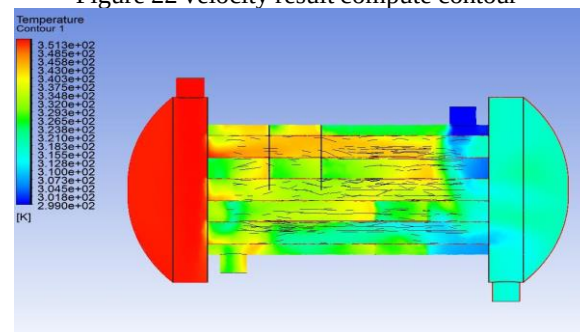


Figure 23 Temperature result compute contour

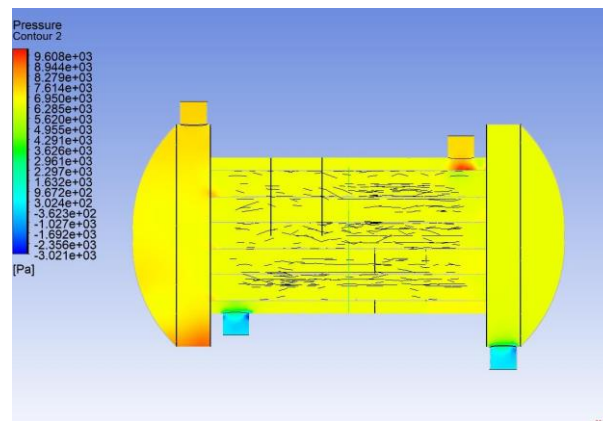


Figure 24 Pressure result compute contour

3.3 Case 3: Advanced Optimized Baffle Arrangement (3rd Iteration) – Detailed Explanation

In the third iteration, the baffle configuration is further refined to create a more complex and highly interactive flow pattern within the shell side of the heat exchanger. In this design, a total of four baffles are strategically distributed across different angular regions to maximize fluid mixing. The first baffle is placed in the 0° – 90° region with a 90 mm offset, the second baffle in the 90° – 180° region with a 40 mm offset, the third baffle in the 270° – 360° region with a 40 mm offset, and the fourth baffle in the 180° – 270° region with a 90 mm offset on the opposite side. This arrangement creates a non-uniform, staggered pattern that forces the fluid to continuously change direction throughout its flow path. The purpose of this advanced configuration is to generate maximum turbulence and multi-directional flow, thereby enhancing the interaction between the shell-side fluid and the tube surfaces. As the fluid passes through each baffle, it is redirected across different sections of the tube bundle, increasing the effective contact area and promoting better heat transfer. The alternating offsets of 90 mm and 40 mm ensure that the flow does not follow a predictable path, which helps in breaking thermal boundary layers more effectively than simpler designs. This results in improved mixing and more uniform temperature distribution across the shell.

