World Journal of Advanced Engineering Technology and Sciences

eISSN: 2582-8266

Cross Ref DOI: 10.30574/wjaets

Journal homepage: https://wjaets.com/

(Research Article)

Check for updates

Simulation of shell and tube type heat exchanger for square geometry using computational fluid dynamics method

Ida Bagus Alit * and IGNK Yudhyadi

Department of Mechanical Engineering, Faculty of Engineering, University of Mataram, Jl. Majapahit No. 62 Mataram-Nusa Tenggara Barat 83125, Indonesia.

World Journal of Advanced Engineering Technology and Sciences, 2023, 09(02), 345–350

Publication history: Received on 17 July 2023; revised on 25 August 2023; accepted on 28 August 2023

Article DOI: https://doi.org/10.30574/wjaets.2023.9.2.0242

Abstract

Heat exchangers have various types of design models, one example is the shell and tube type heat exchanger. This tool is useful for raising or lowering the temperature. One of its uses is in the hydroelectric power generation industry, namely as an oil cooler bearing. This study aims to design and simulate a shell and tube type heat exchanger with tube inner diameters of 20 mm and 25 mm and also tube arrangement square and rotated square. From the design results obtained a total length of 1318.59 mm, 300 mm outer diameter of the shell and 320 mm inner diameter of the shell. The simulation results show that variations in tube diameter and tube arrangement greatly affect the performance of the heat exchanger, the highest value for the overall heat transfer coefficient is found in the tube square arrangement variation with the number of tubes 61 of 112.15 W/m2.K and the lowest value in the tube arrangement variation square with the number of tubes 55 is 106.48 W/m2.K, the highest value of pressure drop on the shell side is found in the variation of the tube square arrangement with the number of tubes 55.

Keywords: Shell and Tube Heat Exchanger; Bearing; Heat Transfer Coefficient; Pressure Drop

1. Introduction

In industrial companies, factories, and power plants play a very important role in the demand for heat exchangers. The demand for heat exchangers is increasing due to the high production volume needed in that industry. The rapid development of the fast-moving industry will also demand a significant energy supply. One of the power plant centers that produce electrical energy is a hydroelectric power plant. This power plant is one of the facilities that harness the flow of water to be transformed into electrical energy. This power plant operates by converting the kinetic energy of flowing water (from dams or waterfalls) into mechanical energy (assisted by water turbines), and then converting the mechanical energy into electrical energy with the help of a generator. The heat exchange process between two fluids at different temperatures, separated by a solid wall, occurs in many engineering applications [1], [2]. One engineering application that utilizes heat exchangers in power plants is bearings, which play a role in maintaining the position of the turbine shaft along a single axis. In the maintenance of turbine bearings, a cooling system is implemented for the turbine bearings, utilizing oil lubrication [3]. For the cooling of lubricating oil, a cooling system employed is the shell and tube heat exchanger type [4]. A heat exchanger consists of a shell, which is in the form of a casing, and tubes that are pipes placed inside that shell. Fluid flowing at different temperatures in these two parts results in the transfer of heat [5]. The direction of flow of the two fluids can occur in parallel, counterflow, crossflow, or a mixture [6]. Parallel flow occurs when both fluids enter from the same direction, flow in the same direction, and exit in the same direction as well. Counterflow happens when both fluids enter from opposite directions, flow in opposite directions, and exit through opposite outlets [7]. In a shell-and-tube heat exchanger, one fluid flows inside the tubes, while another fluid flows outside the tubes. The tube pipes are designed to be within a cylindrical space called the shell, arranged in such a way that these tube pipes are parallel to the axis of the shell [8]. Baffles are designed on the shell side to increase the speed

* Corresponding author: Ida Bagus Alit

and effectiveness of the fluid flow outside the tubes. Pressure drop and heat transfer can be more accurately predicted for the tube section [9]. Pressure loss or pressure drop in fluid flow occurs due to friction between the pipe walls and the flowing fluid inside them. Various methods that have been developed to analyze heat transfer in heat exchangers are actual experiments and simulations. In previous studies, a short heat exchanger with and without baffles had been tested for flow patterns [10], the effect of the number of tubes and baffles on the effectiveness of shell and tube heat exchangers [11]. In his research, 3 variations of the number of tubes and 3 variations of the number of baffles were applied with 3 variations of the fluid flow rate in each variation. From the research results it is known that variations in the number of tubes and the number of baffles greatly affect the overall heat transfer coefficient and the effectiveness of the heat exchanger. The weakness of experimental research is that it takes a long time and costs a lot.

