PENA CENDEKIA PUBLISHING
NURTURING IDEAS, ENRICHING KNOWLEDGE

Pena Journal of Flow Dynamics

Journal homepage:
https://penacendekia.com.my/index.php/pjfd/
ISSN: 3093-6802

# Optimizing Heat Exchanger Performance: A Comparative Analysis of Flowing and Cooling Fluids using Numerical Simulations

Muhamad Fraja Dwi Putra[1], Muh.Farhan Atha Farid[1], Rakha Aribawa Rahman[1], Muhammad Aisyur Rifky Maulana[1], Mochammad Brillian Bima Wicaksono[1], Singgih Dwi Prasetyo[1,*]

[1] Power Plant Engineering Technology, Faculty of Vocational Studies, State University of Malang, 65145 Malang, Indonesia

**ARTICLE INFO**

***Article history:***
Received 7 October 2025
Received in revised form 30 October 2025
Accepted 1 November 2025
Available online 2 November 2025

***Keywords:***
Computational Fluid Dynamics; heat exchanger; heat transfer; working fluid; numerical simulation; SolidWorks Flow simulation; thermal efficiency

**ABSTRACT**

Heat exchangers are one of the main devices in heat transfer systems that are widely used in industries such as power plants, chemical processes, and cooling systems. The performance of heat exchangers is greatly influenced by the thermophysical properties of the fluids used, both as hot-side working fluids and cold-side coolant fluids. This study uses a SolidWorks Flow Simulation-based numerical simulation method to analyze the differences in heat exchanger performance with variations in three flowing fluids (water, ethanol, argon) and three cooling fluids (nitrogen, oxygen, R134a) at an initial temperature of 78 °C. The simulation results show significant differences in the average temperature of each fluid. Water proved to be the most effective flowing fluid with an average temperature of 48.04 °C, while R134a was the most optimal cooling fluid with an average temperature of 36.29 °C. The combination of these two fluids provides the best heat transfer performance, making them the ideal pair for improving heat exchanger system efficiency. These findings emphasize the importance of selecting the right fluid to support the reliability and efficiency of industrial applications.

## 1. Introduction

Heat exchangers are critical components in various industrial heat transfer applications such as power generation, chemical processing, and cooling systems. Their performance depends heavily on the thermophysical properties of the working and cooling fluids, which determine both operational stability and overall system efficiency [1]. They are widely applied in power generation, air-conditioning, and chemical processing industries to improve energy efficiency and thermal reliability [2]. As global industries pursue compact and energy-efficient systems, enhancing the performance of heat exchangers has become a crucial engineering goal [3]. Achieving this improvement requires

* *Corresponding author.*
*E-mail address: singgih.prasetyo.fv@um.ac.id*

https://doi.org/10.37934/pjfd.2.1.4758

not only structural innovation but also a clear understanding of how different fluid properties influence heat transfer [4,5]. Therefore, developing practical and efficient methods to optimize heat exchangers remains an ongoing research priority [6].

Previous studies have mostly concentrated on geometric enhancements and the use of nanofluids to increase heat transfer rates [7,8]. Although these approaches can significantly improve performance, they often lead to higher costs, complex manufacturing, and limited industrial scalability [9]. In contrast, the role of conventional fluids in determining heat exchanger performance under identical boundary conditions has not been systematically studied [10,11]. This gap limits the ability of engineers to make practical decisions about fluid selection in existing thermal systems. Hence, investigating how standard fluids perform in similar configurations is both scientifically and industrially relevant [12].

To address this issue, this study uses SolidWorks Flow Simulation to analyze the performance of a heat exchanger with several combinations of working and cooling fluids. The research focuses on how variations in thermophysical properties—such as density, viscosity, specific heat, and thermal conductivity—affect heat transfer rate, pressure drop, and temperature distribution [13,14]. The simulations are conducted under steady-state conditions to ensure consistency and accuracy [15]. In addition, a one-way ANOVA test is applied to determine the statistical significance of performance differences between fluids. This combined computational and statistical approach enhances the reliability and depth of the findings [16,17].

Beyond its scientific contribution, this research also provides practical guidance for industrial applications [18]. The results offer insights into selecting efficient and cost-effective fluid combinations without major design modifications [19]. Moreover, the study considers environmental aspects by noting that while R134a performs well thermally, its high Global Warming Potential (GWP) presents sustainability concerns [20]. The outcomes therefore emphasize the need for balancing thermal efficiency with environmental responsibility in fluid selection [21]. These insights are expected to support future efforts in designing efficient and sustainable heat exchanger systems [22].

**Table 1**
State of the art of previous research related to heat exchangers

| Reference | Method | Fluid Used | Main Results |
|---|---|---|---|
| [23] | Theoretical calculations (LMTD & Kern method) + CFD simulation (ANSYS) | Hot water & cold water (working fluids in shell-and-tube) | CFD is capable of predicting performance with a deviation of 1.05% from theory; design optimization improves the heat transfer coefficient while maintaining control over pressure drop. |
| [24] | Experiments + CFD modeling | Air (as HTF) + Organic PCM A27 inside the tube | Adding fins & increasing PCM conductivity reduces melting time by 46% and solidification time by 35%; energy efficiency increases significantly. |
| [25] | 3D CFD (SolidWorks Flow Simulation) | Olive paste (inner tube) + hot water (jacket) | Pre-heating olive paste with HE reduces malaxation time, maintains oil quality, and improves extraction process efficiency. |
| [26] | CFD (ANSYS Fluent) | Nanofluid: water + Cu nanoparticles / R22 (chloro-difluoromethane) | Nanoparticles significantly increase the Nusselt number at high Re; turbulence from the turbulator enhances heat transfer. |
| [27] | CFD (Finite Volume, k-ω SST) + AI (ANN, RSM, Genetic Algorithm) | Water (working fluid in inner & outer twisted tubes) | Design optimization results in a 16.35% increase in Performance Evaluation Criterion (PEC); twisted tri-lobe geometry is more efficient than oval or other multi-lobe geometries. |

