

# Analysis of Heat Transfer in Shell and Tube Heat Exchanger (STHE) using Counter and Parallel Flow

Muhammad Danial Aiman Zaidi Mohd Yazid<sup>1</sup>, Nor Mazlana Main<sup>1\*</sup>

<sup>1</sup> Faculty of Mechanical and Manufacturing Engineering,  
Universiti Tun Hussein Onn Malaysia, 86400, Parit Raja, Batu Pahat, Johor,  
MALAYSIA

\*Corresponding Author: [mazlana@uthm.edu.my](mailto:mazlana@uthm.edu.my)

DOI: <https://doi.org/10.30880/rpmme.2024.00.05.01.053>

## Article Info

Received: 15 January 2024

Accepted: 15 July 2024

Available online: 15 Sept. 2024

## Keywords

Heat Exchanger, Shell and Tube,  
Parallel Flow, Counter Flow, CFD  
analysis, Solidworks

## Abstract

A heat exchanger is a device specifically engineered to transfer thermal energy between two fluids with different temperatures. A shell and tube heat exchanger (STHE) is a widely used heat exchanger that facilitates the transfer of heat between a gas or liquid medium. The efficiency of heat exchangers is vital in the thermal industry. This study aims to investigate the efficiency of shell and tube heat exchangers, with a particular focus on parallel and counter-flow arrangements when two different fluids are utilized. Design and simulations are conducted using SolidWorks2020 and computational fluid dynamics (CFD), taking into consideration operational parameters such as flow rates, inlet temperatures, fluid properties, boundary conditions and others. The results reveal that counter-flow produces a higher temperature difference, improving heat transfer efficiency near the outlet. However, calculated values indicate higher heat transfer in parallel flow, contradicting graphical results, which are 86,073.65W for counter flow and 94,975.46W for parallel flow. This discrepancy may stem from various factors. The practical design adheres to Tubular Exchanger Manufacturers Association (TEMA) requirements, providing useful insights into the thermal efficiency of STHE.

## 1. Introduction

STHE is widely preferred in industrial operations due to its exceptional heat transfer capabilities, which are further enhanced through the use of analytical and computational approaches [1]. STHEs are essential components in a wide range of sectors, such as chemical, pharmaceutical, petrochemical, oil and gas, refrigeration, air conditioning, and power generation [2]. STHE are available in several designs, including a variety of shells, tube bundles, and front and rear-end head structures [3]. The Tubular Exchanger Manufacturer's Association (TEMA) is a recognized industry standard for the construction of heat exchangers. TEMA employs a three-letter notation system to classify various types of STHE based on the types of front-end head, shell, and rear-end head [4]. The instrument functions as a closed system, allowing thermal energy transfer while preventing the physical mixing of fluids. The standard heat exchanger operates based on the idea of heat transfer, which can be achieved through either a parallel or counter-flow configuration [5]. The parallel flow arrangement refers to the situation where two fluids move in the same direction, and both reach the heat exchanger at one end [5], while the counter flow arrangement occurs when fluids enter the heat exchanger at opposite ends and flow in opposite directions to one other [6]. Researchers in the field of Heat Exchangers can utilize Computational Fluid Dynamics (CFD) to develop their systems [7]. CFD simulations enable engineers to utilize virtual modeling tools and acquire precise solutions for evaluating heat exchanger systems [8]. Choosing suitable mathematical and physical models that

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



accurately correspond to the real conditions is crucial when performing computational fluid dynamics (CFD) simulations [9]. The logarithmic mean temperature difference (LMTD) is a crucial factor in heat exchangers' theoretical and practical aspects. Their design incorporates several aspects, such as performance computation, steady-state analysis, dynamic modelling, simulation, stability-limit analysis, and control synthesis [10]. The LMTD approach is applicable when the temperatures at the inlet and outlet of both the hot and cold fluids are known [4]. The formula for LMTD was given by:

$$T_m = \frac{(T_1 - T_2)}{\ln\left(\frac{T_1}{T_2}\right)} \quad (1)$$

Where the  $\Delta T_1$  and  $\Delta T_2$  depends on the flow involves in the STHE.

