Research Progress in Mechanical and Manufacturing Engineering Vol. 4 No. 1 (2023) 447-456

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

The Evaluation of Shell-and-Tube Heat Exchanger using Computer Simulation for Oil and Gas Industry

Alexia Luke Laiti[1], Muhd Hafeez Zainulabidin[1*]

Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Pahat 86400 Johor, MALAYSIA.

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2023.04.01.051
Received 19 July 2022; Accepted 30 Oct 2022; Available online 01 June 2023

Abstract: Heat exchangers brings the definition of a component used to transmit heat between two or more mediums. This study will be on the evaluation of shell-and-tube heat exchanger using Computer Simulation for Oil and Gas Industry where it is to model the shell-and-tube heat exchanger by using FEA-CFD software and the effect of Malaysia's surrounding temperature using CFD Software. It is scoped to the surrounding ambient temperature according to Malaysia's surrounding temperature at the industry, with consideration of the mesh element sizing only limits 512,000 cells/nodes. The procedure in this study will be using Fluid Flow (Fluent) using ANSYS Simulation as it is able to create more uniform mesh for improved accuracy where it can use for internal and external characterization of fluid behavior and heat flow. The shell-and-tube heat exchanger is 1m long with an inner diameter of 100mm and outer diameter of 150mm, with copper as the material which viewed as deemed fit. The study's result uses 100 iterations, active energy equation with mesh element sizing of 0.04m. The working pressure was assumed in environmental condition, 101.325kPa with hot inlet velocity of 10m/s and cold inlet at 0m/s. Besides that, it shows that there is near similar reading of temperature between previous studies which signifies that the hot and cold outlets proof to be compatible as the evaluation using Computer Simulation goes. In those past literature study, they mathematically model the heat exchanger using system identification methods, with the usage of Auto Regressive-Moving-Average model with eXogenous inputs (ARMAX) model obtained from Pseudo Random Binary Signal (PRBS). The other study used Computational Fluid Dynamics (CFD) with geometrical design in INVENTER PROFESSIONAL. It showed that the cooling agent (H^2O) flowing inside work effectively to regulate the intense temperature inside the shell-and-tube heat exchanger. The impact of temperature and surface region is highly dependent on the heat transfer rate. However, different approach could be considered such as using more finer mesh element sizing for more accurate result. Besides that, variation of diameters on heat transfer area and other approach can be used to apply in this study.

Keywords: Computer Simulation, Shell-And-Tube Heat Exchangers, Oil And Gas Industry, Heat Transfer Rate

1. Introduction

**Corresponding author: hafeez@uthm.edu.my*

There are a variety of options of heat exchangers that fluids may be single or multiple, separated or in proximity. The ongoing improvement on the thermal analysis with respect to the shell-and-tube heat exchangers generally circles on feed temperature and tension, shell measurement, number of tubes, tube geometry, baffle and cutting spacing [1]. Based on past studies conducted, Abd and Naji [2] stated that the overall heat transfer coefficient by using different bundle diameters depends on the shell diameter. In Shinde and Chavan [3] theory, it was said that baffle spacing, and pitch type can play a significant role in enhancing heat transfers into exchangers. On the other hand, Dizaji, Jafarmadar, and Asaadi [4] cross examined experimentally using a corrugated shell and tube instead of smooth shell and tube to improve the heat transfer through the exchanger. Heat exchangers are an integral part of the oil and gas processing process which will be in high demand in the next twenty years as the human population grows. Wasted industrial energy can be conserved using heat exchangers as it acts as waste heat recovery and this study is determined to analyze and simulate the shell-and-tube heat exchanger using Fluid Flow (Fluent) ANSYS.

The objective of this study is to model the shell-and-tube heat exchanger by using FEA-CFD software, and the effect of Malaysia's surrounding temperature at an oil and gas industry to the heat exchanger performance using CFD Software. The scope and limitation of this study is the surrounding ambient temperature under steady-state condition and the meshing size which only allow 512,000 cells/nodes. The significancy of this study intends to examine and simulate the environment's temperature using shell-and-tube heat exchanger in an oil and gas industry. As a result, the research will potentially assist the oil and gas industry in conducting essential test using remote analysis that can help to preserve the environment.