## 2. Methodology

This research methodology uses a numerical approach based on Computational Fluid Dynamics (CFD) with SolidWorks Flow Simulation software. The research stages began with a literature review to obtain a theoretical basis regarding the working principles of heat exchangers and the results of previous studies. Next, the heat exchanger geometry was modeled using SolidWorks CAD, then boundary conditions were set in the form of a fluid inlet temperature of 78 °C and the selection of fluids used, namely three cooling fluids (Nitrogen, Oxygen, R134a) and three flowing fluids (Argon, Water, Ethanol). The thermophysical properties of each fluid were input into the software according to the literature data. The model was then *meshed* to obtain an optimal grid division so that the simulation could produce accurate results with good computational efficiency.

The next stage was CFD simulation for each combination of cooling fluid and flowing fluid. The simulation results in the form of temperature distribution, flow velocity, and pressure drop were extracted for analysis. The average temperature data of each fluid was then processed using one-way ANOVA statistical analysis to test for significant differences between fluids. Descriptive analysis is also performed through graphs, tables, and contour visualizations to illustrate heat transfer patterns. From these results, conclusions can be drawn about the performance of each fluid, thereby obtaining recommendations for selecting the most optimal fluid to improve heat exchanger efficiency. CFD simulations were performed for each combination of cooling fluid and flowing fluid. The results obtained were temperature distribution, flow velocity, and pressure drop, which were then extracted for analysis. The average temperature data was processed using one-way ANOVA statistical analysis to test for significant differences between fluids. Descriptive analysis was performed using graphs, tables, and contour visualizations to illustrate heat transfer patterns. The study does not clearly explain the assumptions, simplifications, or possible sources of error that may exist in the SolidWorks Flow Simulation process. Clarifying these aspects is important to understand the reliability and limitations of the numerical results.

Numerical analysis in this study was performed using SolidWorks Flow Simulation, which applies the principles of Computational Fluid Dynamics (CFD). The model used describes a counterflow heat exchanger consisting of two concentric pipes—the inner pipe functions as a hot-side working fluid channel, while the outer pipe is used as a cooling fluid channel. The geometry was created with SolidWorks CAD and divided using a structured mesh to ensure smooth and accurate temperature and flow velocity distribution. Based on the Reynolds number calculation results (less than 2300 for all fluids tested), the flow was classified as laminar.

The boundary conditions were set as follows: the hot fluid inlet temperature was 78 °C and the cooling fluid inlet temperature was 25 °C. Each fluid was given a constant mass flow rate according to its physical properties. The outlet boundary was modeled using a constant static pressure condition, while the walls were considered adiabatic except in the heat transfer area between the two fluids.

To increase credibility and ensure that the simulation results are reproducible, a mesh independence test was performed by refining the grid until the temperature variation between iterations was less than 1%. The numerical model was also validated with empirical heat transfer correlations from the Dittus–Boelter and Sieder–Tate equations. The validation results showed a deviation of less than 5%, proving that this model is accurate and reliable for analyzing the performance of heat exchangers with various types of fluids.

