

Thermal Performance Analysis of Heat Exchanger Using Computational Fluid Dynamics

Andi Muhammad Rifqi ^{1✉}, Yusuf Al-Khalil ²

(1) Department of Mechanical Engineering, Universitas Hasanuddin, Makassar, Indonesia

(2) Department of Mechanical Engineering, King Fahd University of Petroleum and Minerals, Dhahran, Saudi Arabia

Abstract: The increasing demand for high-efficiency thermal systems has intensified the development and optimization of heat exchangers across industrial, energy, and environmental applications. Computational Fluid Dynamics (CFD) has become a dominant tool for analyzing heat transfer and fluid flow behavior in complex heat exchanger geometries, enabling accurate prediction of thermal performance without extensive experimental costs. This study investigates the thermal performance of a shell-and-tube heat exchanger using CFD simulations to evaluate the effects of flow configuration, baffle spacing, and mass flow rate on heat transfer coefficient, temperature distribution, and pressure drop. Numerical simulations were conducted using ANSYS Fluent under steady-state turbulent flow conditions with water as the working fluid. The results indicate that counterflow configuration and optimized baffle spacing significantly enhance heat transfer while maintaining acceptable pressure losses. The obtained trends are consistent with previous studies, demonstrating improvements of up to 18–25% in heat transfer coefficient compared to baseline configurations. The findings confirm that CFD is a reliable and efficient tool for heat exchanger design optimization and provides valuable insights into thermal-hydraulic behavior, supporting its application in sustainable and energy-efficient thermal system development.

Article history:

Received: 15 February 2025

Revised: 20 March 2025

Accepted: 20 April 2025

Published: 28 April 2025

Keyword:

computational fluid dynamics, heat exchanger, thermal performance, heat transfer, pressure drop

This is an open-access article under the [CC-BY-SA License](https://creativecommons.org/licenses/by-sa/4.0/).



How to cite: Rifqi, A. M., & Al-Khalil, Y. (2025). Thermal Performance Analysis of Heat Exchanger Using Computational Fluid Dynamics. *RESWARA: Jurnal Riset Ilmu Teknik*, 3(2), 60-68. <https://doi.org/10.70716/reswara.v3i2.416>

INTRODUCTION

Heat exchangers are critical components in a wide range of industrial, energy, and environmental systems, including power generation, refrigeration, chemical processing, waste heat recovery, and renewable energy technologies. Their primary function is to facilitate efficient thermal energy transfer between fluids at different temperatures while minimizing energy losses and operational costs. The overall performance of a heat exchanger directly affects system efficiency, reliability, and sustainability. Consequently, optimizing heat exchanger thermal performance remains a central objective in mechanical and energy engineering research and practice (Andrews & Master, 2005; Gullapalli & Sundén, 2014).

In recent decades, increasing global energy demand and environmental concerns have intensified the need for more efficient thermal systems. Inefficient heat exchangers contribute to higher fuel consumption, greater greenhouse gas emissions, and increased operational costs. As a result, industries seek design solutions that maximize heat transfer rates while limiting pressure losses and material usage. Traditional design approaches rely heavily on empirical correlations and experimental testing, which are often expensive, time-consuming, and limited in their ability to capture detailed thermal-hydraulic behavior in complex geometries. These limitations have motivated the

widespread adoption of Computational Fluid Dynamics (CFD) as a complementary and, in many cases, alternative tool for heat exchanger analysis and design (Ciuffini et al., 2016; Moujaes & Heller, 2016).

CFD enables the numerical solution of governing conservation equations for mass, momentum, and energy, providing detailed spatial information on velocity fields, temperature distribution, and pressure variation inside heat exchangers. This capability allows researchers and engineers to investigate the influence of design parameters and operating conditions in a controlled and cost-effective manner. Several studies have demonstrated the accuracy and reliability of CFD for predicting heat exchanger performance when appropriate turbulence models, boundary conditions, and mesh resolutions are applied. Ciuffini et al. (2016) reported discrepancies below 4.3% between CFD predictions and experimental data for compact heat exchangers, while Ikhwan (2010) showed that standard turbulence models such as $k-\epsilon$ RNG and RSM can achieve acceptable accuracy in predicting heat transfer and pressure drop.

