International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 13, Issue 7 [July. 2024] PP: 91-96

Utilizing Computational Fluid Dynamics (CFD) Methodology to Simulate a Triangular Shell and Tube Heat Exchanger.

Ida Bagus Alit*, IGNK Yudhyadi
Department of Mechanical Engineering, Faculty of Engineering, University of Mataram, Jl. Majapahit No. 62
Mataram-Nusa Tenggara Barat 83125, Indonesia.
Corresponding Author: ***alit.ib@unram.ac.id***

ABSTRACT:
Various heat exchanger designs exist, with the shell and tube type being one example. This equipment has proven its usefulness in temperature control and is applied in various sectors, including hydroelectric power plants, especially in oil cooling bearing capacities. The aim of this research is to develop and simulate a shell and tube heat exchanger with tube inner diameters of 20 mm and 25 mm, using a rotating triangular and triangular tube arrangement. The design specifications outline the main dimensions: total length 1318.59 mm, outer diameter of the shell 300 mm, and inner diameter of the shell 320 mm. Simulation findings highlight the important influence of tube diameter and arrangement on heat exchanger performance. In particular, the heat exchanger with 61 tubes showed the highest overall heat transfer coefficient, recorded at 109 W/m^2.K. Meanwhile, the lowest value is found in the triangular tube arrangement with a total of 55 tubes, measuring 105.5 W/m^2.K. Apart from that, the maximum pressure drop on the shell side occurs in a triangular tube arrangement with a total of 55 tubes.

Date of Submission: 07-07-2024 Date of acceptance: 21-07-2024

I. INTRUDUCTION

In the realm of industrial enterprises, the significance of factories and power plants is paramount in driving the demand for heat exchangers. The increasing need for heat exchangers stems from the high production volumes required in this industry. The swift progress of the fast-paced industrial sector further intensifies the demand for a substantial energy supply. Among the facilities generating electrical energy, hydroelectric power plants stand out. These plants leverage the flow of water to convert it into electrical energy. The process involves transforming the kinetic energy of flowing water (from dams or waterfalls) into mechanical energy through water turbines, which is then converted into electrical energy with the assistance of a generator. The exchange of heat between two fluids at different temperatures, separated by a solid wall, is a common occurrence in various engineering applications [1], [2]. In power plants, a specific engineering application utilizing heat exchangers is in the context of bearings, which play a crucial role in maintaining the position of the turbine shaft along a single axis. To facilitate the maintenance of turbine bearings, a cooling system is implemented, utilizing oil lubrication [3]. The cooling of lubricating oil employs a shell-and-tube heat exchanger [4], where the heat exchange occurs between a shell (casing) and tubes (pipes) carrying fluids at different temperatures [5]. The flow direction of the two fluids can be parallel, counterflow, crossflow, or a combination [6]. In a shell-and-tube heat exchanger, one fluid flows inside the tubes, while another fluid flows outside the tubes. The tubes are arranged within a cylindrical space called the shell, parallel to its axis [7]. Baffles on the shell side enhance the speed and effectiveness of fluid flow outside the tubes, aiding in more accurate prediction of pressure drop and heat transfer for the tube section. Pressure loss in fluid flow results from friction between the pipe walls and the flowing fluid. To analyze heat transfer in heat exchangers, methods such as actual experiments and simulations have been developed. Previous studies have investigated the impact of factors like the number of tubes and baffles on the effectiveness of shell and tube heat exchangers [8]. Experimental research, while valuable, has drawbacks such as time consumption and high costs. Computational Fluid Dynamics (CFD) is a widely used method for simulating flow patterns. Numerical simulations based on CFD offer the advantage of obtaining testing parameters without the need for physical experiments [9]. In this context, it becomes necessary to conduct simulations on shell and tube heat exchangers using SolidWorks [10]. The design of the heat exchanger, including tube diameter and arrangement, leads to unique heat transfer characteristics. This study involves heat transfer simulations with varying diameters and tube arrangements using SolidWorks, which can provide results closely aligned with actual test outcomes, thereby reducing costs and time associated with experimental tests. The material data for this research is derived from the material libraries within the SolidWorks application.