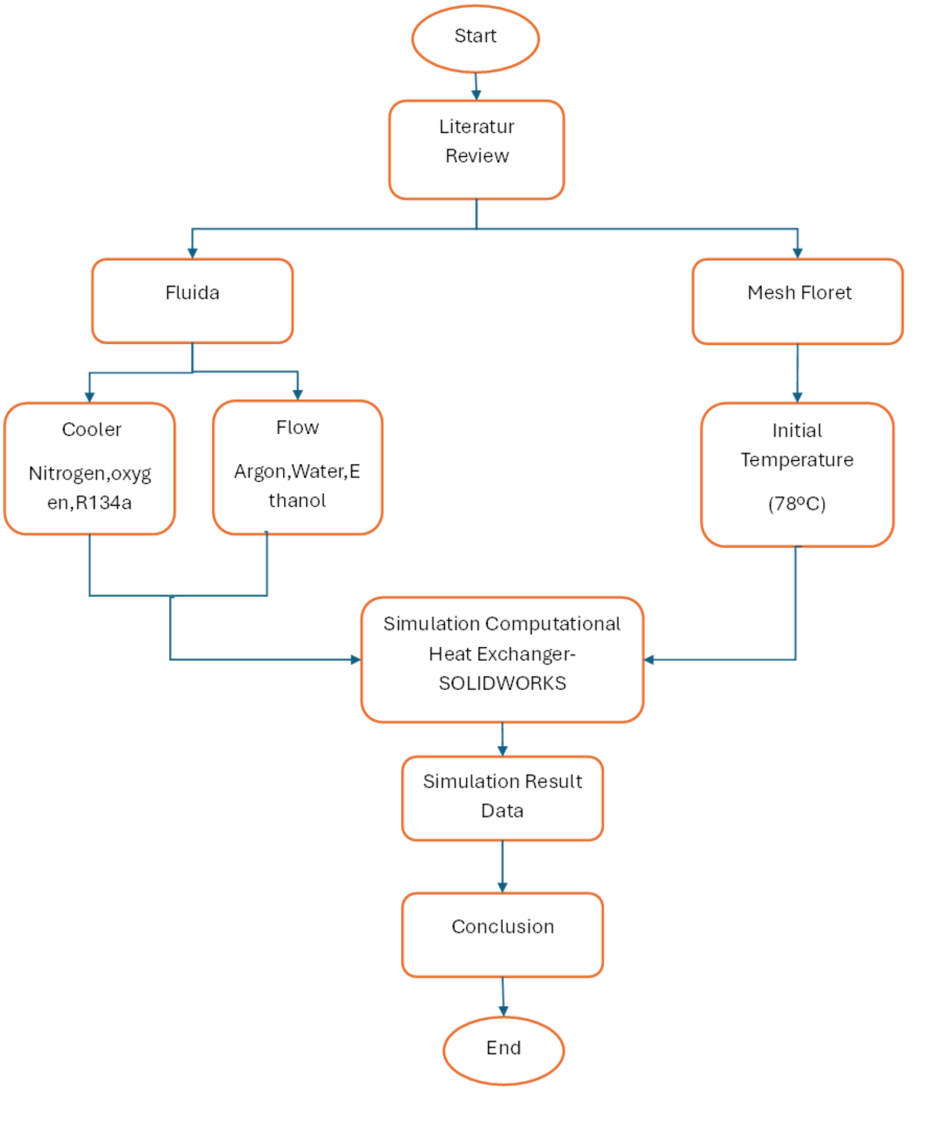



**Fig.1.** Flow diagram

*2.1 Equations*

The calculation method for *heat exchanger* analysis is generally based on the basic relationships of mass and energy conservation and selected convective heat transfer correlations [28]. This section presents key formulas that are often used in numerical simulation and verification before or after *CFD* simulation. All equations are given standard notation; consistent units (SI) are expected when entering values. If necessary, these formulas can be applied repeatedly for each combination of flowing fluid and cooling fluid that you compare [29]. The following is a collection of formulas and brief explanations [30].

1. Continuity (mass, incompressible)

$$\nabla \cdot \mathrm{u} = 0 \qquad (1)$$

Description: Ensures mass conservation for incompressible flow. Used in all flow simulations as a basic equation.[31]

2. Momentum — Navier–Stokes (incompressible, Newtonian)

$$\rho(\partial \boldsymbol{u}/\partial t + \boldsymbol{u}\cdot\nabla\boldsymbol{u}) \;= -\nabla p + \mu\nabla^{\wedge}2\,\boldsymbol{u} + \rho\boldsymbol{g} \quad (2)$$

Description**:** Determines the distribution of velocity and pressure. For steady-state simulations,[32]

3. Energy — heat convection–diffusion equation

$$\rho C\_p\;(\partial T/\partial t + \boldsymbol{u}\cdot\nabla T) = k\nabla^{\wedge}2\;T + \quad (3)$$

Note: Phi($\phi$) is viscous dissipation (often neglected if small). Use to predict the temperature distribution in the fluid domain.[33]

4. Convective heat rate (Newton) + Nusselt definition

$$Q^{\cdot} = hA\Delta T, "Nu" = \frac{hD_h}{k} \quad (4)$$

Note: Calculate $h$ from the Nusselt correlation (e.g., Dittus–Boelter for turbulent flow) then use $Q^{\cdot}$ for the transferred energy. $D\_h$ = hydraulic diameter, $k$ = fluid conductivity.[34]

5. Pressure drop — Darcy–Weisbach

$$\Delta P = f\,\frac{L}{D_h} \times \frac{\rho V^2}{2} \quad (5)$$

Note: Friction factor Important for assessing pressure penalty when selecting fluid (viscosity, Re).[35]

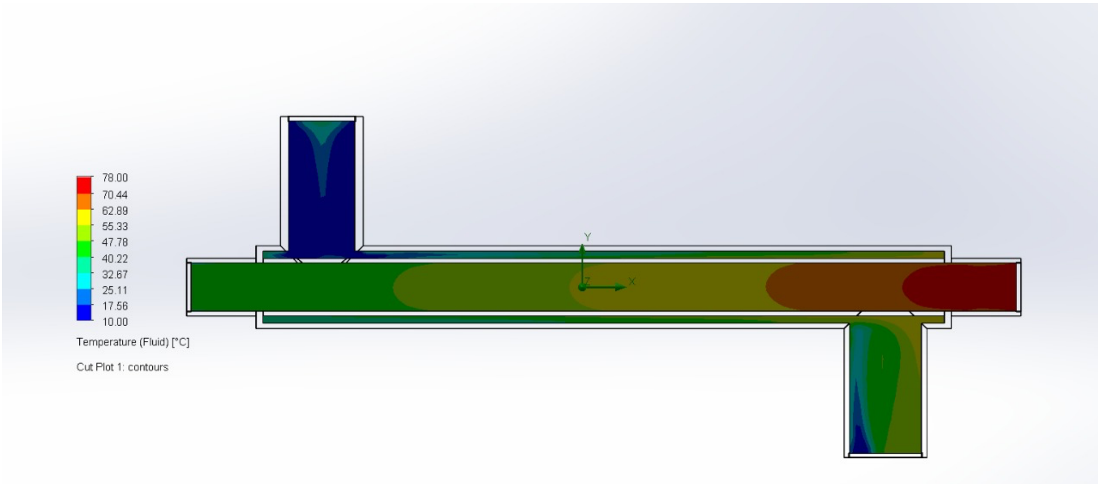
Table 2 shows the thermophysical properties of several fluids used, including density, thermal conductivity, specific heat, and viscosity at 0 °C. The data shows that water has significant advantages with its high density (998.2 kg/m³), thermal conductivity (0.6 W/m·K), and very large specific heat capacity (4182 J/kg·K), making it the most effective medium in heat transfer processes. In contrast, gaseous fluids such as argon, nitrogen, and oxygen have relatively low density and heat capacity, making them less efficient as thermal energy storage media. Ethanol has a fairly high specific heat (2380 J/kg·K), but its viscosity is high (1.78 mPa·s), which can increase pressure losses in the flow. As for R134a, despite its low density, it is still widely used in cooling systems because its thermal properties support the refrigeration process.

