Efficiency Analysis of Full Rotation Counter Flow of Shell and Tube Heat Exchanger using CFD Tool

Muhammad Naim Ramli[1], Norrizal Mustaffa[1*]

¹ *Department of Mechanical Technology, Faculty of Mechanical and Manufacturing Engineering,*
   Universiti Tun Hussein Onn Malaysia, Parit Raja, 86400 Johor, MALAYSIA

*Corresponding Author: norrizal@uthm.edu.my

Article Info

Abstract

A heat exchanger (shell and tube) is an important piece of equipment and is often used in industries, especially those involving processes. This equipment can be used for both heating and cooling systems. This research aims to investigate the correlation between the arrangement of baffles in heat exchanger design with the resultant temperature differences (delta T) and analyse the flow dynamics inherent in the full rotation cross-flow design. The configuration made in this study is a baffles arrangement that is different from the conventional segmental design. Modelling and simulation were done in Ansys DesignModeler and Ansys FLUENT. Different heat exchangers were compared, which are segmental, full rotational, and helical heat exchangers. The segmental baffles were arranged in a zigzag pattern. This design will produce less heat transfer between the two mediums because recirculation zones will form in front of the baffles. The helical design could solve the issues, but the fabrication process was quite challenging. The full rotation heat exchanger design tries to combine the helical features into the segmental design, reducing recirculation zones by creating the swirl effect. The result of the full rotation configuration can reduce the negative effects of the existence of recirculation zones but cannot completely cope with the same as the helical heat exchanger. The helical has a better heat transfer rate than the full rotation, and the segmental heat exchanger was the least efficient.

1. Introduction

Heat exchangers play an important part in operations such as electricity generation, oil refining, HVAC systems, food processing, and many other industrial applications by applying heat transfer principles [1]. Heat exchangers commonly used in industry are plate heat exchangers, tube-in-tube heat exchangers, shell and tube heat exchangers and double-tube heat exchangers. Shell and tube heat exchangers are essential components that are widely utilized in a wide range of industries and sectors of work to assist efficient heat transfer between two or more fluid streams.

2. Methodology

This part will cover the explanation of the methodology that is being used to achieve the research objective. This analysis will be conducted using CFD software, ANSYS, that can simulate heat transfer. All the parameters were

taken from past research [2]. Detailed 3D models must then be created, and conditions for accurate simulations must be determined.

2.1 Heat exchanger 3-Dimension (3D) geometric models

The 3D models of the three heat exchangers below were designed using AnsysModeler software. Modelling in Ansys becomes simpler because only focuses on the important parts.

2.1.1 3D model of full rotation heat exchanger

The diameter of the shell side is 161 mm, and the tube side is 17 mm each and in a hexagonal arrangement with a total length of 1400 mm. This configuration arrangement is at 90°,135°,180, 225° ,270° and 315° at z-axis. The distance between the baffles is 193 mm. For detail, refer to Fig. 1.
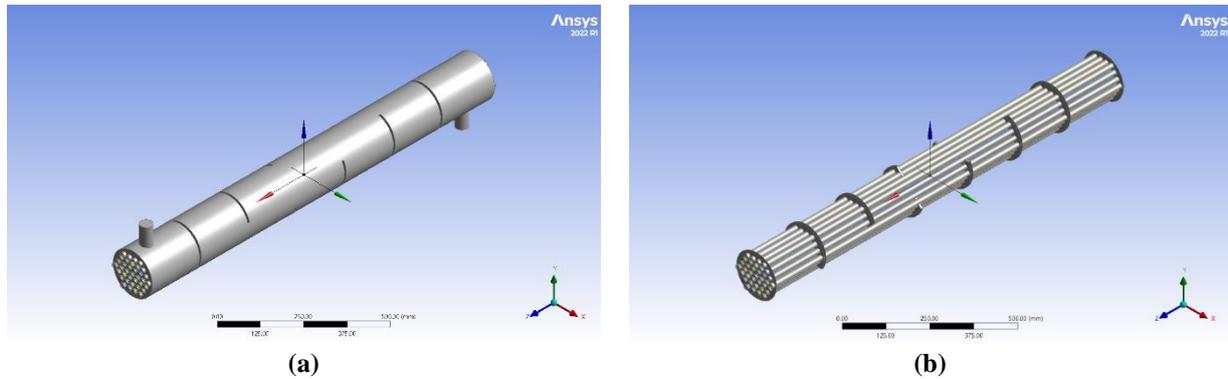
(a) **(b)**

