RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING
e-ISSN: 2773-4765

RPMME

Vol. 6 No. 1 (2025) 393-402
https://publisher.uthm.edu.my/periodicals/index.php/rpmme

Comparison Investigation of Thermal Performance of Shell and Tube Heat Exchanger with Helical Baffles

Muhammad Haziq Mohd Zamri[1], Nurhafizzah Hassan[1]*

[1] *Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400, Parit Raja, Batu Pahat, Johor, MALAYSIA*

*Corresponding Author: hafizzaah@uthm.edu.my
DOI: https://doi.org/10.30880/rpmme.2025.06.01.043

Article Info

Received: 15 December 2024
Accepted: 15 March 2025
Available online: 31July 2025

Keywords

Shell and tube heat exchanger, single segment baffles, helical baffles, heat transfer coefficient, heat transfer rate, Logarithmic Mean Temperature Difference

Abstract

This study was a comparison investigation of the influence of helical baffles on the thermal performance of heat in shell and tube exchangers, utilizing the SolidWorks 2023 software. A 3D model was generated and computational fluid dynamics (CFD) simulations were conducted to compare the heat transfer rates between single segment and helical baffles using the Logarithmic Mean Temperature Difference (LMTD) method. Water and olive oil were used as hot fluids while water was used as the cold fluid. The findings showed significant improvement with the implementation of helical baffles. The water and water simulations indicated a result on the heat transfer rate (Q) for single segment baffles, 623.435 kW and helical baffles, 676.663 kW. Helical baffles improve the mixing of fluids and increase turbulence which in turn enhanced heat transfer. Conversely, olive oil and water simulation gained a lower heat transfer rate. The study emphasizes the need to select optimal baffles and consider fluid properties to achieve maximum performance in heat exchangers. Heat transfer rates were additionally influenced by other factors such as temperature, pressure and tube properties.

1. Introduction

Heat exchangers were essential components in several applications. They allowed thermal energy to be transmitted between two fluids at different temperatures without mixing them [1]. This idea was utilized in several domains such as vehicle radiators and power plants. The heat transfer rate of a heat exchanger was influenced by the temperature differential between the fluids at different points along the exchanger which varies continually. Shell and tube heat exchangers were the most used and versatile among many heat exchanger designs. The exchangers were comprised of a cylindrical shell that contained many tubes. Hot fluid passed through the tubes while cooler fluid circulated in the shell. The heat was transferred through the tube walls via conduction and inside each fluid by convection. Ongoing developments improved the heat transfer efficiency of existing designs [2].

Industrial operations widely use shell and tube heat exchangers for various purposes. Their design allowed for production in multiple sizes and flow arrangements that provided great adaptability. Various factors such as fluid temperature, fluid pressure, shell diameter, tube shape, tube number and baffle spacing influenced the rate of heat transfer in heat exchangers [3]. Baffles were internal components that had dual functions. They provided structural support for the tubes to maintain correct alignment during assembly and operation and to avoid vibrations caused by fluid movement. Baffles guided the fluid flow on the shell side in a zigzag pattern across the tube bundle, increasing fluid speed and heat transmission efficiency [4].

This is an open access article under the CC BY-NC-SA 4.0 license.

The relationship between shell and tube baffles was crucial for optimizing heat transfer efficiency using Computational Fluid Dynamics (CFD) analysis in SolidWorks software. CFD simulations enabled engineers to assess the influence of various baffle configurations on fluid flow patterns, temperature distribution and heat transfer efficiency. This was important for enhancing and creating shell and tube heat exchangers in SolidWorks. SolidWorks utilized the finite element method (FEM) as a computational approach to address the intricate issues of heat exchanger shells. FEM broke down the shell structure into smaller and more essential pieces to facilitate its structural study [5].

The Logarithmic Mean Temperature Difference (LMTD) methodology was an essential way of calculating the characteristic dimensions of a shell and tube heat exchanger. The LMTD approach was beneficial when the initial and final temperatures of both hot and cold fluids were specified which made it easier to compute the LMTD [6]. Temperature is an important factor in heat exchanger design problems. The focus was usually on the fluids' entrance and exit temperatures and the intended ultimate temperature of one fluid. The LMTD was an essential metric to calculate the temperature difference driving force in flow systems. CFD simulation data may be used in the LMTD formula for heat transfer calculations by first specifying the shell's geometry and then moving on to thermal analysis.

