LOGIC
Journal of Engineering Design and Technology
Vol. 26 No. 1 March 2026 ; p. 39 - 47

p-ISSN : 1412-114X
e-ISSN : 2580-5649
http://ojs2.pnb.ac.id/index.php/LOGIC

EXPERIMENTAL AND CFD ANALYSIS OF A LABORATORY-SCALE PARALLEL FLOW SHELL AND TUBE HEAT EXCHANGER

1) Mechanical Engineering Study Program, Faculty of Engineering, Universitas Muhammadiyah Gresik, Indonesia
2) Department of Mechanical Engineering, Politeknik Negeri Malang, Indonesia

Correponding email [1)*] :
alvianihesthi@umg.ac.id

Alviani Hesthi Permata Ningtyas [1)*], Siti Duratun Nasiqiati Rosady [2)], Ilham Arifin Pahlawan [1)], Rilo Chandra Muhamadin [1)]

Abstract. A heat exchanger is a device that functions to change the temperature of a fluid by utilizing the heat transfer mechanism from a high-temperature fluid to a lower-temperature fluid. This study was conducted to design and analyze the performance of a shell and tube heat exchanger with a parallel flow configuration on a laboratory scale. The design process focused on determining the main dimensions and components, such as tube length and the ratio between shell and tube diameters, to ensure optimal operation for laboratory experiments. After the device was successfully fabricated, experiments were carried out to obtain temperature data of the hot and cold fluids at both the inlet and outlet. These experimental data were then compared with the results of numerical simulations using ANSYS Fluent software based on the Computational Fluid Dynamics (CFD) method. The simulation was used to visualize the flow pattern and temperature distribution within the heat exchanger, as well as to calculate heat transfer efficiency. The results showed good agreement between the simulation and experimental data, the deviation is 15%, where the inlet temperature of the hot fluid was 65°C and the outlet temperature was 38°C, indicating the validity of the numerical model used. From this study, it can be concluded that the combination of experimental design and CFD simulation analysis provides a more comprehensive understanding of the temperature distribution and efficiency of a shell and tube heat exchanger with a parallel flow configuration.

Keywords : heat exchanger; simulation; design; fabrication.

1. INTRODUCTION

Heat transfer is an essential element in various engineering processes, including in industries that utilize heating, cooling, and power generation systems[1]–[6]. One of the key tools that supports this process is the heat exchanger[7]–[13]The type of heat exchanger commonly used in industry is the shell and tube type, due to its ability to handle various types of fluids and high operating pressures. Hot fluid flows through the tubes, while cold fluid moves within the shell around the tubes. This system has several variations in fluid flow direction: parallel flow, counterflow, and crossflow[14]–[17]. The parallel flow configuration is characterized by fluids flowing from the inlet to the outlet side in parallel, both in the tubes and the shell. The main advantage of this design is its simplicity, but its drawback is the rapid decrease in temperature difference between the two fluids, resulting in relatively low heat transfer efficiency[17], [18]. A study by [11] discussed the design of shell and tube heat exchangers. In this research, the shell tube has an outer diameter of 300 mm and an inner diameter of 293.65 mm with a length of 1000 mm. The obtained heat transfer rate is 7117.5 watts. Based on TEMA standards and the application of HTRI, the required surface area is 8.27 m^2, while actual calculations show a surface area of 9.09 m^2. The research by [19]discusses a design with one shell pass and two tube passes. The tube length is 2.7 m with a diameter of 13 mm, while the shell measures 1.35 m in length and 70 mm in diameter. The material for the tube is copper with a conductivity of 385 W/mK, whereas the shell is made of aluminum with a conductivity of 205 W/mK. The results of this study show a maximum efficiency of 35.4040%. The research conducted by [14]focused more on the design of the heat exchanger dimensions. In that study, a tube length of 1.2 m, a shell length of 70 cm, and a total of 6 tubes were obtained. In the mixed temperature tests, the measured values ranged from 50°C to 25°C. Testing using CFD Ansys is often applied to understand temperature distribution and efficiency in heat exchangers [20]–[27]. The aim of this study is to design and evaluate the performance of a shell-and-tube type heat exchanger with parallel flow on a laboratory scale. This research includes the design of dimensions and main elements so that the device can operate optimally, experimental testing to obtain temperature data of hot and cold fluids, as well as numerical analysis using ANSYS Fluent based on CFD to visualize flow patterns and temperature distribution[28], [29]. The simulation results are then compared with experimental data to verify the numerical model and assess heat transfer efficiency, with the aim of obtaining a comprehensive understanding of the thermal performance of the heat exchanger.