II. EXPERIMENTAL SETUP

The geometry employed adheres to the standard set by the Tubular Exchanger Manufacturers Association (TEMA). Two tube arrangement variations, specifically triangular and rotated triangular, were considered in this study [11]. The subsequent descriptions outline the geometric configurations of the tube arrangements designed using SolidWorks for this investigation:

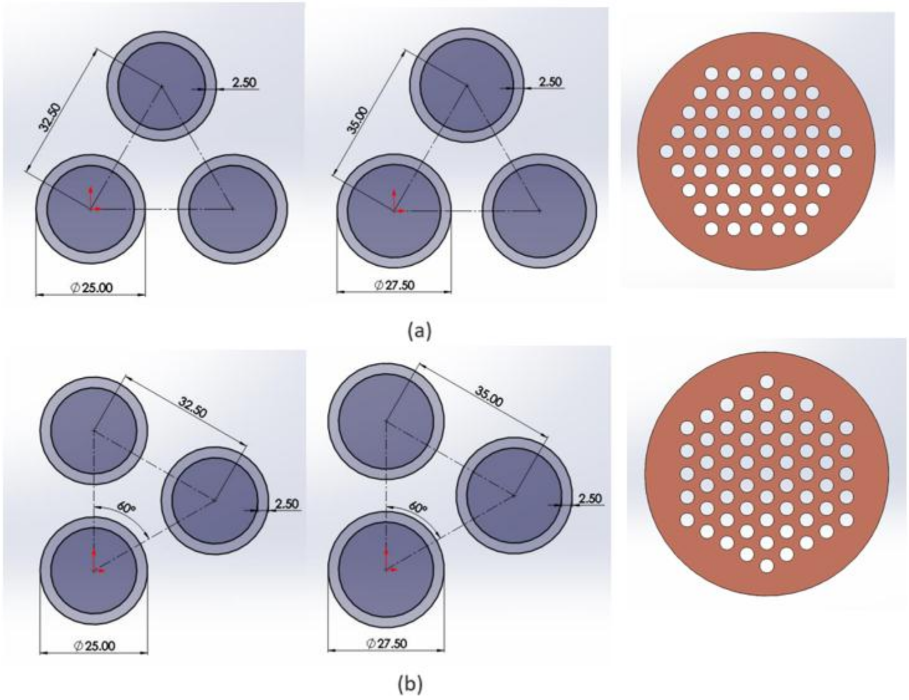


Figure 1. Tube arrangement

In the heat exchanger planning process, a heat load is required to be charged to the heat exchanger. The following operational data will be used as a heat exchanger load.

Table 1. Operational data of heat exchangers

Cooled fluid	Lubricating oil
inlet temperature	59^0 C
outlet temperature	35^0 C – 45^0C
Flow rate	0,2 kg/s
Cooled fluid	water
inlet temperature	26^0 C
Flow rate	0,5 kg/s

Meshing involves the division of objects into smaller elements, which will subsequently undergo simulated modeling. This process is executed according to the geometry of the control volume. The mesh employed in this case is of the hexahedral type. The focus of the flow analysis is on the fluids circulating within the shell and outside the tube. The modeling process progresses by generating a mesh, effectively breaking down the model into smaller components for analysis. Before finalizing the mesh size, it is essential to conduct a convergence test to determine an optimal mesh size that yields stable results in the testing phase.

