REKAYASA MESIN V13 N3
eISSN 2477-6041 artikel 8, pp. 701 – 706, 2022

PERFORMANCE SIMULATION ON THE SHELL AND TUBE OF HEAT EXCHANGER BY ASPEN HYSYS V.10

Erlinda Ningsih
Tenaga Pengajar
Institusi Teknologi Adhi Tama Surabaya
Jurusan Teknik Kimia
erlindaningsih84@itats.ac.id

Isa Albanna
Tenaga Pengajar
Institusi Teknologi Adhi Tama Surabaya
Jurusan Sistem Informasi
isaalbanna@itats.ac.id

Aita Pudji Witari
Mahasiswa S1
Institusi Teknologi Adhi Tama Surabaya
Jurusan Teknik Kimia
pudjiwitari@gmail.com

Gistanya Lindar Anggraini
Mahasiswa S1
Institusi Teknologi Adhi Tama Surabaya
Jurusan Teknik Kimia
gistanyalindari@gmail.com

Heat exchanger type shell and tube, which is the most commonly used heat exchanger in various industries. The efficiency of heat exchangers can be seen from their performance to affect its economy from a process. The purpose is to determine the influence of the hot fluid flow rate and the cold fluid on the overall heat transfer coefficient (U) and log mean temperature difference (ΔT_{LMTD}) values. This simulation is done using Aspen HYSYS V.10 applications and obtained data of the total heat transfer coefficient (U) and ΔT_{LMTD} values. The heat exchanger shell and tube used type 1-2 with single segment type 4 baffle, triangular tube layout, and shell length 1000mm. This simulation results in a hot fluid flow rate compared to reverse with the overall heat transfer coefficient and a cold fluid relative to the overall heat transfer coefficient, with the two best fluid flow rates at 2100 kg/h hot fluid and 1800 kg/h cold fluid at 10400 Kj/°C.h. The influence of the hot fluid flow rate on ΔT_{LMTD} is relative to the straight, while the cold fluid flow rate is relative to the reverse, with the value of the second-best fluid flow rate at the 2100 kg/h hot fluid and the 1800 kg/h cold fluid at 26.25°C

Keywords : *Performance, Simulation, Shell and Tube, Heat Exchanger, Aspen HYSYS.*

1. INTRODUCTION

The energy needed in the industry is closely related to the production process, so it needs a tool that can increase energy use more efficienly. The shell and tube heat exchange instrument has a combined cylindrical shape on the outside and an arrangement of small tubes on the inside[1,2]. This heat exchanger is often used to refine the industrial process because it has advantages such as high efficiency, simple construction, and easy maintenance [3,4].

Heat exchanger performance improvement with optimization and conducting trial-error procedures. Analysis of the performance of heat exchangers to achieve efficiency has been carried out experimentally for laboratory scale [5] and industry [6]. Experimentally, the performance of the heat exchanger is performed by replacing the baffle type [7,8], the length and diameter change shell [9], the tube design [10], which can increase the rate of heat transfer and efficiency by 50% [6]. The factor that influences the performance of the heat transfer is the rate of fluid transfer [11]. The heat transfer rate is affected by the distance of the baffle, the temperature and pressure of the feed, the diameter of the shell, the number of tubes, and the geometry of the tube.

Development of efforts to obtain optimization in the evaluation of heat exchanger performance, some researches are carried with simulation method [12] and numerical [13,14]. [15] simulations with CFD that conclude the maximum heat transfer rate is using a segmental baffle. He [16] did a numerical investigation with baffle configuration and concludes that the type of flower baffle can reduce the pressure drop. Based on the literature studies done, the efforts to measure the performance of shell and tube heat exchanger for laboratory scale, a simulation is made using Aspen HYSYS V.10. The performance of the heat exchanger is seen based on the changes of cold fluid flow rate and hot fluid flow rate against the overall heat transfer coefficient (U) and a value of log mean temperature difference (ΔT_{LMTD}).