**Table 2**
Thermophysical properties of fluids used [8,14,36]

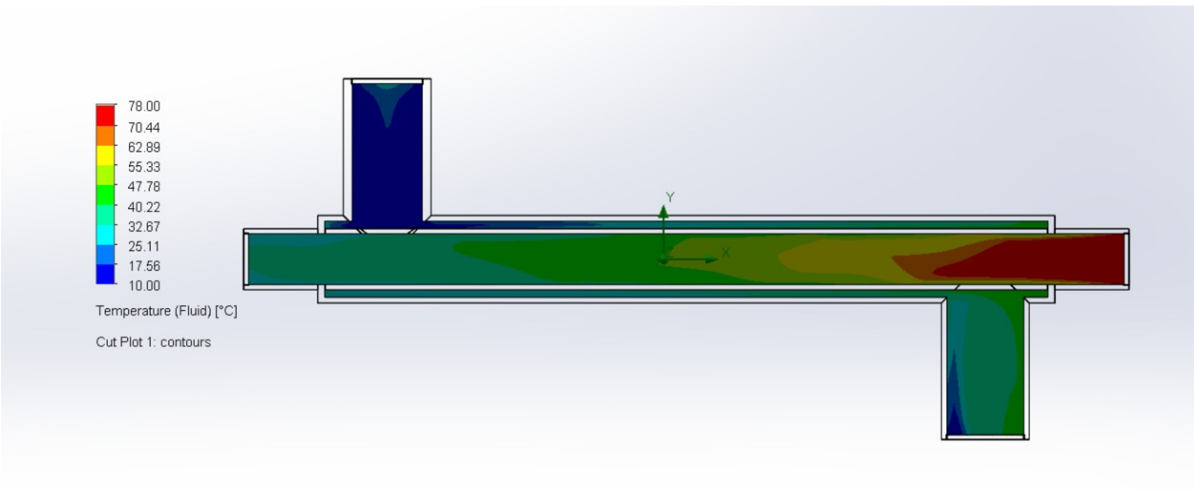
| Fluid | Density (ρ) kg/m³ | Thermal Conductivity (W/m · K) | Specific Heat (Cp) (J/kg · K) | μ at 0 °C (mPa·s) (Ns/m²) |
|---|---|---|---|---|
| Argon | 1.6228 | 0.0158 | 520.64 | 0.0212 |
| Ethanol | 1.263 | 0.0207 | 2380 | 1.78 |
| Water | 998.2 | 0.6 | 4182 | 1.79 |
| Nitrogen | 1.138 | 0.0242 | 1040 | 0.0166 |
| Oxygen | 1.2999 | 0.0246 | 918 | 0.0192 |
| R134a | 1.207 | 0.081 | 1420 | 0.326 |

## 3. Results and Discussion

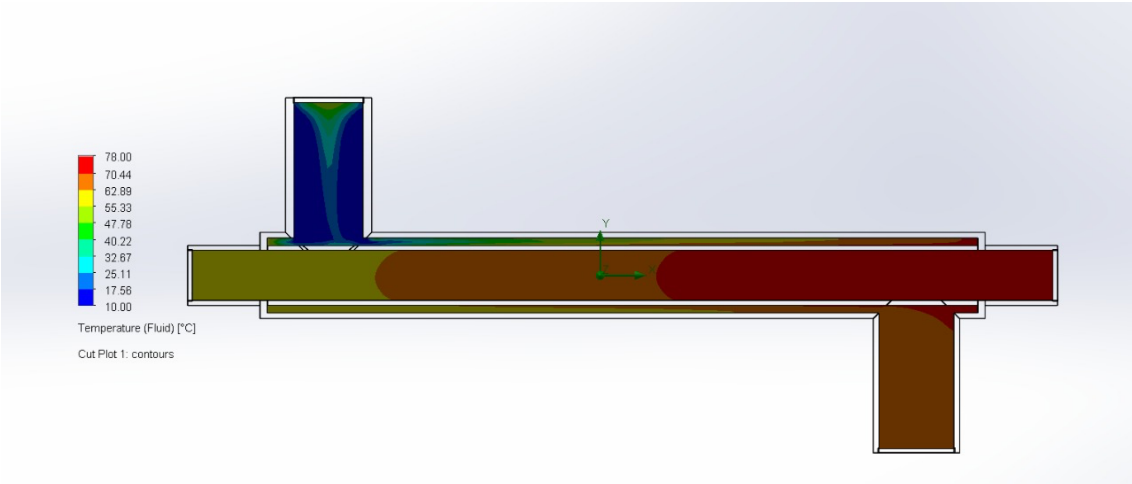
Figures 2 a, b, and c show the temperature distribution of the fluid in the heat exchanger with variations in fluid type, namely Nitrogen, Oxygen, and R134a. Figure 2a shows that the fluid experiences a gradual increase in temperature from the inlet to the outlet, indicating a uniform heat transfer process throughout the channel. Meanwhile, in Figure 2b, the temperature distribution tends to be uneven, where the fluid remains at a low temperature until the middle of the pipe and only experiences a significant increase at the outlet. This condition indicates that heat transfer in the fluid is less than optimal along the flow path. In Figure 2c, the heat distribution appears more dominant and spreads evenly along the channel. Yellow to red colors appear from the middle of the pipe, indicating that the fluid is able to absorb heat energy faster than the previous two conditions. These simulation results show differences in heat transfer capabilities between each fluid. The average temperatures of nitrogen, oxygen, and R134a fluids, where R134a shows the highest average temperature.




