JOURNAL OF MECHANICAL ENGINEERING, SCIENCE, AND INNOVATION
ITATS
Research Article

Simulation Performance Analysis of Shell and Tube Heat Exchanger Using Comsol Multiphysics 5.6 Software

Journal of Mechanical Engineering, Science, and Innovation
e-ISNN: 2776-3536
2023, Vol. 3, No. 1
DOI: 10.31284/j.jmesi.2023.v3i1.3544
ejurnal.itats.ac.id/jmesi

Yety Setyo Ningrum[1], Ririn Martin Erinda[1], Made Arsana[2], Erlinda Ningsih[1], and Anthony Francis Rajan[3]

[1]Adhi Tama Institute of Technology Surabaya, Indonesia
[2]Surabaya State University, Indonesia
[3]Alphasys 3D Printing Solutions, India

Corresponding author:
Erlinda Ningsih
Adhi Tama Institute of Technology Surabaya, Indonesia
Email: erlindaningsih84@itats.ac.id

Abstract

A heat exchanger is a very important tool in the fields of engineering and industry, especially in energy conversion. This study was aimed at determining the effects of hot and cold fluid flow velocity on the overall heat transfer coefficient (UA) and effectiveness in shell and tube heat exchangers using COMSOL Multiphysics 5.6 software. Another research objective was to observe the phenomena of heat transfer in the shell and tube type heat exchanger at each hot and cold fluid flow velocity. The heat exchanger was designed with a total length of 800 mm and equipped with 18 tubes having a diameter of 2 in and a length of 600 mm. The material used for tube and shell construction was stainless steel. A simulation was carried out using COMSOL Multiphysics 5.6 software to determine the performance of the designed heat exchanger. The results of this simulation indicated that the effects of hot and cold fluid velocity were directly proportional to the value of UA. The heat exchanger have result the smallest UA value of 80.062 W/m^2.K, meanwhile the highest UA value of 174.950 W/m^2.K. The heat exchanger have result the minimum effectiveness value of 22.305% and the maximum effectiveness value of 52,047%. The second result is phenomenon stating that the surface temperature of the shell and tube would change along with the increasing velocity of both hot and cold fluids, signifying the heat transfer such as conduction and convection from the fluid to the shell or tube.

Keywords: *Simulation, COMSOL Multiphysics, Heat exchanger, Velocity*

Received: September 20, 2022; Received in revised: April 11, 2023; Accepted: April 14, 2023
Handling Editor: Rina
This article was presented at the ICONIC-RS 2022, 31 March-1 April 2022, Jakarta, Indonesia

INTRODUCTION

Heat exchanger is a very important equipment in industry and engineering, especially in terms of energy conversion and is widely used in almost every process [1-2]. Equipment that is useful for transferring a certain amount of energy in the form of heat from one fluid to another which has a temperature ratio [3-4]. This type of exchanger designed with various configurations based on the application [5-6]. Of the several types of heat exchangers that are known in general is the shell and tube type. The fluid flow in the shell and tube heat exchanger is either cocurrent or countercurrent [7-8].

In optimizing efficient design work, a method is needed. The methods commonly used are the Log Mean Temperature Difference (LMTD) method and the number of heat transfer units (NTU) [9-10], but this method has the disadvantage that the process takes a long time and is expensive [11]. Therefore, it is necessary to conduct a study related to improving the design of a more advanced heat exchanger. Optimization of designs that are concise and effective based on modern computers that can be useful in various ways [12-13]. The CFD approach can be very attractive because it can test the performance of exchanger design results without having to make prototypes and can solve the system completely, can solve equations numerically, and can be applied to any type of heat exchanger with a variety of models [14].

Studies related to the use of approaches with CFD are increasing, this is due to the development of modern computing to solve numerical techniques [15-17]. Studies that have been carried out on the effect of flow patterns on the performance of the heat meter [18], baffle [19], design [20], nozzle [21]. Advanced computer science, mathematics, and engineering have resulted in the CFD or Computational Fluid Dynamics program. The CFD method is capable of visualizing fluid flow phenomena and processes in them such as chemical reactions, heat transfer, multiphase, and others [18]. The results of the temperature contour simulation in this study show that the water temperature increases with the path length [22]. The temperature distribution is as follows: the water temperature on the inlet side is 30°C, the water temperature on the outlet side of line 1 is 30.61°C, the water temperature on the outlet side of line 2 is 30.75°C, and the water temperature on the outlet side. 3 of 31.05°C [23]. Based on the previous research that has been described, this study will simulate the effect of flow velocity on the overall heat transfer coefficient (UA) and the effectiveness of shell and tube heat exchangers using Comsol Multiphysics 5.6 software.