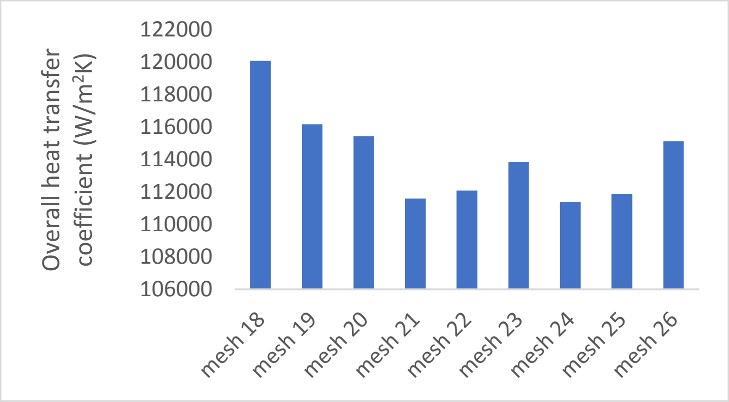
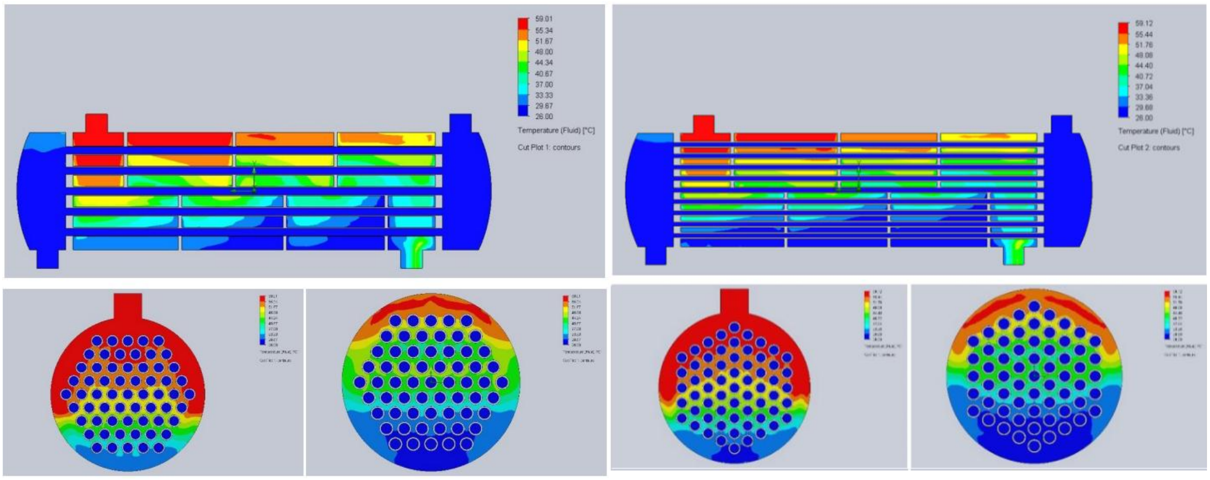


Figure 2. Mesh Convergence Test Results

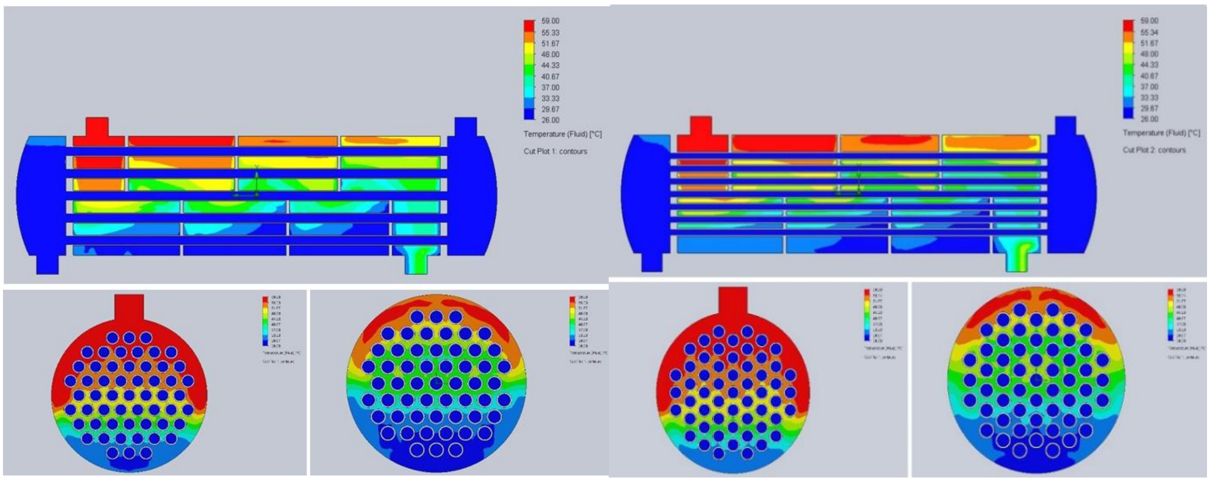
III. RESULT AND DISCUSSION

The figure below illustrates a visualization derived from the simulation outcomes of the temperature distribution cut plot, considering the tube arrangement, at an inlet mass flow rate of 0.5 kg/s for the cold fluid and 0.2 kg/s for the hot fluid.