Fig. 1 *3-D Model of full rotation in DesignModeler (a) Shell side; (b) Tube side*

2.1.2 3D model of segmental heat exchanger

The difference is the way the baffle is arranged. The segmental is arranged at the upper and bottom sides that will, creating a zigzag fluid flow as shown in Fig. 2. The number of baffles was 6, and the distance of each of them was 193 mm.
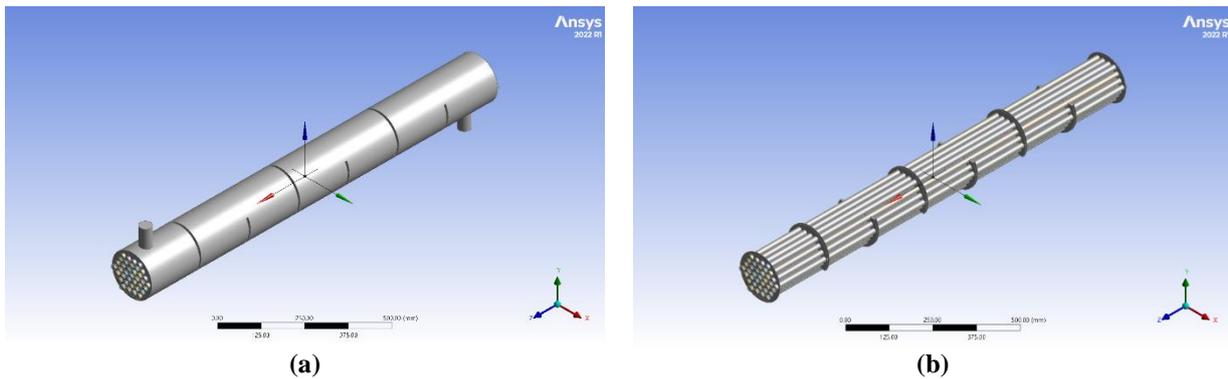
(a) **(b)**

Fig. 2 *3-D Model of segmental in DesignModeler (a) Shell side; (b) Tube side*

2.1.3 3D model of full rotation heat exchanger

The helical design does not use a number of baffles because it uses continuous baffles. The difference is the use of the number of turns to form a helical shape. Fig. 3 show there were 6 turns that used on helical baffles.
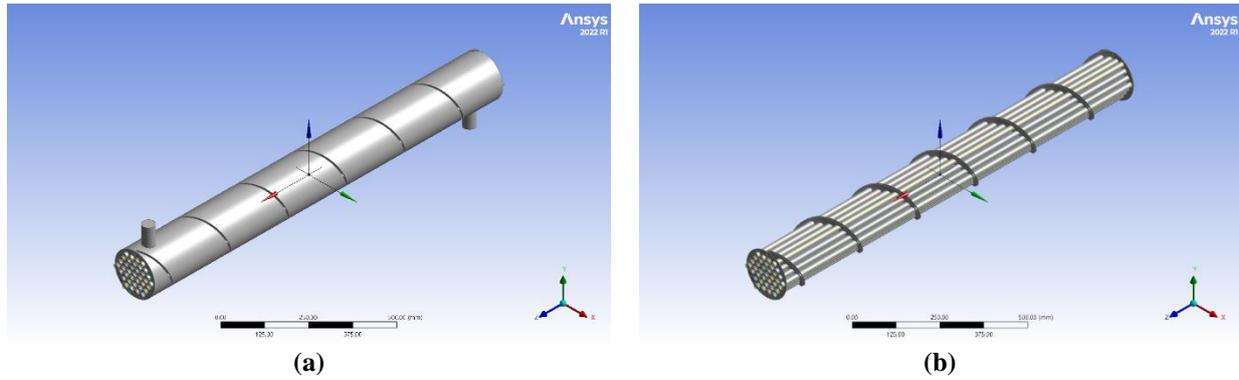
(a)   **(b)**