2. METHODS

This research was conducted through three main stages: design, fabrication, and experimental testing of a laboratory-scale shell-and-tube heat exchanger with a parallel flow configuration. During the design stage, the primary geometric parameters were determined, including tube length, shell diameter, number of tubes, and material selection. These parameters were defined based on heat transfer considerations, structural integrity, and ease of assembly. The complete configuration was modeled using three-dimensional CAD software to ensure dimensional consistency and proper alignment of the parallel flow direction between shell- and tube-side fluids..

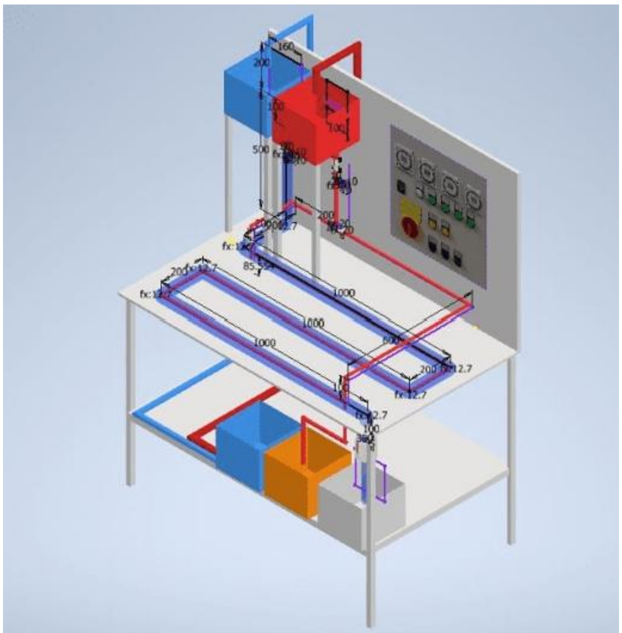

Figure 1. Heat Exchanger Design

Experimental testing was conducted using the dual-reservoir circulation system shown in Figure 1. The setup consists of two elevated tanks serving as hot- and cold-water supply reservoirs and two lower tanks for fluid collection and recirculation. Hot water was prepared in the upper reservoir using an electric heater prior to testing, while cold water was supplied from a separate reservoir to maintain controlled inlet conditions. The fluids were circulated through the shell-and-tube heat exchanger via a closed-loop piping system equipped with control valves, and the volumetric flow rate was maintained at 800 L/h as monitored by a calibrated flowmeter. After passing through the heat exchanger, both fluids were collected in the lower tanks and recirculated to ensure continuous operation.

Table 1. Shell and Tube Specifications

Description	Shell	Tube
Diameter	22 mm	6,35 mm
Length	3400 mm	3400 mm
Material Pipe	Acrylic	Aluminum

Temperature measurements at the hot and cold fluid inlets and outlets were obtained using calibrated K-type thermocouples installed at the respective pipe sections. The thermocouple readings were recorded manually at 5-minute intervals using a digital temperature display, while a stopwatch was used to ensure consistent time tracking throughout the 45-minute experimental duration. This procedure ensured systematic data collection while maintaining stable operating conditions for evaluating the thermal performance of the parallel-flow heat exchanger.

Although the system is referred to as a shell-and-tube heat exchanger, the laboratory-scale configuration used in this study consists of a single small copper tube in the inner section through which hot water flows, while cold water flows directly in contact with the outer surface of the inner tube within the surrounding cylindrical outer pipe. Geometrically, this arrangement more closely resembles a concentric (double-pipe) heat exchanger rather than a conventional multi-tube shell-and-tube system equipped with baffles for flow distribution. In the present study, no thermal insulation was applied to the outer surface of the external pipe, allowing heat transfer to occur primarily through conduction across the copper tube wall and convection on both fluid sides, with potential natural heat loss to the surrounding environment. The term “shell” is retained due to the presence of the external cylindrical enclosure; however, only a single tube pass is employed without the complex flow distribution characteristics typical of industrial shell-and-tube exchangers. This clarification is important to avoid ambiguity in interpreting flow distribution behavior, effective heat transfer area, and performance comparisons with industrial-scale shell-and-tube heat exchangers.