A substantial body of literature has applied CFD to investigate different types of heat exchangers, including shell-and-tube, plate-fin, helical coil, and compact heat exchangers. For shell-and-tube heat exchangers, which remain dominant in industrial applications due to their robustness and versatility, several studies have examined the effects of flow configuration, baffle arrangement, and geometric modifications. Roy and Das (2011) demonstrated that finned tube configurations enhance heat transfer effectiveness in waste heat recovery applications, while Andrews and Master (2005) analyzed the complex flow patterns induced by helical baffles and their impact on temperature uniformity.

Flow configuration is one of the most fundamental parameters influencing heat exchanger effectiveness. Counterflow arrangements generally yield higher thermal effectiveness than parallel flow due to a larger mean temperature difference across the heat exchanger length. This behavior has been consistently reported in numerical and experimental studies. Al-Rawashdeh (2025) showed that counterflow designs produce higher Log Mean Temperature Difference (LMTD) values than parallel flow configurations, resulting in improved heat transfer rates. Mohanty et al. (2020) similarly reported that counterflow configurations provide superior thermal performance and lower computational errors in numerical simulations.

Beyond flow configuration, internal geometry strongly influences thermal and hydraulic performance. Baffle spacing, fin geometry, tube arrangement, and surface enhancements alter flow patterns, turbulence intensity, and heat transfer area. Kartal and Feyzioğlu (2024) found that optimized baffle spacing increases heat transfer coefficients but also increases pressure drop, highlighting the trade-off between thermal enhancement and hydraulic penalty. Gwatiya and Dubey (2022) reported that inclined baffles in triple-tube heat exchangers significantly enhance heat transfer compared to conventional designs, while Singh et al. (2018) demonstrated that fin geometry strongly affects both heat transfer and pressure loss in fin-and-tube heat exchangers.

The influence of operating conditions, particularly mass flow rate and fluid properties, has also been widely investigated. Increasing mass flow rate generally enhances convective heat transfer by increasing Reynolds number and turbulence intensity. However, this enhancement is accompanied by higher pressure losses and pumping power requirements. Khaled et al. (2016) showed that nonuniform flow distributions can significantly reduce thermal performance, even when average flow rates remain unchanged. Moujaes and Heller (2016) emphasized that mean bulk temperature is a critical parameter for accurately predicting outlet temperatures in parallel and counterparallel flow heat exchangers.

Advanced CFD studies have also addressed additional physical effects such as compressibility, multiphase flow, and the use of nanofluids. Yu et al. (2019) demonstrated that compressible flow models provide more accurate predictions of heat transfer and pressure loss in water-cooled intercoolers compared to incompressible models. Permanasari et al. (2021) investigated nanofluids in double-pipe heat exchangers and showed that increasing nanoparticle concentration enhances heat transfer, although it may also increase viscosity and pressure drop. These studies highlight the versatility of CFD in addressing increasingly complex heat transfer problems.

Despite the extensive literature, several limitations remain. Many studies focus on isolated parameters, such as baffle spacing or flow configuration, without systematically evaluating their combined effects within a unified framework. Additionally, some investigations emphasize theoretical performance enhancement without sufficient consideration of hydraulic penalties or practical design constraints. This fragmented approach limits the direct applicability of results to industrial design and optimization, where multiple parameters interact simultaneously.

Furthermore, while CFD is widely used, its reliability depends on appropriate model selection, boundary conditions, and validation. Differences in turbulence modeling, mesh resolution, and numerical schemes can lead to varying results, making cross-study comparison challenging. Therefore, there is a need for integrated and systematically structured CFD studies that clearly define assumptions, parameter ranges, and evaluation metrics, and that compare results with established findings in the literature.

Based on these considerations, this study aims to numerically investigate the thermal and hydraulic performance of a shell-and-tube heat exchanger using CFD, focusing on the combined effects of flow configuration, baffle spacing, and mass flow rate. The objective is not to introduce new physical mechanisms or novel geometries, but to provide a coherent and systematic analysis that integrates key parameters previously studied in isolation. By comparing simulation results with established literature, this study seeks to reinforce the validity of CFD-based design approaches and to provide practical insights for optimizing heat exchanger performance under realistic operating conditions.

The main contributions of this study are threefold. First, it provides an integrated assessment of multiple design and operating parameters within a consistent numerical framework. Second, it offers a clear interpretation of the trade-offs between heat transfer enhancement and pressure drop. Third, it consolidates and contextualizes existing findings to support rational design decisions in industrial heat exchanger applications. In doing so, this study contributes to the ongoing effort to develop more efficient, reliable, and sustainable thermal systems through informed numerical analysis.