Fig. 3 *3-D Model of helical in DesignModeler (a) Shell side; (b) Tube side*

2.2 Meshing Geometry

Meshing methods are used to produce grid point collections that determine the accuracy of a CFD simulation. Entering the appropriate element size that obtained from the grid independence test. The selection part will determine the inlet and outlet for both hot water and cold water. The cold water will enter the shell side while the hot water inlet enters the tube section.

2.3 Setup and Solution Geometry

K-epsilon was the most widely used computational fluid dynamics (CFD) model for simulating flow characteristics under turbulent flow situations. K-epsilon (2 eqn) was chosen as a turbulent model [1,2,3,4]. Create or add new material in Fluent database material, water-liquid (h2o < l >) as a fluid selection that has been used in this simulation. At the boundary conditions, the parameters' values, such as velocity input and temperature input, are entered for both the shell and tube sides [2].

Table 1 *Input Parameters*

Location (input)	Temperature (^{o}C)	Velocity (ms^{-1})
Shell	20.0	0.38
Tube	122.5	0.80

After all the parameters have been entered and set, the model must be analysed in the initialization process. This initialization is crucial for defining the initial conditions, material properties, boundary conditions, and other relevant settings that govern the behaviour of the simulation. Failed initialization shows that there is no problem with part selection, mesh, or boundary condition. After the initialization succeeds, click Calculate to generate the result.

2.4 Grid Independent Test (GIT)

Based on Table 2, the grid independence test, where different element sizes were evaluated (0.8 m, 0.7 m, and 0.65 m) with an increment of 100,000 elements per iteration, shows that sample 2 stands out as the most suitable mesh configuration. Sample 2 was selected as the best mesh configuration, demonstrating that mesh resolution and computational efficiency were the most balanced. GIT 2 produced accurate predictions and trustworthy simulation results despite not having the best resolution of all the tested configurations.

Table 2 *Grid Independence Test*

Number of GIT	Element size (m)	Element
1	0.8	348, 056
2	0. 7	457, 399
3	0.65	577, 655

2.5 Graphical presentation

The simulated domain's behaviour of the modelled system is portrayed through these graphical representations, which illustrate how different parameters change across the simulated domain [5].

2.5.1 Contour display parameters

Describing different ways to change contour plots, like using colour maps, contour levels, and range settings. Fig. 4 shows an example of the contour display parameter setting. Explaining the changes made makes it easier to see and understand the results of the simulation.

Fig. 4 *Details of Contour Setting*

2.5.2 Streamline display parameters

Deciding where streamlines begin can help visualize the movement of fluids. Selecting different entry points makes it easier to identify fluid flow patterns. Knowing what to focus on is important to see how things move in a fluid. This helps understand how different parts of a system influence the behaviour of the fluid.
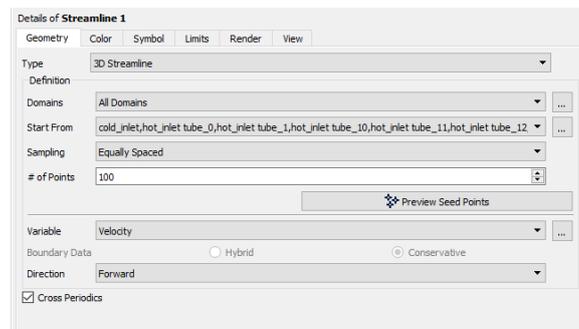
Fig. 5 *Details of Streamline Setting*