Corresponding Author: erlindaningsih84@itats.ac.id Received on: November 2021 Accepted on: November 2022
DOI: https://doi.org/10.21776/jrm.v13i3.1078

2. EXPERIMENTAL METHODS

This research is a simulation of the performance of a shell and tube type 1-2 heat exchanger, counter current flow with its specifications. The Heat Exchanger design is presented in Figure 1 and the specifications used are as follows:

- Inside diameter shell : 102.3 mm
- Outside diameter shell : 114.3 mm
- Number of tubes per shell : 18
- Number of baffles : 4
- Layout tube : Triangular
- Inside diameter tube : 12.52 mm
- Outside diameter tube : 17.15 mm
- Thickness : 0.8 mm
- Shell length : 1000 mm
- Material : Stainless steel
- Cut ratio : 20%
- Baffle type : Single Segmental

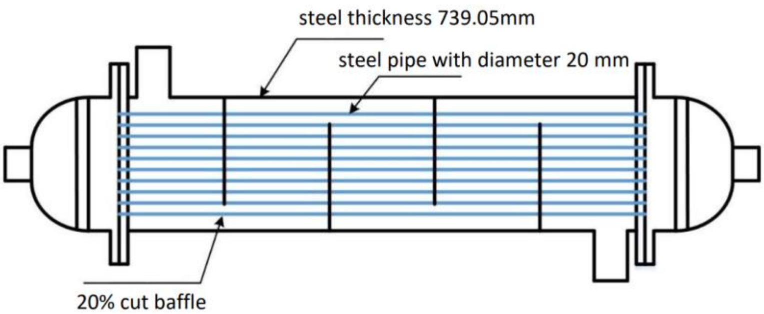


Figure 1. Shell and tube-type heat exchanger design

2.1 Research Variable

The research variables divided into fixed variables are the inlet temperature of hot fluid 95^0 C and for cold fluid is 30^0 C, while the outlet temperature for cold liquid is 65^0 C and for hot liquid is 50^0 C. The performance test was carried by changing the hot fluid flow rate, which is 2100 kg/h, 3000 kg/h, 4500 kg/h, 6000 kg/h, and 10000 kg/h and the cold fluid flow rate are 1000 kg/h, 1200 kg/h, 1400 kg/h, 1600 kg/h, and 1800 kg/h.

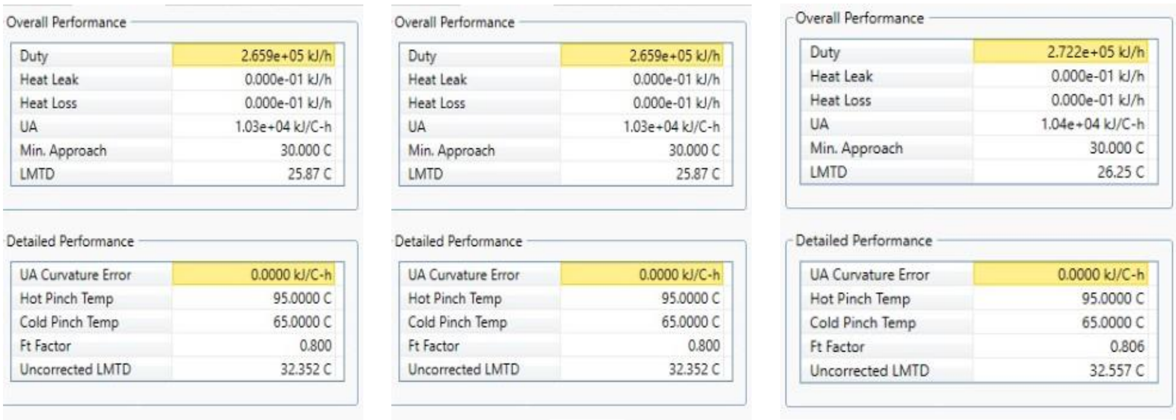
2.2 Simulation Procedure

1. On the Simulation Basis Manager menu (Aspen HYSYS), input fluid type and other properties.
2. On the Property Package menu, use Peng-Robinson as a fluid package Aspen HYSYS. In the fluid package selection, it must be consistent with the fluid characteristic.
3. After setting the fluid package, enter the simulation then select the type of instrument that will be used is shell and tube heat exchanger.
4. Add four flow sources consisting of two flow sources as inputs and two flow sources as outputs.
5. Double click on each stream, then input the existing data.
6. If all data has been entered and there is already a green okay at the bottom of the page, then see the results by choosing the performance menu.

3. RESULTS AND DISCUSSION

Shell and tube heat exchanger consist of a shell and several tubes that are in the shell. Shell and tube heat exchangers are usually equipped with a bulkhead commonly called a baffle. The first simulation is done with variable flow rates of hot fluid and cold fluid. This cold fluid later is heated to a temperature of 65ºC. The temperature of the incoming hot fluid is 95ºC, and the cold fluid is 30ºC. Based on the simulation carried out, a graph can be made between the flow rate with ΔT_{LMTD} and U.