For counter flow,

$\Delta T_1 = T_{h, \text{enter}} - T_{c, \text{exit}}$  and  $\Delta T_2 = T_{h, \text{exit}} - T_{c, \text{enter}}$

For parallel flow,

$\Delta T_1 = T_{h, \text{enter}} - T_{c, \text{enter}}$  and  $\Delta T_2 = T_{h, \text{exit}} - T_{c, \text{exit}}$

The effectiveness of heat exchangers is crucial in the thermal industry. When dealing with thermal issues, it is crucial to carefully choose a combination of characteristics that will result in high efficacy [11]. Chand *et al.* (2019) used Ansys Fluent software for realistic calculations, examining temperature drops in STHE with transformer oil as hot and water as cold. Models of both parallel flow and counter flow heat exchangers were created and simulated to conduct a comprehensive examination. In this research, a practical STHE will be designed using SolidWorks 2020 to analyze the parallel and counter flow by using the flow simulation tools. Heat transfer rate calculations will be performed using mathematical equations, with numerical processes based on equations mentioned by Taware *et al.* (2017). The fluid involved in this study was water as cold fluid and olive oil as hot fluid.

## 2. Material and Methods

The computational fluid dynamic analysis in this study comprises three main simulation tasks: pre-processing, processing, and post-processing. The three-step approach is iterated multiple times using various boundary conditions and process parameters to generate a set of findings that can be compared and analyzed. The software employed in this study for these methods is SolidWorks, as it allows for the creation and simulation of models using flow simulation.

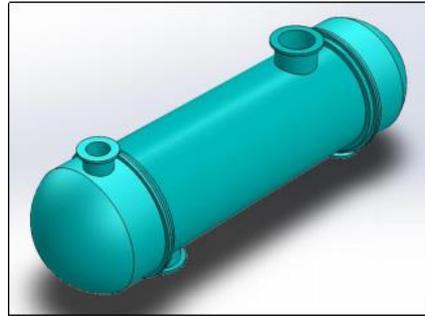
### 2.1 Pre-processing

The initial stage of pre-processing involves creating a model of the shell and tube heat exchanger domain by accurately specifying and generating its geometry and corresponding mesh. For the geometry model, the shell and tube heat exchanger (STHE) is a modified version of Asadbeigi *et al.* (2023) experimentally validated work, which was developed in accordance with the Tubular Exchanger Manufacture Association (TEMA) guidelines. Table 1 provides a concise overview of the overall structure and specific design characteristics of the model.

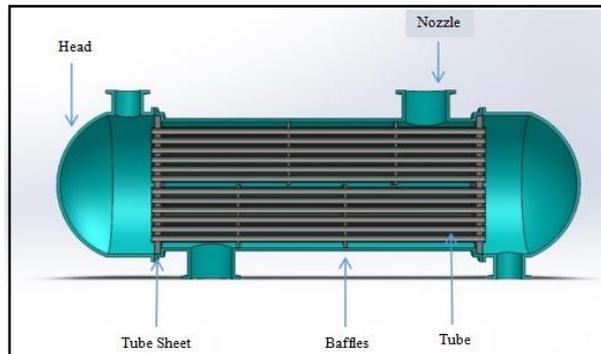
**Table 1** Geometric parameters of shell and tube heat exchanger

Parameter	Value	Unit
Shell diameter	400	mm
Inner diameter of tubes	19	mm
Outer diameter of tubes	25	mm
Number of tubes	124	pcs
Number of baffles	4	pcs

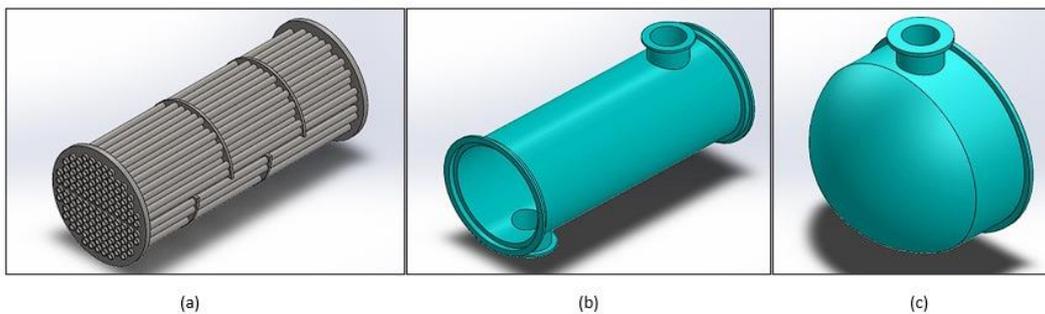
The heat exchanger illustrated in Fig. 1 is a scaled-down version designed in SolidWorks 2020 while Fig. 2 shows the cross-section area of the model. The design is known as the fixed tube heat exchanger. It comprises six components: head, shell, baffles, tube sheets, nozzle and tube bundles as shown in Fig. 3.



