Study the heat exchanger through ANSYS Fluent

Sufian, Muhammad
Department of Physics, Government Islamia College Civil Lines

Muhammad Javaid Afzal
Department of Physics, Government Islamia College Civil Lines

Javaid, Farah
Department of Physics, Government APWA College (W) Lahore

Tayyaba, Shahzadi
Department of Computer Engineering, The University of Lahore

他

https://doi.org/10.5109/7157976

出版情報：Proceedings of International Exchange and Innovation Conference on Engineering & Sciences (IEICES). 9, pp.226-231, 2023-10-19. 九州大学大学院総合理工学府
バージョン：

Study the heat exchanger through ANSYS Fluent

Muhammad Sufian[1], Muhammad Javaid Afzal[1,*], Farah Javaid[2], Shahzadi Tayyaba[3], Muhammad Waseem Ashraf[4],

[1]Department of Physics, Government Islamia College Civil Lines, Lahore 54000, Pakistan
[2]Department of Physics, Government APWA College (W) Lahore, Pakistan
[3] Department of Computer Engineering, The University of Lahore, Lahore 54000, Pakistan
[4] Department of Physics (Electronics), GC University, Lahore 54000, Pakistan
Correspondance : *javaidphy@gmail.com,

Abstract: *A heat exchanger is a tool used to move heat from one fluid to another without letting the fluids mix. The fluids can be liquids or gases, flowing in separate channels or passages separated by a solid wall. In order to get the cold fluid to the desired temperature, the heat exchanger converts thermal energy from the hot fluid to the cold fluid. They are commonly used in a number of businesses and industries to recover waste heat, cool or heat process fluids, and maintain or adjust temperatures. ANSYS Fluent software was used to model and simulate the heat exchanger in this investigation. The simulation results of the heat exchanger of velocity at the inlet and temperature at the outlet are 3.43 m/s and 3.4×10^2 K, respectively. The study aimed to understand better the heat exchanger's fluid flow and heat transfer properties to design it for better performance. A wide range of working situations was simulated, including various inlet velocities, outlet pressures, and temperatures. Experimental data were used to validate the simulation model, and the findings revealed good agreement between the two. For better findings, copper material was employed in the simulation in this study rather than aluminium in the work of other researchers. Copper is a better heat conductor than aluminium because it has a higher thermal conductivity. Because of this, it is a strong option for applications where heat transfer must happen swiftly and effectively. Copper is a superior material for applications that will be exposed to aggressive environments since it is more corrosion-resistant than aluminium. The simulation results were examined to determine how different factors affected the heat exchanger's output, including its heat transmission and fluid flow capabilities. The simulation was then used to improve the heat exchanger's design for improved performance.*

Keywords: ANSYS Fluent, Copper Material, Fluids, Gases, Heat Exchanger, Industrial Areas, pipes

1. INTRODUCTION

A heat exchanger device moves heat between two fluids of various temperatures [1]. The fluids, which can be either gases or liquids, are usually kept apart by a solid wall to prevent mixing. Without allowing the two fluids to mix or come into contact with one another, the heat exchanger helps the thermal energy move from the hotter fluid to the cooler fluid. Applications for heat exchangers include industrial operations, power generation, refrigeration and air conditioning systems, heating and cooling systems for buildings, and power generation. Depending on the particular application and the needed heat transfer properties, they are available in various sizes, forms, and patterns. There are two distinct fluid streams, the hot fluid and the cool fluid, in a conventional heat exchanger. While a different fluid travels through the other heat exchanger component, one fluid flows through its side as the fluids move through it. A solid wall or heat transfer surface separates the fluids and transmits heat from the hot fluid to the cold one [2]. Conduction, convection, and radiation are the three basic ways heat transfers. Convection is the movement of the fluid itself, whereas conduction is heat transmission via the heat exchanger's solid wall. When heat is exchanged between fluids by electromagnetic radiation, radiation takes place.

This study's goal is to investigate the significance of heat exchangers in the process of transferring thermal energy across fluids. There are some specific questions that this study is looking forward to investigating. These questions are given as follows.

How well is ANSYS Fluent able to simulate the fluid flow and heat transfer characteristics of the selected heat exchanger configuration? What impact do different coolant flow rates have on the spiral-wound heat exchanger that Deeb (2023) examined in terms of energy transfer rate and thermal performance? The heat and fluid performance of the plate heat exchanger with rectangular channels investigated by Mehdi et al. (2023) is impacted by the plate shape and flow rate. Can ANSYS simulations verify the thermal-hydraulic performance of the additively made heat exchangers mentioned by Inderjot Kaur and Prashant Singh (2021), especially in terms of their rough surfaces and microchannels?