(a) Triangular (b) Rotated triangular

Figure 3. Cut plot of temperature distribution with 61 tubes



(a) Triangular (b) Rotated triangular

Figure 4. Cut plot of temperature distribution with 55 tubes

The deep blue hue serves as an indication of the minimum bottom temperature value. The cold fluid's inlet temperature is set at 260°C. As the cold fluid travels within the tube, it undergoes friction with the inner surface of the pipe, initiating a heat transfer process. Simultaneously, the hot fluid, with a temperature of 590°C, flows outside the pipe, transferring heat from the tube's outer wall. This heat is then conveyed to the cold fluid on the inner side of the tube, a phenomenon referred to as conduction.

The hot fluid near the inlet exhibits a higher temperature compared to the fluid closer to the shell outlet. Consequently, the heat absorbed by the hot fluid undergoes convection, propagating to the cold fluid in a one-dimensional manner.

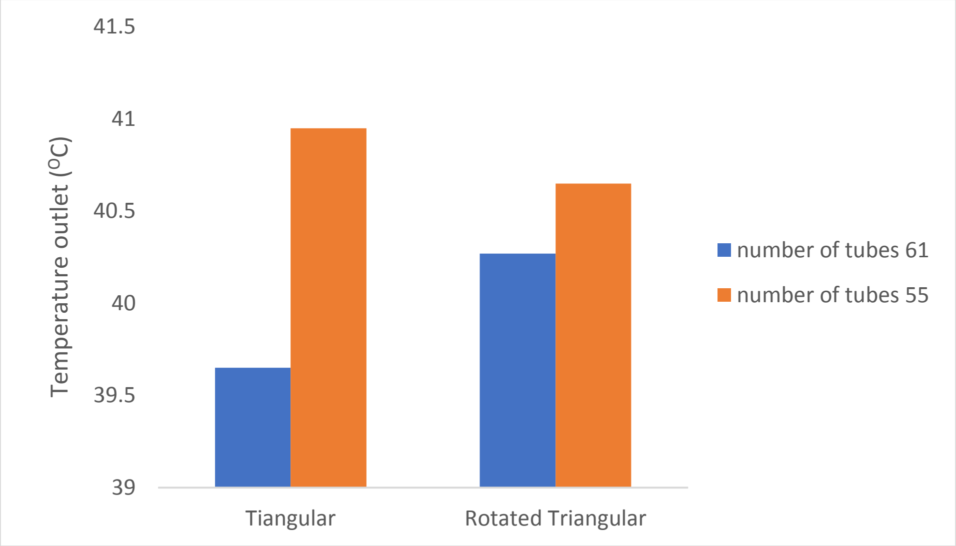


Figure 5. Graph of hot fluid outlet temperature distribution

The simulation results show that the number of tubes 61 produces a lower exit temperature in both the triangular and rotated triangular arrangements. The highest temperature was obtained in a triangular arrangement of 55 tubes, namely 40.95°C, while the lowest temperature occurred in a triangular arrangement with 61 tubes, namely 39.65°C. This observation is supported by the visual representation in Figure 3 and Figure 4, where the heat propagation in the heat exchanger with 61 tubes is shorter compared to the 55 tube arrangement. The increased turbulence in the 61-tube configuration contributes to this phenomenon, leading to more efficient heat transfer.