2. Computational Fluid Dynamics (CFD) analysis in SolidWorks

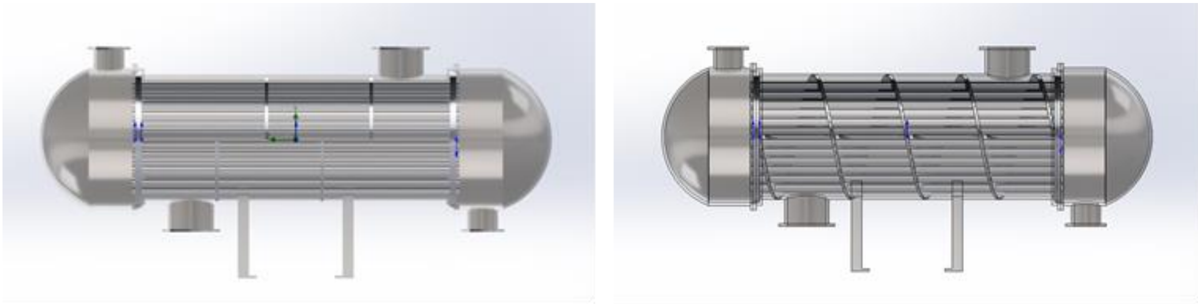
The Computational Fluid Dynamics (CFD) analysis in SolidWorks is crucial for studying fluid flow processes in engineering and scientific simulations. This guide outlines the essential phases of conducting a CFD analysis on SolidWorks 2023 including setting objectives, creating a precise shell and tube heat exchanger geometry, generating a mesh, specifying boundary conditions and selecting physics models. The solver setup involves configuring numerical algorithms and convergence criteria. The simulation and post-processing stages provide an in-depth understanding of fluid dynamics, aiding optimization and improvement. The reliability and applicability of the results are enhanced through validation against experimental data and comprehensive documentation.

2.1 Geometry Modelling

Table 1 *The geometry of the shell and tube heat exchanger*

Component	Value	Unit
Inner shell diameter	400	mm
Outer shell diameter	440	mm
Shell length	1000	mm
Inner diameter of tubes	22.5	mm
Outer diameter of tubes	25	mm
Tubes length	1000	mm
Number of tubes	124	pcs
Number of baffles	4	pcs
Baffle distance	250	mm

Table 1 shows the geometry of the shell and tube heat exchangers. Shell and tube heat exchangers were precisely developed to maximize heat transfer efficiency. It adhered to strict guidelines established by the Tubular Exchanger Manufacturers Association (TEMA) and incorporated innovations from recent studies such as those conducted by Bichkar et al. The design procedure included separating the exchanger into essential components. Each played an important role in optimizing efficiency as the main body of the exchanger is in the tube bundle. This system of pipelines is frequently fitted with turbulence-inducing obstacles that act as the main pathway for one of the fluids. The baffles enhanced heat transfer efficiency by inducing a turbulent flow pattern inside the tubes while the shell surrounded the tube bundle. This durable part created a safe and sealed space under pressure for the heat transfer. The shell has been designed to withstand the pressure of the fluids by holding, guaranteeing secure and dependable performance. Sealing and access were crucial components of a well-constructed shell and tube heat exchanger. The shell cover and head flange, along with a gasket, form a seal that prevents leaking. The design provided simple access for maintenance operations using a detachable head cover. While in use, the saddles raise and steady the whole exchanger, reducing strain and vibrations. This guaranteed ideal heat transfer conditions by avoiding accidental fluid mixing. This guaranteed the exchanger's durability and dependable performance in various industrial settings. Fig. 1 The shell and tube heat exchanger have (a) single segment baffles and (b) helical baffles.



(a) (b)

Fig. 1 *The shell and tube heat exchanger (a) single segment baffles; (b) helical baffles*

2.2 Fluid Properties and Material

Table 2 *The material and fluid materials used in CFD flow simulation*

Component	Hot fluid (Water)	Hot fluid (Olive oil)	Cold fluid (Water)
Inlet lids placement	Tubes	Tubes	Shell
Inlet temperature (˚C)	90	90	20
Density (kg/m^3)	998	916	998
Flow rate (kg/s)	8	8	8
Operating pressure (bar)	1.013	1.013	1.013
Material	Stainless steel	Stainless steel	Stainless steel

Table 2 displays the materials and fluids used in the CFD flow simulation. CFD simulations rely significantly on specific materials and parameters to forecast fluid behavior in a particular situation. Density, flow rate, thermal condition and pressure were critical in determining how materials and fluids reacted to external forces, heat transfer and flow. Incorporating these characteristics into the simulation, CFD in SolidWorks 2023 software can accurately simulate the fluid's movement around objects, pressure variations, and heat transfer.

2.3 Fluid Subdomains, Boundary Conditions, Simulation Goals and Mesh

Fluid subdomains in SolidWorks Flow Simulation enabled the simulation of numerous fluids in a single model. This was beneficial for scenarios where the design had various fluids interacting such as a heat exchanger with water on one side and an air or gas combination on the other. In this investigation, fluid subdomain 1 is a hot-water inlet, whereas fluid subdomain 2 is a cold-water inlet.

Boundary conditions in SolidWorks Flow Simulation were crucial for specifying fluid behavior at the boundaries of the shell and tube heat exchanger models. The conditions simulate real-world situations by defining features such as fluid inlets, fluid outlets and barriers (types of pressure). The opportunity to specify the pressure and flow rate at the inlets and outlets established no-slip conditions for walls where the fluid sticks. Implementing these circumstances may generate precise simulations of interior or exterior fluid flow, ensuring dependable outcomes in the simulation.

Simulation goals are a special goal setting in the heat exchanger simulation that allows customization of the desired results. This setting concentrates on four essential key performance indicators (KPIs) that regulate the exchanger's performance. It specifies the desired temperatures for the hot and cold fluids entering and exiting the system and the intended flow rate inside the exchanger. Furthermore, it enhanced flexibility by defining extra targets by setting and adding the calculation goals for the LMTD formula.