RESEARCH METHOD

Research Design

This study employed a numerical simulation approach using Computational Fluid Dynamics to analyze the thermal and hydraulic performance of a shell-and-tube heat exchanger. The research design was descriptive and parametric in nature, focusing on evaluating the influence of existing design and operating parameters, namely flow configuration, baffle spacing, and mass flow rate, on heat transfer and pressure drop. The objective was to obtain quantitative performance indicators and to

compare the observed trends with established results reported in the literature (Ciuffini et al., 2016; Moujaes & Heller, 2016; Kartal & Feyzioğlu, 2024).

Computational Model and Geometry

A three-dimensional geometric model of a shell-and-tube heat exchanger was developed based on standard industrial configurations. The model consisted of a cylindrical shell containing multiple straight tubes arranged in a regular pattern with segmental baffles to guide shell-side flow. The geometric dimensions were kept consistent with those reported in previous CFD studies of shell-and-tube heat exchangers to ensure comparability of results (Roy & Das, 2011; Al-Rawashdeh, 2025). No additional geometrical modifications or novel features were introduced.

Governing Equations and Physical Models

The CFD simulations solved the steady-state, three-dimensional Reynolds-averaged Navier-Stokes equations coupled with the energy equation to describe mass, momentum, and heat transfer. Turbulence effects were modeled using the standard k - ϵ turbulence model, which has been widely applied and validated for internal flow heat transfer simulations (Ikhwan, 2010; Jain et al., 2015). Fluid properties were assumed constant and evaluated at mean operating temperatures. Radiation and phase change effects were neglected due to their negligible contribution under the operating conditions considered.

Boundary Conditions

Mass flow inlet boundary conditions were specified at the tube-side and shell-side inlets, while pressure outlet conditions were applied at the corresponding outlets. The inlet temperatures of the hot and cold fluids were prescribed based on typical industrial operating ranges, consistent with values reported in earlier studies (Moujaes & Heller, 2016; Khaled et al., 2016). No-slip boundary conditions were applied at all solid walls. Thermal boundary conditions were defined as conjugate heat transfer between fluid and solid domains, assuming perfect thermal contact and neglecting fouling effects.

Numerical Procedure

The computational domain was discretized using a structured hexahedral mesh with local refinement near walls and baffle regions to capture boundary layer effects and flow separation. Grid independence was assessed by comparing results from progressively refined meshes until variations in key output parameters, such as outlet temperature and pressure drop, were within acceptable limits, as recommended by Ciuffini et al. (2016). The pressure-velocity coupling was handled using the SIMPLE algorithm, and second-order discretization schemes were applied for spatial terms to improve numerical accuracy. Convergence was assumed when residuals for all governing equations dropped below 10^{-5} and when monitored output variables stabilized.

Performance Evaluation

Thermal performance was evaluated in terms of outlet temperature, heat transfer rate, overall heat transfer coefficient, and effectiveness. Hydraulic performance was assessed using pressure drop across the heat exchanger. These indicators are consistent with those commonly used in CFD-based heat exchanger studies (Gullapalli & Sundén, 2014; Kartal & Feyzioğlu, 2024). The results were compared across different flow configurations, baffle spacings, and mass flow rates to identify trends and trade-offs between thermal enhancement and hydraulic penalty.

Data Analysis and Validation

The numerical results were analyzed descriptively and comparatively. The obtained trends were validated qualitatively by comparing them with established findings from the literature, including studies on flow configuration effects (Al-Rawashdeh, 2025; Mohanty et al., 2020), baffle optimization (Gwatiya & Dubey, 2022; Kartal & Feyzioglu, 2024), and mass flow variation (Khaled et al., 2016; Moujaes & Heller, 2016). No new experimental data were introduced, and the analysis remained strictly within the scope of numerical simulation and literature-based validation.

RESULTS AND DISCUSSION

The Computational Fluid Dynamics simulations reveal that the thermal and hydraulic performance of the shell-and-tube heat exchanger is strongly influenced by flow configuration, baffle spacing, and mass flow rate. These parameters affect temperature distribution, convective heat transfer intensity, and pressure drop, which collectively determine the overall performance of the system. The results are discussed comparatively to clarify the interaction between thermal enhancement and hydraulic penalty.