As the campus is journey towards sustainability, it is crucial to apply sustainable practices in reports, research, etc. The sustainable development goals in this study relies on SDG9 which makes sense of on the advancement of versatile foundation, the advancement of comprehensive and reasonable industry, and the encouragement of innovation while SDG13 surrounds on the climate change.

2. Methodology

In this part, the research strategy, research technique, research methodology, data collection methods, sample selection, research process, data analysis, ethical considerations, and project study constraints are covered in this section. The study is conducted by gathering information from articles on the internet and other studies completed in the literature review to obtain the study's information then determining all the mechanisms that is needed and simulate using ANSYS and finally collecting and analyzing all the data from the simulation.

The characterization of this study is revolving around the pertinent analysis regarding shell-and-tube heat exchanger in an oil and gas industry. The Fluid Flow (Fluent) starts up by setting up the workbench in a standalone style by clicking on Fluid Flow (Fluent), meshing, setting up and solving the boundary conditions, post processing and lastly, analyze the result based on desired parameters.

During setting up of workbench, few parameters in the engineering need to be set first such as the material, which in this case uses copper; material properties such as density, isotropic thermal conductivity, and specific heat constant pressure. In meshing, the smaller the mesh element size, the more accurate the result will be. For this study, the mesh element size does not exceed 512,000 due to the fact that the workbench is running on student's version. Before importing the geometry into ANSYS, it is first drawn at Solidworks using suitable parameters and constraints. The tubes inside the shell have an inner diameter of 100mm and outer diameter of 150mm, and geometry which consists of shell wall, tube domain, shell domain, and tube wall. Fluid flow could create more uniform meshes for improved accuracy where it can be used for both internal and external characterization of fluid and heat flows in shell-and-tube heat exchanger. The simulation runs once the problem has been properly described and defined, and the analysis progress and convergence can be tracked by selecting a goal plot. The material is being set using a manual in accordance with 1998 ASME BPV Code, Section 8 with the density of 7850kg/m^3, and the result is tabulated accordingly. However, in the boundary conditions, since the study is about heat analysis, a definition of temperature must be set whereby in this study the initial startup temperature is at 32℃. To specify them, a model check runs in the Model command which will then generate tabular data and insert the intent temperature. In post processing, temperature gradient was chosen for this study to view the thermal analysis that is taking place on the surface of the shell-and-tube heat exchanger. In this simulation, there are few colors for different temperatures depending on the number of contours that will show the diverse temperature on the running data, and when the temperature changes over time, the indication is plotted on a graph. As for the fluid flow testing method, it is usually fast and accurate to determine the mean heat transfer coefficients of compact heat transfer surfaces [5]. The first goal in post processing is to analyze the performance of shell-and-tube heat exchangers using fluid flow on the effect of heat transfer and taking in consideration of fluid as a cooling medium in it. Since this study based on a steady-state condition, time is constant. A graph is presented in the results and discussion column on the temperature that have been analyzed. Using the data, it substantively determines the performance of the shell-and-tube heat exchanger for its heat worthiness on the subject body [6] and at the end of the day, conclusively proof that thermal plays a significant role in heat exchangers.

The characterization of this study lies on the method of Fluid Flow (Fluent) as a study medium. The liquid that should be warmed or cooled is contained in one bunch of cylinders in a shell-and-tube heat exchanger. The subsequent liquid runs over the cylinders that are being warmed or cooled, giving, or retaining heat on a case-by-case basis. Fluid Flow (Fluent) is chosen because in a shell-and-tube heat exchanger, there is a medium which regulates it which is water. The medium may vary as a heat transfer fluid (HTF) as it can be water, synthetic oil, and molten salt [7]. HTF primarily utilized as a heat transfer fluid between a heat source and other heat demands and it carries heat to the storage tank and then to a steam generator. Due to its large thermal capacity and low viscosity, water is an excellent heat transfer fluid. It is inexpensive to utilize since it is used in direct steam generation, which reduces the costs. Water is a feasible option as oil although has a higher tolerance on liquid temperature compared to water, hydrocarbons break down faster if heated to within 400°C while for molten salt the restriction lies as it is at risk of freezing of salt in the longer length of the receiver [8]. To summarize, the method narrative is used to apply in this study. The result is displayed in the table in the next part.