Based on the literature review that has been carried out, it is necessary to conduct an assessment related to the performance of heat exchangers that have been designed on a lab scale with shell and tube 1-2 types. With the Comsol Multiphysics 5.6 CFD approach to be more effective and efficient.

METHODS

In this study the heat exchanger used was a type of Shell and Tube 1-2 Exchanger with Shell and Tube materials made of Stainless Steel. For the fluid flowing in the heat exchanger, both the hot fluid and the cold fluid are water. The dimensions of the heat exchanger based on a literature study for the laboratory scale are shown in Figure 1. The following are the heat exchanger specifications presented in Table 1:

Variable of Riset

Inlet velocity of hot fluid : 0,02468 m/s; 0,03455 m/s; 0,04443 m/s; 0,04936 m/s.

Cold fluid inflow velocity : 0,00987 m/s; 0,01975 m/s; 0,02962 m/s; 0,03949 m/s.

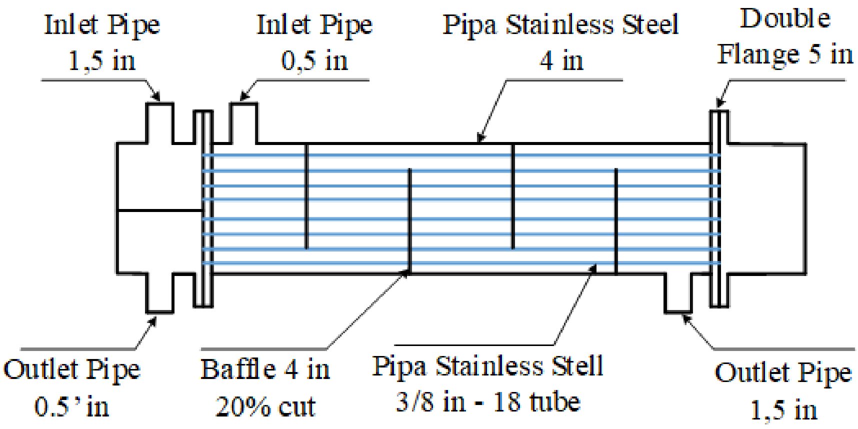

Figure. 1. Heat Exchanger Design

Table 1. Spesification of Heat Exchanger.

Parameter	Description
Shell and Tube Type	1-2 Exchnger
Shell and Tube Material	Stainless Steel
Shell, Tube Size	2 in sch 40, 3/8 in
Inlet and Outlet Pipe	1,5 in
Shell Length	600 mm
Number of Tubes	18 pieces
Shell fluid	Water (70°C)
Tube fluid	Water (30°C)

CFD analysis

A software package that models fluid flow and heat transfer with a main focus on optimization through studies of thermal performance, pressure drop, fouling, and fluid maldistribution using computational fluid dynamics (CFD) (1,5,11-14 in 4807). This CFD software allows modeling and simulating various fluid processes, fluid flow, heat transfer, turbulence (11 in 8822).

RESULTS AND DISCUSSIONS

Simulations were carried out using Comsol Multiphysics 5.6 software, the results of the total heat transfer coefficient (UA) on the shell and tube heat exchanger for each variable velocity of hot and cold fluid flow can be seen in Table 3 :

Velocity Effect on UA

The first simulation was carried out to determine the effect of hot and cold fluid flow rates on the UA in a shell and tube heat exchanger using Comsol Multiphysics 5.6 software. The variable velocity of the hot fluid used is 0,02468 m/s; 0,03455 m/s; 0,04443 m/s; 0,04936 m/s with cold fluid velocity variations are 0,00987 m/s; 0,01975 m/s; 0,02962 m/s; 0,03949 m/s for each hot fluid velocity. Based on Figure 3, it is found that the velocity of hot fluid and cold fluid is directly proportional to the value of UA. From Figure 3, it can be seen that the highest UA value occurs when the velocity of the hot fluid is 0,04936 m/s and the velocity of the cold fluid is 0,03949 m/s, which is 174,950 W/m^{2}K. While the