2.5.3 Volume rendering parameters

The volume rendering parameters can adjust various settings to improve the visualization of these flow properties. These parameters include colour mapping, opacity, transfer functions and lighting effects, which can be adjusted to highlight specific flow characteristics or areas of interest within the simulation.
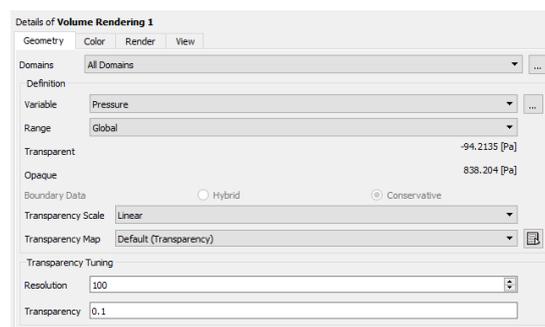
Fig. 6 *Volume Rendering Display Parameter Setting*

3. Result and Discussion

Inputs and parameters in all three models are similar to each other. Back to the objective of this project, to study the relationship of baffle design configuration with the delta T. The temperature difference is significant output data in this study. Besides, to analyse the flow characteristic of full rotational cross flow design. This chapter will prove the objective of this project has been achieved.

3.1 Comparative thermal performance of diverse heat exchanger types

Table 3 shows that the helical heat exchanger obtains the highest temperature output on the shell side, which is 70.36 °C, followed by full rotation baffles at 67.12 °C. The segmental heat exchanger is the lowest at 65.4 °C. The temperature differences for full rotation were 47.12 °C, higher than the segmental which is 45.4 °C. This shows a quiet lot of heat transfer occurs in the full rotation heat exchanger.

Table 3 The Temperature Obtained at Shell Side

Types of heat exchanger	Segmental heat exchanger	Helical heat exchanger	Full rotational heat exchanger
Temperature Location	Temperature		
Input	20 °C	20 °C	20 °C
Output	65.4 °C	70.36 °C	67.12 °C
ΔT	45.4 °C	50.36 °C	47.12 °C

Table 4 show the temperature output at the tube side. The three of heat exchangers had similar temperature input at 122.5 °C. At the tube side, the difference seems insignificant. The difference was about 1 °C to 2 °C. The use to raise the temperature on the cold fluid medium is more suitable for this small size of heat exchanger.

*Table 4* The Temperature Obtained at Tube Side

Types of heat exchanger	Segmental heat exchanger	Helical heat exchanger	Full rotational heat exchanger
Temperature Location	Temperature		
Input	122.5 °C	122.5 °C	122.5 °C
Output	116.58 °C	115.07 °C	115.98 °C
ΔT	5.92 °C	7.43 °C	6.52 °C

### 3.2 Contour plots of the temperature distribution.

Contour simulations, as shown in Fig. 7 for the shell sides of the heat exchangers, are crucial for understanding and optimising its performance. Within the exchanger, these models visually illustrate the temperature distributions.