A good shell-and-tube heat exchanger exhibit a good performance of temperature distribution along them. The operating pressure is at 101.325kPa with velocity magnitude of 10m/s for the hot inlet and 0m/s at the hot outlet. Based on Figure 1, it depicts the meshing condition. It is necessary to generate mesh because the accuracy of the solution as it is sufficiently refined to obtain the viable accuracy, and it can use discretization or meshing to reduce the degrees of freedom (DOF) from infinite to finite. The mesh sizing used is 0.04m, it is the most viable size utilizing in this meshing as lesser element size that causing the nodes and elements to exceed 512,000 which do not support under ANSYS Student's Version.

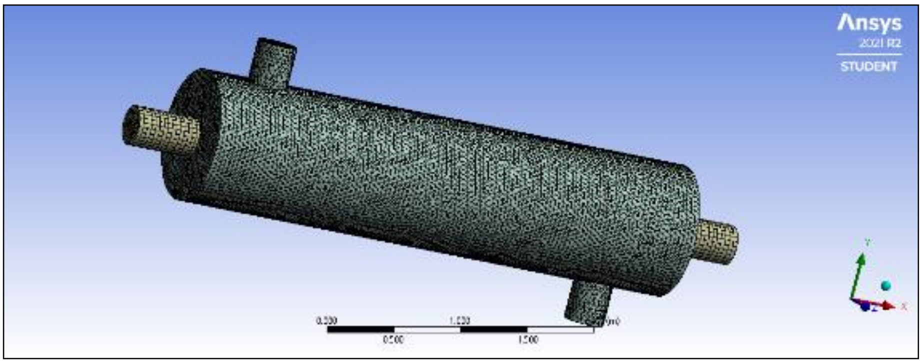

Figure 1 Meshing in Shell-and-Tube Heat Exchanger

Nodes and elements are the very backbones of Finite Element Analysis (FEA) [12]. In this shell-and-tube heat exchanger, it is categorized into shell fluid domain, tube fluid domain, shell wall, and tube wall and all of these will be absorbing and dissipating hot and cold medium. In Table 1, the nodes and elements numbers for the simulation ran.

Table 1 Nodes and Elements of Each Domain

Domain	Nodes	Elements
Shell fluid domain	13215	65913
Shell wall	4527	15242
Tube fluid domain	11914	10240
Tube wall	10656	7203
All domain	40312	98598

There are 100 iterations for this analysis with active energy equation. Energy equations must be switch on because the study involves heat transfer and compressible flows, and it is necessary as the temperature flowing is not constant. Both walls of the shell and tube utilized copper due to it being excellent thermal conductivity along with its relatively low specific heat. Table 2,3 and 4 depicts the result for material, thermal, and fluid properties after running the meshing result. The data generated was obtained from the tab of Fluent Database Properties to identify the parameters of shell-and-tube heat exchanger.

Table 2 Material Properties

Description	Notation	Data
Structural		
Young's Modulus	E(Pa)	$2x10^{11}$
Poisson Ratio	υ	0.3
Bulk Modulus	k(Pa)	$1.6667x10^{11}$
Shear Modulus	G(Pa)	$7.6923x10^{11}$
Isotropic Secant Coefficientof Thermal Expansion	α(1/K)	$1.2x10^{-5}$
Compressive Ultimate Strength		0
Compressive Yield Strength		$2.5x10^{8}$Pa
Tensile Ultimate Strength	σmax(Pa)	$4.6x10^{8}$
Tensile Yield Strength	σ(Pa)	$2.5x10^{8}$

Table 3 Thermal Properties

Description	Notation	Data
Thermal		
Initial Temperature	T (°C)	32
Isotropic Thermal Conductivity	W/m.°C	60.5
Specific Heat Constant Pressure	Cp(J/kg°C)	434

Table 4 Fluid Properties

Description	Notation	Data
Fluid		
Density	ρ(kg/m^3)	998.2
Viscosity	ɳ (Pa.s)	0.001003
Latent Heat	J/kg	2.2631x10^6
Vaporization Temperature	°C	12

