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

RPMME

Vol. 5 No. 2 (2024) 268-273
https://publisher.uthm.edu.my/periodicals/index.php/rpmme

The Study of Baffles Effect on Temperature in Shell and Tube Heat Exchanger

Siti Nur Hidayah Abd Razak[1], Azwan Sapit[1]*

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

*Corresponding Author: azwans@uthm.edu.my
DOI: https://doi.org/10.30880/rpmme.2024.05.02.030

Article Info

Received: 21 May 2024
Accepted: 20 August 2024
Available online: 31 Dec 2024

Keywords

Shell and tube heat exchanger, STHE, Single segmental baffles, Heat transfer rate, Solidworks flow simulation, CFD analysis, Efficiency.

Abstract

Heat exchangers find wide usage in a variety of industry applications. It is a device that works by facilitating the transfer of thermal energy between two fluids at different temperatures with circulate through separate pathways to avoid direct contact. Shell and tube is one of the most common type of heat exchanger used in the industry. The basic design consists of a series of tubes enclosed within a cylindrical shell. Baffles are placed in the shell side to create turbulence in the shell-side fluid, which improves heat transfer efficiency. This study investigates how the presence or absence of baffles affects heat transfer rates within the exchanger, exploring the intricate dynamics of fluid flow and heat exchange. SolidWorks 2023 software has been utilized to design and simulation analysis of the shell and tube heat exchanger. The goal of this study is to measure the impact of baffles on heat transfer rates, fluid dynamics, and total energy efficiency through analysis of computational fluid dynamic (CFD) simulation. The result of this study are expected to enhance the efficiency and effectiveness of shell and tube heat exchanger.

1. Introduction

A heat exchanger is a critical device in many industrial processes, facilitating the transfer of heat between two or more fluids of different temperatures. Typically, these fluids are separated by a wall that allows heat transfer through conduction and convection without the exchange of external thermal energy or work. Heat exchangers find extensive application in industries such as power generation, petroleum refining, chemical engineering, and air conditioning, with the shell and tube heat exchanger being the most prevalent type[1].

In a shell and tube heat exchanger, two fluids of differing temperatures circulate through the system without mixing, as they are separated by a dividing wall. The hot fluid flows through the tubes, which may be fitted with baffles, while the cold fluid passes through the shell side. To ensure efficient heat transfer, these exchangers must exhibit low pressure loss, strong heat transfer capabilities, and sufficient turbulence, which can be augmented by the insertion of baffles[2]. Baffles perform several essential functions such as support the tubes, maintain the desired velocity of the shell-side fluid, prevent tube vibration, and enhance heat transfer by increasing the heat transfer coefficient and fluid velocity through the redirection of the shell-side flow over the tube bundle. The efficiency of a heat exchanger is crucial for maximizing energy savings and thermal performance. Thus, understanding the efficiency variations among different designs is vital for improving heat exchanger performance under diverse operating conditions[3].

Previous research has demonstrated that altering the shape of baffles significantly impacts flow characteristics and heat transfer on the shell side. The number and type of baffles affect pressure drop, temperature distribution, turbulence, flow rate, and overall heat transfer rate[4]. This study aims to examine the fluid flow processes in shell and tube heat exchangers with and without baffles, focusing on how baffle shape, type, and spacing influence heat transfer rates and characteristics.

The objectives of this study are to evaluate and quantify the impact of baffles on the overall heat transfer within the heat exchanger, to visualize the fluid flow affected by baffles and study its effect on heat transfer, and to compare the efficiency of shell and tube heat exchangers with different baffle spacing. This study involves developing a 3D model of the heat exchanger in SolidWorks, including baffles and other components. Using SolidWorks software, simulations are performed to visualize contour plots, streamlines, and temperature profiles. Boundary conditions for the tube and shell fluid flows are defined, including inlet velocities and temperatures. The analysis examines the heat exchanger with hot fluid at 95°C, cold fluid at 20.05°C, and baffles spacing of 0, 4, and 8. Simulations are conducted with mass flow rates of 0.2, 1,2,3,4 and 5 kg/s.