Table 3. Research Result Data

Hot Fluid Flow Rate (m/s)	Cold Fluid Flow Rate (m/s)	Overall Heat Transfer Coefficient (UA) (W/m²K)	Effectiveness (Ɛ) (%)
0,02468	0,00987	80,062	52,047
	0,01975	107,890	36,077
	0,02962	123,670	28,004
	0,03949	134,950	22,684
0,03455	0,00987	86,768	49,838
	0,01975	120,400	34,585
	0,02962	140,420	27,146
	0,03949	155,190	22,305
0,04443	0,00987	91,014	50,107
	0,01975	128,720	34,842
	0,02962	151,890	27,259
	0,03949	169,360	22,487
0,04936	0,00987	92,603	50,712
	0,01975	131,910	35,277
	0,02962	156,380	27,654
	0,03949	174,950	22,867

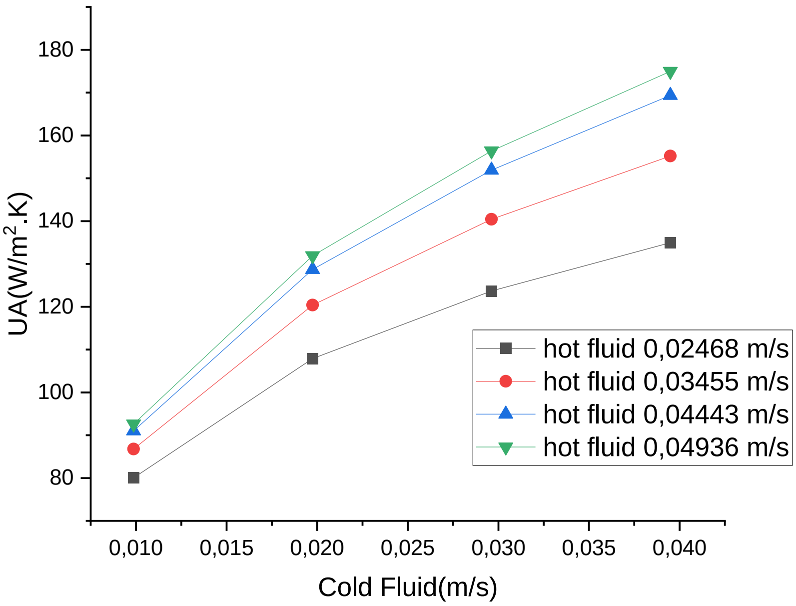

Figure 3. Graph of Velocity Effect on UA

smallest UA value is obtained when the hot fluid velocity is 0,02468 m/s and the cold fluid velocity is 0,00987 m/s, which is 80.062 W/m².

In previous studies, the smallest heat transfer coefficient is obtained when the cold fluid velocity is 1,19 m/s, which is 668,13 W/m².°C. While the highest heat transfer coefficient is obtained when the cold fluid velocity is 2,91 m/s, which is 1367,88 W/m².°C. From these data it can be concluded that the higher the cold fluid flow rate, the greater the heat transfer coefficient. So that the results shown in Figure 3 are in accordance with the existing theory [24].

Effect of velocity on effectiveness (Ɛ)

In Figure 4 it can be seen that there was a consistent decline. The optimal effectiveness value is at a cold fluid flow velocity of 0,00987 m/s and a hot fluid flow velocity of 0,02468 m/s which is 52,047%. This speed is an efficient condition in heat transfer events because it is able to strengthen the contact between hot fluid and cold fluid, so that this speed variation is suitable for use when operating a shell and tube type heat exchanger.

In his research [25], proves that the effectiveness will decrease drastically along with the increase in mass flow of fluid on the tube side. In this statement, it can be seen that the effectiveness of the tool decreases along with the increase in the velocity of the fluid flow in the tube and in this study the tube part is flowed by cold fluid so that the results shown in Figure 4 are in accordance with the applicable theory. Based on these data, it can be concluded that the effectiveness value is inversely proportional to the fluid flow velocity, the higher the fluid flow velocity, the lower the effectiveness of the tool. The minimum effectiveness value is at a cold fluid flow velocity of 0,03949 m/s and a hot fluid flow velocity of 0,03455 m/s which is 22,305%.