Based on the contour plot, the shell side is where the cold-water input is entered at the temperature of 20 °C. Dark blue colour represents the lowest temperature in the range of 18.70 °C to 24.99 °C. The shift from cool hues to warmer tones visually demonstrates the movement of thermal energy. This movement aligns with the basic principles of heat transfer convection [7]. In the simulation, the cold water (shell side), shown by the darker blue, undergoes a heat exchange process with the hotter water (tube side).
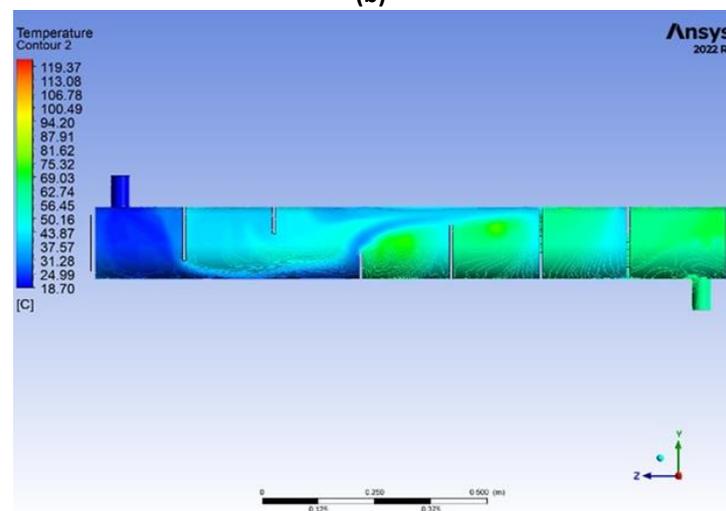
The cold water absorbs the heat from the tube side and increases its temperature through the convection process. A warmer tone (cyan-greenish) is shown at the cold-water output as the temperature has risen. The temperature at the output is in the range of 62.74 °C to 71.00 °C. This change may be affected by the time travel and the distance [8].
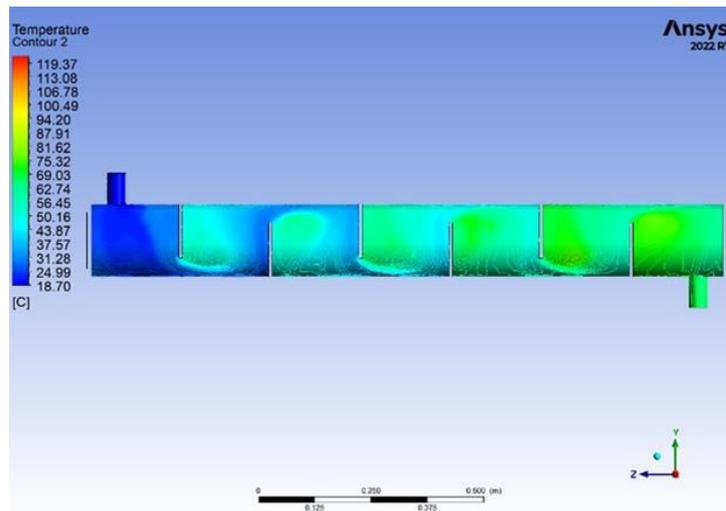
(a)
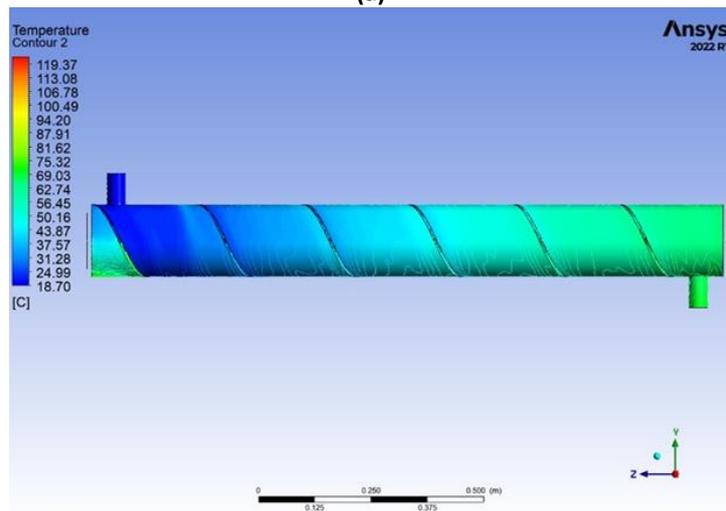
(b)

(c)

Fig. 7 *Temperature Contour on The Shell Side (a) Segmental; (b) Helical; (c) Full rotation*

3.3 Streamline visualization of the fluid flow

Fig. 8 shows the streamlined visualization of the models that also show the fluid flow inside. The full rotation baffles arrangement describes the fluid flow quite differently from segmental baffles, which do a fully zig-zag pattern [9]. The fluid flow pattern forms as if it were a swirl pattern that helps reduce the recirculation zones. This effect is especially advantageous because it increases the fluid's momentum, which prevents the formation of low-velocity regions [10]. By adding more energy to the flow, the swirl effect keeps the flow moving more continuously and dynamically, lessening the chance of fluid stagnation and recirculation zone formation. Because of the baffle's configuration arrangement, the recirculation zone can be reduced. Minimizing the recirculation zones is important to increase the heat transfer and reduce the pressure drop inside the shell side [11].