Table 3 *The mesh testing data on shell and tube heat exchanger.*

Component	Hot fluid temperature (˚C)	Cold fluid temperature (˚C)
Mesh 1	31.56	26.23
Mesh 2	31.53	26.02
Mesh 3	31.50	25.56
Mesh 4	31.51	25.31
Mesh 5	31.49	25.28
Mesh 6	31.49	25.24
Mesh 7	Undefined	Undefined

Table 3 shows the output data after running the CFD simulation in SolidWorks for this investigation. SolidWorks Flow Simulation used a mesh of small 3D cells to evaluate fluid flow in the shell and tube heat exchanger model. This meshing procedure was crucial for accuracy. The program used a finite volume approach to partition the fluid space into cells for computations. The ability to globally manipulate the mesh by establishing a certain element size for the whole shell and tube heat exchanger model. Global mesh settings provide more refinement in select regions. Various factors such as the complexity of flow and computational capabilities influenced mesh configuration to achieve ideal outcomes. The researcher has set mesh level 5 for this flow simulation because from mesh testing, the data shows the value of the mesh result starting to converge at 25 ˚C.

2.4 Logarithmic Mean Temperature Difference (LMTD)

The logarithmic mean temperature difference (LMTD) was a fundamental concept employed in the design and analysis of heat transfer equipment, namely heat exchangers. The LMTD methods were widely used in the field of chemical engineering. These methods involved applying a correction factor to the LMTD to accurately determine the average temperature difference between the fluids involved in heat exchange. The LMTD was employed because of the prevailing belief that its magnitude was greater when two opposing streams were in countercurrent flow than when the same streams were in concurrent flow [8]. The temperature differences between T1 and T2 between the two fluids were evaluated from the temperatures at the shell and tube heat exchanger terminals (counter flow). The data taken from the CFD simulation in SolidWorks applied to this equation. The LMTD can be obtained by using mass and energy conservation principles in Equations 1, 2 and 3:

$$T_{LM} = \frac{T1 - T2}{\ln\left(\frac{T1}{T2}\right)} \tag{1}$$

$$T1 = T_{h\,in} - T_{c\,out} \tag{2}$$

$$T2 = T_{h\,out} - T_{c\,in} \tag{3}$$

Where;

T_{LM} = LMTD

$T_{h\,in}$ = Hot inlet temperature

$T_{h\,out}$ = Hot outlet temperature

$T_{c\,in}$ =Cold inlet temperature

$T_{c\,out}$ = Cold outlet temperature

2.5 Heat Transfer Rate

The investigation relies on three crucial equations. Equation 4 illustrates the rate at which heat is transferred, whereas Equation 5 computes the surface area of the shell and tube heat exchanger. Equation 1 ultimately established the LMTD. These equations were interrelated. The equations will be utilized throughout the simulation to compute the heat transfer rate for each test by inputting all the available data. This investigation offered vital insights into the system's heat transfer performance.

Research Progress in Mechanical and Manufacturing Engineering Vol. 6 No. 1 (2025) p. 393-402

$$Q = U \times A \times T_m \quad (4)$$

$$A = L\pi d \quad (5)$$

Where;
Q = Heat transfer rate
U = Heat transfer coefficient
T_m = T_{LM} = LMTD
A = Area of shell and tube heat exchanger
L = Length of shell and tube heat exchanger
d = diameter of shell and tube heat exchanger

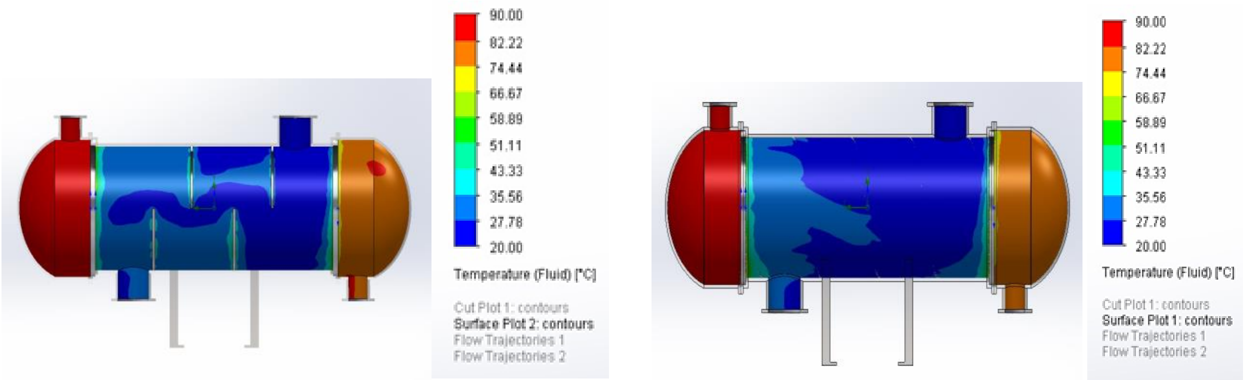
3. Result and Discussion

Efficient heat transfer was the top priority in shell and tube heat exchangers in biodiesel facilities. Baffles which were fixed plates located inside the shell have an important purpose. These devices channel the hot fluid on the shell side to efficiently transfer heat to the cold tubes, but they also serve as structural support to minimize tube vibration and damage [9]. Nevertheless, the baffle's design has a significant impact on performance. Single segmental baffles which were often used in industry provide enough support and decrease efficiency in slower flow regions. With their curved arrangement, Helical baffles provide a rotational movement in the fluid to enhance mixing and heat transfer, increasing pressure drop across the exchanger. Researchers used computational fluid dynamics (CFD) simulations in SolidWorks 2023 to investigate how single segmental and helical baffles affect the heat transfer rate in a shell and tube heat exchanger. Two sets of fluid conditions have been simulated: the first set tested with water (hot fluid) and water (cold fluid) and later the second set tested with olive oil (hot fluid) and water (cold water).