2. Methodology

Setting the boundary conditions is crucial to achieving the study's goals. This chapter discusses the geometry of the shell and tube heat exchanger, the baffle design, and the various parameters and boundary conditions. After designing the heat exchanger and baffles, parameters such as fluid temperatures for hot and cold streams, mass flow rates, and velocities are set. The simulations are then conducted using SolidWorks. Once the results are obtained, mathematical calculations for the heat transfer coefficient and efficiency are performed. Finally, the results are analyzed and visualized to understand the fluid flow patterns.

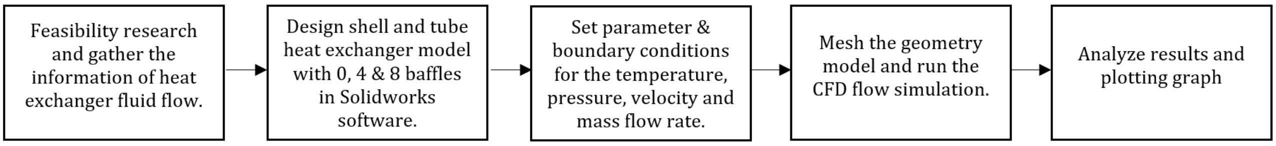

Fig. 1 *Overall flow of heat exchanger effect study*

2.1 Geometry Model of Shell and Tube Heat Exchanger

The dimensions of geometry model of the shell and tube heat exchanger (STHE) has been set up following with the boundary condition for the fluid flow. In order to accomplish the required increase in heat transfer, the boundary condition that is supposed to be applied is applied in terms of the fluid flow rate and temperature. However, this comes with the loss of a low-pressure drop at the outlet of the shell and tube. Detailed information on the geometrical properties is provided in the table 1.

Table 1 *Geometry model parameters*

Geometry component	Value
Shell diameter (mm)	450
Shell length (mm)	3000
Tube diameter (mm)	20
Number of tube	128
Inlet and outlet diameter (hot temperature) (mm)	80
Inlet and outlet diameter (cold temperature) (mm)	130
Total heat exchanger length (mm)	3600
Number of baffles	0, 4, 8

2.2 Boundary Conditions and Parameter Set Up

After set up the model geometry of the heat exchanger, the parameter and boundary condition need to be set up. In order to accomplish the required increase in heat transfer, the boundary condition that is supposed to be applied is applied in terms of the fluid flow rate and temperature. However, this comes with the loss of a low-pressure drop at the outlet of the shell and tube. Detailed information on the geometrical properties is provided in the table 2.

Table 2 *Parameter setting and boundary condition*

Description	Shell side	Tube side
Fluid	Cold water	Hot water
Inlet mass flow rate, ṁ (kg/s)	1.2	0.2, 1, 2, 3, 4, 5
Pressure outlet	Atmospheric pressure	Atmospheric pressure
Inlet fluid temperature (°C)	20.05	95

In order to accomplish the required increase in heat transfer, the boundary condition that is supposed to be applied is applied in terms of the fluid flow rate and temperature. However, this comes with the loss of a low-pressure drop at the outlet of the shell and tube. Detailed information on the geometrical properties is provided in the table 3.2.

The formula that been used to find the heat transfer rate are by using specific heat formula:

$$Q = \dot{m} \times Cp \times \Delta T$$

Where:
- Q = heat flow rate (W)
- ṁ = mass flow rate (kg/s)
- Cp = Specific heat capacity (J/kg.°C)
- ΔT = Temperature difference (°C)

In this study, the specific heat value is 4200 J/kg.°C since the fluid usage is water. For the temperature difference, the formula is temperature hot inlet minus temperature hot outlet. As the specific heat increases, a greater amount of energy is necessary to induce a modification in temperature. In opposition to substances with low specific heat, those with high specific heats require a greater amount of heat energy to decrease in temperature.