Both working fluids were modeled as incompressible water under steady-state and single-phase conditions. Thermophysical properties were assumed constant and evaluated at the respective mean bulk temperatures (ρ = 997 kg/m³, Cp = 4182 J/kg·K, k = 0.6 W/m·K, μ = 0.001 Pa·s). The realizable k–ε turbulence model with enhanced wall treatment was employed to accurately resolve near-wall thermal gradients. The computational mesh was refined in the boundary-layer region to maintain y^+ values below 5, ensuring appropriate resolution of the viscous sublayer and improved prediction of convective heat transfer coefficients. A conjugate heat transfer (CHT) approach was implemented by explicitly modeling the aluminum tube wall as a solid domain, allowing conductive heat transfer between the hot and cold fluid regions to be simultaneously solved with the governing energy equations. This approach improves thermodynamic consistency and provides a more realistic representation of the coupled convection–conduction mechanism within the heat exchanger.
.

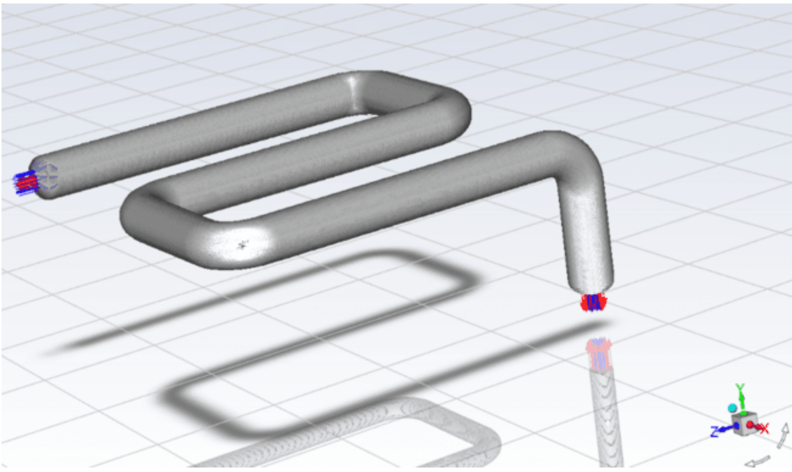

Figure 2. Meshing in ANSYS

Figure 2 shows the meshing results in ANSYS with inlet and outlet conditions, along with a total of 3,754,832 faces and 928,676 nodes. Figure 3 illustrates the schematic flow of fluid in the heat exchanger system with a parallel flow configuration. The cold fluid and hot fluid flow in opposite directions through two different paths, each marked by blue and red colors respectively. The cold fluid enters from the top left and exits at the bottom right, while the hot fluid also enters from the top left but exits at the bottom right through a different path. This schematic is equipped with temperature measurement points numbered 1 to 17, depicting the locations of temperature monitoring along the flow. This design allows for a more efficient heat transfer process through a consistent temperature difference between the two fluids. The realizable k–ε turbulence model with enhanced wall treatment was applied. Velocity inlet boundary conditions were specified according to the experimental flow rate (800 L/h) and measured inlet temperatures, while pressure outlet conditions were imposed at both exits. Convergence criteria were set to 10^{-6} for the energy equation and 10^{-4} for continuity and momentum equations. Mesh quality Ansys Metric in this simulation is 0.9 that means acceptable. The thermal performance of the heat exchanger was evaluated using the effectiveness–NTU method and the Log Mean Temperature Difference