3.1 Heat Exchanger Simulation of Hot Fluid (Water) and Cold Fluid (Water)

3.1.1 Surface Plot

Fig. 2 shows the surface plot of fluid temperature for both baffles. These figures were used to analyze the fluid temperature condition on the surface of the shell and tube heat exchanger with baffles in SolidWorks Flow Simulation. The hot water input reached the maximum temperature (100 ˚C) and then decreased to a lower temperature at the hot fluid outlet at 81 ˚C (single segment baffles) and 80 ˚C (helical baffles). On the other hand, the cold inlet started with the lowest temperature (20 ˚C). Then it increased to a higher temperature at the cold fluid outlet at 26 ˚C (single segment baffles) and 27 ˚C (helical baffles).

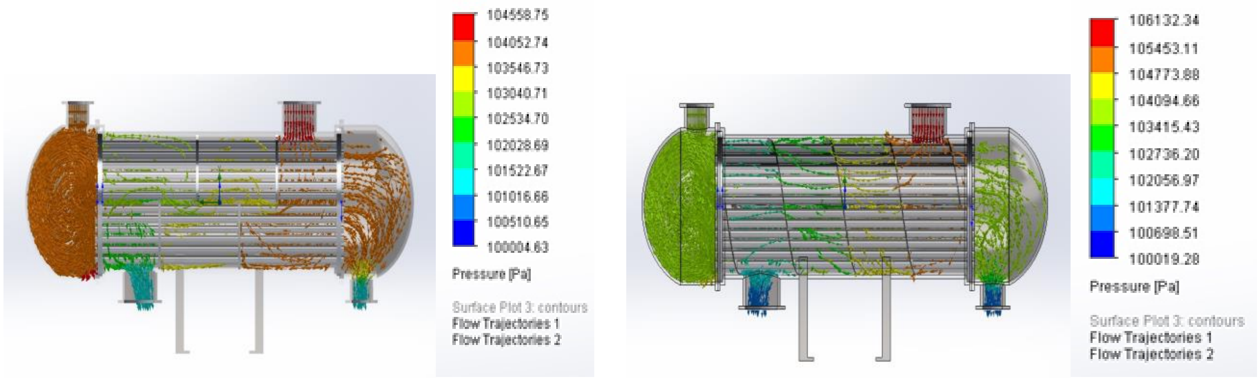


(a) (b)

Fig. 2 *The surface plot of fluid temperature (a) single segment baffles; (b) helical baffles*

3.1.2 Pressure Trajectory

The pressure trajectory was a crucial aspect of this simulation as it illustrates the pressure flow path of both the hot fluid in tubes and the cold fluid at the baffles within the shell and tube heat exchanger. Fig. 3 shows the path followed by the pressure of hot fluid and cold fluid in a shell and tube heat exchanger with single segment baffles and the trajectory of the pressure of hot fluid and cold fluid in a shell and tube heat exchanger with helical baffles. The trajectory of single segment and helical baffles may have distinct effects on the pressure drop and heat transfer processes in shell and tube heat exchangers. These baffles contribute to increased pressure drop and improved heat transfer efficiency. From Fig. 3, the pressure started entering the baffles with 104.558 kPa and dropped to 101.016 kPa in single segment baffles while the pressure entered the baffles with 106.132 kPa and dropped to 100.698 kPa.



(a) (b)

Fig. 3 *The pressure trajectory (a) single segment baffles; (b) helical baffles*

3.1.3 Velocity vs. Heat Transfer Coefficient

As depicted in Fig 4, the velocities increase from 0.13 m/s to 0.28 m/s, the value of the heat transfer coefficient decreases from 2515.433 W/m^2/K to 924.279 W/m^2/K (single segment baffles) and 2271.272 W/m^2/K to 1005.772 W/m^2/K (helical baffles). Initially, turbulence increased when velocity increased, resulting in improved mixing and reduced heat transfer coefficient. A comparison between heat transfer coefficient and velocity for single segment and helical baffle layouts emphasizes the importance of detailed investigation. At this stage, although there may be a considerable increase in pressure drop on the shell side which results in a faster heat transfer rate, there might be a modest fall in the overall heat transfer coefficient [10]. In this regard, SolidWorks CFD simulations are highly effective tools. Flow visualization techniques allow us to observe and analyze fluid movement patterns, detect areas where flow may become stagnant and enhance the design of baffles and operational parameters to create an optimal trade-off between heat transmission and pressure drop. Fig. 4 illustrates the graphical correlation between velocity and heat transfer coefficient based on the SolidWorks CFD simulation.