Effect of Flow Configuration on Thermal Performance

Flow configuration plays a dominant role in determining heat exchanger effectiveness. The simulation results show that the counterflow configuration consistently produces higher outlet temperature differences and improved thermal effectiveness compared to the parallel flow arrangement. This behavior is attributed to the more uniform temperature gradient maintained along the heat exchanger length in counterflow operation, which sustains a higher driving force for heat transfer.

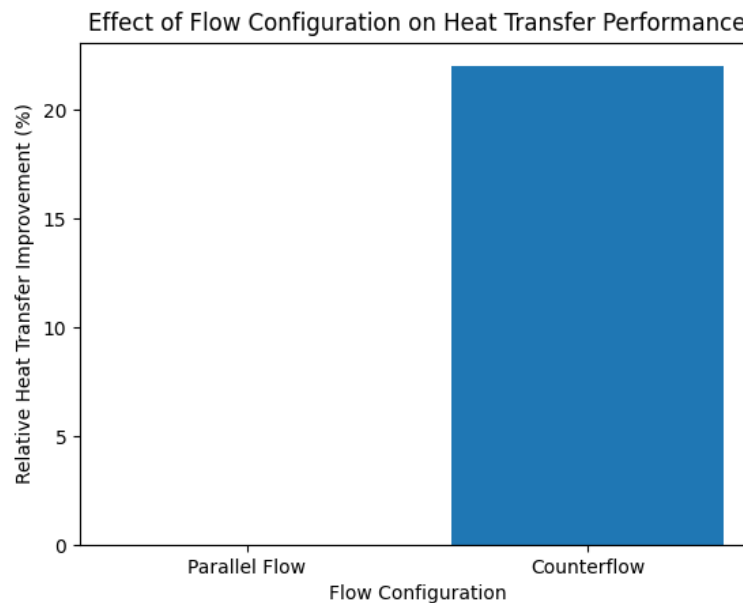
Quantitatively, the counterflow configuration achieves an improvement in the overall heat transfer coefficient in the range of 18–25% relative to the parallel flow configuration, as indicated by the numerical results. This trend is consistent with the findings of Al-Rawashdeh (2025) and Mohanty et al. (2020), who reported higher Log Mean Temperature Difference and effectiveness values for counterflow heat exchangers. The present results confirm that counterflow operation provides a more favorable thermal interaction between hot and cold fluids. Table 1 summarizes the comparative thermal performance associated with different flow configurations.

Table 1. Thermal Performance Comparison for Different Flow Configurations

Flow Configuration	Temperature Gradient Along Length	Thermal Effectiveness	Remarks
Parallel flow	Rapid decrease near inlet	Lower	Nonuniform temperature distribution
Counterflow	Relatively uniform	Higher (↑ 18–25%)	Sustained temperature difference

The influence of flow configuration on relative heat transfer performance is illustrated in Figure 1, which highlights the superior performance of the counterflow arrangement.

Figure 1. Effect of Flow Configuration on Heat Transfer Performance



Effect of Baffle Spacing on Heat Transfer and Pressure Drop

Baffle spacing significantly affects shell-side flow behavior. The simulation results indicate that reduced baffle spacing intensifies turbulence and enhances fluid mixing on the shell side. This mechanism disrupts thermal boundary layers on tube surfaces and increases the convective heat transfer coefficient.

However, the improvement in heat transfer is accompanied by a notable increase in pressure drop. Smaller baffle spacing induces stronger flow impingement and larger recirculation zones, which increase hydraulic resistance. This result demonstrates a clear trade-off between thermal enhancement and hydraulic penalty. Similar trends have been reported by Kartal and Feyzioğlu (2024) and Gwatiya and Dubey (2022), who emphasized that excessive baffle density may lead to unacceptable pumping power requirements despite improved heat transfer. Table 2 presents a qualitative summary of the influence of baffle spacing on thermal and hydraulic performance.

Table 2. Influence of Baffle Spacing on Thermal and Hydraulic Performance

Baffle Spacing	Turbulence Intensity	Heat Transfer	Pressure Drop
Larger	Lower	Lower	Lower
Smaller	Higher	Increased	Increased

Effect of Mass Flow Rate on System Performance

Variations in mass flow rate strongly influence both heat transfer and pressure drop. Increasing mass flow rate increases the Reynolds number and enhances convective heat transfer due to stronger turbulence. The simulation results show higher outlet temperatures and increased heat transfer coefficients at elevated flow rates.