The method widely used to analyze flow simulation is computational fluid dynamics (CFD). The application of numerical simulation based on computation with the assistance of CFD is utilized, among other reasons, due to its capability to obtain testing parameters without conducting actual experiments [12]. This numerical simulation aids in understanding the temperature distribution patterns occurring within the shell-and-tube heat exchanger and their impact on different tube arrangements, namely triangular, rotated triangular, square, and rotated square. From the background above, it is necessary to test shell and tube heat exchangers using solidworks simulations. As a result of the design of the heat exchanger, the diameter and arrangement of the tubes, heat transfer occurs which is different from other heat exchangers. In this study, heat transfer simulations with different diameters and tube arrangements were carried out on the effectiveness of shell and tube heat exchangers using solidworks. Solidworks can provide results that are close to the actual test results from the data. Simulated testing can also reduce the cost and time of experimental tests making them more efficient. The material data used in this research is taken from the material libraries in the Solidworks application.

2. Material and methods

The geometry used is standard from the Tubular Exchanger Manufacturers Association (TEMA) standard. There are 2 variations of the tube arrangement, namely square and rotated square [13]. The following are the geometric shapes of the tube arrangement variations designed using Solidworks in this study:

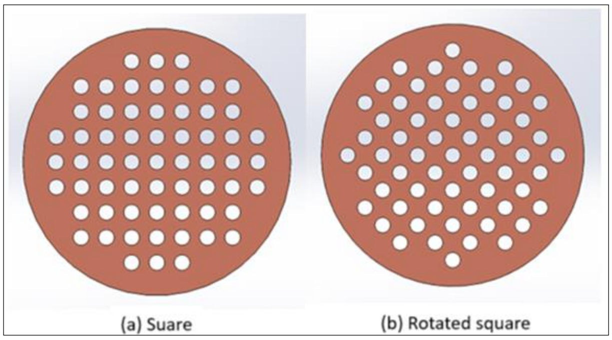

Figure 1 Tube arrangement

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 ^{0}C
Flow rate	0,2 kg/s
Cooled fluid	water
inlet temperature	26 ^{0}C
Flow rate	0,5 kg/s

Meshing is dividing objects into smaller elements which will later be simulated modeling. The meshing process is operated based on the control volume geometry. The mesh used is a hexaedral type. The flow characteristics to be analyzed are the fluids flowing inside the shell and outside the tube. Modeling is continued by creating a mesh or dividing the model into smaller parts for analysis. Before determining the mesh size, it is necessary to carry out a convergence test to obtain an effective mesh size with stable results in the test.

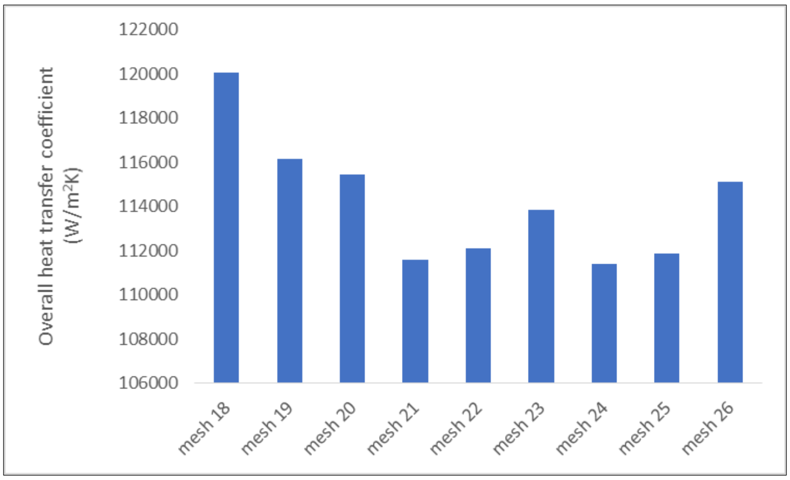

Figure 2 Mesh Convergence Test Results

3. Discussion

Visualization based on the simulation results of the temperature distribution cut plot with the tube arrangement at the inlet mass flow rate of cold fluid of 0.5 kg/s and the mass flow rate of hot fluid of 0.2 kg/s is presented in the figure below.