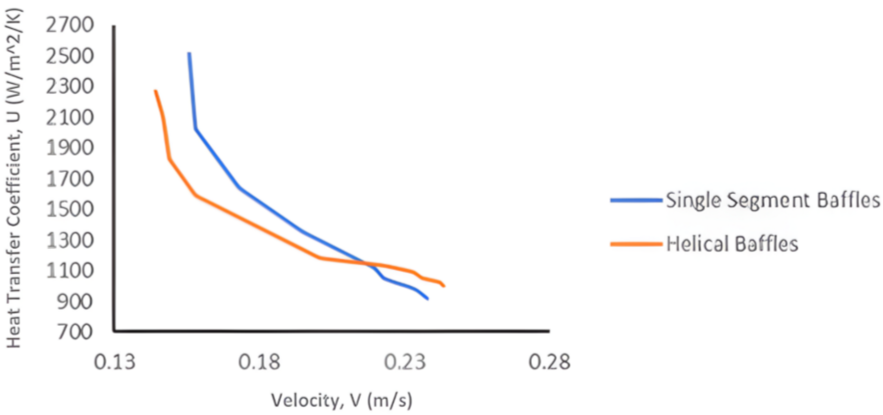
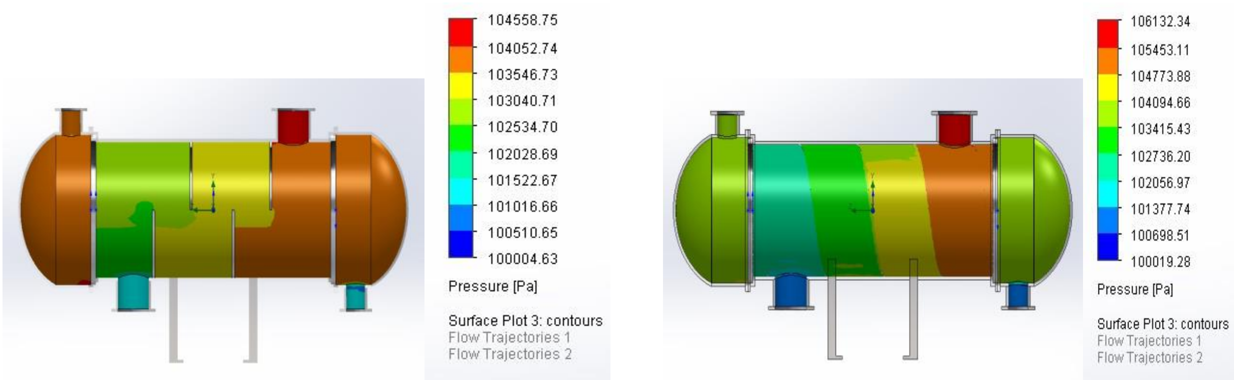


Fig. 4 *The relationship between Velocity and Heat Transfer Coefficient*

3.1.4 Pressure Distribution

The pressure on the shell side (cold fluid subdomain) dropped when the fluid passed through both baffles. As shown in Fig. 5, the single segment baffles started with high pressure (104.558 kPa) and ended with low pressure (101.522 kPa) while helical baffles started with high pressure (106.132 kPa) and dropped at the end with lower pressure than single segment baffles (101.377 kPa). However, the helical design created a more complex flow pathway for the fluid, resulting in an increase in resistance. The pressure drop was greater when using helical baffles and the benefit is that it promotes a better heat transfer coefficient than single segment baffles. The helical design induced a swirling motion in the cold fluid subdomain (Fig. 3), hence improved mixing and turbulence. This enhanced blending results in higher heat transfer efficiency between the hot and cold fluids. Although helical baffles still get similar outlet temperatures and LMTD as single segment baffles. The effect occurred due to the higher heat transfer coefficient value associated with helical baffles compensating for the lower heat transfer coefficient value of single segment baffles. A higher heat transfer coefficient value enabled efficient heat transfer even with a slightly higher pressure drop. It also demonstrated the differences in the hot fluid subdomain between the two baffles. Fig. 5 illustrates the pressure distribution in the shell and tube heat exchanger with baffles during SolidWorks 2023 CFD flow simulations.



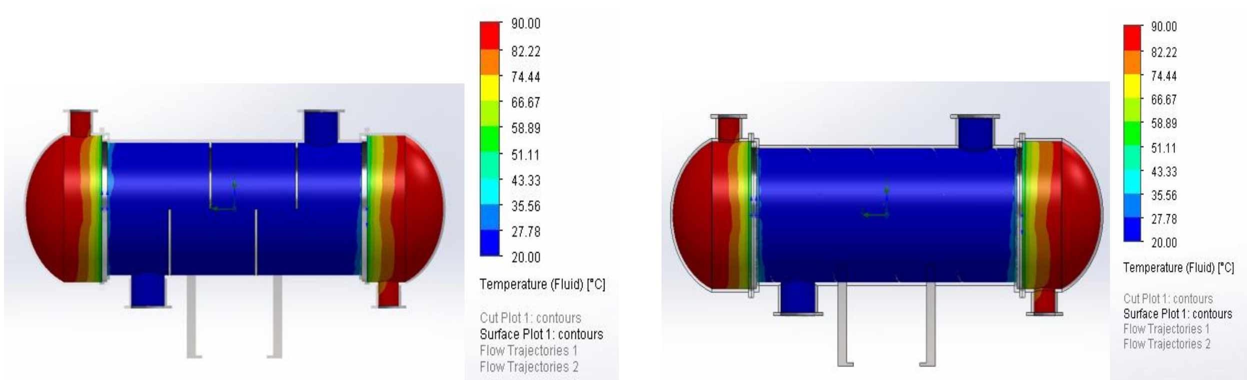
(a) (b)