2.3 Mesh in Solidworks

After designing the model geometry and defining the boundary conditions, the next step is to mesh the assembly. SolidWorks meshing software is used for this process, dividing the component into smaller parts to ensure even load distribution. The Watertight Geometry workflow streamlines mesh production, allowing users to complete all CFD simulation phases in a single window. Meshing is crucial in finite element analysis, enabling the simulation and evaluation of structures and components under various conditions, which aids in system design and optimization.

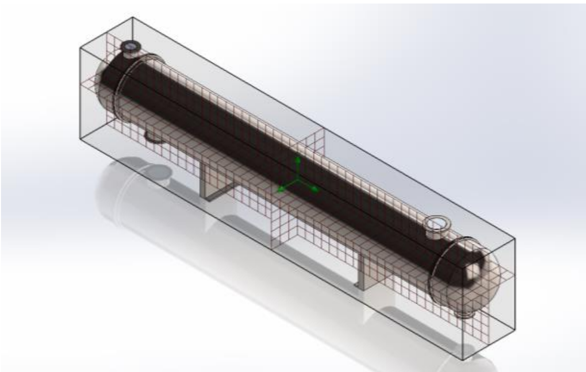

Fig. 2 *Basic mesh in Solidworks*

3. Result and Discussion

After simulation, the result of heat transfer coefficient, velocity and the pressure drop are become clearer. Analysis of tube and shell heat exchanger is utilized to validate the case studies, ensuring that the obtained results are independent of mesh size. The geometries are simulated using three different heat exchanger designs, each varying in the number of baffles, different spacing and mass flow rates.

3.1 Heat Exchanger with 0, 4 & 8 Baffle

The simulation of the shell and tube heat exchanger without baffles demonstrated that increasing the mass flow rate of the hot fluid enhances heat transfer performance by improving thermal mixing and convective heat transfer. The absence of baffles resulted in a relatively low pressure drop, providing a balance between heat transfer efficiency and pumping power requirements. These findings indicate that optimizing the mass flow rate is crucial for achieving the desired heat transfer outcomes in shell and tube heat exchangers without baffles.