**Fig. 1** 3D model of SHTE



**Fig. 2** Cross section of the SHTE



**Fig. 3** (a) Baffles and tube, (b) Shell and (c) Head

The precision of a computational fluid dynamics (CFD) simulation is often strongly correlated with the quality of the mesh. A tiny mesh with a precisely organized arrangement might produce more accurate results, particularly in locations with complex flow patterns or slopes.

## 2.2 Processing

The processing stage ensues, wherein the problem's governing equations, fluid properties, lid creation, fluid subdomains, boundary conditions and goals are determined.

### 2.2.1 Governing equations

The analysis of the results in this study is conducted using the fundamental equation of the heat exchanger, referred to as equation (2). LMTD, or Logarithmic Mean Temperature Difference, is the term used to describe the average temperature difference between the hot and cold feeds at each end of the heat exchanger. This is the driving force behind the heat transfer between the two fluids in a single-pass heat exchanger.

$$Q = U \times A \times T_m \quad (2)$$

Where,

U = the overall heat transfer coefficient,

A = total area of heat transfer and

T<sub>m</sub> = Logarithmic Mean Temperature Difference (LMTD)

### 2.2.2 Fluid Properties

The performance and effectiveness of heat exchangers are influenced by the density, viscosity, thermal conductivity, and specific heat of the fluids used. Table 2 contains the parameters mentioned in this work, together with additional specifications for hot and cold fluids. These parameters are applied to both parallel and counter flow simulation.

**Table 2** Specific hot and cold fluids parameters

Parameter	Tube Side (olive oil)	Shell Side (water)
Inlet temperature (T <sub>h,in</sub> , °C)	180°C	mm
Inlet temperature (T <sub>c,in</sub> , °C)	19	40°C
Density (kg/m <sup>3</sup> )	916	998
Fluid flow rate (kg/s)	2	1
Heat Specific (J/g. °C)	2	4.19
Operating pressure (bar)	1.013	1.013
Ambient temperature	27°C	27°C
Material	Stainless steel	Stainless steel

### 2.2.3 Lid Creation

All parameters and boundary conditions must be defined before conducting the simulation to ensure that there are no errors when the simulation begins. A lid must be created first under the flow simulation tab to fill the heat exchanger with fluid. This phase is crucial because it determines whether the fluid entering the heat exchanger through the nozzle fills all the available space in the design.

### 2.2.4 Fluid Subdomains

The fluid subdomains of the fluids that enter the nozzles must then be defined. In this section, two nozzles (fluid inlet) will be chosen, and the temperatures of the hot and cold fluids will be set to 180°C and 40°C, respectively. The hot fluid will enter through the same nozzle, while the cold fluid will enter through a different nozzle depending on whether the flow is parallel or counter.

### 2.2.5 Boundary Conditions

The fluid's boundary condition, such as mass flow rate, can then be defined using Table 2, which is 2 kg/s for hot fluid and 1 kg/s for cold fluid for the inlet. The hot and cold outlets, on the other hand, will be set to environmental pressure.

### 2.2.6 Goals

This setting is used to specify the desired outcome. The result will be achieved by the employment of 4 goals, which are hot and cold temperatures for the inlet and outlet of the nozzle.