Before proceeding into simulation, the geometry of models, dimensions, and material of the heat exchanger was set to:

Table 5 Dimensions and Material on the Tube's Diameter

Tube	Dimensions
Outer Diameter of Tube	0.3m
Inner Diameter of Tube	0.25m
Length of Tube	5m
Material of Tube	Copper

Table 6 Dimensions and Material on the Shell's Diameter

Shell	Dimensions
Outer Diameter of Shell	1m
Inner Diameter of Shell	0.95m
Length of Shell	4m
Material of Shell	Copper

A good shell-and-tube heat exchanger exhibit a good performance of temperature distribution along them. In this study, the temperature is set between ambient temperature on the heat exchangers at oil and gas industry, with constant pressure. The inlets and outlets of the tube were as such:

Table 7 Inlet and Outlet Dimensions

Inlet	Outlet
Velocity: 10m/s Pressure: 101.325 kPa	Velocity: 0m/s Pressure: 101.328kPa
Fluid Used:	Water
Inlet Temperature of Cold Water:	15℃
Inlet Temperature of Hot Water	90℃

In Figure 2, the result computed as such where it indicates the hot and cold inlet, red indicates hot while blue indicates cold inlet. Since shell-and-tube heat exchanger is a component that involves intense heat capacity in a petrochemical industry, the exchange of fluids is a two-way flow outside and through the tubes.

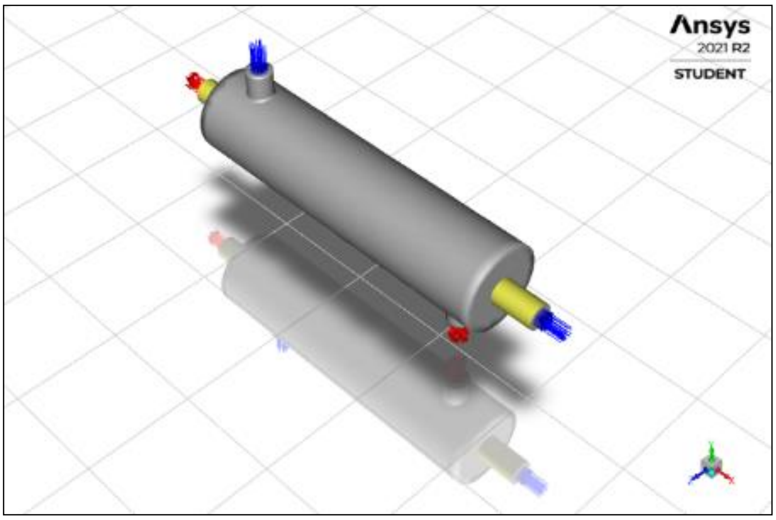

Figure 2 Top View of the Shell-and-Tube Heat Exchanger

It is known that the effect of surface area is directly proportional to the overall heat transfer coefficient which does not based solely on thermal conductivity and wall thickness of the material, but also on the surface area in the outside wall of the shell-and-tube heat exchanger. The selection of material is important because it determines how the material behave under certain conditions. Despite the way aluminum is essentially more lightweight, adaptable, and flexible than copper, it has a lower hang obstruction, making it more inclined to deformity during brazing and after intense cycles therefore it is infeasible to involve aluminum as the main material. Copper, however, on the other hand, is preferable at heat transfer and is joined by brazing compared to welding which serves as the last option because of its great hanging opposition and the ability to diminish distortion during brazing.

3. Results and Discussion

The results and discussion section presents data and analysis of the study. To recap the objectives of this study, it is to investigate the effect of surface area of the heat exchanger on the heat transfer rate and Malaysia's surrounding temperature at an oil and gas industry using Computer Simulation. The outcome was extracted using ANSYS Workbench specifically Fluid Flow (Fluent) and was classified accordingly to surface area and surrounding temperature of the heat exchanger, to determine the heat analysis performance with water (H^2O) as the medium.

According to past studies, there were two studies done regarding shell-and-tube heat exchangers via different approach. The study used CFD where several shell-and-tube heat exchangers with segmental baffles, where it aims to study the flow and temperature inside the shell-and-tubes with various design and tube [9]. The study depends also on CFD considered flexibility in design because of the core geometry can be varied easily by changing the tube diameter, length, and arrangement. The author also carried out the analysis to review different material and based on the results made which provide better heat transfer rate [10].