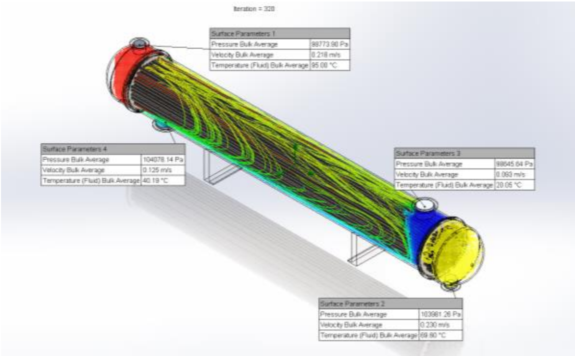

Fig. 2 *Flow simulation result for 0 baffles*

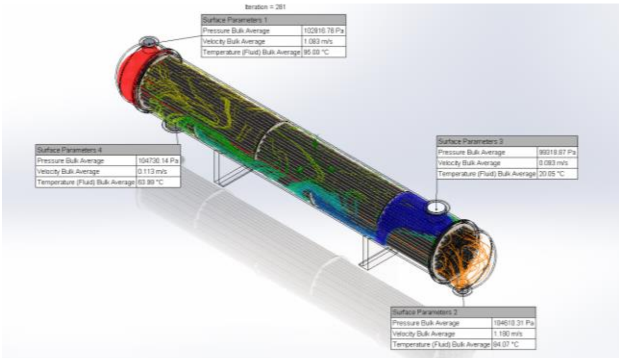

Fig. 3 *Flow simulation result for 4 baffles*

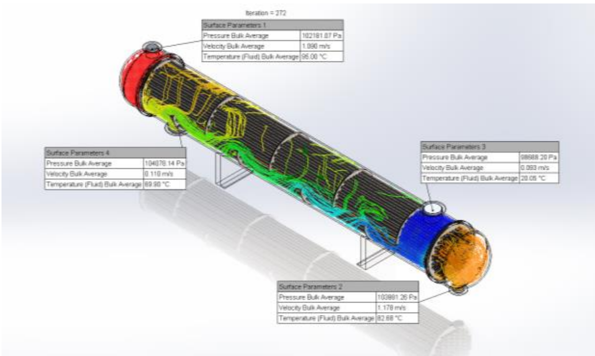

Fig. 4 *Flow simulation result for 8 baffles*

Table 3 *Result for 0 baffle heat exchange*

Description	0.2 kg/s	1 kg/s	2 kg/s	3 kg/s	4 kg/s	5 kg/s
Cold Outlet Temperature (°C)	26.67	37.39	42.11	44.27	45.99	47.15

Hot Outlet Temperature (°C)	79.46	69.42	78.35	82.84	85.19	86.91
Heat transfer rate, Q̇ (KW)	13.05	107.44	139.86	153.22	164.81	169.89
Efficiency, ε (%)	3.46	28.44	37.02	40.56	64.04	44.97

Table 4 *Result for 4 baffle heat exchange*

Description	0.2 kg/s	1 kg/s	2 kg/s	3 kg/s	4 kg/s	5 kg/s
Cold Outlet Temperature (°C)	28	46.37	55.96	59.92	61.51	63.88
Hot Outlet Temperature (°C)	77.61	62.70	72.92	78.51	81.97	84.06
Heat transfer rate, Q̇ (KW)	14.61	135.66	185.47	207.77	218.90	229.74
Efficiency, ε (%)	3.87	35.91	49.1	55	57.95	60.82

Table 5 *Result for 8 baffle heat exchange*

Description	0.2 kg/s	1 kg/s	2 kg/s	3 kg/s	4 kg/s	5 kg/s
Cold Outlet Temperature (°C)	29.08	48.90	59.83	64.62	67.49	69.80
Hot Outlet Temperature (°C)	78.24	61.06	70.98	76.98	80.6	82.85
Heat transfer rate, Q̇ (KW)	14.08	142.55	201.77	227.05	241.92	255.15
Efficiency, ε (%)	3.73	37.74	53.41	60.11	64.04	67.55

3.2 Heat Exchanger Efficiency

The flow simulation results for the shell and tube heat exchanger (STHE) with different numbers of baffles show clear trends in performance metrics such as outlet temperatures, heat transfer rate, and efficiency. As the number of baffles increases from 0 to 8, the cold outlet temperature rises, and the hot outlet temperature decreases, indicating more effective heat transfer. For instance, with a hot fluid mass flow rate of 5 kg/s, the cold outlet temperature increases from 47.15°C for 0 baffles to 69.80°C for 8 baffles, while the hot outlet temperature drops from 86.91°C to 82.85°C.

The heat transfer rate also improves significantly with more baffles. At a hot fluid mass flow rate of 5 kg/s, the heat transfer rate rises from 169.89 kW for 0 baffles to 255.15 kW for 8 baffles. Efficiency follows a similar trend, increasing from 44.97% for 0 baffles to 67.55% for 8 baffles. This improvement is due to the increased turbulence and mixing caused by the baffles, enhancing heat exchange.

The hot fluid mass flow rate significantly impacts the performance of the heat exchanger. At 0.2 kg/s, the heat transfer rate and efficiency are low across all baffle configurations. As the flow rate increases from 1 to 5 kg/s, both metrics improve dramatically. As result, at a hot fluid mass flow rate of 5 kg/s, the efficiency increases from 3.46% for 0 baffles to 35.91% for 4 baffles and reaches 67.55% for 8 baffles. This dramatic increase underscores the importance of optimizing both the number of baffles and the mass flow rate to maximize the efficiency and performance of the shell and tube heat exchanger.