(LMTD) approach[2]. The actual heat transfer rate was calculated from the energy balance:

$$Q = \dot{m}_h c_{p,h}(T_{h,in} - T_{h,out}) \tag{1}$$

$$Q = \dot{m}_c c_{p,c}(T_{c,out} - T_{c,in}) \tag{2}$$

where $\dot{m}$ is the mass flow rate (kg/s), c_p is the specific heat capacity (J/kg·K), and T represents fluid temperatures (°C or K).
The heat exchanger effectiveness (ε) is defined as:

$$\varepsilon = \frac{Q_{actual}}{Q_{max}} \tag{3}$$

where

$$Q_{max} = C_{min}(T_{h,in} - T_{c,in}) \tag{4}$$

and

$$C = \dot{m}c_p \tag{5}$$

$$C_{min} = \min(C_h, C_c) \tag{6}$$

The number of transfer units (NTU) is calculated as:

$$NTU = \frac{UA}{C_{min}} \tag{7}$$

where U is the overall heat transfer coefficient (W/m²·K) and $A$is the total heat transfer area (m²).
For a parallel-flow heat exchanger, the analytical effectiveness relation is:

$$\varepsilon = \frac{1 - \exp[-NTU(1 + C_r)]}{1 + C_r} \tag{8}$$

with

$$C_r = \frac{C_{min}}{C_{max}} \tag{9}$$

The overall heat transfer coefficient $U$was determined using the LMTD [1]method:

$$Q = UA\Delta T_{lm} \tag{10}$$

where the log mean temperature difference for parallel flow is:

$$\Delta T_{lm} = \frac{(T_{h,in} - T_{c,in}) - (T_{h,out} - T_{c,out})}{\ln\left(\frac{T_{h,in} - T_{c,in}}{T_{h,out} - T_{c,out}}\right)} \tag{11}$$

These formulations provide a rigorous thermodynamic basis for evaluating the experimental results and validating the CFD predictions of the parallel-flow shell-and-tube heat exchanger.

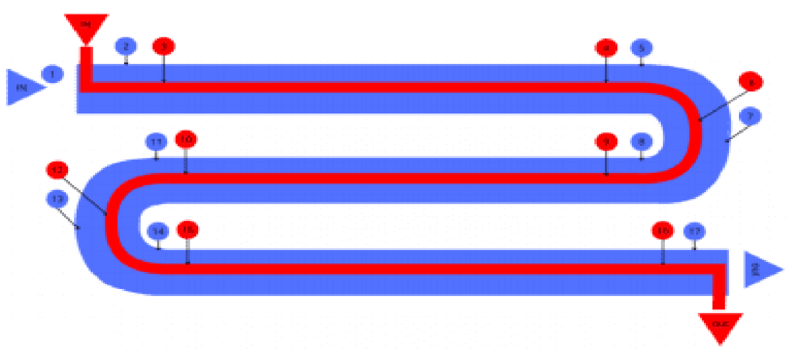

Figure 3. Fluid flow rate scheme

Figure 3 illustrates the schematic configuration of the parallel-flow heat exchanger. Both hot and cold fluids enter from the same axial direction and flow parallel along separate channels. Temperature measurement points (1–17) indicate monitoring locations used for experimental validation and numerical comparison. The red inner passage represents the hot fluid, while the surrounding blue domain corresponds to the cold fluid, separated by a solid wall that facilitates heat transfer primarily through conduction across the wall and convection within each fluid stream.

3. RESULTS AND DISCUSSION

The test results of the shell and tube type heat exchanger with a parallel flow configuration that has been completed, as shown in Figure 4. The testing was carried out to observe how well this device transfers heat by monitoring the temperature changes of the hot and cold fluids at the inlet and outlet at certain time intervals. The data in Table 2 show a significant temperature change in both fluids, indicating that the heat transfer process occurred as expected during the tests. These temperature changes were then analyzed to determine the effectiveness of the device, the thermal behavior of the system during operation, and serve as a basis for comparison with theoretical calculations and CFD simulation results in the following section.

Figure 4. Fabrication of heat exchanger

Figure 4 shows the heat exchanger that has been fabricated. The fabrication was carried out according to the design that had been calculated.