The simulation process begins with the input fluid flowing[17]. Namely, the cold fluid is air with a temperature of 30°C, and the hot fluid is 95°C. Next, choose a fluid package type consisting of NRTL, UNIQUAQ, and Peng Robinson. In this process, Peng Robinson was chosen because it produces the best overall heat transfer coefficient value. Figure 2. The difference in heat transfer coefficient results for the three types of fluid packages.



(a) b (c c

Figure 2. Overall Performance Results with difference types of fluid packages, a) NRTL (non-random two liquids), b) UNIQUAQ model, c) Peng-Robinson.

The selection of types and data input of hot and cold fluid flow rates, as well as flow selection is carried out after the input process for the type of fluid packages, namely Peng Robinson, this is shown in Figure 3.

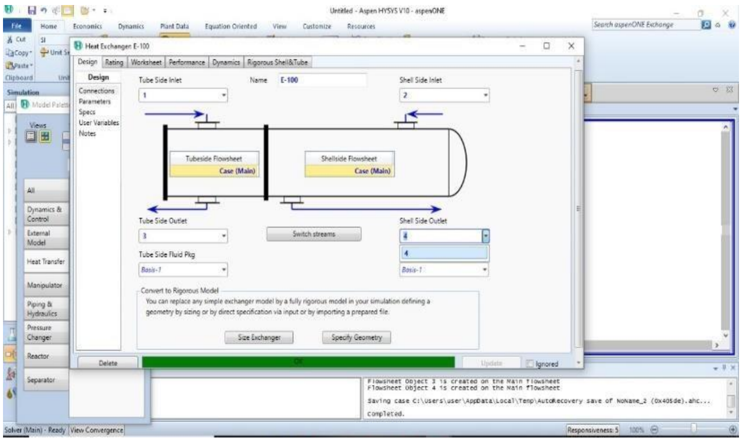


Figure 3. Numeric parameter input process on Aspen HYSYS.

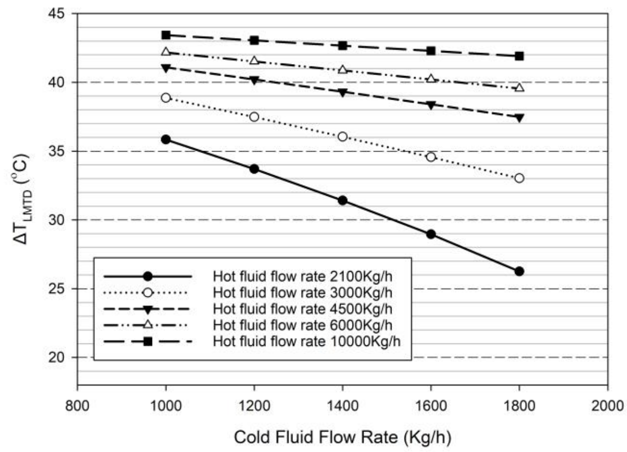
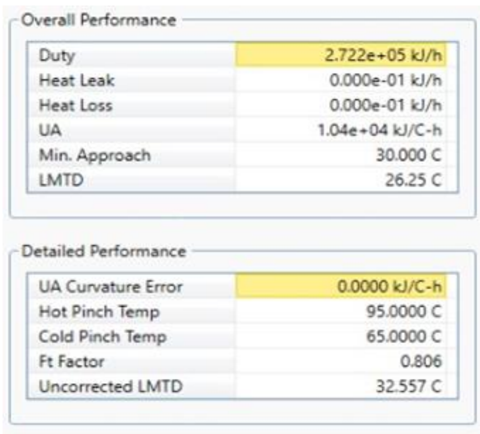


Figure 4. Graph ΔT_{LMTD} effect of hot and cold fluid flow rate

Based on Figure 4. is obtained, when the hot fluid flow rate increase, it makes the ΔT_{LMTD} value increase too and when the cold fluid flow rate increase, it makes the ΔT_{LMTD} value relatively down. From Figure 4., the value can be seen on Figure 5. From simulation, the result of the highest ΔT_{LMTD} value is 43.43°C when the hot fluid flow rate is 1000 Kg/h and the hot fluid flow rate is 10000 Kg/h. The lowest ΔT_{LMTD} value is 26.25°C when the hot fluid flow rate is 2100 Kg/h and the cold fluid flow rate is 1800 Kg/h. The most optimal value is 26.25°C when the hot fluid flow rate is 2100 Kg/h and the cold fluid flow rate is 1800 Kg/h, which can be show in Figure 5.