(a)




(b)




(c)

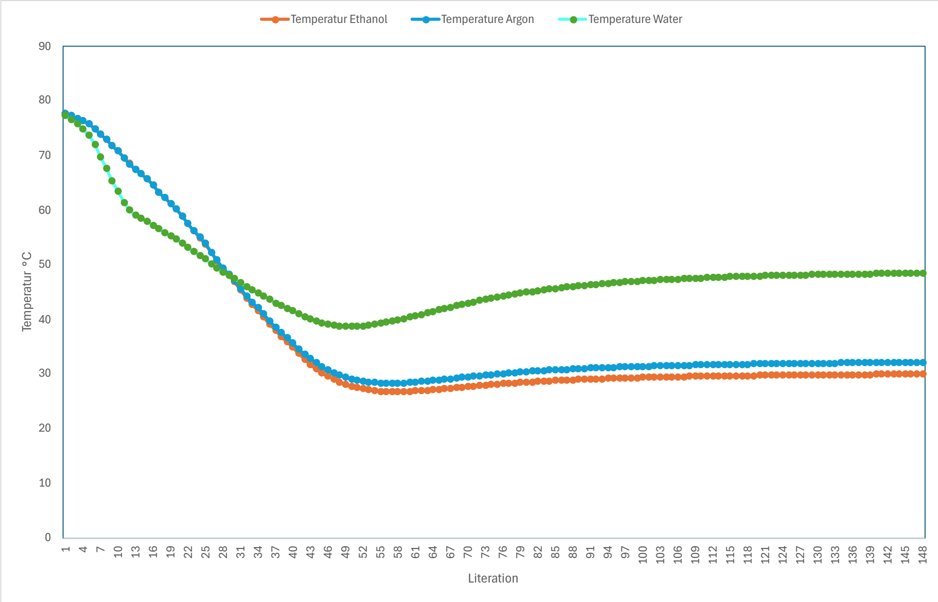
**Fig. 2.** Contour of the heat exchanger's effect on fluids: (a) Nitrogen-Ethanol (b) Argon-Oxygen (c) Water-R134a

Figure 3a shows the temperature changes of ethanol, argon, and water over iterations. The three fluids were initially at a high temperature of around 75–80 °C, then experienced a significant decrease to a low point in the range of 25–35 °C before finally stabilizing. From the pattern observed, water maintained a higher temperature than ethanol and argon, indicating that water has a greater specific heat capacity (Cp) and is therefore able to store heat energy longer. Conversely, ethanol and argon experienced a faster decrease in temperature, indicating that heat transfer occurs more easily in these two fluids.

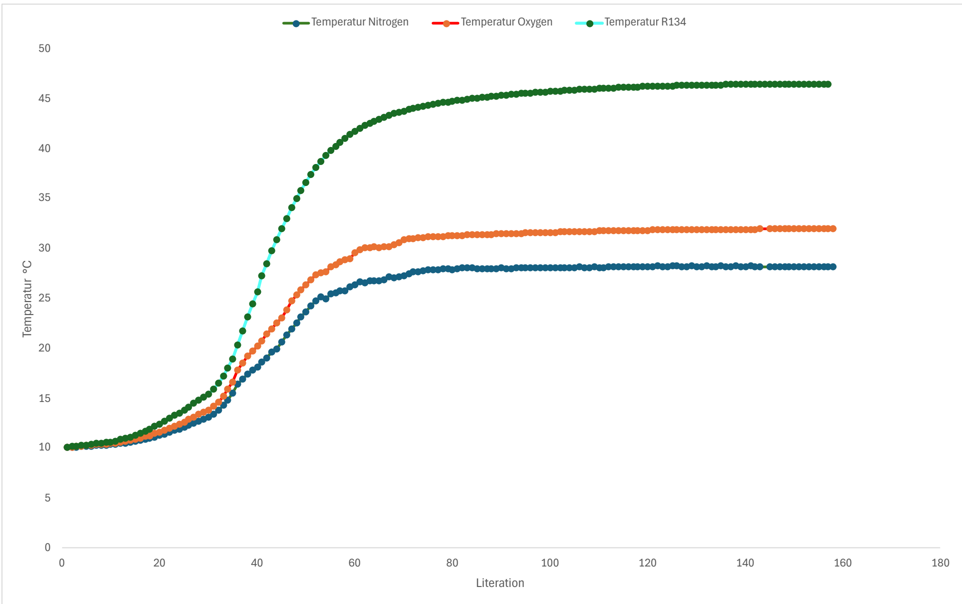
Meanwhile, Figure 3b shows the temperature changes in nitrogen, oxygen, and R134a with an upward pattern from the initial condition of around 10 °C to a steady state in the range of 25–47 °C. Of the three fluids, R134a showed the highest temperature increase, followed by oxygen, while nitrogen was at the lowest position. This proves that R134a has a greater heat absorption capacity than the other two fluids. These differences in thermal characteristics are in line with the results of the ANOVA analysis, which shows a significant difference in the average temperature between fluids.