Table 2. Test result data

Time (m)	Temperature (°C)			
	Cold		Hot	
	Inlet	Outlet	Inlet	Outlet
5	19.3	26.3	54.8	45.7
10	20.2	25.9	59.7	44.6
15	19.6	25.4	59.6	43.2
20	22.3	27.6	65.6	42.2
25	21.1	26.1	69.3	38.4
30	20.1	26.9	54.7	38.3
35	22.8	27.4	55.6	37.6
40	23.6	26.5	62.4	36.7
45	20.8	26.3	67.3	45.7

Table 2 shows how the temperatures of cold water and hot water changed during testing from the 5th minute to the 45th minute. The cold water, which initially ranged from 19–23°C, appeared to rise to around 25–27°C as it exited the device, indicating that the cold water successfully absorbed heat. Meanwhile, the hot water entering at 54–69°C dropped to 37–46°C upon exit, showing that the heat transferred to the cold water. Although there were slight fluctuations in some data points, the overall pattern remained clear: the cold water became warmer, the hot water became cooler, indicating that the heat exchanger worked effectively.

Table 3. Calculated Thermal Performance Parameters

Parameter	Symbol	Equation	Value	Unit
Heat Capacity Rate	$C = \dot{m}c_p$	0.222×4180	927.96	W/K
Actual Heat Transfer Rate	Q_{actual}	$\dot{m}c_p(T_{c,out} - T_{c,in})$	5,010	W
Maximum Heat Transfer Rate	Q_{max}	$C_{min}(T_{h,in} - T_{c,in})$	37,000	W
Effectiveness	ε	Q_{actual}/Q_{max}	0.135	–
Temperature Difference 1	ΔT_1	$T_{h,in} - T_{c,in}$	39.89	°C
Temperature Difference 2	ΔT_2	$T_{h,out} - T_{c,out}$	14.88	°C
Log Mean Temperature Difference	ΔT_{lm}	LMTD formula	25.34	°C
Overall Heat Transfer Coefficient	U	$Q/(A\Delta T_{lm})$	2,916	W/m²K
Number of Transfer Units	NTU	UA/C_{min}	0.21	–
Capacity Ratio	C_r	C_{min}/C_{max}	1.0	–

Table 3 summarizes the calculated thermal performance parameters of the parallel-flow shell-and-tube heat exchanger under laboratory-scale operation. The heat capacity rate was determined as 927.96 W/K, resulting in an actual heat transfer rate of 5,010 W compared to a theoretical maximum of 37,000 W, yielding an effectiveness of 0.135. The measured terminal temperature differences (ΔT_1 = 39.89°C and ΔT_2 = 14.88°C) produced an LMTD of 25.34°C. Based on these values, the overall heat transfer coefficient was calculated as 2,916 W/m²K, with an NTU of 0.21 and a capacity ratio (C_r) of 1.0. A comparison with CFD simulation results can be seen in Figure 5 below.