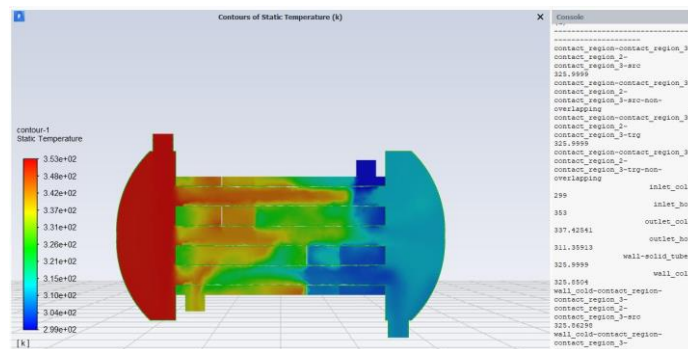


Figure 25 Temperature result compute contour

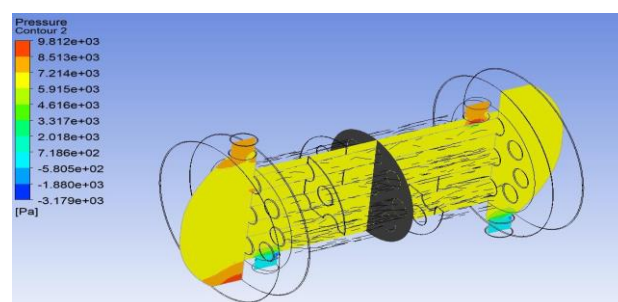


Figure 26 Pressure result compute contour

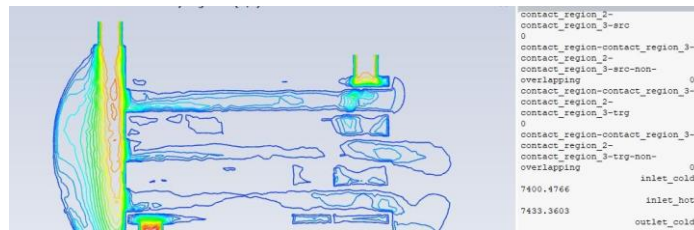


Figure 27 pressure result compute from surface integral

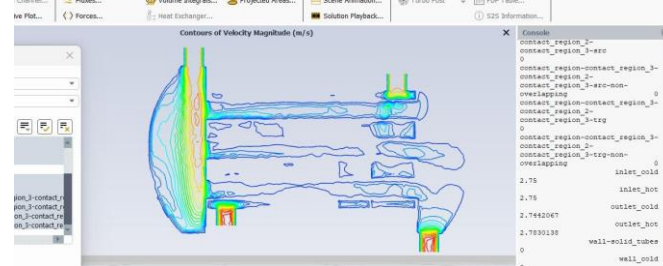


Figure 28 velocity result compute from surface integral

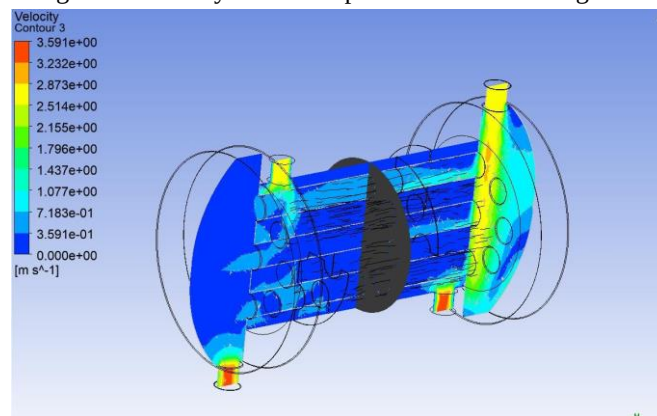


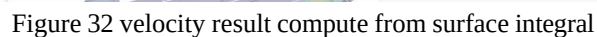
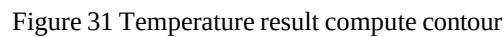
Figure 29 velocity result compute contour

However, the increased complexity of the flow path also introduces certain challenges. The frequent changes in direction and higher turbulence levels lead to an increase in flow resistance, which can result in a higher pressure drop compared to simpler configurations. From the simulation results, the tube outlet temperature is observed to be 311.35 K, while the shell outlet temperature is 337.42 K. Although heat transfer is enhanced compared to the baseline design, it is slightly lower than the second iteration. The pressure at the shell inlet is 9.81×10^3 Pa, which is higher than the optimized second case, indicating increased resistance due to the complex flow pattern.

This outcome suggests that while turbulence is essential for improving heat transfer, excessive flow disturbance may not always lead to better performance. In this case, the highly complex baffle arrangement causes additional energy losses, which partially offsets the benefits of increased mixing. As a result, the overall efficiency is slightly lower than the second iteration, even though the design is more advanced.

3.4 Case 4: Effect of Changing Working Fluid to Air (4th Iteration)

In the fourth iteration, the focus shifts from geometric optimization to working fluid variation, where the shell-side fluid is changed from water to air while keeping the baffle arrangements unchanged. Two cases are analyzed under this iteration: one with the baseline (1st iteration) baffle design and another with the advanced optimized (3rd iteration) baffle configuration. This approach helps in understanding how fluid properties influence heat exchanger performance independently of geometry.



In the second case of this iteration, the optimized 3rd iteration baffle arrangement is used with air as the working fluid. The complex baffle configuration enhances turbulence and promotes better mixing of the fluid within the shell. However, due to the inherently poor thermal properties of air, the improvement in heat transfer remains limited. The results show a tube outlet temperature of 311.05 K and a shell outlet temperature of 338.20 K, which are very close to the baseline air case. The pressure remains low at approximately 12.5 Pa, confirming that air continues to offer minimal flow resistance even in a more complex geometry.