Fig. 5 *The pressure distribution (a) single segment baffles; (b) helical baffles*

3.2 Heat Exchanger Simulation of Hot Fluid (Olive Oil) and Cold Fluid (Water)

3.2.1 Surface Plot

Surface plots of the fluid temperature for both baffles were generated using SolidWorks Flow Simulation. These plots enabled the investigation of the temperature distribution throughout the surface of the shell and tube heat exchanger. The olive oil achieved its maximum temperature (90 ˚C) and gradually cooled as it moved towards the exit (87 ˚C), with significant amounts reaching a lower temperature (58 ˚C), indicating effective heat transmission. Conversely, the cold fluid began at its minimum temperature (20 ˚C) and remained at that temperature throughout the exchanger. A visual comparison of these plotted with those for the hot water outlet in the orange region in Fig. 2 revealed that the surface plot effectively showed the faster cooling with olive oil that still has a red temperature region as in Fig. 6. Fig. 6 shows the surface plot of fluid temperature in the shell and tube heat exchanger with baffles.

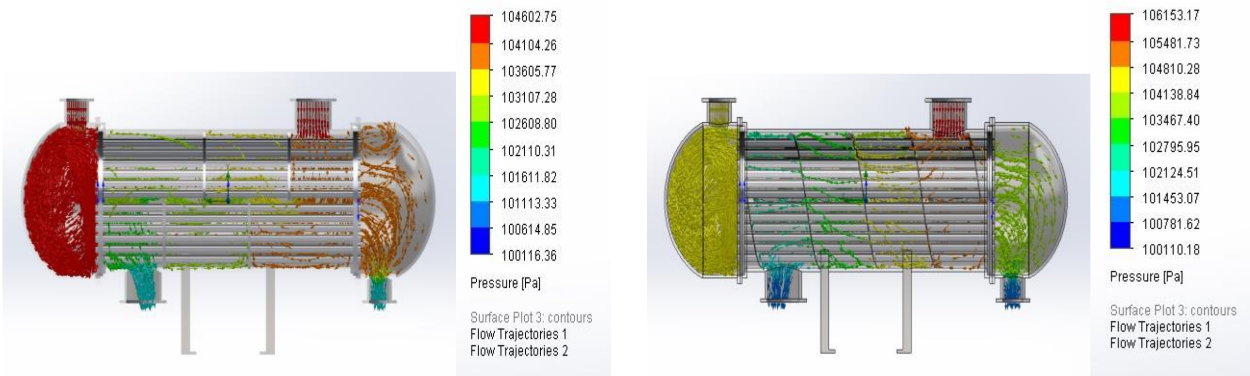


(a) (b)

Fig. 6 *The surface plot of fluid temperature (a) single segment baffles; (b) helical baffles*

3.2.2 Pressure Trajectory

SolidWorks Flow Simulation was essential for investigating the fluid dynamics inside the shell and tube heat exchanger. These visualizations revealed the influence of baffle design on pressure reduction and heat transfer efficiency improvement. Although the hot fluid direction within the tubes stayed consistent regardless of the type of baffle used, the baffles substantially impacted the movement of the cold fluid within the shell side. The variability in the path of the cold fluid directly impacts the efficiency of the interaction between the hot and cold fluids, affecting both the pressure drop and the total heat transfer rate. From Fig. 7, the pressure started entering the single segment baffles with 104.602 kPa and dropped to 101.611 kPa while the pressure entered the helical baffles with 106.153 kPa and dropped to 100.781 kPa. Fig. 7 displayed the pressure paths followed by hot and cold fluids in two different baffle arrangements: a single segment and a helical.



(a) (b)

Fig. 7 *The pressure trajectory (a) single segment baffles; (b) helical baffles*

3.2.3 Velocity vs. Heat Transfer Coefficient

The graphical representation of velocity and heat transfer coefficient data obtained from a CFD SolidWorks simulation. As depicted in Fig. 8, the velocities increase from 0.14 m/s to 0.26 m/s, the value of the heat transfer coefficient decreases from 1861.694 W/m^2/K to 860.770 W/m^2/K (single segment baffles) and 1628.029 W/m^2/ to 910.132 As W/m^2/K (helical baffles). Additionally, greater velocities increase pressure drop within the shell and tube heat exchanger [10]. Even with different types of fluids, helical baffles give the same result: a greater pressure differential enhanced the heat transfer rate as shown in Fig. 3 (b) and 7 (b). Fig. 8 displayed a graph that compares the data obtained for both baffles.