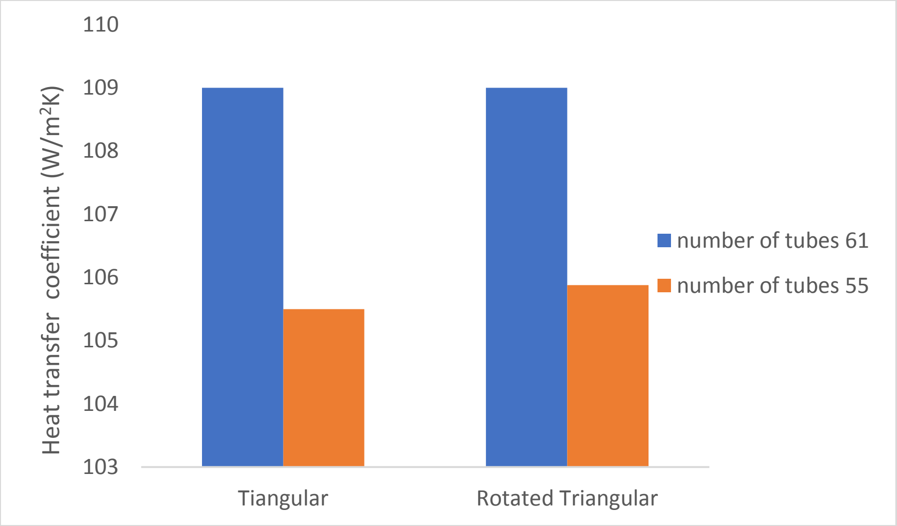


Figure 6. Graph of the overall heat transfer coefficient

Figure 6, a graph illustrates the overall heat transfer coefficient across different tube arrangements and tube numbers. Within the triangular and rotated triangular arrangements, the configuration with 61 tubes exhibits a higher coefficient value compared to the one with 55 tubes. This difference is attributed to the heightened turbulence experienced by the configuration with 61 tubes in comparison to the one with 55 tubes. The overall heat transfer coefficient is influenced by factors such as high heat transfer in both hot and cold fluids, fluid flow rate, tube cross-sectional area, and average temperature (ΔTlm).

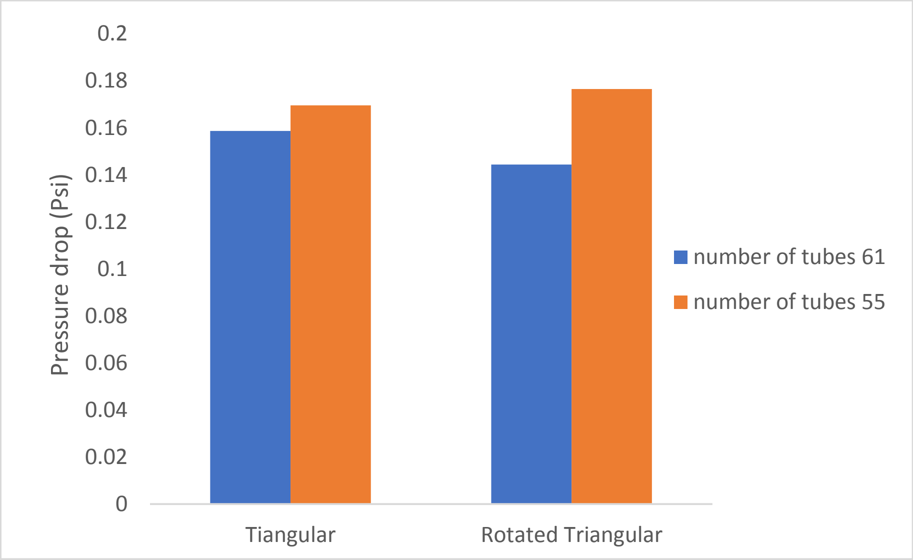


Figure 7. Graph of pressure drop on the shell side

Figure 7 depicts a graph illustrating the pressure drop on the shell side. In the tube rotated triangular arrangement with 55 tubes, the highest recorded pressure drop is 0.1766 psi, whereas the lowest pressure drop is observed in the rotated triangular tube arrangement with 61 tubes at 0.1444 psi. The magnitude of the pressure drop is influenced by factors such as the friction factor value, the number of tubes, and the flow area. A larger flow area results in a smaller pressure drop, and vice versa.

The graph clearly illustrates that the configuration with 55 tubes experiences a higher pressure drop compared to the configuration with 61 tubes. This discrepancy may be attributed to the larger flow area in the arrangement with 61 tubes, leading to increased resistance in the configuration with 55 tubes.