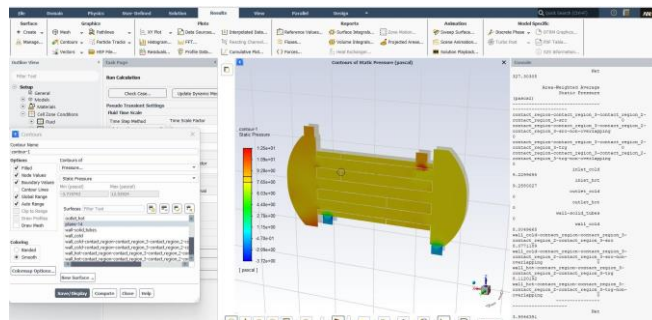


Figure 33 pressure result compute from surface integral

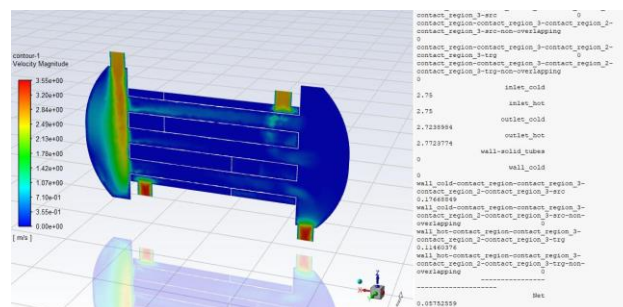


Figure 34 velocity result compute from surface integral

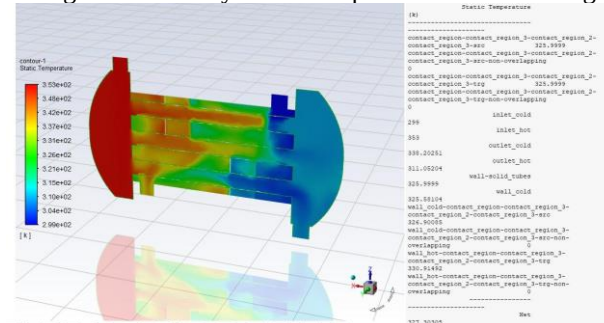


Figure 35 Temperature result compute contour

These analyses prove that although modifications in the baffle shapes can help improve the flow path, the thermal performance will depend highly on the characteristics of the working fluid itself. For instance, since air has a relatively poor heat capacity and thermal conductivity, no matter how complicated the shape of the baffles gets, the amount of heat transferred will not increase significantly. However, one of the advantages of using air is that it will reduce the pressure drop substantially and minimize the pumping losses, so if energy efficiency outweighs heat transfer efficiency, this would be the best option to consider.

To conclude, based on the results obtained during the fourth iteration, it becomes evident that the working fluid is the crucial parameter influencing the effectiveness of the heat exchanger. Water provides excellent heat transfer, while air demonstrates superior flow characteristics.

3.6 Case 5: Effect of Material Change and Ethylene Glycol (5th Iteration)

During the fifth experiment, the influence of the material of the tubes and the shell-side fluid in combination with the optimal arrangement of baffles is evaluated. In this case, aluminum is replaced by copper for the construction of the heat exchanger tubes and the shell-side fluid is switched to ethylene glycol, keeping the third-generation baffle configuration constant. The chosen parameters of materials and fluid create an opportunity to observe how enhanced thermal conductivity of the material in combination with new fluid affect heat exchanger performance.

The material of the tubes used in this configuration, namely copper, is crucial for increasing heat transfer efficiency. With significantly greater thermal conductivity, copper promotes more rapid heat transfer from the hot fluid in the tubes to the cold fluid in the shell side due to reduced thermal resistance between two

media. Thus, the efficiency of energy transfer and the resulting performance are improved in comparison with aluminum materials.