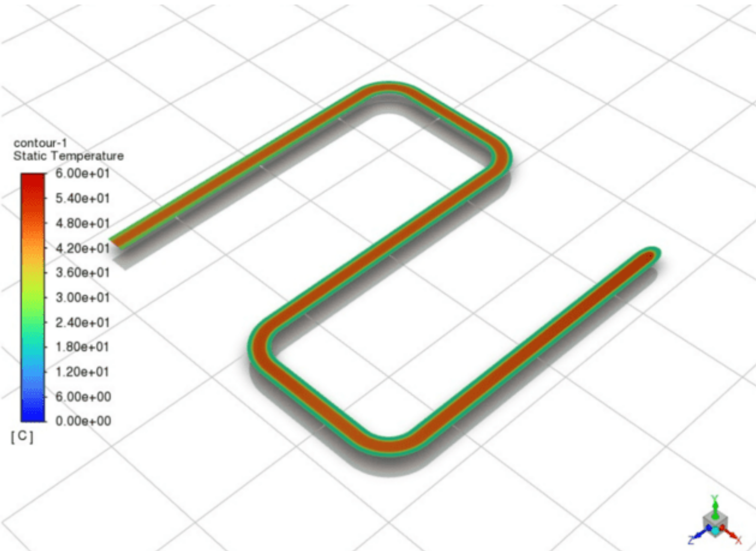

Figure 5. Temperature distribution in ANSYS software

Figure 5 shows the simulation results of the temperature distribution in a shell and tube type heat exchanger with a parallel flow configuration using ANSYS Fluent software. The color contours illustrate the decrease in temperature of the hot fluid from around 60°C at the inlet side (red color) to about 19.3°C at the outlet side (blue color). The uniform color gradient pattern indicates that an effective heat transfer process occurs along the fluid flow path. Experimental test results show that the heat transfer process in the shell and tube type heat exchanger with a parallel flow configuration is effective, as indicated by significant temperature changes in both the hot and cold fluids at each observation interval. The inlet temperature of the hot fluid ranges from 54.8°C to 69.3°C, while its outlet decreases to the range of 36.7–45.7°C. Meanwhile, the temperature of the cold fluid increased from 19.3C to about 27.6°C by the end of the test. The average temperature difference between the hot and cold fluids indicates that the conduction mechanism through the tube wall and convection on both sides of the fluid are working simultaneously and efficiently. Numerical simulations using ANSYS Fluent show a stable flow pattern and temperature distribution consistent with experimental results. The model, using the standard k-epsilon turbulence and second-order upwind discretization, is capable of accurately representing heat transfer phenomena. Simulation results indicate a temperature difference of 38°C at the hot fluid outlet and 26°C at the cold fluid outlet, with only about a 15.7% deviation from experimental results. The temperature contour from the simulation shows a linearly decreasing temperature gradient along the flow direction, indicating that the heat transfer rate is dominated by forced convection mechanisms.

The temperature contour presented in Figure 5 indicates localized regions where the hot-fluid temperature approaches the lower temperature range. It is important to emphasize that these values correspond to local minima within the computational domain rather than the mass-averaged outlet temperature. The bulk outlet temperature predicted numerically was approximately 38°C, which is in close agreement with experimental measurements and remains thermodynamically consistent with parallel-flow heat exchanger theory. Since no external heat loss was imposed in the numerical model, the reduction in hot-fluid temperature corresponds directly to the heat absorbed by the cold fluid, satisfying overall energy conservation. Furthermore, considering the relatively low NTU value (0.21) and effectiveness (0.135), the moderate outlet temperature difference observed is physically reasonable and aligns with established heat exchanger performance relations for parallel-flow configurations.

4. CONCLUSION

This study successfully designed and evaluated a laboratory-scale shell-and-tube heat exchanger with a parallel-flow configuration using experimental testing and CFD simulation in ANSYS Fluent. The results showed that the hot-fluid temperature decreased from approximately 65°C to around 38–41°C, while the cold-fluid temperature increased to about 26–27°C, confirming effective heat exchange. Thermal analysis yielded an average heat transfer rate of 5.01 kW, effectiveness (ε) of 13.5%, overall heat transfer coefficient (U) of 2916 W/m²K, and NTU of 0.21, indicating moderate thermal performance under the tested conditions. The deviation between experimental and numerical results was approximately 15.7%, demonstrating good model validation. For future research, it is recommended to vary the mass flow rate and compare other flow configurations, such as counter flow and cross flow, to obtain a broader range of heat transfer characteristics. Additionally, the use of tube materials with different thermal conductivities and variations in the number of baffles can be analyzed to optimize system efficiency. Further validation can be conducted by expanding the simulation domain and using more complex turbulence models, such as Realizable k-ε or k-ω SST, so that the simulation results more closely approximate real conditions.

5. ACKNOWLEDGEMENT

I would like to thank DPPM UMG for their assistance in completing this research.

6. REFERENCES

[1] K. P. Im, R. Allifia, E. Dia, P. Sari, and D. Pratiwi, "Pengaruh Laju Alir Fluida Panas dan Fluida Dingin terhadap besarnya Transfer Panas pada Alat Heat Exchanger," *Semin. Nas. Sains dan Teknol. Terap. X*, pp. 5–11, 2022.

[2] D. D. Saputra *et al.*, "Perancangan Alat Penukar Kalor Untuk Pemanas Air dengan Memanfaatkan Kondensat Panas Bumi Dieng PT XYZ," pp. 1855–1861, 2024.

[3] Suwarso, Y. Mariah, Nuradi, and S. Atti, "Perawatan Penukar Panas / Heat Exchanger Untuk Meningkatkan Kinerja Mesin Pendingin pada Kapal Niaga." pp. 11586–11597, 2024.

[4] F. S. D. Anggoro, "Rancang Bangun Helical Fin Pada Penukar Panas Double Pipe RANCANG BANGUN FIN TIPE HELICAL PADA PENUKAR PANAS DOUBLE PIPE Fajar Septiawan Dwi Anggoro I Made Arsana Abstrak," vol. 10, no. 2015, pp. 35–40, 2022.