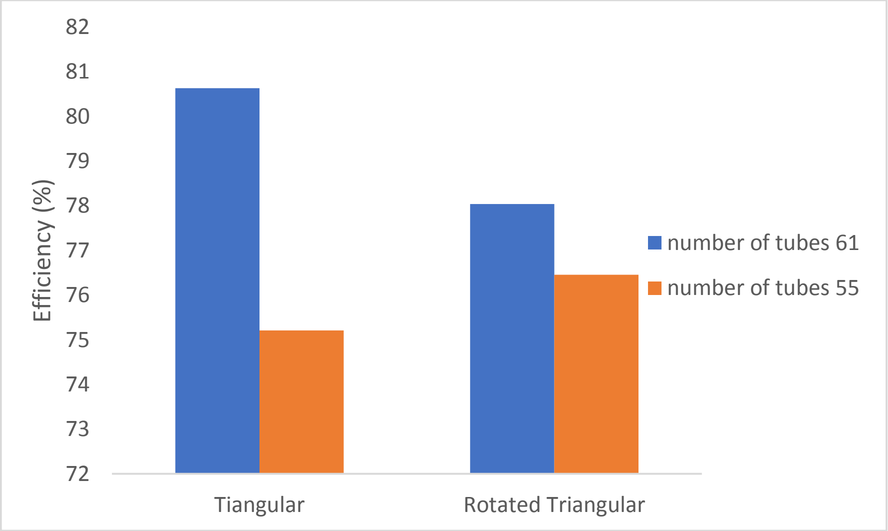


Figure 8. Graph of shell and tube heat exchanger effectiveness

The depicted figure presents the effectiveness values of the heat exchanger for different tube arrangements and quantities. The simulation results indicate that both the tube arrangement and the number of tubes play a significant role in influencing the effectiveness of shell and tube heat exchangers. Heat exchangers with 61 tubes provide higher efficiency compared to heat exchangers with 55 tubes in triangular and rotating triangle arrangements. The highest efficiency was obtained in a triangular heat exchanger with 61 tubes of 80.63%.

IV. CONCLUSION

Variations in tube arrangement and number of tubes greatly influence of overall heat transfer coefficient. The simulation results show that the number of tubes with a triangular arrangement of 61 has the highest overall heat change coefficient. The number of tubes 61 produces a lower exit temperature compared to the number of tubes 55 in both the triangle and rotating triangle arrangements. Variations in tube arrangement and number of tubes greatly influence the pressure drop value. The pressure drop value on the shell side is highest in the variation of the rotated triangular tube arrangement with a number of tubes of 55. The highest heat exchange efficiency occurs in the triangular tube arrangement with a number of tubes of 61.

REFERENCES

[1]. Selbaş, R., Kızılkan, Ö., & 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: Process Intensification, 45(4), 268-275.

[2]. De, D., Pal, T. K., & Bandyopadhyay, S. (2017). Helical baffle design in shell and tube type heat exchanger with CFD analysis. International journal of heat and technology, 35(2), 378-383.

[3]. Sembiring, A.F.B., Siahaan, Johan, C., Siahaan, S. (2022). Analysis of the horizontal francis turbine bearing cooling system unit 1 with 6.7 MW power and 750 rpm rotation at PLTA Pakkat PT. Synergy Sakti Sentosa. JITM 03 (01)

[4]. Incropera, Frank P., Dewitt, David P., Bregman, Theodore L., Lavine, Adrienne S. (2007). Fundamentals of heat and mass transfer 7thedition. United State of America. John Wiley & sons, Inc.

[5]. Arani, A. A. A., & Moradi, R. (2019). Shell and tube heat exchanger optimization using new baffle and tube configuration. Applied Thermal Engineering, 157, 113736.

[6]. Prithiviraj, M., & Andrews, M. J. (1998). Three dimensional numerical simulation of shell-and-tube heat exchangers. Part I: foundation and fluid mechanics. Numerical Heat Transfer, Part A Applications, 33(8), 799-816.

[7]. Prasad, A. K., & Anand, K. (2020). Design Analysis of Shell Tube Type Heat Exchanger. Int. J. Eng. Res. Technol, 9(01).

[8]. Pal, E., Kumar, I., Joshi, J. B., & Maheshwari, N. K. (2016). CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. Chemical engineering science, 143, 314-340.

[9]. Affandi, Nurlan. (2018). Performance simulation of a shell and tube type heat exchanger with helical baffles and disk and donut baffles. JTM 06 (01), 61-68

[10]. Alit,I.B.,& IGNK Yudhyadi. (2023). Simulation of Shell and Tube type Heat Exchanger for square geometry using Computational Fluid Dynamics Method. World Journal of Advanced Engineering Technology and Sciences. 9(2), 345-350.

[11]. Byrne, Richard C. (2019). Standards of Tubular Exchanger Manufacturers Association 10th (TEMA). Tubular Exchanger Manufacturers Association, Inc.