In other cases of helical baffle, these recirculation zones are never formed. The flow was created as a beautiful swirl flow. The temperature is distributed uniformly inside the helical heat exchanger. Swirling flow can create vortices or eddies within the shell, affecting pressure and temperature distribution [11]. High-velocity swirling flows might lead to the formation of vortices, occur low-pressure drops at specific locations and potentially influence the heat transfer [4].
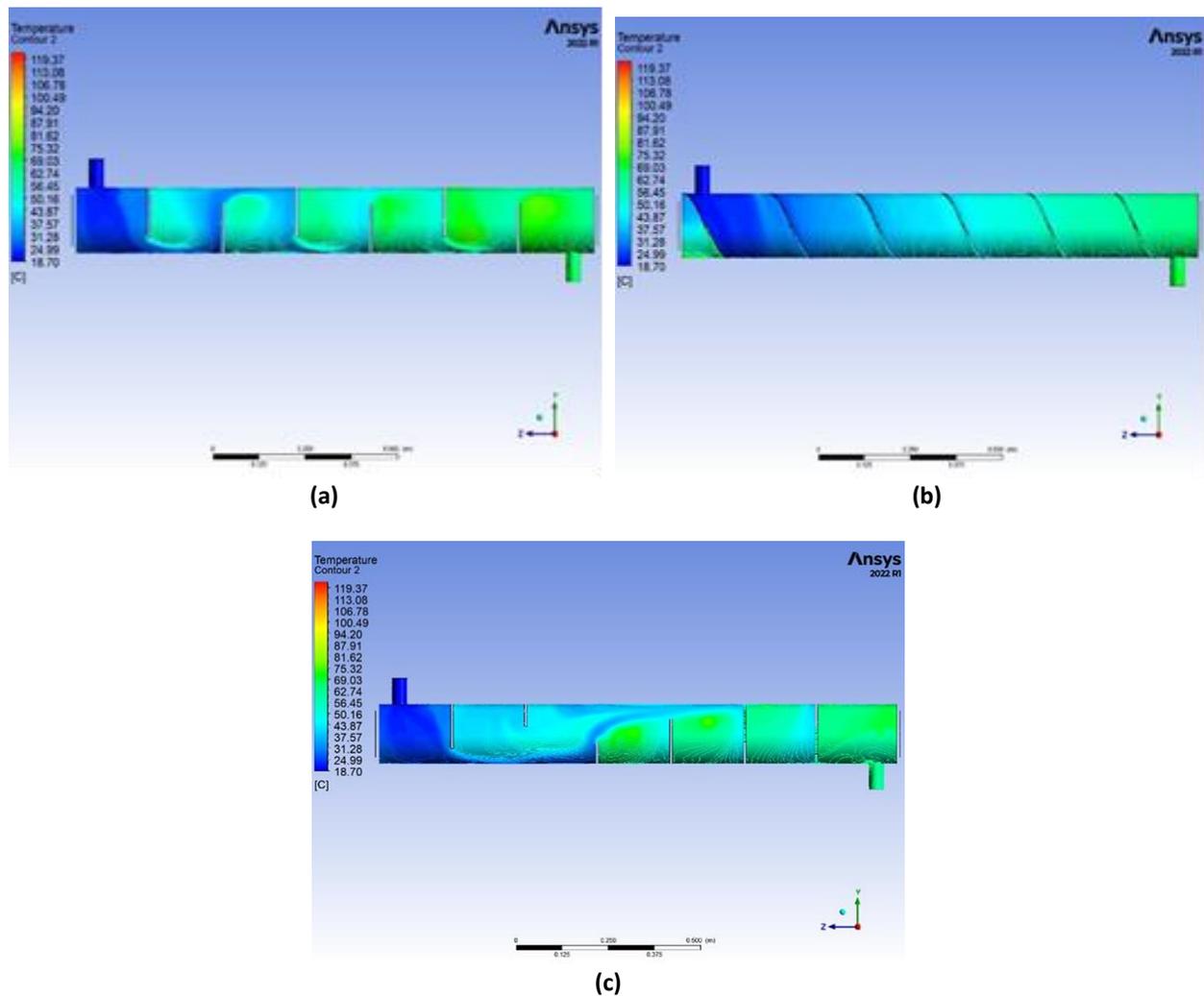