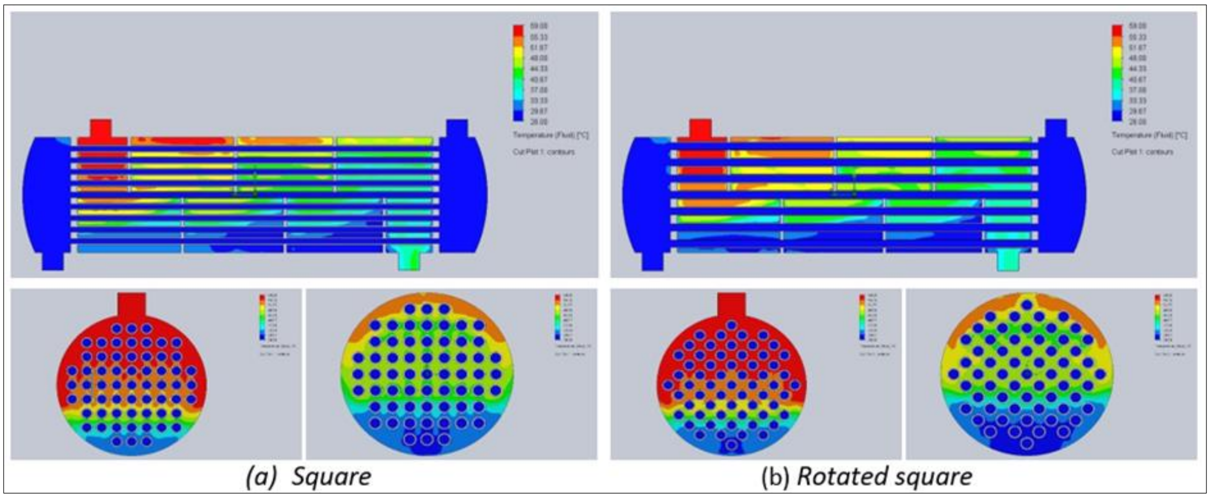

Figure 3 Cut plot of temperature distribution with 61 tubes

Dark blue as an indicator of the minimum bottom temperature value. The cold fluid inlet temperature setting has the same value of 260C. when cold fluid flows inside the tube it will experience friction with the inner surface of the pipe. When friction occurs, a heat transfer process occurs, where the hot fluid flows outside the pipe with a fluid temperature of 590C and will transfer heat from the outer wall of the tube to be forwarded so that the cold fluid flows on the inside of the tube receiving heat that is channeled from the outer wall of the tube or also known as conduction phenomenon. The hot fluid closest to the inlet flow has a higher temperature than the fluid closer to the shell outlet flow. Meanwhile, because of that, the heat received by the hot fluid will propagate to the cold fluid that exists in one dimension through the convection process.