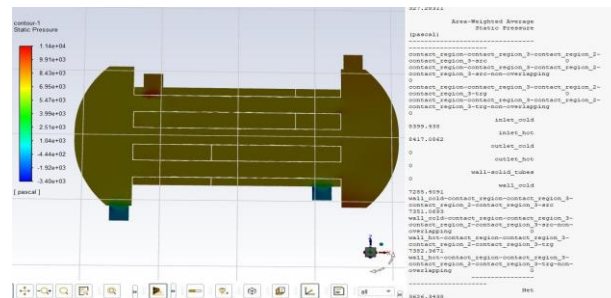


Figure 36 pressure result compute from surface integral

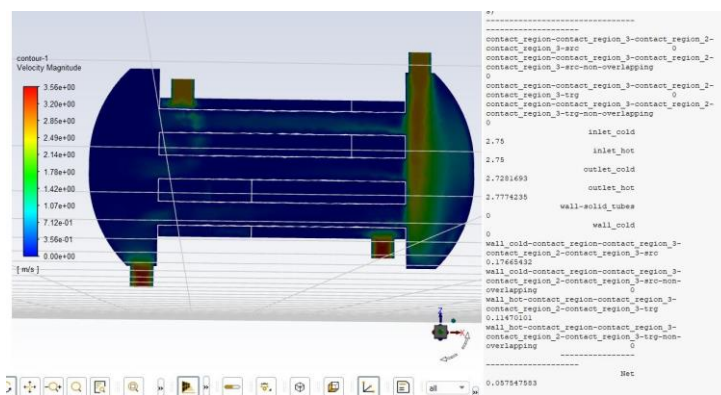


Figure 37 velocity result compute from surface integral

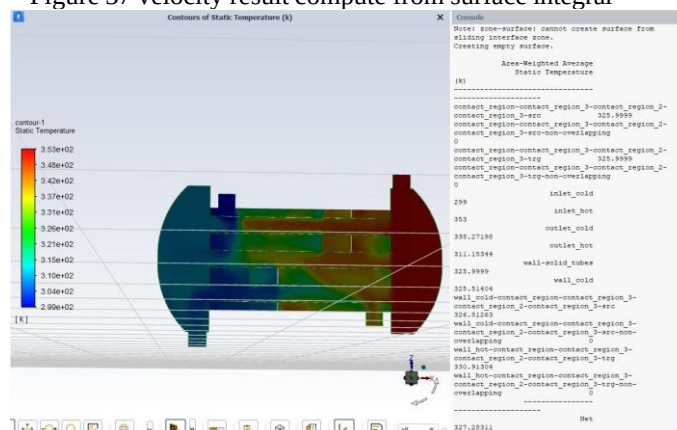


Figure 38 Temperature result compute contour

At the same time, the working fluid is changed to ethylene glycol, which has different thermophysical properties compared to water and air. Ethylene glycol has higher viscosity and lower thermal conductivity than water, but better heat retention capability. Due to its higher viscosity, the fluid experiences greater resistance while flowing through the shell, leading to an increase in pressure drop. From the results, the pressure at the shell inlet is 1.14×10^4 Pa, which is higher than both water and air cases. The velocity is observed to be 3.56 m/s, indicating stable but more resistive flow conditions. In terms of thermal performance, the tube outlet temperature is 311.15 K, and the shell outlet temperature is 338.2 K. These values indicate moderate heat transfer, but not as high as the optimized water case. While copper improves heat conduction, the overall performance is influenced by the fluid's ability to transfer heat. Since ethylene glycol has lower thermal conductivity than water, the heat transfer enhancement due to copper is

partially offset by the fluid's limitations.

This case highlights the trade-off between material enhancement and fluid properties. Although copper significantly improves heat conduction, the higher viscosity of

ethylene glycol increases flow resistance and reduces convective heat transfer efficiency. However, ethylene glycol is advantageous in applications requiring freeze protection and stable thermal behavior, making it suitable for specific industrial conditions.

3.7 Comparison of All Cases and Design Changes

The overall comparison of all six cases clearly demonstrates the combined influence of baffle arrangement, working fluid, and material selection on the thermal and hydraulic performance of the shell-and-tube heat exchanger. Each iteration introduces a specific modification, and the results show how these changes affect outlet temperatures, pressure drop, and flow behavior. A systematic comparison reveals that both geometric optimization and fluid properties play critical roles, but their effectiveness depends on how well they are balanced.

Starting with the baseline design using water (Case 1), the heat exchanger shows stable but moderate performance. The shell outlet temperature of 335.87 K and tube outlet temperature of 310.45 K indicate a reasonable level of heat transfer. However, the conventional baffle arrangement leads to non-uniform flow distribution, causing dead zones and reducing efficiency. The pressure drop of 9.90×10^3 Pa reflects moderate resistance due to the zig-zag flow pattern. This case serves as a reference point, highlighting the limitations of traditional designs.

When the baffle arrangement is optimized in Case 2 (water with 2nd iteration design), a significant improvement in performance is observed. The tube outlet temperature increases to 315.72 K, which is the highest among all cases, indicating maximum heat transfer. Simultaneously, the shell outlet temperature decreases to 331.70 K, showing better heat absorption by the cold fluid. The pressure drop reduces slightly to 9.60

$\times 10^3$ Pa, which means that the optimized flow path not only enhances heat transfer but also improves hydraulic performance. This case demonstrates that proper baffle positioning can effectively eliminate dead zones and improve flow uniformity, making it the best-performing configuration.