### 3. Result and Discussion

#### 3.1 Post Processing

##### 3.1.1 Counter Flow Result

Fig. 4 and Fig. 5 illustrate the graph of fluid temperature versus simulation repetition (iteration) to determine the probable outcomes for the counterflow. This graphical representation depicts the temperature distribution of hot and cold fluids across the heat exchanger. As the fluids travel in opposite directions via the heat exchanger, the temperature of the hot fluid drops, and the temperature of the cold fluid rises. This temperature change occurs gradually and continuously along the length of the heat exchanger. The temperature of the hot fluid should be equal to (or almost equal to) the temperature of the cold fluid, according to the graph reported by Abeykoon [6], however this does not occur in this study. This is due to the total number of simulation iterations is just 300, despite the large temperature differential between the inlet hot and cold fluids.

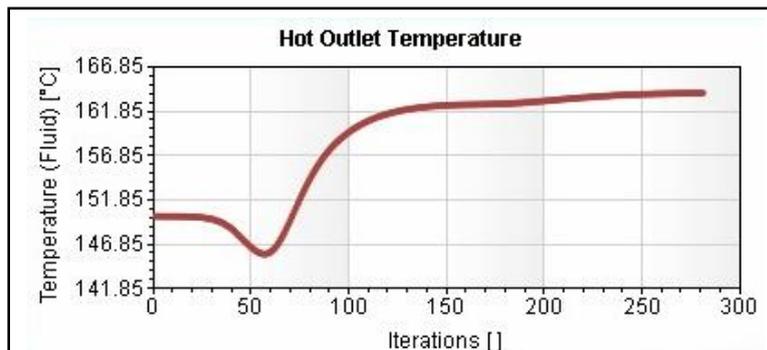


Fig. 4 Hot outlet temperature for counterflow

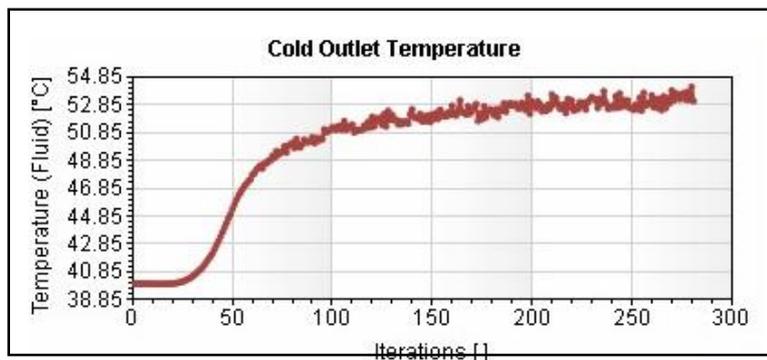


Fig. 5 Cold outlet temperature for counter flow

Table 3 shows the average, minimum and maximum value of the temperature outlet for the counter flow. According to Table 3, the simulation was a success because the progress could be reached to 100% without any errors or crashes. The maximum and minimum values from Table 3 show the simulation's highest and lowest potential values. In this study, the average that was use as the result and the temperature difference of hot and cold fluid is 29.7°C and 23.77°C respectively. This result indicates that the hot fluid is losing more heat energy to the cold fluid which demonstrates effective heat absorption by the cold fluid.

Table 3 Overall result gained for the counterflow

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
V max	[m/s]	0.6566137	0.657297708	0.652699536	0.662932527	100
Hot Inlet Temperature	[°C]	180	180	180	180	100
Hot Outlet Temperature	[°C]	150.29061	150.3047965	150.15426	150.8199067	100
Cold Inlet Temperature	[°C]	40	40	40	40	100
Cold Outlet Temperatre	[°C]	64.004296	63.7665262	63.30191138	64.41532359	100

### 3.1.2 Calculation for Counter Flow

To compute the heat transfer rate between the fluids, equation (2) was utilized, with all of the components, such as heat transfer coefficient, area transfer, and LMTD, first being determined. For the overall heat transfer coefficient, the value was used from Taware [13], which is  $U=38.0869 \text{ W/m}^2 \text{ K}$ .