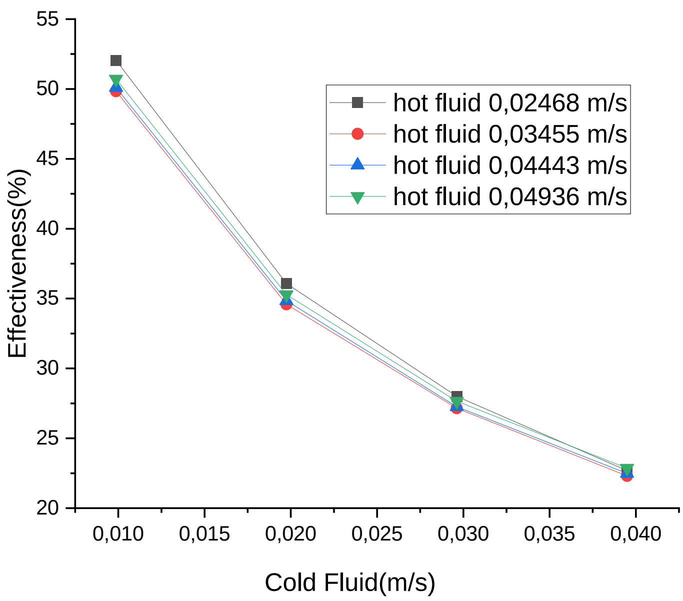

Figure 4. Graph of the Effect of Velocity on Effectiveness (Ɛ)

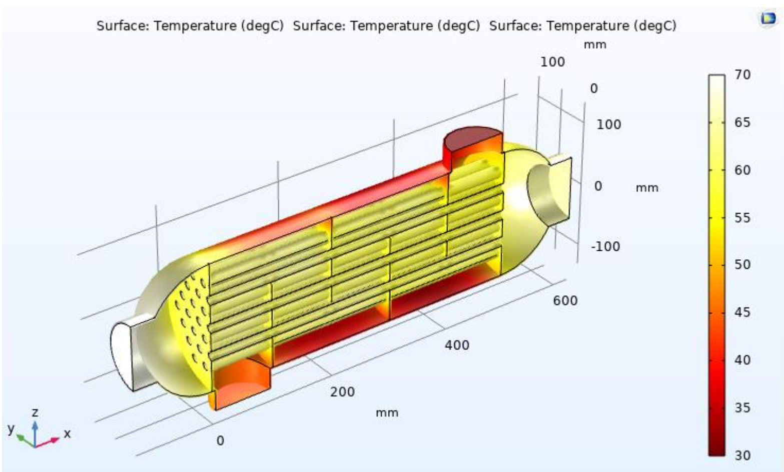

Figure 5. Temperature profile on the heat exchanger surface

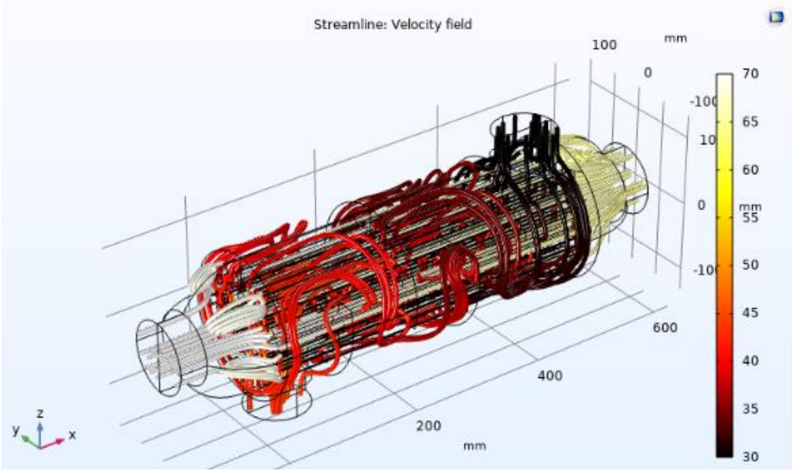

Figure 6. Temperature profile of the fluid

Parallel Slice Plot of Temperature and Velocity Distribution on Shell and Tube Heat Exchanger

From the solution that has been simulated by comsol software, it is obtained that the research variable data will be studied. The hot fluid is simulated at an inlet temperature of 343.15 K, while the cold fluid is simulated at an inlet temperature of 303.15 K.