In Case 3 (3rd iteration with water), the design introduces a more complex baffle arrangement aimed at increasing turbulence. While this results in improved mixing, the performance does not surpass Case 2. The tube outlet temperature is 311.35 K, which is lower than the optimized Case 2, and the shell outlet temperature rises to 337.42 K, indicating less efficient heat transfer. The pressure increases to 9.81×10^3 Pa, showing higher resistance due to excessive flow disturbances. This comparison reveals that increasing turbulence beyond an optimal level can reduce overall efficiency, as the energy losses begin to outweigh the benefits of enhanced mixing. The impact of working fluid change is clearly observed in Case 4 (baseline with air). Compared to water, air results in a drastic reduction in pressure drop to 12.4 Pa, due to its low density and viscosity. However, the heat transfer performance decreases, with a tube outlet temperature of 311.80 K. Although this is slightly higher than the baseline water case, the shell outlet temperature of 336.62 K indicates less effective heat exchange. This highlights that air provides excellent flow characteristics but poor thermal performance, making it suitable only where low energy consumption is more important than heat transfer efficiency.

In Case 5 (optimized 3rd design with air), the improved baffle configuration is applied to air flow. The expectation is that increased turbulence might compensate for the poor thermal properties of air. However, the results show minimal improvement, with a tube outlet temperature of 311.05 K and shell outlet temperature of 338.20 K. The pressure remains very low at 12.5 Pa. This confirms that fluid properties dominate over geometric modifications, and even advanced designs cannot significantly enhance heat transfer when using a low-conductivity fluid like air.

Finally, in Case 6 (optimized 3rd design with ethylene glycol and copper), both material and fluid are modified simultaneously. The use of copper enhances heat conduction, but the high viscosity of ethylene glycol increases resistance, resulting in the highest pressure drop of 1.14×10^4 Pa among all cases. The tube outlet temperature is 311.15 K, and the shell outlet temperature is 338.2 K, indicating moderate heat transfer. Compared to water, glycol shows lower thermal performance but higher resistance. This case demonstrates the trade-off between improved material conductivity and increased viscous losses of the fluid.

When comparing all cases collectively, it is evident that Case

2 (optimized baffle design with water) provides the best overall performance. It achieves the highest heat transfer with a relatively low pressure drop, making it the most efficient configuration. Water consistently outperforms air and glycol in terms of heat transfer due to its superior thermophysical properties. Air offers

the lowest pressure drop but fails to deliver adequate heat exchange, while ethylene glycol provides stable but less efficient performance with higher resistance.

The comparison also highlights that baffle arrangement plays a crucial role in controlling flow distribution, but its effectiveness is strongly influenced by the working fluid. While optimization improves performance, excessive complexity (as in Case 3) can lead to diminishing returns. Similarly, material enhancement alone cannot compensate for poor fluid properties, as seen in the glycol case.

In conclusion, the detailed comparison confirms that the best heat exchanger performance is achieved through a balanced combination of optimized geometry, suitable working fluid, and appropriate material selection. Among all parameters, working fluid and baffle design have the most significant impact, while material selection provides additional improvement. This study successfully demonstrates the importance of multi-objective optimization in achieving maximum efficiency in heat exchanger systems.

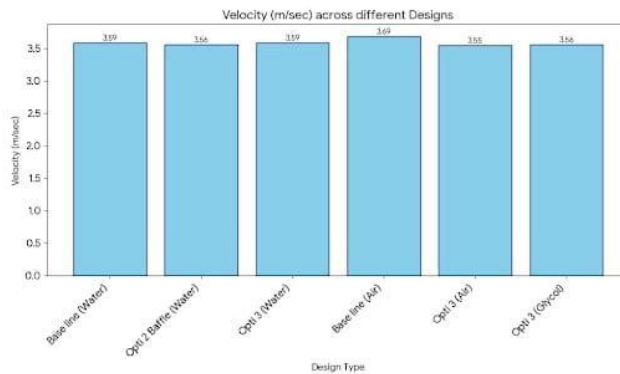
49 Result Table

Sr . No	Type	Outlet Temperat ure in K at shell	Outlet Temperat ure in K at Tube	Pressu re in Pa shell side at inlet	Veloci ty in m/sec
1.	Base line Desig n with water	335.87	310.450	9.90E 3	3.59
2.	Opti 2	331.70	315.72	9.60E 3	3.56