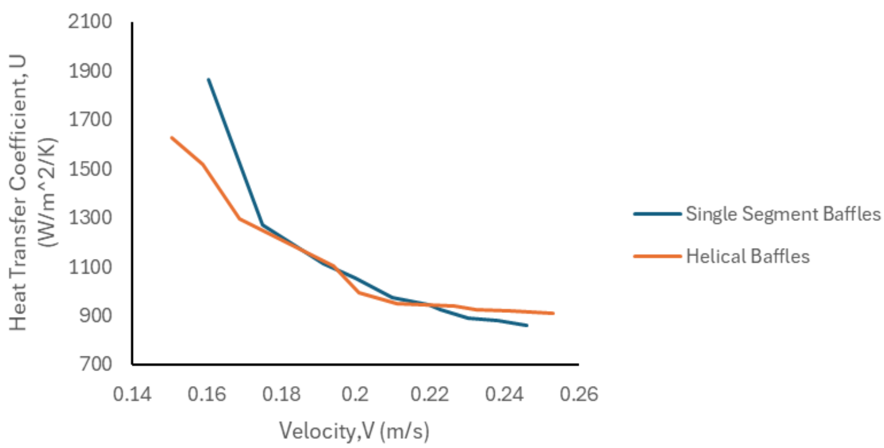
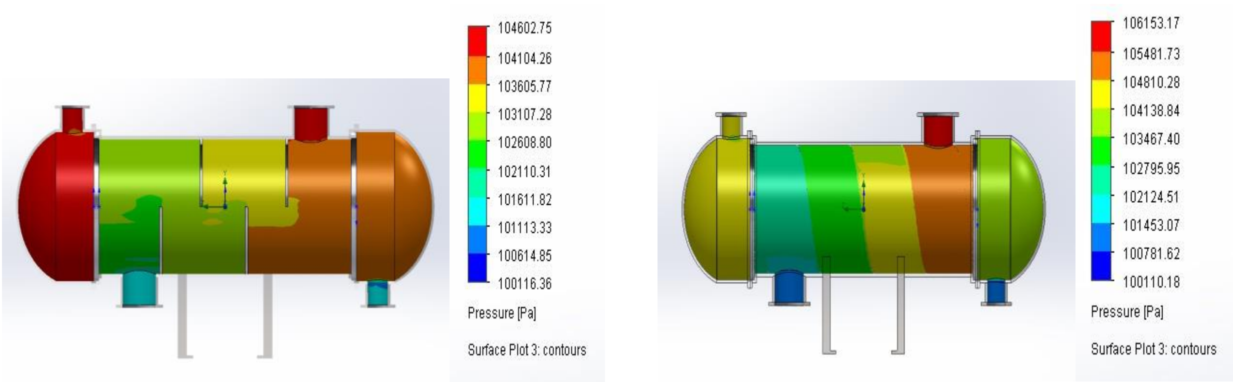


Fig. 8 *The relationship between Velocity and Heat Transfer Coefficient*

3.2.4 Pressure Distribution

Fig. 9 shows the pressure distribution in the shell and tube heat exchanger using single segment baffles. Single segmental baffles create a zigzag flow pattern that results in stagnant areas, swirls and fluid particles mixing in the opposite direction. Therefore, heat transmission is inefficient. For helical baffles, the fluid particles follow a helical route. The removal of dead zones ensured that there was no stagnant fluid and the mixing of fluid particles decreased pressure loss which led to a decrease in pumping power [7]. The pressure area distribution is similar to the distribution in Fig. 5. The pressure on the shell side (cold fluid subdomain) dropped when the fluid passed through both baffles. As shown in Fig. 9, the single segment baffles started with high pressure (104.602 kPa) and ended with low pressure (101.611 kPa) while helical baffles started with high pressure (106.153 kPa) and dropped at the end with lower pressure than single segment baffles (101.453 kPa).



(a) (b)

Fig. 9 *The pressure distribution (a) single segment baffles; (b) helical baffles*

3.3 Heat Transfer Rate

Table 4 *The heat transfer rate for water and water simulations.*

Parameter	LMTD, T_{LM} (Conversion)		Heat transfer coefficient, U (W/m^2/K)	Area, A (m^2)	Heat transfer rate, Q (kW)
	˚C	K			
Simulation 1	62.29	335.41	924.279	2.011	623.435
Simulation 2	61.40	334.55	1005.772	2.011	676.663

Table 4 illustrates the calculated heat transfer rate for water and water simulations. The first set in Table 4 tested with water (hot fluid) and water (cold fluid); Simulation 1 for single segment baffles and Simulation 2 for helical baffles. To calculate the heat transfer rate between the fluids, the equation was utilized, considering all relevant factors such as the heat transfer coefficient, transfer area and logarithmic mean temperature difference (LMTD) which were calculated beforehand. The entire heat transfer coefficient was determined using a CFD simulation in SolidWorks. The LMTD value utilized in this computation corresponds to the values of the hot and cold fluids obtained from Simulations 1, 2, 3 and 4. Next, the area of the tube was determined to determine the area through which the fluids were exchanged. Equations 1, 2, 3, 4, and 5 were used in this calculation. Simulation 1 obtained 623.435 kW from the calculation and Simulation 2 obtained 676.663 kW.