The simulation results show clear temperature variations among the fluids tested. These differences are mainly due to the differences in the thermophysical properties of each fluid, including density, viscosity, specific heat capacity, and thermal conductivity. Water has high specific heat (4182 J/kg·K) and high thermal conductivity (0.6 W/m·K), enabling it to absorb and transfer heat more effectively. In contrast, the higher viscosity of ethanol increases flow resistance, resulting in greater pressure loss and lower convection efficiency. Argon, with its low density and specific heat, has limited heat storage capacity, causing it to cool down more quickly. For refrigerants, R134a exhibits good heat absorption due to its thermal conductivity and favorable phase change potential, while nitrogen and oxygen, as light gases, have low heat capacity that limits heat transfer. Thus, the observed temperature difference is directly related to the thermophysical behavior of each fluid, not simulation uncertainty. This explanation confirms that the superior performance of water and R134a stems from their high specific heat and efficient energy transfer characteristics.

Therefore, it can be concluded that the thermophysical properties of each fluid directly affect the effectiveness of heat transfer in a heat exchanger system.




(a)




(b)

**Fig. 3.** Average output temperature for fluids: (a) Flow (b) Coolant

Table 3 shows the results of the one-way ANOVA analysis on temperature variation show that the calculated F value (72.5635) is much greater than the critical F value (3.01499) with a very small p-value (3.55E-28) that is far below the significance level of 0.05. This indicates that there is a significant difference between the average temperatures of the three fluid groups, namely Nitrogen, Oxygen, and R134a. From the summary results, it is known that the highest average temperature is found in the R134a fluid (36.286 °C), followed by Oxygen (26.039 °C), and the lowest is Nitrogen (23.361 °C). The different *variance* values also reinforce that each group has a distinctive data distribution. Thus, it can be concluded that the type of fluid has a real effect on temperature variation in the system, where R134a tends to produce higher temperatures than other fluids.

**Table 3**
Analysis of average temperature in coolant fluid

| *Groups* | *Count* | *Sum* | *Average* | *Variance* | |
|---|---|---|---|---|---|
| Nitrogen Temperature | 157 | 3667.601 | 23.36052 | 47.12355 | |
| Oxygen Temperature | 157 | 4088.127 | 26.03903 | 66.82996 | |
| R134 Temperature | 157 | 5696.977 | 36.28648 | 188.1586 | |
| *Source of Variation* | *SS* | *df* | *MS* | *F* | *P-value* | *F crit* |
| Between Groups | 14614.88 | 2 | 7307.438 | 72.5635 | 3.55E-28 | 3.01499 |
| Within Groups | 47,129.49 | 468 | 100.704 | | | |
| Total | 61,744.37 | 470 | | | | |

Table 4 shows the the results of a one-factor ANOVA test on fluid temperature variations, a calculated F value of 34.30275 was obtained, which is much greater than the critical F value of 3.016175. In addition, the p-value (1.42E-14) is very small and far below the significance level of 0.05. This indicates that there is a significant difference in average temperature between the three fluids, namely Ethanol, Argon, and Water. From the data summary, it is known that the fluid with the highest average temperature is Water (48.043 °C), followed by Argon (38.284 °C), and the lowest is Ethanol (36.865 °C). The different variance values for each fluid also show a characteristic variation in data distribution. Thus, it can be concluded that the type of fluid has a significant effect on temperature variation, where Water tends to have a higher temperature than Ethanol and Argon. However, reporting only the *average temperature* is insufficient to fully justify the performance of the heat exchanger. Additional parameters such as the *overall heat transfer coefficient (U)* and *thermal effectiveness (ε)* should be analyzed to provide a more comprehensive and reliable evaluation of the system's heat transfer performance.

**Table 4**
Analysis of average temperature in flowing fluids

| *Groups* | *Count* | *Sum* | *Average* | *Variance* | | |
|---|---|---|---|---|---|---|
| Ethanol Temperature | 148 | 5455.989 | 36.86479 | 218.8434 | | |
| SArgon Temperature | 148 | 5665.967 | 38.28356 | 198.1623 | | |
| Water Temperature | 148 | 7,110.419 | 48.04337 | 62.39734 | | |
| *Source of Variation* | *SS* | *df* | *MS* | *F* | *P-value* | *F crit* |
| Between Groups | 10963.23 | 2 | 5481.614 | 34.30275 | 1.42E-14 | 3.016175 |
| Within Groups | 70,472.24 | 441 | 159,801 | | | |
| Total | 81,435.47 | 443 | | | | |