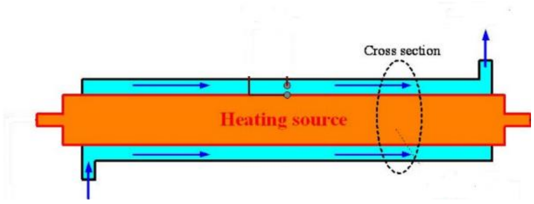

Fig. 1. Basic diagram of Heat Exchanger.
https://doi.org/10.3390/app11178139

The solid wall and, later, the cold fluid on the other side of the heat exchanger both get thermal energy as the hot fluid moves through it. The variance in temperature between the two liquids, the heat exchanger's heat transfer factor, and the measurements of the surface used for heat transfer all impact how quickly heat is transferred.

The efficiency and effectiveness of the heat exchanger are influenced by the materials used in construction, fluid flow rates, and heat exchanger design [3]. Depending on the particular application, many heat exchanger types, such as shell-and-tube, plate-and-frame, and spiral, each have their own distinct qualities and benefits. Fig. 1 presents the heat exchanger's fundamental diagram.

When the exchange of thermal energy between two fluids is necessary, a wide variety of applications call for the usage of heat exchangers. The following are some typical uses for heat exchangers:

Heat exchangers are used to move heat from indoor to outside air streams in heating, ventilation, and air conditioning systems. Preheating or precooling the incoming air increases energy efficiency and lowers energy costs [5]. To move heat from the refrigerant to the surrounding air, refrigeration and air conditioning systems need heat exchangers. This makes it possible for the refrigerant to absorb or release heat as needed, which is crucial for dehumidifying and cooling indoor spaces [6]. A wide range of industrial operations, including the production of chemicals, the refining of petroleum, and the production of electricity, all use heat exchangers. The internal structure of the heat exchanger is given in Fig. 2.

Deeb (2023) looked into the work, which presents a numerical analysis of a rectangular-cross-section spiral-wound heat exchanger. A high-temperature gas stream

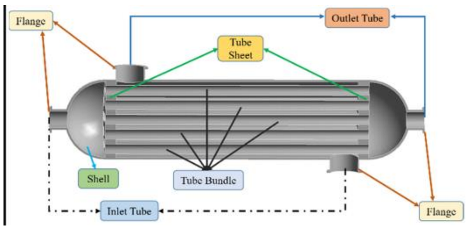

Fig. 2. Internal Structure of Heat Exchanger.
https://doi.org/10.3390/pr10051003

can be cooled by transferring the heat to a coolant fluid running through the heat exchanger. Computational fluid dynamics (CFD) simulations were used in the study to examine the heat exchanger's circulation and transfer of heat properties under various working circumstances. The outcomes demonstrated that the coolant flow rate and gas velocity impacted the energy transfer rate. The study also examined and contrasted the heat exchanger's pressure drop and thermal performance with that of other kinds of heat exchangers. A spiral wound with a rectangular cross-section for its heat exchanger is used to cool a high-temperature gas stream by transferring heat to a coolant fluid flowing through the exchanger. A numerical analysis of this heat exchanger is provided in the paper. The study examined heat transfer and flow in the heat exchanger properties under various operating situations using CFD models, and it also evaluated the pressure drop and thermal performance [8]. Mehdi et al. (2023) researched the design and performance evaluation of a plate heat exchanger with rectangular channels included in the study. The efficient and speedy heat transmission between two fluids is made possible by using heat exchangers. Numerical simulations were done in the study to evaluate the heat and fluid performance of the heat exchanger at various flow rates and structural designs. The results showed that the flow rate significantly affected the plate geometry and rate of heat transfer and that the heat exchanger had high thermal efficiency and minimum pressure loss. The study of a plate heat exchanger with rectangular channels used to efficiently and compactly transfer heat between two fluids concludes the essay. The results of the study's numerical simulations, which assessed the heat exchanger's thermal and hydraulic performance at various flow rates and geometrical configurations, showed that it had a high thermal efficiency and a low-pressure drop [9].

Inderjot Kaur and Prashant Singh (2021) investigated the production of heat exchangers has been revolutionized by advances in additive manufacturing (AM), which have significantly changed conventional approaches. Complex and unorthodox designs that were previously impossible to produce using traditional manufacturing methods are now possible because of advances in AM technology. Additionally, compared to traditional methods, AM has benefits including less weight, volume, lower manufacturing costs, and higher load-bearing capabilities. The advantages of using additive manufacturing to create heat exchangers come with certain inherent difficulties related to things like process parameters, surface quality, and material choice. Understanding the quality of the resultant HX surfaces thoroughly is essential for clarifying the ensuing flow and thermal characteristics. The primary goal of this experiment is to assess the thermal-hydraulic effectiveness of additively made HXs, taking into account aspects such as rough surfaces, microchannels, and surface area. According to a thorough review of the literature, intrinsic surface roughness emerges as the key issue to take into account regardless of the exact form of the heat exchanger produced using metal additive manufacturing. Particularly when the dimensions get closer to the production capabilities' limits, significant dimension variations from the planned design are detected. However, there is a bright future for heat exchanger development as AM technologies continue to improve final product surface quality, dimensional precision, and the capacity to realize lower sizes accurately. This review article offers insights into the changing AM landscape and its consequences for heat exchanger progress, making it an essential point of reference in this respect.