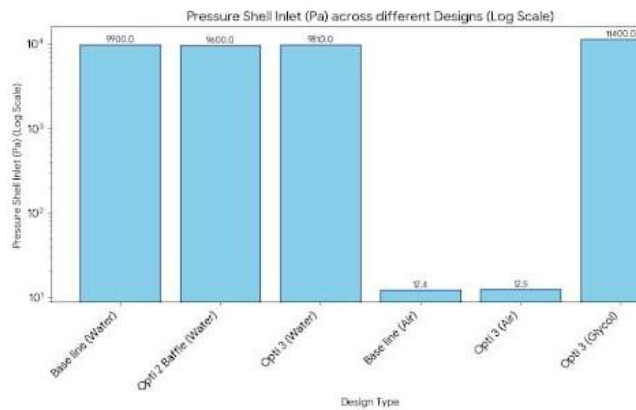
	Baffl e plate Desig n with water				
3.	Opti 3 Desig n with water	337.42	311.35	9.81E 3	3.59
4.	Base line Desig n with air	336.62	311.80	12.4	3.69
5.	Opti 3 Desig n with Air	338.20	311.05	12.5	3.55

	Opti 3 Desig n with Glyc ol	338.2	311.15	1.14E 4	3.56
--	---	-------	--------	------------	------

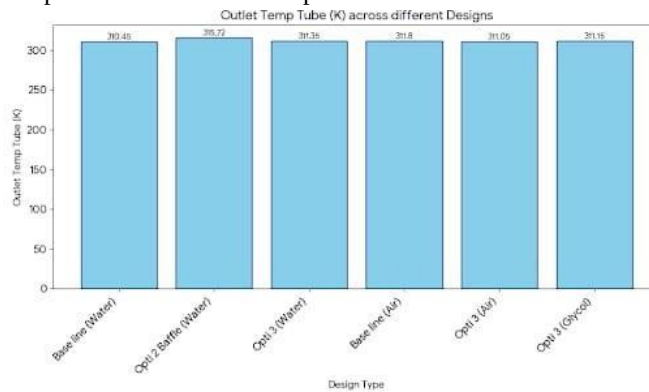
Table 2 overall comparison of results



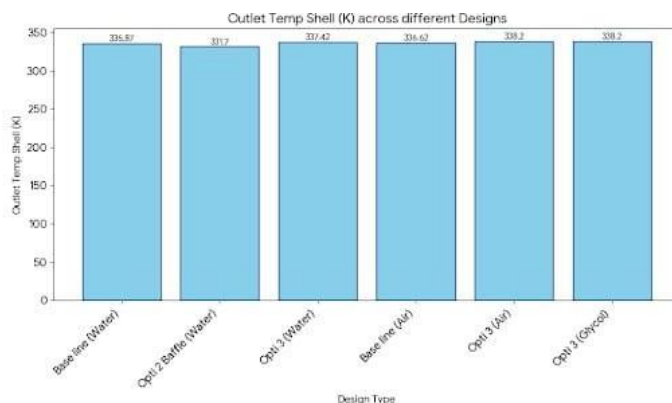
Graph 1 velocity result comparison



Graph 2 Pressure result comparison



Graph 3 Outlet temperature result comparison at tube side



Graph 4 Outlet temperature result comparison at Shell side

4. CONCLUSION & FUTURE SCOPE

4.1 CONCLUSION

The current work explores the design of the shell and tube heat exchanger with an investigation into the CFD modeling of performance improvement by means of multi-objective optimization. The research is based on six cases performed for identical conditions regarding boundary, geometrical and fluid characteristics and aimed at the optimization of the baffle arrangement and working fluid, as well as material adjustment. This study provides a detailed analysis of how parameters influence the heat exchanger performance in terms of temperature distribution and pressure drop.

The findings prove that for case 1 when using water as a working fluid, the performance is stable, but it is moderate as a result of conventional baffles that cause poor distribution of the flow in the tubes and even dead zones. Case 2 demonstrates significant increase in performance due to optimized baffle design since temperature inside the tubes is significantly elevated, while shell outlet temperature is decreasing, which indicates efficient exchange. Pressure drop becomes reduced as well; therefore, it is evident that optimization of baffle position positively affects both heat transfer and pressure drop and is the most beneficial solution.

In case 3, the baffles are more complex in their structure to induce turbulence, but the findings confirm that increasing turbulence leads to an increment of pressure drop and no additional heat transfer effect, which suggests existence of a threshold of flow disturbance. Thus, excessive turbulence does not guarantee efficiency improvement.