3.1 Results

Since the average temperature of a shell-and-tube heat exchanger is significantly high, therefore the heat dissipation around the area is quite high. Based on the graph that is generated in Figure 3, it shows that the curve from when the shell-and-tube heat exchanger starting to heat up during operation. The graph depicts the temperature against the whole length of shell-and-tube heat exchanger, where it stated off as lower temperature and it goes higher as it started working towards the end of the shell-and-tube heat exchanger, which then the heat will dissipate out from it after mixing with the cold medium that is coming in from the end of the shell-and-tube heat exchanger as in Figure 2 (as shown above). Besides that, it shows also regarding the increment of temperature as it passes through the heat exchanger with the help of water as the cooling medium inside the heat exchanger that is exchanging in the tubes. A comparison of temperature difference on parallel flow study by Chinta [11] and Singh [12] are depicted in Table 8 and is plotted on the graph in Figure 3.

Table 8 Table of Comparison from Previous Literature Study and Current Study

Author	Hot Inlet Temperature (K)	Hot Outlet Temperature (K)	Cold Inlet Temperature (K)	Cold Outlet Temperature (K)
Chinta, S. R. et. al. [11]	329	320	300	306
Singh, R. et. al. [12]	363	319.62	299	311.62
Current Study	363	319	288	310

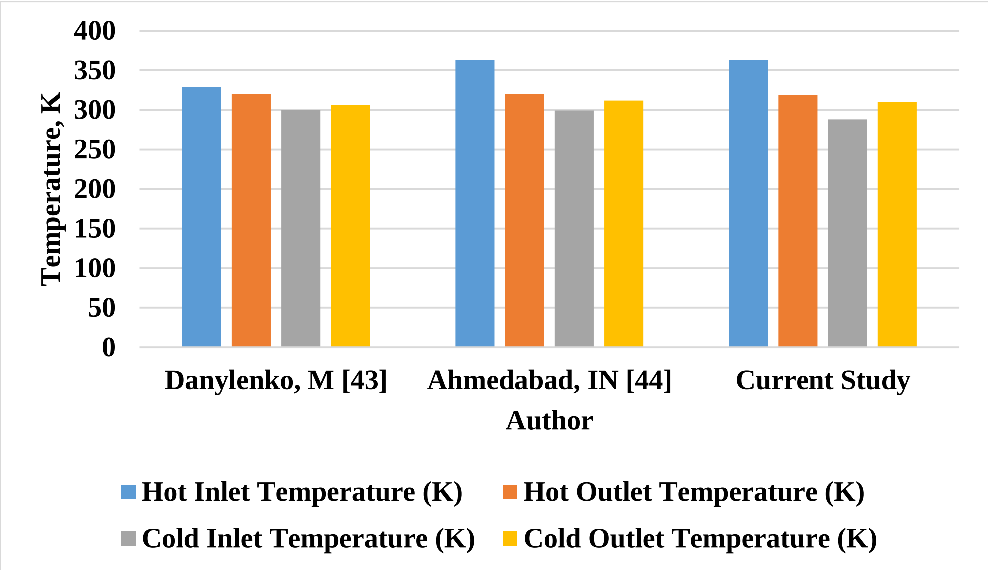

Figure 3 Comparison of Temperature on Shell-and-Tube Heat Exchanger

Based on the current study, the cold inlet is set at 288K, while hot inlet is set at 363K. The hot inlet's temperature is high, but it simmered down at the outlet while the cold inlet's temperature increased during the simulation. This shows that the heat exchanger is in an effective state because it acts as waste heat recovery and most importantly the fluid medium—H^2O is fully functional as it lowers the temperature of the hot temperature. The waste of energy or heat stream that is being exhausted, heat exchangers can be used to recover this heat. Besides that, it is crucial that the heat decreases because it had to optimize the cooling process and make it more efficient where resources and energy can be saved. In an oil and gas industry, it plays an important role in processing oil and gas in thermal management. It might be impossible to eliminate the whole wastage of heat, but at least it starts locally on energy conservation and world preservation. They are used in the refining process in cracking units as well as in the liquefaction of natural gas. The result that simulated also proved to be viable because when heat exchangers are used in preheating air before entering a furnace system, it takes some of the stress away from the furnace, allowing it to use less fuel and energy to heat the air itself which significantly serve the purpose of the SDG9 and SDG that was aforementioned.