## 4. Conclusion

It's important to remember that judging performance only by higher average temperatures can be deceptive. In most heat exchanger applications, a lower outlet temperature of the working or cooling fluid generally indicates more effective heat removal and improved thermal performance. Therefore, the analysis should focus on the temperature difference (ΔT), overall heat transfer coefficient (U), or effectiveness (ε) rather than just the average temperature to accurately assess system performance. Evaluating heat exchanger performance based only on average temperature can be misleading. In most cases, a lower outlet temperature of the working or cooling fluid indicates better heat removal and overall system efficiency. Therefore, performance assessments should also include parameters such as the temperature difference (ΔT), overall heat transfer coefficient (U), and effectiveness (ε). These parameters provide a more complete and quantitative understanding of system behavior. By including them, the evaluation becomes more reliable and less dependent on single-temperature indicators. The simulation and one-way ANOVA results confirm that the type of fluid significantly affects heat transfer performance. For cooling fluids (Nitrogen, Oxygen, and R134a), the calculated F-value (72.56) is much greater than the critical F-value (3.01), with a p-value of $3.55 \times 10^{-28}$, far below the 0.05 significance level. Similarly, for flowing fluids (Ethanol, Argon, and Water), the F-value (34.30) exceeds the critical F-value (3.02), with a p-value of $1.42 \times 10^{-14}$, also below 0.05. These findings confirm that the performance differences among the tested fluids are statistically significant. Thus, the claim of significant variation in fluid performance is well supported by quantitative evidence. Fluids with higher specific heat capacity and thermal conductivity, such as water, can absorb and transfer heat more efficiently. In contrast, fluids with higher viscosity experience greater flow resistance, resulting in lower convective heat transfer. Density differences also influence the energy-carrying capacity of each fluid within the system. Collectively, these thermophysical properties determine how effectively a fluid can absorb, store, and transfer heat energy. This relationship explains why certain fluids demonstrate superior or inferior performance in the heat exchanger setup. Among all tested fluids, water as the working fluid and R134a as the coolant showed the most stable and efficient heat transfer performance. Water's high specific heat and conductivity make it effective for storing and transferring energy. Meanwhile, R134a demonstrated strong heat absorption and stable flow characteristics, making it an effective coolant under the tested conditions. However, its high Global Warming Potential (GWP) poses serious environmental concerns. This limitation must be acknowledged to ensure that future recommendations remain environmentally responsible.

Future studies should therefore explore low-GWP refrigerants that can achieve both high thermal performance and sustainability. Additionally, including parameters such as the overall heat transfer rate (Q), convective coefficient (h), and Nusselt number will strengthen performance evaluations. Combining experimental validation with simulation data would also improve model accuracy. These steps can ensure that conclusions are not only statistically valid but also practically applicable. Ultimately, this approach supports the design of efficient and environmentally conscious heat exchanger systems. Considering these aspects, the combination of water and R134a can still be considered a balanced pair in maintaining operational stability and thermal effectiveness. While R134a offers effective cooling performance, it's important to recognize its high Global Warming Potential (GWP), which raises environmental and sustainability concerns. Recommending R134a as the optimal coolant without addressing this limitation may overlook its environmental impact. Therefore, future research should also explore alternative low-GWP refrigerants that balance thermal efficiency with environmental responsibility. Nevertheless, further research with experimental validation and parametric analysis is recommended to strengthen the conclusions obtained. However, the suggestion that water and R134a can be recommended as the optimal pair should be interpreted with caution. This conclusion may be overgeneralized if applied without considering the specific operating context. The suitability of this fluid combination depends on several factors, including operating pressure, material compatibility, safety requirements, and environmental regulations. These variables can significantly influence the thermal and operational behavior of the heat exchanger. Therefore, recommendations regarding optimal fluid pairs should be made within the framework of the intended application rather than as a universal conclusion.

## References

[1] Chongxu Wang dan Zitong Zhang. "The Aerodynamic Optimization of New Energy Vehicles." *Highlights in Science, Engineering and Technology*, Vol. 96 (2024): **29–39**. https://doi.org/10.54097/jmzwas23 -

[2] Xia, Zhijie, and Mengjun Huang. "Optimizing the Aerodynamic Efficiency of Electric Vehicles via Streamlined Design: A Computational Fluid Dynamics Approach." *International Journal of Heat & Technology* 42, no. 3 (2024). https://doi.org/10.18280/ijht.420315 -

[3] Paramasivan, Shankarsana. "Analysis of Tesla CyberTruck Speed on the Velocity and Pressure Distribution Using SimFlow Software." *Advanced and Sustainable Technologies (ASET)* 3, no. 1 (2024): 71-79.https://doi.org/10.58915/aset.v3i1.802 -

[4] Afianto, Darryl, Yu Han, Peiliang Yan, Yan Yang, Anas FA Elbarghthi, and Chuang Wen. "Optimisation and efficiency improvement of electric vehicles using computational fluid dynamics modelling." *Entropy* 24, no. 11 (2022): 1584. https://doi.org/10.3390/e24111584 -

[5] Valencia, Alvaro, and Nicolas Lepin. "Effect of Spoilers and Diffusers on the Aerodynamics of a Sedan Automobile." *International Journal of Heat & Technology* 42, no. 4 (2024). https://doi.org/10.18280/ijht.420406. -

[6] Badu, Prajash, Rahul Panjiyar, Kushal Guragain, and Pankaj Yadav. "CFD Analysis of Rear Spoiler Effects on Vehicles Aerodynamic Performance." *OODBODHAN* 7 (2024): 16-22.https://doi.org/10.3126/oodbodhan.v7i1.75760. -

[7] Buscariolo, Filipe F., Gustavo RS Assi, and Spencer J. Sherwin. "Computational study on an Ahmed Body equipped with simplified underbody diffuser." *Journal of Wind Engineering and Industrial Aerodynamics* 209 (2021): 104411.https://doi.org/10.1016/j.jweia.2020.104411. -

[8] Huminic, Angel, and Gabriela Huminic. "Aerodynamics of curved underbody diffusers using CFD." *Journal of Wind Engineering and Industrial Aerodynamics* 205 (2020): 104300. https://doi.org/10.1016/j.jweia.2020.104300. –

[9] Park, Gwanyong, Changmin Kim, Minhyung Lee, and Changho Choi. "Building geometry simplification for improving mesh quality of numerical analysis model." *Applied Sciences* 10, no. 16 (2020): 5425.https://doi.org/10.3390/app10165425

[10] Xu, Fusuo, Jianzhi Yang, and Xiaowei Zhu. "A comparative study on the difference of CFD simulations based on a simplified geometry and a more refined BIM based geometry." *Aip Advances* 10, no. 12 (2020). https://doi.org/10.1063/5.0031907