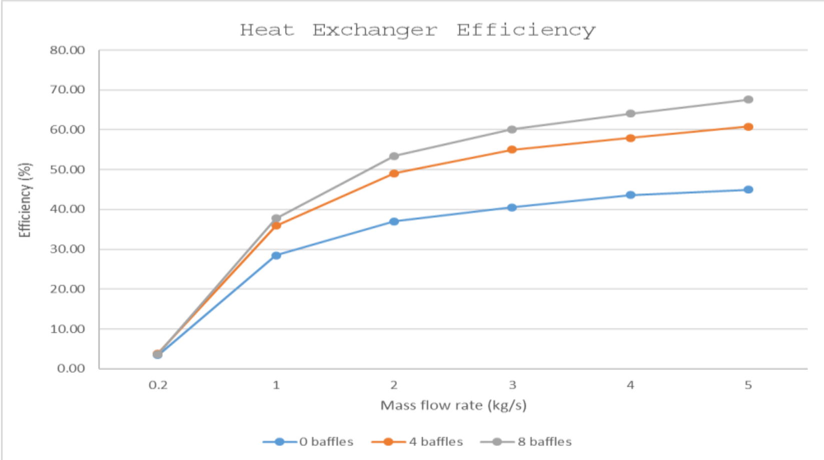

Fig. 5 *Graph heat exchanger efficiency*

4. Conclusion

The flow simulation experiment conducted on the shell and tube heat exchanger using SolidWorks with different numbers of baffles and different mass flow rates for hot and cold fluids reveals several key conclusions. Firstly, the inclusion of baffles in the heat exchanger design significantly enhances heat transfer efficiency across all tested simulation. Thermal transmission between the hot and cold fluids is more effectively achieved with the help of baffles, which also improve fluid mixing and expand the surface area accessible for heat exchange.
Furthermore, the heat exchanger's effectiveness is impacted by the mass flow rates of the fluids, whereas elevated flow rates often result in amplified heat transfer and efficiency. On the other hand, at certain flow rates, the efficiency against flow rate relationship could show unpredictable changes. When choosing the ideal baffle layout, practical factors like cost, complexity, and particular performance needs must be taken into account.
In conclusion, this stud effectively met its objectives by rigorously examining the impact of baffles on heat transfer, visualizing fluid flow patterns, and comparing heat exchanger efficiency across different configurations. These findings hold promising implications for the design and optimization of shell and tube heat exchangers, offering opportunities for improved efficiency and effectiveness in real-world applications.

Acknowledgement

The author express gratitude to Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its invaluable support and resources during this research.

References

[1] H. N. Purnamasari, T. Kurniawan, A. Bayu, and D. Nandiyanto, "DESIGN OF SHELL AND TUBE TYPE HEAT EXCHANGER FOR NANOFIBRIL CELLULOSE PRODUCTION PROCESS," 2021.

[2] Amirul Haiqal bin Anif, "The Hydraulics and Thermal Performance Assessment of Segmental Baffles of a Shell-Side of a Shell-and-Tube Heat Exchangers," *Fac. Mech. Manuf. Eng. Univ. Tun Hussein Onn Malaysia*, 2018.

[3] E. Akpabio, I. Oboh, and E. O. Aluyor, "The effect of baffles in shell and tube heat exchangers," *Adv. Mater. Res.*, vol. 62–64, pp. 694–699, 2009, doi: 10.4028/www.scientific.net/amr.62-64.694.

[4] M. Petrik and G. L. Szepesi, "Shell side CFD analysis of a model shell-and-tube heat exchanger," *Chem. Eng. Trans.*, vol. 70, pp. 313–318, 2018, doi: 10.3303/CET1870053.