The study also emphasizes the dominant role of working fluid properties. When water is replaced with air, a drastic reduction in pressure drop is observed due to lower density and viscosity. However, due to low thermal conductivity and low specific heat of the air, the heat transfer capability is significantly reduced. With properly adjusted baffles, there is still a limited enhancement of the heat transfer, and it becomes obvious that physical characteristics of fluid play an important role rather than geometric changes in baffle design. In case of using ethylene glycol and copper pipes, the improved thermal conductivity of copper promotes better heat conduction, while increased viscosity causes an increase in pressure drops in the system. Thus, the final performance of the process is not significantly higher and does not overcome the performances in water cases.

On the whole, from the analysis of the performed experiments it can be concluded that the geometry of baffles and fluid type are key factors affecting heat exchanger performance, whereas material selection plays only the second role. Thus, the most optimal solution for the task is realized using water in the second iteration where the highest level of heat transfer efficiency and minimum pressure drop are reached. As follows from the presented results, it is necessary to perform the multi-criteria optimization of heat exchangers rather than to optimize one criterion.

4.2 Future Modifications and Scope

The present study provides a strong foundation for improving heat exchanger performance through CFD-based optimization; however, there are several opportunities for further enhancement and advanced

research. One important area for future modification is the use of advanced baffle designs, such as helical, segmental, or perforated baffles. These designs can provide smoother flow paths, reduce pressure drop, and improve heat transfer simultaneously compared to conventional flat baffles. Additionally, optimizing parameters such as baffle spacing, cut ratio, and orientation using numerical optimization techniques can further enhance performance.

Another significant scope lies in the application of advanced working fluids, particularly nanofluids. Fluids containing nanoparticles such as Al_2O_3 , CuO , or TiO_2 have shown the potential to significantly improve thermal conductivity and heat transfer rates. Future studies can focus on combining nanofluids with optimized baffle configurations to achieve higher efficiency. Moreover, the use of hybrid nanofluids can also be explored to balance thermal performance and viscosity effects.

The study can also be extended by considering temperature-dependent material and fluid properties instead of constant values. This would provide more realistic results, especially for systems operating under a wide temperature range. In addition, transient (time-dependent) analysis can be performed to study the dynamic behavior of the heat exchanger during start-up and varying load conditions, which is important for real industrial applications.

Further improvement can be achieved by exploring advanced materials such as composite materials, coated tubes, or corrosion-resistant alloys. The use of additive manufacturing (3D printing) can enable the development of complex geometries like lattice structures and enhanced surface textures, which can significantly improve heat transfer performance.

From a design perspective, future work can include multi-objective optimization techniques such as genetic algorithms or machine learning methods to simultaneously optimize heat transfer, pressure drop, and cost. Experimental validation of CFD results is another important scope, as it helps in verifying simulation accuracy and improving model reliability.

4.3 Final Conclusion (Points)

- The performance of the heat exchanger is strongly influenced by baffle arrangement, working fluid, and material selection
- The baseline design shows stable but moderate performance due to flow maldistribution and dead zones
- The 2nd iteration (optimized baffle design) significantly improves heat transfer and reduces pressure drop
- The 3rd iteration increases turbulence but leads to higher pressure drop with no major gain in heat transfer
- Water is the most effective working fluid due to its high specific heat and thermal conductivity
- Air provides the lowest pressure drop but has poor heat transfer capability
- Ethylene glycol offers stable performance but results in higher pressure drop due to high viscosity
- Copper enhances heat transfer due to high thermal conductivity but increases cost significantly
- Aluminum provides a good balance between cost and performance
- Excessive turbulence does not always improve performance; optimal flow distribution is more important
- Multi-objective optimization is necessary to achieve the best thermo-hydraulic performance

References

- [1] M. Aydin *et al.*, "Optimization and CFD Analysis of a Shell-and-Tube Heat Exchanger with Multi-Segmental Baffle," 2020.
<https://www.researchgate.net/publication/346123036>
- [2] P. Sahoo *et al.*, "Design and Analysis of Heat Exchanger for Determined Heat Transfer Rate," *IJERT*, 2020.
<https://www.ijert.org/design-and-analysis-of-heat-exchanger-for-determined-heat-transfer-rate>
- [3] S. Kumar *et al.*, "Review on Design and Optimization of Shell and Tube Heat Exchanger," 2020.
<https://www.researchgate.net/publication/343311331>
- [4] R. Shukla *et al.*, "Investigation on Shell and Tube Heat Exchanger Using CFD," 2022.
<https://www.researchgate.net/publication/360872993>
- [5] M. Abdelhamid *et al.*, "Comparative Performance of Shell-and-Tube Heat Exchangers,"