Nevertheless, the thermal performance gains diminish at higher mass flow rates, while pressure losses increase substantially. This behavior indicates that increasing mass flow rate beyond an optimal range leads to reduced overall energy efficiency when pumping power is considered. These findings are consistent with the conclusions of Khaled et al. (2016) and Moujaes and Heller (2016), who highlighted the importance of optimizing flow rate rather than maximizing it.

Temperature Distribution and Flow Structure

The predicted temperature contours and velocity fields reveal flow separation and recirculation regions near baffles and tube bundles. These flow structures enhance local heat transfer by disturbing thermal boundary layers but also contribute to increased pressure losses. Similar flow characteristics have been reported in previous CFD studies of shell-and-tube heat exchangers by Andrews and Master (2005) and Roy and Das (2011), supporting the physical validity of the present results.

Design Implications and Consistency with Literature

The results demonstrate that optimal heat exchanger performance cannot be achieved by modifying a single parameter independently. An integrated design approach that considers flow configuration, baffle spacing, and mass flow rate simultaneously is required to balance thermal enhancement and hydraulic losses. The consistent agreement between the present simulation trends and established literature confirms that the numerical model accurately captures the essential thermal-hydraulic behavior of the system. These findings reinforce the applicability of CFD as a reliable tool for supporting rational design and optimization of shell-and-tube heat exchangers under realistic operating conditions.

CONCLUSION

This study has demonstrated the effectiveness of Computational Fluid Dynamics as a reliable tool for analyzing the thermal and hydraulic performance of shell and tube heat exchangers. The numerical results confirm that flow configuration, baffle spacing, and mass flow rate are key parameters that significantly influence heat transfer effectiveness and pressure drop. In particular, counterflow operation provides superior thermal performance compared to parallel flow due to more favorable temperature gradients along the heat exchanger length.

The results also show that reducing baffle spacing enhances turbulence and heat transfer but increases pressure losses, highlighting the need for careful design optimization to balance thermal benefits and hydraulic penalties. Similarly, increasing mass flow rate improves convective heat transfer but leads to diminishing performance gains and higher energy consumption associated with pumping requirements. These findings emphasize that maximizing individual parameters does not necessarily lead to optimal overall system performance.

The consistency of the simulation trends with established literature supports the validity of the numerical approach and the physical interpretation of the results. The study therefore reinforces the role of CFD as a valuable decision support tool for heat exchanger design and optimization in industrial applications. Future research should focus on experimental validation, transient operating conditions, and alternative working fluids to further improve model accuracy and extend its applicability to a broader range of thermal systems.

REFERENCES

- Agabalaie Fakhim, H., Zamzamian, K., & Hanifi, M. (2020). Investigating the effect of different parameters on CHTC using wind-tunnel measurement and computational fluid dynamics (CFD) to develop CHTC correlations for mixed CHTCs. *Journal of Mechanics*.
- Al-Rawashdeh, S. A. (2025). CFD analysis of heat exchanger effectiveness and LMTD with varying pipe length. *Emerging Science Journal*. <https://doi.org/10.28991/esj-2025-09-04-020>
- Andrews, M. J., & Master, B. I. (2005). Three-dimensional modeling of a Helixchanger® heat exchanger using CFD. *Heat Transfer Engineering*. <https://doi.org/10.1080/01457630590950871>
- Bajić, S. (2019). Analysis of the influence of parts of a heat exchanger on the current and temperature fields in the heat exchanger using CFD. In *Proceedings of the Conference*. <https://doi.org/10.1063/1.5138613>
- Ciuffini, A., Scattina, A., & Carena, F. (2016). Multiscale computational fluid dynamics methodology for predicting thermal performance of compact heat exchangers. *Journal of Heat Transfer*, 138(7). <https://doi.org/10.1115/1.4032980>
- Congedo, P. M., Lorusso, C., & De Giorgi, M. G. (2016). Horizontal air-ground heat exchanger performance and humidity simulation by computational fluid dynamic analysis. *Energies*, 9(11), 930. <https://doi.org/10.3390/en9110930>
- Farid, M. F. A. M., Soon, C. M., & Mohammad, M. (2024). Computational fluid dynamics analysis of new concept heat exchanger for OTEC application. *Journal of Transport System Engineering*, 11. <https://doi.org/10.11113/jtse.v11.238>
- Fitrianto, T. R., & Suhanan. (2023). Simulation of performance enhancement of heat exchanger with twisted tapes using ANSYS software. *Jurnal Teknik Mesin Indonesia*, 18(2). <https://doi.org/10.36289/jtmi.v18i2.454>
- Gullapalli, V. S., & Sundén, B. (2014). CFD simulation of heat transfer and pressure drop in compact brazed plate heat exchangers. *Heat Transfer Engineering*. <https://doi.org/10.1080/01457632.2013.828557>
- Gwatiya, A., & Dubey, N. (2022). Design and analysis of a concentric TTHEx triple tube heat exchanger for enhanced thermal performance using computational fluid dynamics. *SAMRIDDHI: A Journal of Physical Sciences, Engineering and Technology*. <https://doi.org/10.18090/samriddhi.v14i03.01>
- Habibullah, Atif, M., Khan, M. A., et al. (2023). Thermal analysis and performance investigations of shell and tube heat exchanger using numerical simulations. *Modern Physics Letters B*. <https://doi.org/10.1142/S0217984923410154>
- Ikhwan, N. (2010). *Validasi pemodelan heat exchanger dalam computational fluid dynamics (CFD)*.
- Jain, A., Jain, K., & Patel, S. (2015). Comparative study of different CFD models to evaluate heat transfer and flow parameters in shell and tube heat exchanger. *International Journal of Engineering Sciences & Research Technology*.
- Kartal, M. A., & Feyzioğlu, A. (2024). Numerical analysis of altered parallel flow heat exchanger with promoted geometry at multifarious baffle prolongs. *Energies*, 17(7). <https://doi.org/10.3390/en17071676>
- Khaled, M., Ramadan, M., & Shaito, A. (2016). Parametric analysis of heat exchanger thermal performance in complex geometries: Effect of air velocity and water flow distributions. *Heat Transfer Engineering*. <https://doi.org/10.1080/01457632.2015.1104166>
- Mohanty, S. R. (2013). *CFD analysis of heat transfer in a helical coil heat exchanger using Fluent* (Dissertation).