(a)

(b)

(c)

Fig. 8 *Temperature Streamline within The Models (a) Segmental; (b) Helical; (c) Full rotation*

4. Conclusion

The findings suggest that the helical heat exchanger demonstrated superior heat transfer capabilities compared to both the full rotation and segmental heat exchangers. Specifically, the helical design effectively managed heat transfer between hot and cold fluids, showcasing optimized performance. The segmental heat exchanger design showed inefficiencies in heat transfer due to the presence of recirculation zones induced by its zig-zag flow configuration. In contrast, the full rotation heat exchanger exhibited improved performance over the segmental

design by reducing recirculation zones. Its S-shaped flow pattern played a key role in minimizing recirculation effects, offering enhanced heat transfer efficiency.

The results emphasize the promising potential of the full rotation heat exchanger in lessening the recirculation zones and improving heat transfer efficiency compared to conventional segmental designs. Although the helical heat exchanger outperformed the full rotation and segmental designs in this study, the full rotation configuration showed significant promise in addressing the shortcomings of recirculation zones.

4.1 Guidelines

The research follows clear steps to analyze the efficiency of a full rotation heat exchanger using CFD tool. It uses ANSYS to simulate fluid flow and heat transfer accurately. The process includes creating the mesh, setting boundaries, configuring the solver, and post-processing, following standard CFD practices.

Acknowledgement

References

[1] Kumar Gupta, R., & Pawar, D. (2021). Design and Analysis of Shell and Tube Heat Exchanger using different tubes materials (Vol. 8). www.jetir.org

[2] Aydin, A., Yaşar, H., Engin, T., & Büyükkaya, E. (2020). Optimization and CFD Analysis of A Shell-and-Tube Heat Exchanger with A Multi Segmental Baffle.

[3] Kishan, R., Singh, D., & Sharma, A. K. (2020). CFD Analysis of Heat Exchanger Models Design Using Ansys Fluent. *International Journal of Mechanical Engineering and Technology (Ijmet)*, *11*(2). https://doi.org/10.34218/ijmet.11.2.2020.001

[4] Li, Y., Wu, J., Wang, H., Kou, L., & Tian, X. (2012). Fluid Flow and Heat Transfer Characteristics in Helical Tubes Cooperating with Spiral Corrugation. *Energy Procedia*, *17*, 791–800. https://doi.org/10.1016/j.egypro.2012.02.172

[5] Howard H. HU. (1987). *FLUID MECHANICS FIFTH EDITION*. www.AnnualReviews.org.

[6] Gowthaman, P. S. (2013). Analysis of Segmental and Helical Baffle in Shell and tube Heat Exchanger. *International Journal of Current Engineering and Technology*, *2*(2), 625–628. https://doi.org/10.14741/ijcet/spl.2.2014.119

[7] Boz, Z., Erdogdu, F., & Tutar, M. (2014). Effects of mesh refinement, time step size and numerical scheme on the computational modelling of temperature evolution during natural-convection heating. *Journal of Food Engineering*, *123*, 8–16. https://doi.org/10.1016/j.jfoodeng.2013.09.008

[8] Hernandez, H. (2019). *On the Relationship between Time and Temperature*. https://doi.org/10.13140/RG.2.2.36103.34727

[9] Wang, Q., Chen, G., Chen, Q., & Zeng, M. (2010). Review of Improvements on Shell-and-Tube Heat Exchangers with Helical Baffles. *Heat Transfer Engineering*, *31*(10), 836–853. https://doi.org/10.1080/01457630903547602

[10] Zitek Pavel, & Valenta Vaclav. (2015). Various methods to improve heat transfer in heat exchangers. EPJ web of conference, EDP Sciences, 92, 02119, https://doi.org/10.1051/epjconf/20159202119

[11] Ahmed, A., Ferdous, U., & Saha, S. (2017). Comparison of Performance of Shell-and-Tube Heat Exchangers with Conventional Segmental Baffles and Continuous Helical Baffle.