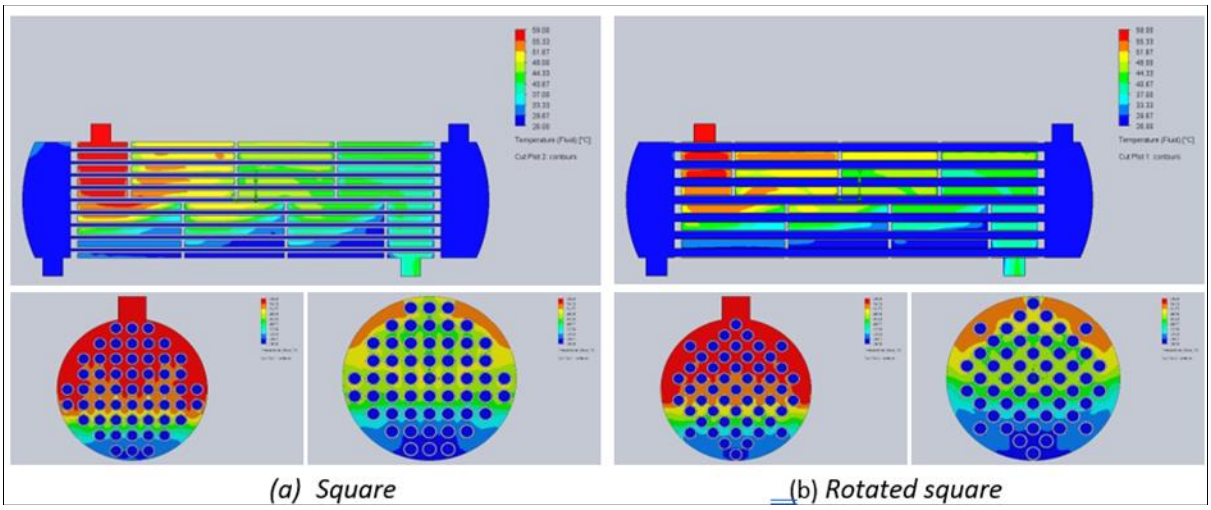

Figure 4 Cut plot of temperature distribution with 55 tubes

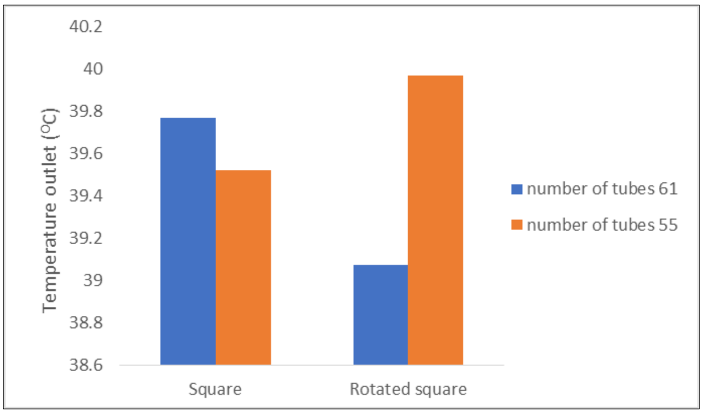

Figure 5 Graph of hot fluid outlet temperature distribution

The simulation results show that the highest outlet temperature value for the rotated square arrangement with 55 tubes is 39.970C and the lowest output temperature value is 39.07 ⁰C for the rotated square arrangement with 61 tubes. This phenomenon is also reinforced by Figure 3 and Figure 4 where the heat propagation in the rotated square arrangement of tubes with the number of tubes 61 is shorter than the rotated square arrangement with the number of tubes 55. This is because the turbulence that occurs in the number of tubes 61 is greater than the number of tubes 55.

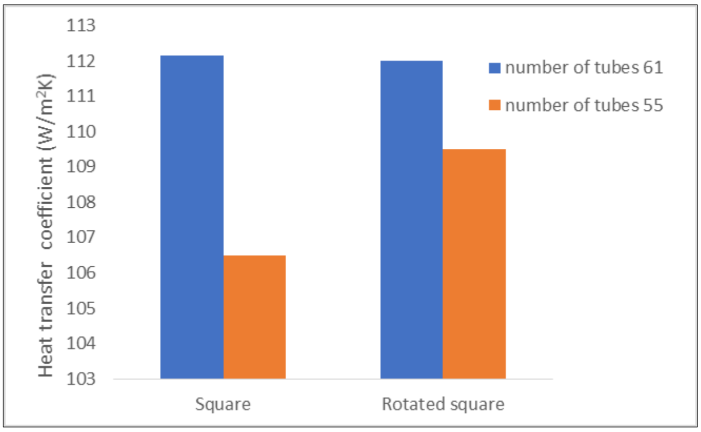

Figure 6 Graph of the overall heat transfer coefficient

Figure 6 shows a graph of the overall overall heat transfer coefficient for various tube arrangements and the number of tubes. Based on variations in the tube square and rotated square arrangement, the number of tubes 61 that has a greater coefficient value than the number of tubes 55. This is because the number of tubes 61 experiences considerable turbulence compared to the number of tubes 55. The overall heat transfer coefficient is affected by heat transfer which is high occurs in both hot and cold fluids, fluid flow rate, tube cross-sectional area and average temperature (ΔTlm).