Overall Performance

Duty	2.722e+05 kJ/h
Heat Leak	0.000e-01 kJ/h
Heat Loss	0.000e-01 kJ/h
UA	1.04e+04 kJ/C-h
Min. Approach	30.000 C
LMTD	26.25 C

Detailed Performance

UA Curvature Error	0.0000 kJ/C-h
Hot Pinch Temp	95.0000 C
Cold Pinch Temp	65.0000 C
Ft Factor	0.806
Uncorrected LMTD	32.557 C

Figure 5. Simulation Results of the Effect of Hot and Cold Fluid Flow Rate on ΔT_{LMTD}

Previous research stated that at a constant hot fluid flow rate of 0.11681 kg/s and variations in the cold fluid flow rate of 0.055 kg/s, 0.50 kg/s, 0.044 kg/s, the largest ΔT_{LMTD} value was obtained when the flow rate was higher. cold fluid flow 0.044 kg/s. This condition can occur because, the greater the flow rate of the cold fluid, the smaller the heat transfer received by the cold fluid, so that the ΔT_{LMTD} is getting bigger [18].

From Figure 6. the highest heat transfer coefficient occurs when the hot fluid flow rate is 2100 kg/hour and the cold fluid flow rate is 1800 kg/hour, which is 10400 Kj/°C·hour. While the lowest heat transfer coefficient is obtained when the hot fluid flow rate is 10000 kg/hour and the cold fluid flow rate is 1000 kg/hour, which is 3480 Kj/°C·hour. This can happen because the greater the flow rate, the greater the fluid flow rate, then the contact between the two fluids will be shorter, so that the heat transfer that occurs is not optimal and the effectiveness of the shell and tube heat exchanger will decrease. So, the most optimal overall heat transfer coefficient occurs when the hot fluid flow rate is 2100 kg/hour and the cold fluid flow rate is 1800 kg/hour, which is 10400 Kj/°C which can be seen in Figure 7.

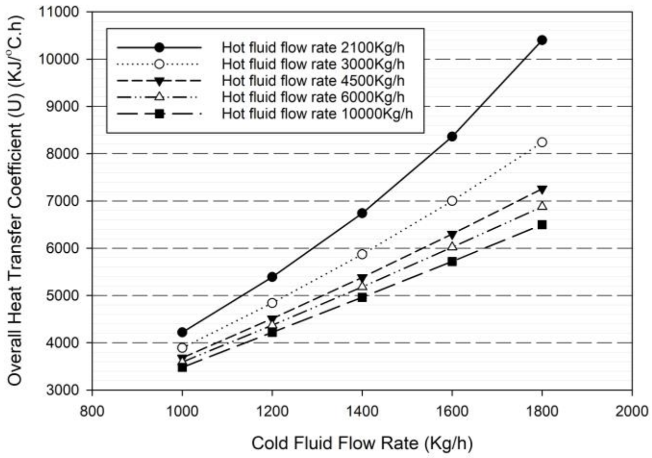


Figure 6. Graph Effect of Hot and Cold Fluid Flow Rate on Overall Heat Transfer Coefficient (U)

In previous studies, according to [11], the lowest coefficient of heat transfer is obtained at a cold fluid flow rate of 1.19 m/s, which is 668.13 W/m2.ºC. While the highest heat transfer coefficient is obtained at a cold fluid flow rate of 2.91 m/s, which is 1367.88 W/m^2·ºC. Based on that data, the higher the cold fluid flow rate, the greater the coefficient of heat transfer. Fluid flow rate changes significantly affect the efficiency of heat exchangers. At the time of the heat transfer process there is a quick flow of hot fluids and a quick flow of cold fluids. The flow rate or flow rate can determine the type of flow as a laminar flow or turbulence turbulence flow due to its height the flow rate can increase Reynold's number and can increase the heat transfer in a convex so that the heat transfer coefficient increases along with the efficiency of the shell and tube heat exchanger[18].

4. CONCLUSION

1. The effect of the hot fluid flow rate on the overall heat transfer coefficient is inversely proportional, effect of the cold fluid flow rate is directly proportional to the most optimal overall heat transfer coefficient value is 10400 Kj/^{0} C.hour when the hot fluid flow rate is 2100 kg/hour and cold fluid 1800 kg/hour.
2. The effect of hot fluid flow rate on ΔT_{LMTD} is directly proportional, while the effect of cold fluid flow rate is inversely proportional to the most optimal value of ΔT_{LMTD} which is 26.25^0 C when the hot fluid flow rate is 2100 kg/hour and the cold fluid flow rate is 1800 kg/hour.

5. ACKNOWLEDGEMENTS

This research received funding from LPPM ITATS and YPTS.