The area of the tube was calculated for the area transfer between the fluids, and the equation is as follows.

$$A=L\pi d$$

Which length of tube = 1060 mm

Inner diameter = 19 mm

Outer diameter = 25 mm

$$\text{So, } A = [(1060) (\pi) (25)] - [(1060) (\pi) (19)] = 19,980.52 \text{ mm}^2$$

$$\text{Or } A = 19.98 \text{ m}^2$$

While for the LMTD, the value that was used for this calculation is an average value from Table 3.1.

$$T_m = (T_1 - T_2) / (\ln[(T_1/T_2)])$$

$$T_m = ((180-64) - (150.30-40)) / (\ln [(180-64)/(150.30-40)])$$

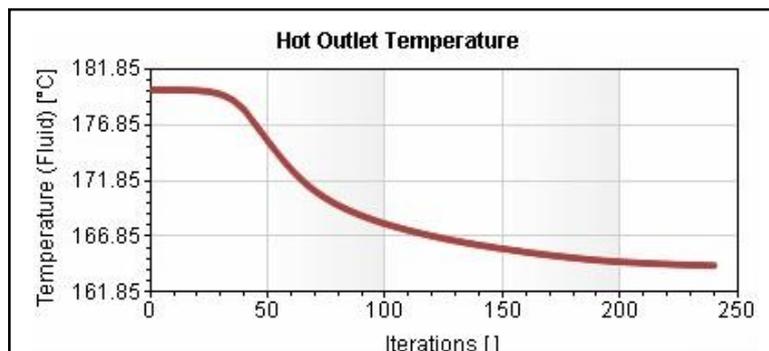
$$T_m = 113.13 \text{ }^\circ\text{C}$$

With all the data that was gained, it then can be inserted into the equation (2).

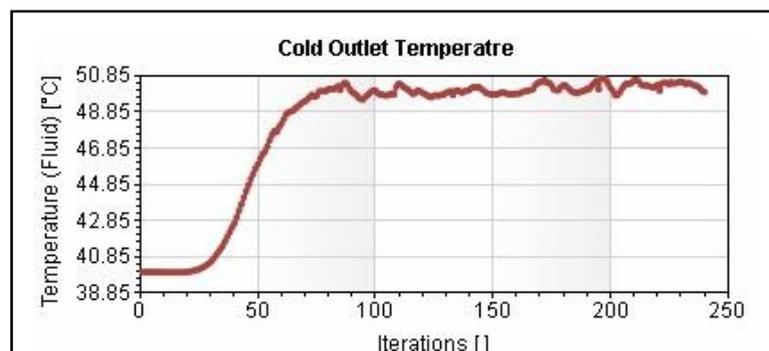
$$Q = (38.08) (19.98) (113.13) = 86,073.65 \text{ W}$$

### 3.1.3 Parallel Flow Result

The graphs in Fig. 6 and Fig.7 illustrate the relationship between fluid temperature and simulation repetition (iteration) to analyse potential outcomes for parallel flow. Because the hot and cold fluids travel in the same direction, the highest temperature difference between the two fluids occurs at the heat exchanger's entrance. In parallel flow, unlike in counter flow, there is no temperature crossover point. Instead, the fluids maintain their relative temperature difference throughout the exchanger's length. The graphical representation illustrates that the outcome complies to the Saari [3], as the hot and cold fluids approach without ever intersecting.



**Fig. 6** Hot outlet temperature for parallel flow



**Fig. 7** Cold outlet temperature for parallel flow

Table 4 shows the average, minimum and maximum value of the temperature outlet for the parallel flow. According to Table 4, the simulation was a success because the progress could be reached to 100% without any

errors or crashes. The maximum and minimum values from Table 4 show the simulation's highest and lowest potential values. The temperature differential for the hot fluid is 16.16°C, but for the cold fluid it is 13.16°C. The result outcome is solely explained by the temperature distribution being limited to the entry of the heat exchanger.

**Table 4** Overall result gained for the parallel flow

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]
V max	[m/s]	0.7866928	0.78672539	0.78664026	0.787219641	100
Hot Inlet Temperatre	[°C]	180	180	180	180	100
Hot Outlet Temperature	[°C]	163.916	163.8398934	163.7237061	163.9179777	100
Cold Inlet Temperature	[°C]	40	40	40	40	100
Cold Outlet Temperature	[°C]	53.120618	53.15650772	52.4463955	54.15391889	100

### 3.1.4 Calculation for Parallel Flow

To compute the heat transfer rate between the fluids, equation (2) was utilized, with all of the components, such as heat transfer coefficient, area transfer, and LMTD, first being determined. For the overall heat transfer coefficient, the value will be use from Taware [13], research which is  $U=38.0869 \text{ W/m}^2 \text{ K}$