2022.
<https://www.researchgate.net/publication/367583933>
- [6] M. Marzouk *et al.*, “Heat Transfer Enhancement Methods in Shell-and-Tube Heat Exchangers,” 2023.
<https://www.researchgate.net/publication/371126245>
- [7] R. Kumar *et al.*, “Analysis of Shell-and-Tube Heat Exchanger Using Kern’s Method and CFD,” 2024.
<https://www.researchgate.net/publication/381843628>
- [8] M. Şahin *et al.*, “CFD Analysis of Heat Exchanger with Nanofluid,” 2025.
<https://link.springer.com/article/10.1007/s10973-025-14276-8>
- [9] J. Estupiñán-Campos *et al.*, “Numerical Simulation of Heat Transfer in Shell-and-Tube Heat Exchanger,” *Energies*, 2024. <https://www.mdpi.com/1996-1073/17/3/691>
- [10] A. Şahin *et al.*, “Biogas Nanofluid Heat Transfer Enhancement in Heat Exchanger,” 2025.
<https://link.springer.com/article/10.1007/s10973-025-14276-8>
- [11] M. Ali *et al.*, “Nanoparticle-Based Heat Transfer Enhancement in Heat Exchanger,” 2026.
<https://link.springer.com/article/10.1186/s44147-026-00896-w>
- [12] M. Alhulaifi *et al.*, “CFD Analysis of Double Pipe Heat Exchanger Using Nanofluids,” *MDPI*, 2024.
<https://www.mdpi.com/2411-9660/8/3/39>
- [13] T. Tahmid *et al.*, “CFD Study on Twisted Tube Heat Exchanger,” 2025.
<https://scienpg.com/scs/index.php/engineering/article/view/scse.2025.3.139>
- [14] Y. Li *et al.*, “Thermal-Hydraulic Analysis of Heat Exchanger with Supercritical Fluids,” 2025.
<https://www.sciencedirect.com/science/article/pii/S0017931025009561>
- [15] W. Boonloi *et al.*, “Heat Transfer Enhancement Using Modified V-Orifice,” *Frontiers*, 2026.
<https://www.frontiersin.org/articles/10.3389/fmech.2025.1739706>
- [16] M. Marcinkowski and K. Korab, “CFD Analysis of Multi-Row Finned Tube Heat Exchanger,” 2025.
<https://arxiv.org/abs/2509.08611>
- [17] T. Yanagihara *et al.*, “Optimization of TPMS Lattice Heat Exchanger,” 2025.
<https://arxiv.org/abs/2512.10207>
- [18] A. Tutar *et al.*, “CFD Modeling of Industrial Furnace Heat Exchanger,” 2025.
<https://arxiv.org/abs/2501.08660>
- [19] A. Khan *et al.*, “CFD Study of Heat Transfer Enhancement Using Hexagonal Tubes,” *Energies*, 2023.
<https://www.mdpi.com/1996-1073/16/2/880>
- [20] A. Bartosik, “Numerical Heat Transfer and Fluid Flow Review,” *Energies*, 2022.
<https://www.mdpi.com/1996-1073/15/8/2922>
- [21] G. Anjaneya *et al.*, “CFD-Based Study of Microchannel Heat Exchanger Using Nanofluids,” *Scientific Reports*, 2026.
<https://www.nature.com/articles/s41598-026-41225-8>
- [22] S. Al-Rawashdeh, “CFD Analysis of Heat Exchanger Effectiveness and LMTD,” *Emerging Science Journal*, 2025.
<https://www.ijournalse.org/index.php/ESJ/article/view/3165>
- [23] L. Singh *et al.*, “Geometric Modification of Tube for Heat Transfer Enhancement,” *IJERT*, 2024.
<https://www.ijert.org/numerical-study-of-tube-to-increase-the-heat-transfer-performance-by-geometric-modification-using-cfd-method>
- [24] R. Zhu and D. Shin, “Flameless Combustion-Based Heat Exchanger Analysis,” *Energies*, 2023.
<https://www.mdpi.com/1996-1073/16/20/7160>
- [25] G. Swathi and V. Subrahmanyam, “CFD Analysis of Pipe Heat Exchanger Using Copper and Aluminum,” *IJERT*, 2024. <https://www.ijert.org/cfd-analysis-of-a-pipe-in-a-heat-exchanger>