Figure 5 illustrates the temperature profile on the surface of the heat exchanger from the hot fluid entering through the shell side which is depicted in white with a temperature of 70°C, while the cold fluid enters through the side depicted in red which flows directly into the tube. Figure 6 is a velocity profile which is a picture of the movement of fluid flow velocity that occurs on the shell side and tube side, hot fluid enters from the shell which is white then goes straight to the tube and hot fluid exits through the shell side with a yellow color, then there is a decrease in temperature. This is because the hot fluid is in contact with the cold fluid around the tube. Variations were observed gradually from the minimum to the maximum velocity over the cross-sectional area of the shell and tube heat exchanger. The cross-sectional plot shows the same tendency, namely the surface temperature of the shell and tube increases with increasing flow velocity in both hot and cold fluids as the point moves from the inlet side to the outlet.

Figure 5 and Figure 6 indicate that convection occurs on the shell side, conduction from the outer wall to the inner wall of the tube, and convection on the tube side. Convection on the shell side is due to the movement of molecules in the hot fluid which also moves along with the velocity of the hot fluid flow, while conduction from the outer wall to the inner wall of the tube occurs due to heat transfer that propagates through the intermediary wall or in this case stainless steel and convection on the side of the tube. tube due to the movement of hot fluid particles that occur in the tube.

CONCLUSIONS

The conclusions obtained from the simulation results are as follows:

1. The effect of hot fluid velocity and cold fluid velocity is directly proportional to the value of UA, while the effect of hot fluid velocity and cold fluid velocity is inversely proportional to the effectiveness value.
2. The phenomenon that occurs that the temperature on the surface of the shell and tube changes with increasing velocity of both hot and cold fluids which indicates that there is a heat transfer event, namely conduction and convection from the fluid to the shell or tube.

Further studies are needed regarding the performance of this heat exchanger so as to obtain a high effectiveness value. In addition to the flow rate, type, number and number of baffles, as well as the number of passes, pitch, square or triangular.

ACKNOWLEDGEMENTS

DECLARATION OF CONFLICTING INTERESTS

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

FUNDING

The author(s) disclosed receipt of the following no financial support for the research, authorship, and/or publication of this article.

REFERENCES

[1] R. Mukherjee, "s Figure 1. TEMA designations for shell-and-tube heat exchangers," no. February, 2015.
[2] J. Saari, "Faculty of Technology LUT Energy HEAT EXCHANGER," pp. 1–101, 2011.
[3] M. H. Mousa, N. Miljkovic, and K. Nawaz, "Review of heat transfer enhancement techniques for single phase flows," Renew. Sustain. Energy Rev., vol. 137, 2021.
[4] A. Dewan, P. Mahanta, K. S. Raju, and K. P. Suresh, "Review of Passive Heat Transfer Augmentation Techniques," Proc. Inst. Mech. Eng. Part A J. Power Energy, vol. 218, no. 7, pp. 509–527, 2004.
[5] K. Thulukkanam, "Heat Exchanger Design Handbook, Second Edition - Kuppan Thulukkanam," p. 1260, 2013.
[6] E. Ningsih, A. H. Fahmi, M. Riyanando, M. R. Faiz, E. C. Muliawati, and R. Izroiel, "Desain Shell and Tube Heat Exchanger (STHE) Tipe Counter Current dengan Material Stainless Steel", Journal of Industrial Process and Chemical Engineering, Vol.2 (2), pp. 124-130, 2022.
[7] S. S. Yogesh, A. S. Selvaraj, D. K. Ravi, and T. K. R. Rajagopal, "Heat transfer and pressure drop characteristics of inclined elliptical fin tube heat exchanger of varying ellipticity ratio using CFD code," Int. J. Heat Mass Transf., vol. 119, pp. 26–39, 2018.
[8] M. Farnam, M. Khoshvaght-Aliabadi, and M. J. Asadollahzadeh, "Heat transfer intensification of agitated U-tube heat exchanger using twisted-tube and twisted-tape as passive techniques," Chem. Eng. Process. - Process Intensif., vol. 133, pp. 137–147, 2018.
[9] M. M. Aslam Bhutta, N. Hayat, M. H. Bashir, A. R. Khan, K. N. Ahmad, and S. Khan, "CFD applications in various heat exchangers design: A review," Appl. Therm. Eng., vol. 32, no. 1, pp. 1–12, 2012.
[10] Safiullah, A. Ahmed, A. Khuhro, I. Tunio, and K. Nishida, "Non-Vaporizing and Vaporizing Diesel Spray Evaluation with Experimental and Computational Approaches," Quaid-e-Awam Univ. Res. J. Eng. Sci. Technol., vol. 19, no. 2, pp. 114–124, 2021.
[11] E. Ningsih, Fitriana, and D. Pratiwi, "Desain Alat Penukar Panas Tipe Shell and Tube dengan Material Stainless Steel," pp. 81–89, 2022.
[12] S. Jun and V. M. Puri, "3D milk-fouling model of plate heat exchangers using computational fluid dynamics," Int. J. Dairy Technol., vol. 58, no. 4, pp. 214–224, 2005.
[13] E. Ningsih, I. Albanna, A. P. Witari, and G. L. Anggraini, "Performance Simulation on