The area of the tube was calculated for the area transfer between the fluids, and the equation is as follows.

$$A=L\pi d$$

Which length of tube = 1060 mm

Inner diameter = 19 mm

Outer diameter = 25 mm

$$\text{So, } A = [(1060) (\pi) (25)] - [(1060) (\pi) (19)] = 19,980.52 \text{ mm}^2$$

$$\text{Or } A = 19.98 \text{ m}^2$$

While for the LMTD, the value that was used for this calculation is an average value from Table 3.2.

$$T_m = ((T_1 - T_2) / (\ln(T_1 / T_2)))$$

$$T_m = ((180 - 40) - (163.92 - 53.12)) / (\ln [(180 - 40) / (163.92 - 53.12)])$$

$$T_m = 124.83 \text{ }^\circ\text{C}$$

With all the data that was gained, it then can be inserted into the equation (2).

$$Q = (38.08) (19.98) (124.83) = 94975.46 \text{ W}$$

## 3.2 Comprehensive Analysis of Heat Exchanger Efficiency

An in-depth examination of the shell and tube heat exchanger, which includes design, modelling, and mathematical calculations, provides valuable knowledge about the rate at which heat is transferred between two different fluids. SolidWorks, a 3D computer-aided design (CAD) software, is crucial for the design and analysis of heat exchangers. It includes several components such as baffles, tube sheets, tubes, heads, and shells [15]. The counter flow arrangement has excellent heat transfer efficiency, with a higher temperature difference, especially near the exit, and keeping significant variations throughout the length of the exchanger [16]. On the other hand, parallel flow maintains a steady temperature difference but may have smaller overall differences leading to decreased efficiency in transferring heat compared to counter flow [12]. Unfortunately, the calculation result contradicts the graphical outcome, suggesting potential errors in the mathematical analysis, possibly stemming from factors like heat exchanger design, simulation inaccuracies, or boundary condition definitions. It emphasizes the importance of considering these factors for accurate calculations in the design process.

## 4. Conclusion

The research successfully fulfilled its objective of designing the shell and tube heat exchanger using SolidWorks 2020, flow simulation for counter and parallel flow, and calculating heat transfer rate. The study's notable findings demonstrated that SolidWorks could design TEMA-compliant shell and tube heat exchangers. SolidWorks is an excellent tool for creating detailed 3D models while adhering to TEMA standards. Heat transfer rates have been used to demonstrate the thermal efficiency of shell and tube heat exchangers that operate in counter flow and parallel flow designs. The study also emphasises the need of material selection and assigning properties to guarantee that the heat exchanger meets TEMA material, thermal conductivity, and corrosion resistance criteria. The study's findings revealed that the counterflow arrangement had higher heat transfer rates than the parallel flow design. This is consistent with expectations, as the counterflow configuration naturally optimises the

temperature difference between the hot and cold fluids along the length of the heat exchanger. The capacity of counterflow to sustain a greater temperature gradient improves heat transfer efficiency, making it a preferable alternative when obtaining optimal thermal performance is critical.

In contrast, the parallel flow configuration, though simpler in design, displayed lower heat transfer rates due to a smaller temperature difference between the fluids. This emphasises the importance of thoroughly examining the requirements of a given application when selecting the appropriate flow design. The study drew attention to a significant gap in the comparison of counter and parallel flow configurations between mathematical calculations and graphical data produced through simulation analysis. The calculated result for counterflow is 86,073.65W, while the result for parallel flow is 94,975W. The detected discrepancies highlight probable flaws during simulation or errors in the design process. Mathematical calculation errors or differences from theoretical predictions indicate that the study's underlying assumptions and mathematical formulations must be extensively reevaluated. It is vital to identify the underlying causes of these inconsistencies, whether they are caused by theoretical model simplifications or errors in mathematical equation implementation.

## Acknowledgement

The authors would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for the support in accomplishing this research.

## Conflict of Interest

The authors declare that there is no conflict of interest regarding the publication of the paper.