[11] Fu, Chen, Mesbah Uddin, and Chunhui Zhang. "Computational analyses of the effects of wind tunnel ground simulation and blockage ratio on the aerodynamic prediction of flow over a passenger vehicle." *Vehicles* 2, no. 2 (2020): 318-341. https://doi.org/10.3390/vehicles2020018

[12] Wang, Shibo, Terence Avadiar, Mark C. Thompson, and David Burton. "Effect of moving ground on the aerodynamics of a generic automotive model: The DrivAer-Estate." *Journal of Wind Engineering and Industrial Aerodynamics* 195 (2019): 104000.https://doi.org/10.1016/j.jweia.2019.104000

[13] Marcon, Julian, Michael Turner, Joaquim Peiro, David Moxey, Claire Pollard, Henry Bucklow, and Mark Gammon. "High-order curvilinear hybrid mesh generation for CFD simulations." In *2018 AIAA Aerospace Sciences Meeting*, p. 1403. 2018. https://doi.org/10.1016/j.cpc.2019.01.015

[14] Muflikhun, Muhammad Akhsin, Hifni Muktar Ariyadi, and Alfian Muhammad Fadhillah Hidayat. "The Evaluation of Mesh Characteristics of the Car Modeling and Simulation Using CFD Analysis." *Jurnal Rekayasa Mesin* 13, no. 1 (2022): 129-140. https://doi.org/10.21776/ub.jrm.2022.013.01.14

[15] Al-Saadi, Ahmed, Khaled Al-Farhany, Kadhim K. Idan Al-Chlaihawi, Wasim Jamshed, Mohamed R. Eid, El Sayed M. Tag El Din, and Zehba Raizah. "Improvement of the aerodynamic behavior of a sport utility vehicle numerically by using some modifications and aerodynamic devices." *Scientific Reports* 12, no. 1 (2022): 20272. https://doi.org/10.1038/s41598-022-24328-w

[16] Lintermann, Andreas. "Computational meshing for CFD simulations." In *Clinical and biomedical engineering in the human nose: A computational fluid dynamics approach*, pp. 85-115. Singapore: Springer Singapore, 2020. https://doi.org/10.1007/978-981-15-6716-2_6

[17] Guerrero, Alex, Robert Castilla, and Giorgio Eid. "A numerical aerodynamic analysis on the effect of rear underbody diffusers on road cars." *Applied Sciences* 12, no. 8 (2022): 3763. https://doi.org/10.3390/app12083763

[18] Rao, Anirudh N., Jie Zhang, Guglielmo Minelli, Branislav Basara, and Siniša Krajnović. "An LES investigation of the near-wake flow topology of a simplified heavy vehicle." *Flow, Turbulence and Combustion* 102, no. 2 (2019): 389-415.https://doi.org/10.1007/s10494-018-9959-6

[19] Pátý, Marek, Michael Valášek, Emanuele Resta, Roberto Marsilio, and Michele Ferlauto. "Passive Control of Vortices in the Wake of a Bluff Body." *Fluids* 9, no. 6 (2024): 131. https://doi.org/10.3390/fluids9060131

[20] Nebot, Eugeni Pérez, Antim Gupta, and Mahak Mahak. "Investigating the Impact of Structural Features on F1 Car Diffuser Performance Using Computational Fluid Dynamics (CFD)." *Mathematics* 13, no. 9 (2025): 1455. https://doi.org/10.3390/math13091455

[21] Moghimi, P., and R. Rafee. "Numerical and experimental investigations on aerodynamic behavior of the Ahmed body model with different diffuser angles." *Journal of Applied Fluid Mechanics* 11, no. 4 (2018): 1101-1113. https://doi.org/10.29252/jafm.11.04.27923

[22] Podvin, Bérengère, Stéphanie Pellerin, Yann Fraigneau, Antoine Evrard, and Olivier Cadot. "Proper orthogonal

decomposition analysis and modelling of the wake deviation behind a squareback Ahmed body." *Physical Review Fluids* 5,no.6(2020):064612. https://doi.org/10.48550/arXiv.1909.13129

[23] Zhou, Haichao, Qingyun Chen, Runzhi Qin, Lingxin Zhang, and Huiyun Li. "Investigation of wheelhouse shapes on the aerodynamic characteristics of a generic car model." *Advances in Mechanical Engineering* 13, no. 12 (2021): 16878140211066842.doi: https://doi.org/10.1177/16878140211066842

[24] Aulakh, Deepinder Jot Singh. "Effect of underbody diffuser on the aerodynamic drag of vehicles in convoy." *Cogent Engineering* 3, no. 1 (2016): 1230310. doi: https://doi.org/10.1080/23311916.2016.1230310

[25] Ashton, Neil, Danielle C. Maddix, Samuel Gundry, and Parisa M. Shabestari. "AhmedML: High-fidelity computational fluid dynamics dataset for incompressible, low-speed bluff body aerodynamics." *arXiv preprint arXiv:2407.20801* (2024).https://doi.org/10.48550/arXiv.2407.20801

[26] Yuan, Zhiqun, Zhengqi Gu, Yiping Wang, and Xiaoqun Huang. "Numerical investigation for the influence of the car underbody on aerodynamic force and flow structure evolution in crosswind." *Advances in Mechanical Engineering* 10, no. 10 (2018): 1687814018797506. https://doi.org/10.1177/1687814018797506

[27] Alkan, B. (2020). *Aerodynamic Analysis of Rear Diffusers for a Passenger Car Using CFD*. OSF Preprints. https://doi.org/10.31224/osf.io/ntkq4