Table 5 *The heat transfer rate for olive oil and water simulations.*

Parameter	LMTD, T_{LM} (Conversion)		Heat transfer coefficient, U (W/m^2/K)	Area, A (m^2)	Heat transfer rate, Q (kW)
	˚C	K			
Simulation 3	68.00	341.15	860.770	2.011	590.528
Simulation 4	67.98	341.13	910.132	2.011	624.362

Table 5 illustrates the calculated heat transfer rate for olive oil and water simulations. The second set in Table 5 was filled with olive oil (hot fluid) and water (cold water); Simulation 3 was for single segment baffles and Simulation 4 was for helical baffles. Simulation 3 obtained 590.528 kW from the calculation and Simulation 4 obtained 624.362 kW. The calculated data in Tables 4 and 5 shows that the heat transfer rate with helical baffles was greater than single segment baffles.

4. Conclusion

The study utilized CFD flow simulation in SolidWorks 2023 to examine the thermal performance of shell and tube heat exchangers with helical baffles. It found SolidWorks 2023 to be a useful tool for creating complex 3D models while adhering to TEMA regulations. The investigation allowed a thorough examination of heat transfer rates, emphasizing the importance of selecting suitable materials and assigning properties.

The study also used computational fluid dynamics (CFD) to investigate the effect of different baffle configurations on the heat transfer rate of shell and tube heat exchangers with baffles. The results showed that helical baffles resulted in greater heat transfer rates than a single segment baffles design as shown in Tables 4 and 5. This is due to helical baffles' built-in capacity to optimize the temperature distribution between hot and cold fluids over the length of the exchanger. The CFD flow simulation also showed increases in heat transfer rates due to heat transfer coefficient values. The value of the heat transfer rate obtained from the calculation for Simulation 2 (helical baffles) was 676.663 kW which was greater than Simulation 1 (single segment baffles) 623.4353 kW and Simulation 4 (helical baffles) which was 624.362 kW which is also greater than Simulation 3 (single segment baffles) was 590.528 kW.

The study concluded that the investigation effectively achieved all its objectives, despite new factors affecting outlet temperature and LMTD. Enhancing fluid dynamics within the heat exchanger can improve heat transfer efficiency, demonstrating progress in improving heat transfer rates between single segment and helical baffles.

Acknowledgement

The authors expressed their gratitude to the Faculty of Mechanical and Manufacturing Engineering at the Universiti Tun Hussein Onn Malaysia for providing the opportunity and support to conduct this investigation. This research would not have been possible without their resources and guidance.

References

[1] Ghajar, A. J., & Dr, Y. a. C. (2014). *Heat and Mass Transfer: Fundamentals and Applications*. McGraw-Hill Education.

[2] Khatoon, B., Choudhary, V. K., Kumar, R., Alam, M. S., Pandey, S. K., Singh, R., & Naresh, R. (2022). Enhancement of heat transfer rate in shell & tube heat exchanger using CuO/Al2O3-water based nanofluids. *Materials Today: Proceedings*. https://doi.org/10.1016/j.matpr.2022.10.258

[3] Al-Darraji, A. R., Marzouk, S., Aljabr, A., Almehmadi, F. A., Alqaed, S., & Kaood, A. (2024). Enhancement of heat transfer in a vertical shell and tube heat exchanger using air injection and new baffles: Experimental and numerical approach. *Applied Thermal Engineering*, *236*, 121493. https://doi.org/10.1016/j.applthermaleng.2023.121493

[4] Prasad, N. a. K., & Anand, N. K. (2020). Design Analysis of Shell Tube Type Heat Exchanger. *International Journal of Engineering Research and Technology*, *V9*(01). https://doi.org/10.17577/ijertv9is010215

[5] Aydin, A., Yaşar, H., Engin, T., & Büyükkaya, E. (2022). Optimization and CFD analysis of a shell-and-tube heat exchanger with a multi segmental baffle. *Thermal Science/Thermal Science*, *26*(1 Part A), 1–12. https://doi.org/10.2298/tsci200111293a

[6] Selbaş, R., Kızılkan, N., & Reppich, M. (2006). A new design approach for shell-and-tube heat exchangers using genetic algorithms from economic point of view. *Chemical Engineering and Processing*, *45*(4), 268–275. https://doi.org/10.1016/j.cep.2005.07.004

[7] Bichkar, P., Dandgaval, O., Dalvi, P., Godase, R., & Dey, T. (2018). Study of Shell and Tube Heat Exchanger with the Effect of Types of Baffles. *Procedia Manufacturing*, *20*, 195–200. https://doi.org/10.1016/j.promfg.2018.02.028

[8] Cartaxo, S. J., & Fernandes, F. A. (2011). Counterflow logarithmic mean temperature difference is actually the upper bound: A demonstration. *Applied Thermal Engineering*, *31*(6–7), 1172–1175. https://doi.org/10.1016/j.applthermaleng.2010.12.015

[9] El-Said, E. M., & Al-Sood, M. A. (2019). Shell and tube heat exchanger with new segmental baffles configurations: A comparative experimental investigation. *Applied Thermal Engineering*, *150*, 803–810. https://doi.org/10.1016/j.applthermaleng.2019.01.039

[10] Naqvi, S., Elfeky, K., Cao, Y., & Wang, Q. (2019). Numerical analysis on performances of shell side in segmental baffles, helical baffles and novel clamping anti-vibration baffles with square twisted tubes shell and tube heat exchangers. *Energy Procedia*, *158*, 5770–5775. https://doi.org/10.1016/j.egypro.2019.01.553