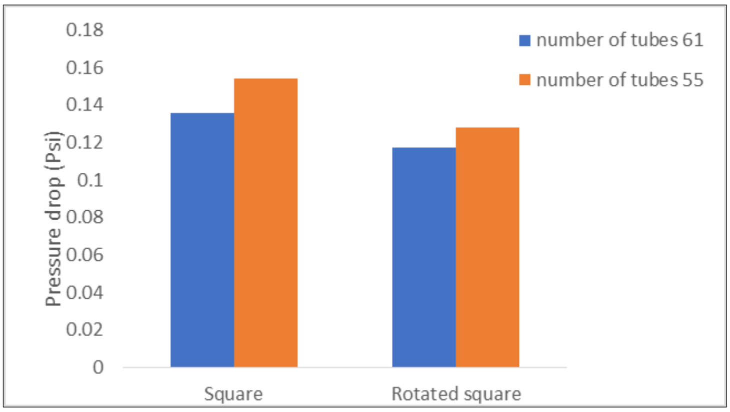

Figure 7 Graph of pressure drop on the shell side

Figure 7 shows a graph of the pressure drop on the shell side. The highest pressure drop value in the variation of the tube square arrangement with the number of tubes 55 is 0.1540 psi. While the smallest pressure drop value is in the variation of the rotated square tube arrangement with the number of 61 tubes of 0.1175 psi. The size of the pressure drop is influenced by the value of the friction factor, the number of tubes and the area of flow, the greater the area of flow, the smaller the value of the pressure drop and vice versa. In the figure above it can be seen that the number of tubes 55 has a greater pressure drop value than the number of tubes 61. This could be due to the area of flow on the number of tubes 61 being larger than the number of tubes 55 so that the resistance occurs is greater on the number of tubes 55 compared to the number of tubes 61.

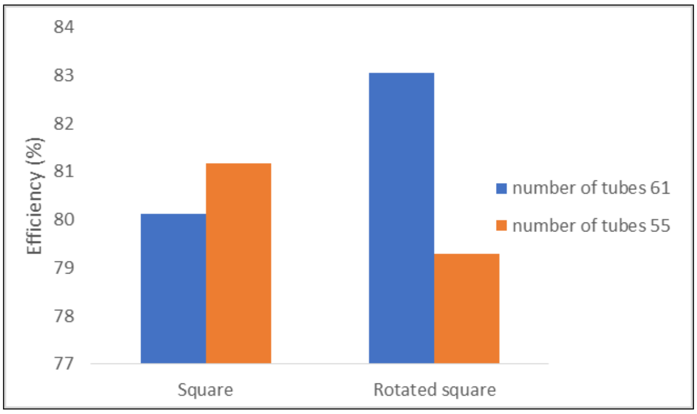

Figure 8 Graph of shell and tube heat exchanger effectiveness

The figure above shows the value of the effectiveness of the heat exchanger for variations in tube arrangement and the number of tubes. Based on the simulation results, it is known that the arrangement of tubes and the number of tubes affect the effectiveness of shell and tube heat exchangers. It can be seen from the graph above that the heat exchanger has the greatest effectiveness value of 83.04% in the rotated square tube arrangement with 61 tubes. While the lowest effectiveness value is in the rotated square arrangement variation with 55 tubes, namely 79.29%. The effectiveness value is directly proportional to the heat transfer rate value, the greater the heat transfer rate value, the effectiveness value will also be greater.

4. Conclusion

The results of the design of the heat exchanger obtained the dimensions of the total length of 1318.59 mm, the inner diameter of the shell is 325 mm, the outer diameter of the shell is 345 mm. Variations in the arrangement of tubes and the number of tubes greatly affect the overall heat transfer coefficient, the highest value is found in the variation of the arrangement of tube squares with the number of tubes 61. Variations in the arrangement of tubes and the number of tubes greatly affect the value of the pressure drop. The highest pressure drop value on the shell side is found in the variation of the tube square arrangement with the number of tubes 55. The highest heat exchanger efficiency occurs in the tube rotated square arrangement with the number of tubes 61.

Compliance with ethical standards

Disclosure of conflict of interest

The Authors proclaim no conflict of interest.

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, Adelia F. Br., Siahaan, Johan Cristian., Siahaan, Sihar. (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. Journal ilmiah teknik mesin 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] Adhitiya, Anggareza dan Ichsani, Djatmiko. (2013). Simulation of the performance of a shell and tube type heat exchanger with double segmental baffles against helical baffles. Journal Teknik POMITS, 2 (3) ISSN: 2337-3539

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

[9] Gustyawan, Bayu Esha. (2017). Helical baffle variation analysis on shell and tube heat exchanger based on Computational Fluid Dynamics (CFD) [Final Project. Sepuluh Nopember Institute of Technology (ITS) Surabaya].

[10] 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.

[11] Irawan, Dwi., Mafruddin., Rian., Wibowo, Mukti., Anggara, Zul. (2020). Effect of the number of tubes and baffles on the effectiveness of the shell and tube heat exchanger. SNPPM-2 Muhammadiyah Metro University. ISBN 978-623-90328-5-2

[12] 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

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