6. REFERENCES

[1] MASTER BI, CHUNANGAD KS, BOXMA AJ, KRAL D, STEHLIK P. Most Frequently Used Heat Exchangers from Pioneering Research to Worldwide Applications Most Frequently Used Heat Exchangers from Pioneering Research n.d.:37–41. https://doi.org/10.1080/01457630600671960.

[2] PATEL VK, RAO R V. Design optimization of shell-and-tube heat exchanger using particle swarm optimization technique. Appl Therm Eng 2010;30:1417–25. https://doi.org/10.1016/j.applthermaleng.2010.03.001.

[3] BELL KJ. 2003 Max Jacob Memorial Award Lecture Heat Exchanger Design for the Process Industries 2013;126. https://doi.org/10.1115/1.1833366.

[4] GULYANI BB. Estimating Number of Shells in Shell and Tube Heat Exchangers : A New Approach Based on Temperature Cross 2016;122:566–71.

[5] ALIMORADI A. Study of thermal effectiveness and its relation with NTU in shell and helically coiled tube heat exchangers. Case Stud Therm Eng 2017. https://doi.org/10.1016/j.csite.2017.01.003.

[6] BUDIMAN A, SYARIEF A, ISWORO H, et al. Analisis Perpindahan Panas dan Efisiensi Efektif High Pressure Heater (HPH) di PLTU Asam-Asam 2014;03:76–82.

[7] WEN J, YANG H, WANG S, XUE Y, TONG X. Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffle. HEAT MASS Transf 2015;84:990–7. https://doi.org/10.1016/j.ijheatmasstransfer.2014.12.071.

[8] GU X, LUO Y, XIONG X, WANG K, WANG Y. International Journal of Heat and Mass Transfer Numerical and experimental investigation of the heat exchanger with trapezoidal baffle. Int J Heat Mass Transf 2018;127:598–606. https://doi.org/10.1016/j.ijheatmasstransfer.2018.07.045.

[9] ALI A, ZAKI S. Case Studies in Thermal Engineering Analysis study of shell and tube heat exchanger for clough company with reselect di ff erent parameters to improve the design. Case Stud Therm Eng 2017;10:455–67. https://doi.org/10.1016/j.csite.2017.10.002.

[10] ABD AA, KAREEM MQ. Author ' s Accepted Manuscript Performance Analysis of Shell and Tube Heat Exchanger : Parametric Study. Case Stud Therm Eng 2018. https://doi.org/10.1016/j.csite.2018.07.009.

[11] HUSEN A, AKBAR TMI, CHOLIS N. Analisis pengaruh kecepatan aliran fluida dingin terhadap efektivitas shell and tube heat exchanger 2020;16:1–10.

[12] SOPURTA A, SIREGAR P. Perancangan Sistem Simulasi HYSYS & Iintegrasi dengan Programmable Logic Controller - Human Machine Interface : Studi Kasus pada Plant Kolom Distilasi Etanol-Air 2014;6:1–9.

[13] MAAKOUL A EL, LAKNIZI A, SAADEDDINE S, METOUI M EL, ZAITE A, MEZIANE M, et al. Numerical comparison of shell-side performance for shell and tube heat exchangers with trefoil-hole, helical and segmental baffles. Appl Therm Eng 2016. https://doi.org/10.1016/j.applthermaleng.2016.08.067.

[14] YOU Y, FAN A, HUANG S, LIU W. Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles. Int J Heat Mass Transf 2012;55:7561–9. https://doi.org/10.1016/j.ijheatmasstransfer.2012.07.058.

[15] SHRIKANT AA, SIVAKUMAR R, ANANTHARAMAN N, VIVEKENANDAN M. CFD simulation study of shell and tube heat exchangers with different baffle segment configurations. Appl Therm Eng 2016. https://doi.org/10.1016/j.applthermaleng.2016.08.013.

[16] HE L, LI P, LI P. Numerical investigation on double tube-pass shell-and-tube heat exchangers with different baffle configurations 2018. https://doi.org/10.1016/j.applthermaleng.2018.07.098.

[17] CHEMMANGATTUVALAPPIL N, CHONG S. Basics of Process Simulation With Aspen HYSYS. Elsevier Inc.; 2005. https://doi.org/10.1016/B978-0-12-803782-9.00011-X.

[18] SHANAHAN R, CHALIM A. Studi Literatur Tentang Efektivitas Alat Penukar Panas Shell And Tube 1-1 Sistem Fluida Gliserin – 2020;6:164–70.