the Shell and Tube of Heat Exchanger By Aspen Hysys V.10," J. Rekayasa Mesin, vol. 13, no. 3, pp. 701–706, 2022.

[14] M. Ravikumar, R. K. Beemaraj, K. M. Arunraja, P. Pandiarajan, and R. Sundaresan, "Heat transfer analysis in fin and tube exchanger using CFD," Mater. Today Proc., vol. 52, pp. 1603–1605, 2022.

[15] S. Freund and S. Kabelac, "Investigation of local heat transfer coefficients in plate heat exchangers with temperature oscillation IR thermography and CFD," Int. J. Heat Mass Transf., vol. 53, no. 19–20, pp. 3764–3781, 2010.

[16] T. Ihli, R. Kruessmann, I. Ovchinnikov, P. Norajitra, V. Kuznetsov, and R. Giniyatulin, "An advanced He-cooled divertor concept: Design, cooling technology, and thermohydraulic analyses with CFD," Fusion Eng. Des., vol. 75–79, no. SUPPL., pp. 371–375, 2005.

[17] C. Abeykoon, "Modelling of Heat Exchangers with Computational Fluid Dynamics," Proc. 8th Int. Conf. Fluid Flow, Heat Mass Transf., no. May, 2021.

[18] C. Abeykoon, "Compact heat exchangers – Design and optimization with CFD," Int. J. Heat Mass Transf., vol. 146, p. 118766, 2020.

[19] E. Pal, I. Kumar, J. B. Joshi, and N. K. Maheshwari, "CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles," Chem. Eng. Sci., vol. 143, pp. 314–340, 2016.

[20] Safiullah, K. Nishida, Y. Ogata, T. Oda, and K. Ohsawa, "Effects of nozzle hole size and rail pressure on diesel spray and mixture characteristics under similar injection rate profile – experimental, computational and analytical studies under non-evaporating spray condition," Proc. Inst. Mech. Eng. Part D J. Automob. Eng., vol. 236, no. 2–3, pp. 310–321, 2022.

[21] Safiullah, K. Nishida, and Y. Ogata, "Evaporation and mixture formation characteristics of diesel spray under various nozzle hole size and injection pressure condition employing similar injection rate profile," Int. Commun. Heat Mass Transf., vol. 123, no. February, p. 105184, 2021.

[22] C. Abeykoon, "Improving the Performance of Shell-and-Tube Heat Exchangers by the Addition of Swirl," Int. J. Process Syst. Eng., vol. 2, no. 3, pp. 221–245, 2014.

[23] C. Damis Widiawaty, R. Gunadi, and D. A. Syuriadi, "Pemodelan dan Analisis Kinerja dengan Metode CFD," Politeknologi, vol. 16, no. 3, pp. 239–244, 2017.

[24] A. Husen, T. M. I. Akbar, and N. Cholis, "Analisis pengaruh kecepatan aliran fluida dingin terhadap efektivitas shell and tube heat exchanger," vol. 16, no. Vc, pp. 1–10, 2020.

[25] R. Shanahan and A. Chalim, "STUDI LITERATUR TENTANG EFEKTIVITAS ALAT PENUKAR PANAS SHELL AND TUBE 1-1 SISTEM FLUIDA GLISERIN –," vol. 6, no. 9, pp. 164–170, 2020.