According to one of the previous studies, who had done a study on shell-and-tube heat exchanger that are extensively utilized in industries, the author pointed out that the temperature of hot and cold fluids travelling through the heat exchanger should be closely monitored and controlled. The author numerically demonstrated the heat exchanger utilizing framework strategies and tentatively assess the viability of two PID regulator tuning techniques like Internal Model Control (IMC) and hand-off auto-tuning for temperature control to assess the gulf and outlet temperature for both hot and cold liquid. In the study, the author's parameters outcome on their temperature range is as in Figure 4.2 [42]. The other study is on the CFD Analysis on the fin of the circular shape with baffles and without baffles making the flow counter and parallel with 400mm segment of the shell provided with 84 fins and 4 baffles with 7 tubes with vary moulding design.

Figure 4 depicts the temperature profile where the temperature is higher at the left side as it is a hot outlet and lower temperature on the right side as the cold inlet is on that side with the influence of water as the cooling agent. The region is covered with blue, green, yellow, and red where the region that is with blue and green indicates the locality is cold and less heat. In vector profile, it provides flow around the module showing details of the geometry. Based on Figure 5, it depicts the "hair strands" like color are vectors without arrowheads. The arrowheads define the flow of the temperature in the shell-and-tube heat exchanger.

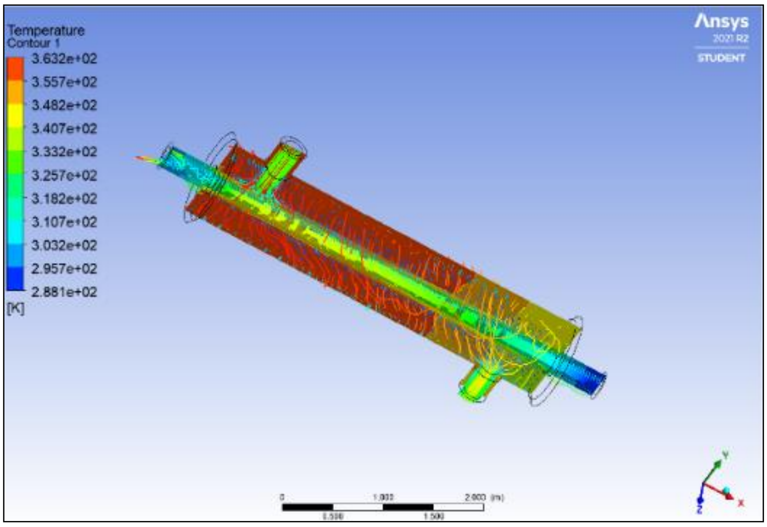

Figure 4 Contour Profile for Temperature

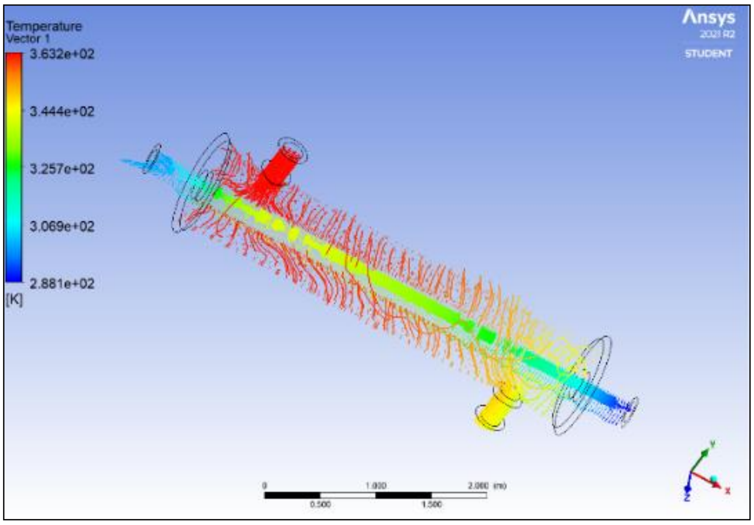

Figure 5 Vector Profile for Temperature

4. Conclusion

The issue of the industry is half of the energy input is wasted, and it is emitted into the environment that will cause unfavorable impacts towards the climate. However, many industries in Malaysia are using heat exchangers as a component to transfer heat from one medium to another, where it tends to be used in both heating and cooling processes. It is briefly shown that the evaluation of shell-and-tube heat exchanger can be done remotely using Computer Simulation. The investigation that is being done has the characteristics such as ambient temperature, water as transfer medium and operating pressure of 101.325kPa. It uses copper as the material with thermal conductivity of $60m^{-1}K^{-1}$, 100 iterations with active energy equation. In recommendations, different approach could be considered such as surface area, pressure characteristics of the flow and the usage of different type of material to observe the heat analysis. To add-on, to predict a more accurate result, a finer mesh with lesser mesh elements can be utilized in the next study as the current one has restricted element size of 512,000. Many approaches can consider improving the performance of the heat exchanger so that the workload can be more efficient and at the same time putting sustainability as top priority to conserve the environment.