## References

- [1] Mohanty, S., & Arora, R. (2020). CFD analysis of a shell and tube heat exchanger with single segmental baffles. *International Journal of Automotive and Mechanical Engineering*, 17(2), 7890–7901. <https://doi.org/10.15282/ijame.17.2.2020.08.0589>
- [2] Ariffin, A. Z., & Ridzuan, P. D. (2012). Development of Heat Exchangers Design Tool. (Degree dissertation, Universiti Teknologi PETRONAS). Universiti Teknologi PETRONAS Research.
- [3] Saari, J. (2023). Heat Exchanger Dimensioning. Studocu; Studocu. <https://www.studocu.com/my/document/universiti-putra-malaysia/chemical-engineering/saari-heat-exchanger-dimensioning/10948398>
- [4] Bhatt, D., & Javhar, P. M. (2014). Shell and Tube Heat Exchanger Performance Analysis. In *International Journal of Science and Research*. [www.ijsr.net](http://www.ijsr.net)
- [5] Nagaraj, S., Nagaraj, A., & Karthik, N. (2020). Thermal performance of a Divided-flow type Shell and Tube Compact Heat Exchanger. *Journal of Physics: Conference Series*, 1473(1). <https://doi.org/10.1088/1742-6596/1473/1/012031>
- [6] Abeykoon, C. (2014). Improving the performance of shell-and-tube heat exchangers by the addition of swirl. *International Journal of Process Systems Engineering*, 2(3), 221. <https://doi.org/10.1504/ijpse.2014.066691>
- [7] Pamuk, M. T. (2020). CFD simulation of a shell and tube heat exchanger. *Proceedings of the World Congress on Mechanical, Chemical, and Material Engineering*, 1–8. <https://doi.org/10.11159/iccpe20.105>
- [8] Cruz, P. A. D., Yamat, E. J. E., Nuqui, J. P. E., & Soriano, A. N. (2022). Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger. *Digital Chemical Engineering*, 3. <https://doi.org/10.1016/j.dche.2022.100014>
- [9] Yin, S., Liu, Q., & Hua, Y. (2019). Transient CFD modelling of space-time evolution of dust pollutants and air-curtain generator position during tunneling. *Journal of Cleaner Production*, 239, 117924–117924. <https://doi.org/10.1016/j.jclepro.2019.117924>
- [10] Zavala-Río, A., Femat, R., & Santiesteban-Cos, R. (2005). Universidad Autónoma Metropolitana Unidad Iztapalapa Distrito Federal. *México Revista Mexicana De Ingeniería Química*, 4, 201–212. <http://www.redalyc.org/articulo.oa?id=62040301>
- [11] Cruz, P. A. D., Yamat, E. J. E., Nuqui, J. P. E., & Soriano, A. N. (2022). Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger. *Digital Chemical Engineering*, 3. <https://doi.org/10.1016/j.dche.2022.100014>
- [12] Chand, S., Subhani, M., Sravani, P., & Ijmtst, E. (2021). The Systematic Comparison on Analysis of Parallel Flow and Counter Flow Heat Exchanger by using CFD and Practicle Methods. *International Journal for Modern Trends in Science and Technology*. 7. 153-161. 10.46501/IJMTST0711026.
- [13] Taware, Mr. S. R., Patil, D., & Arakerimath, R. R. (2017). Design, Fabrication and Testing of Shell and Tube Heat Exchanger for Heat Recovery from Hydraulic Oil. *International Journal of Engineering Research and*. V6. 10.17577/IJERTV6IS070289.

- [14] Asadbeigi, S., Ahmadi, E., Goodarzi, M., & Sagharichian, A. (2023). Analyzing and simulating heat transfer and designing a shell and tube heat exchanger for the pasteurization process of tomato paste: A CFD study. *Heliyon*, 9(11). <https://doi.org/10.1016/j.heliyon.2023.e21593>
- [15] Mishra, S., & Ullas, A. V. (2023). Concept Modelling of Small-Scale Device for Continuous Production of Graphene using Solidworks. *Materials Today: Proceedings*, 79, 345–348. <https://doi.org/10.1016/j.matpr.2022.12.034>
- [16] Mogaji, T. S., Rotimi, I., & Olapojoye, A. O. (2020). Effect of Fluid Types on the Performance of Heat Exchanger Base on Flow Configurations. 2. 102-113.