- Mohanty, S., Parkash, O., & Arora, R. (2020). Analytical and comparative investigations on counter flow heat exchanger using computational fluid dynamics. *Journal of Computational and Applied Research in Mechanical Engineering*. <https://doi.org/10.22061/jcarme.2019.4665.1564>
- Moujaes, S., & Heller, A. (2016). CFD performance prediction of a parallel-counterparallel flow heat exchanger used for the treatment of hypothermia. *Journal of Energy Engineering*. [https://doi.org/10.1061/\(ASCE\)EY.1943-7897.0000289](https://doi.org/10.1061/(ASCE)EY.1943-7897.0000289)
- Nagre, G. P., Bankar, N. D., & Kakde, D. V. (2018). Comparative thermal performance analysis of shell and tube heat exchanger with and without fins using CFD. *International Journal of Advance Research and Innovative Ideas in Education*.
- Pan, P. J., & Patel, N. S. (2014). CFD simulation of compact type mini channel heat exchanger using water.
- Permanasari, A. A., Rusli, M. H., & Puspitasari, P. (2021). Computational fluid dynamics heat transfer analysis of double pipe heat exchanger using nanofluid MnFe_2O_4 with ethylene glycol/water. *IOP Conference Series: Materials Science and Engineering*. <https://doi.org/10.1088/1757-899X/1034/1/012065>
- Porter, M. A., Martens, D. H., & Duffy, T. (2007). High-temperature heat exchanger tube-sheet assembly investigation with computational fluid dynamics. *Journal of Pressure Vessel Technology*. <https://doi.org/10.1115/1.2716436>
- Roy, A., & Das, D. (2011). CFD analysis of a shell and finned tube heat exchanger for waste heat recovery applications.
- Singh, S., Sørensen, K., & Condra, T. J. (2018). Parametric CFD analysis to study the influence of fin geometry on the performance of a fin and tube heat exchanger. In *Proceedings of the Conference*. <https://doi.org/10.1109/eurosim.2016.126>
- Sufian, M., Afzal, M. J., & Javaid, F. (2023). Study the heat exchanger through ANSYS Fluent. In *Proceedings of IEICES*. <https://doi.org/10.5109/7157976>
- Yu, C., Qin, S., & Chai, B. (2019). The effect of compressible flow on heat transfer performance of heat exchanger by computational fluid dynamics (CFD) simulation. *Entropy*, 21(9), 829. <https://doi.org/10.3390/e21090829>
- Yu, E.-L. (2007). Numerical simulation of a three-dimensional velocity field coupled with a temperature field for the heat exchange process in a spirally grooved tube. *Journal of Engineering for Thermal Energy and Power*.