[5] M. I. Kamil and D. A. Sari, "Komparasi Desain Alat Penukar Panas Tipe Air-Cooled," *J. Teknol.*, vol. 16, no. 2, pp. 180–186, 2023.

[6] S. Zhou, S. Yan, W. Zhang, H. Li, P. Niu, and Y. Shi, "Study on the heat discharging performance for a horizontal latent heat thermal energy storage exchanger with movable inner tube ☆," *J. Energy Storage*, vol. 152, no. PB, p. 120783, 2026, doi: 10.1016/j.est.2026.120783.

[7] F. Lubis and S. Lubis, "Analisis Baffle Cut Pada Alat Penukar Kalor Shell and Tube Pada Susunan Tabung Segi Empat," vol. 3, no. 1, pp. 39–44, 2021.

[8] F. Achmad *et al.*, "Efek laju alir dan arah aliran terhadap analisis performa alat penukar panas tipe shell and tube heat exchanger menggunakan SCADA," vol. 17, pp. 148–157, 2023, doi:

10.22146/jrekpros.77376.
[9] K. Astawa, I. G. T. P. Surya, and I. G. N. P. Tenaya, "Analisis Efektivitas Perpindahan Panas pada Alat Penukar Panas Jenis Water to Water," vol. 6, no. 1, pp. 26–30, 2022.
[10] Y. E. Pardede, "Analisa Laju Perpindahan Kalor Pada Alat Penukar Panas Kondensor Pembangkit Listrik Tenaga Uap (PLTU) di . PT XYZ," 2023.
[11] Y. Siswanto, K. W. Wirakusuma, and A. B. Pratama, "Rancang bangun alat penukar panas tipe pipa ganda kajian aliran berlawanan arah," vol. 05, no. 01, pp. 114–121, 2024.
[12] R. Yuniarti, F. Achmad, Y. L. Listyadevi, and L. Angraini, "JURNAL INTEGRASI PROSES Website : http://jurnal.untirta.ac.id/index.php/jip PENGARUH TEMPERATUR DAN ARAH ALIRAN TERHADAP EFEKTIVITAS PENUKAR PANAS NTU (Ɛ -NTU) PADA ALAT PENUKAR PANAS TIPE PLATE AND FRAME 1 Program Studi Teknik Kimia , Institut Teknologi Sumatera Jl . Terusan Ryacudu , Way Huwi , Kec . Jati Agung , Kabupaten Lampung Selatan , Lampung 2 Program Studi Teknik Kimia , Universitas Jenderal Achmad Yani Jl . Terusan Jenderal Sudirman , Cimahi , Jawa Barat 3 BRIN LIPI UPT Balai Pengolahan Mineral Lampung Jalan Ir . Sutami Km . 15 , Tanjung Bintang , Sindang Sari , Kabupaten Lampung Selatan , Lampung," vol. 11, no. 1, pp. 32–39, 2022.
[13] W. Ohoirenan, E. A. Rettob, and H. Dumatubun, "Analisis Perpindahan Panas Pada Aliran Searah (Current Flow) Dengen Pemanfaatan Pemanas Hieter Sistem Heat Exhanger," vol. 6, no. 1, pp. 54–56, 2025.
[14] C. Anuntasethakul and D. Banjerdpongchai, "Design of Supervisory Model Predictive Control for Building HVAC System with Consideration of Peak-Load Shaving and Thermal Comfort," *IEEE Access*, vol. 9, pp. 41066–41081, 2021, doi: 10.1109/ACCESS.2021.3065083.
[15] H. Kim, D. Sumiyoshi, and Y. Choi, "Development of flow rate and equipment simulation model for commercial building HVAC&R system by data-driven method," *J. Build. Eng.*, vol. 44, 2021, doi: 10.1016/j.jobe.2021.103303.
[16] Ahmad Maksum, Gun Gun Ramdlan Gunadi, Faiz Irza Ramadhan, and Candra Damis Widiyawaty, "Desain Fully Welded Plate Heat Exchanger untuk Pemanfaatan Asap Pirolisis dengan Metode Computational Fluid Dynamic (CFD)," *J. Mek. Terap.*, vol. 5, no. 1, pp. 8–15, 2024, doi: 10.32722/jmt.v5i1.6268.