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Long, L., Ning, D., Junbo, S., Na, X., Weimin, G., and Chi-Man, L. W. (2016). Failure Analysis of Tube-to-Tubesheet Welded Joints in a Shell-Tube Heat Exchanger. *Case Study Engineering Failure Analysis*. Page 32-40.

[2] Abd and Naji (2018). Performance Analysis of Shell and Tube Heat Exchanger: Parametric Study. *Case Studies in Thermal Engineering*. Volume 12, Pages 563-568. https://doi.org/10.1016/j.csite.2018.07.009

[3] Shinde S. and Chavan, U. (2018). Numerical and Experimental Analysis on Shell Side Thermo-Hydraulic Performance of Shell and Tube Heat Exchanger with Continuous Helical FRP Baffles. Thermal Science and Engineering Process. Volume 5, Pages 158-171.

[4] Dizaji S.H. (2017). Experimental Exergy Analysis for Shell and Tube Heat Exchanger made of Corrugated Shell and Corrugated Tube. *Experimental Thermal and Fluid Science*. Volume

81, Pages 475-481. https://doi.org/10.1016/j.expthermflusci.2016.09.007

[5] Roetzel, W (2020). Design and Operation of Heat Exchangers and their thermal performance of heat exchangers. *Design and Operation of Heat Exchangers and their Networks*. Pages 391-429. https://doi.org/10.1016/B978-0-12-817894-2.00008-X

[6] Sherak, M (2020). Transient Thermal Study. Chemical Process Equipment (Second Edition, pages 165-224).

[7] Qazi, S. (2017). Solar Thermal Electricity & Solar Insolation. *Standalone Photovoltaic (PV) Systems for Disaster Relief and Remote Areas*. https://doi.org/10.1016/B978-0-12-803022-6.00007-1

[8] Mukund H. B. and Bandyopadyay, S. (2012). Targeting Minimum Heat Transfer Fluid Flow for Multiple Heat Demands. Computer Aided Chemical Engineering. https://doi.org/10.1016/B978-0-444-59507-2.50127-X

[9] Irshad, M., Kaushar, M and Rajmohan, G. (2017). Design and CFD Analysis of Shell and Tube Heat Exchanger. Volume 7. Issue No. 4.

[10] Patel, P. and Amitesh, P. (2012). Thermal Analysis of Tubular Heat Exchanger by Using ANSYS. Volume 1. Issue 8. ISSN: 2278-0181

[11] Chinta, S. R., Sahoo, A. and Radhakrishnan, T.K. (2018). Materials Choice in Heat Exchanger Design. Material & Application. Access on 2nd June 2022. Retrieved from https://matmatch.com/resources/blog/materials-heat-exchanger/fic

[12] Singh, R., Dubey, D. and Thankur, H. (2016). CFD Analysis of Shell-and-Tube Heat Exchanger. *International Journal for Scientific Research and Development*. Vol. 3, Issue 12.