[17] H. Yoel, N. Sinaga, B. Yunianto, and M. F. Soetanto, "Heat Transfer Intensification by Means of Convex-Strip Around Tube in Fin and Tube Heat Exchanger with Field Synergy Principle," *Int. J. Heat Technol.*, vol. 40, no. 4, pp. 1069–1078, 2022, doi: 10.18280/ijht.400425.
[18] A. Aziz, P. D. Rey, P. Studi, T. Mesinfakultas, S. Dan, and T. Universitas, "DESAIN DAN RANCANG BANGUN ALAT PENUKAR KALOR (HEAT EXCHANGER) JENIS SHELL DAN TUBE," vol. 03, no. 1, pp. 53–60, 2021.
[19] Farel H. Napitupulu :, "Rancang Bangun Alat Penukar Kalor Shell And Tube Dengan Satu Laluan Cangkang Dan Dua Laluan Tabung Sebagai Pemanas Air TALENTA Conference Series Rancang Bangun Alat Penukar Kalor Shell And Tube Dengan Satu Laluan Cangkang Dan Dua Laluan Tabung Sebagai Pem," vol. 1, no. 2, 2018.
[20] R. Nanda and N. Ruhyat, "CFD Study on Wire Mesh Inserts Enhancing Heat Transfer in Double Pipe Heat Exchanger," vol. 9, no. 1, pp. 36–49, 2025.
[21] A. Ponikarov, "Modeling of heat exchangers in ANSYS CFX for the digital twins development," vol. 03021, 2024.
[22] M. N. Ramli and N. Mustaffa, "RESEARCH PROGRESS IN MECHANICAL AND RPMME Efficiency Analysis of Full Rotation Counter Flow of Shell and Tube Heat Exchanger using CFD Tool," vol. 5, no. 1, pp. 444–451, 2024.
[23] W. M. Rosni, A. Adam, M. H. Jumaidi, and E. F. Gornica, "Double Straight Tube Counter-Flow Heat Exchanger for Optimum Heat Transfer by Numerical Simulation," vol. 6, no. 2, pp. 1–9, 2024.
[24] B. Jalaali and E. Kumolosari, "Simulasi aliran dua fase pada campuran air-minyak di pipa horizontal dengan parameter termal," *Angkasa J. Ilm. Bid. Teknol.*, vol. 13, no. 2, pp. 113–122, 2021, doi: 10.28989/angkasa.v13i2.1042.
[25] C. Suárez, A. Iranzo, J. A. Salva, E. Tapia, G. Barea, and J. Guerra, "Parametric investigation using computational fluid dynamics of the HVAC air distribution in a railway vehicle for representative weather and operating conditions," *Energies*, vol. 10, no. 8, 2017, doi: 10.3390/en10081074.
[26] P. P. Jati and A. Widyaparaga, "Simulasi CFD Dinamika Kecepatan Gelombang Aliran Stratified Wavy Dua Fasa Minyak-Air pada Pipa Horizontal," *J. Mech. Des. Test.*, vol. 3, no. 1, p. 1, 2021, doi: 10.22146/jmdt.56417.
[27] E. I. Epelle and D. I. Gerogiorgis, "A multiparametric CFD analysis of multiphase annular flows for oil and gas drilling applications," *Comput. Chem. Eng.*, vol. 106, pp. 645–661, 2017, doi: 10.1016/j.compchemeng.2017.08.011.
[28] J. Zhong, Y. Wang, M. Li, G. Pei, and S. Tang, "Heat transfer characteristics of the fin-tube three-fluid

heat exchanger in cold warehouse with cold storage," *Energy*, vol. 347, no. October 2025, p. 140362, 2026, doi: 10.1016/j.energy.2026.140362.

[29] S. Edhy *et al.*, "Results in Engineering Computational fluid dynamic simulation of earth air heat exchanger : A thermal performance comparison between series and parallel arrangements," *Results Eng.*, vol. 24, no. September, p. 102932, 2024, doi: 10.1016/j.rineng.2024.102932.