Dong Ho Nguyen and Ho Seon Ahn (2021) investigated that in the field of energy conversion, improving the thermal-hydraulic efficiency of heat exchangers is of utmost importance. A variety of strategies that may be divided into active and passive techniques can be used to accomplish this purpose. The development of nanotechnology has brought about notable advancements in surface modification, which holds great promise for improving heat transmission in heat exchangers. Empirical studies show that using micro/nanostructured surfaces improves heat transfer results by creating turbulence and fluid mixing in single-phase heat exchangers. The changing of the surface has transformational implications in situations involving phase shift heat exchange. It promotes improved interaction between nucleation sites and speeds up the release of vaporized bubbles, enabling smoother liquid transport during boiling operations. Additionally, by switching from thin film condensation to either dropwise condensation (DWC) or leaping droplet condensation, tailored surfaces in condensers help to better heat transmission. This paper's goal is to provide a comprehensive summary of the most recent advances in using micro/nanoscale surface modification methods to enhance heat transmission in heat exchangers. The report also explores potential directions for future research, including both single-phase and phase-shift heat exchange situations, as well as the processes underlying the improvement of heat transfer.

Bohong Wang et al. (2021) investigated that one method for reducing energy use and increasing efficiency is heat integration using a heat exchanger network (HEN). The layout of a retrofitting HEN can be influenced by the choice of various heat exchanger types and materials, which can have a substantial impact on investment costs. The features of various heat exchanger types, their operating settings, related investment costs, and the most advanced techniques for synthesizing and retrofitting HENs are all thoroughly evaluated in this paper. The objective is to provide a well-organized framework for HEN retrofit that takes into account heat exchanger and material factors. The suggested structure divides the retrofit design process into two different phases: optimization and diagnostic. Two graphical aids for decision-making are presented during the diagnostic phase. These instruments are used to evaluate and diagnose the current HEN and design workable retrofit options using preselected heat exchangers and components. The designs created in the earlier stage are then improved upon during the optimization phase using a limited particle swarm optimization algorithm in an effort to reduce the overall yearly cost. A case study that demonstrates the effectiveness of the suggested technique is provided. This application involves choosing the best heat exchanger types and materials for new installations, leading to a retrofitted design that successfully lowers utility costs by 8.9% in comparison to the pre-existing HEN arrangement. The given methodology provides a simple yet reliable answer for HEN retrofitting, allowing for its useful application in real-world applications.

Vladimír et al. (2023) investigated "Tube Design on the rod-Fin Heat Exchanger's Heat Exchange and Pressure Fall". They discussed the numerical investigation of the impacts of flow configuration and the layout and evaluation of plate-fin heat exchangers. An illustration of a compact heat exchanger is a rod-fin unit. It consists of alternate layers of flat plates and fins piled up to create a network of parallel channels through which fluids can flow. The fins enhance the area for heat transmission and promote turbulence in the fluid flow, while the plates act as a barrier between the channels. Due to its great heat transfer efficiency and small size, the plate-fin heat exchanger is frequently used in refrigeration systems, aeroplanes, and medical devices. In this study, numerical investigations are done to determine the effects of flow configuration and tube focus on the plate-fin heat exchanger's loss of pressure and heat transfer characteristics. The scientists used computational fluid dynamics (CFD) simulations to model the heat exchanger's fluid flow and transfer behaviour. The results were then analyzed to see how different flow configurations and tube designs affected the effectiveness of pressure drop and heat transfer [10].

Harsh et al. (2023) investigated "Experimental investigation of a novel A unique heat exchanger's construction and operation for high-temperature gas-cooled reactors covered in "Heat exchanger design for high-temperature reactors." Nuclear power plants known as graphite are used in moderation, and the gas known as helium is used as a coolant in high-temperature gas-cooled reactors (HTGRs). A heat exchanger transfers the reactor’s heat to a working fluid, such as helium or supercritical carbon dioxide. Because it carries heat from the reactor to the system that turns it into electricity, the heat exchanger is an essential component of the reactor system. An experimental assessment of a novel heat exchanger design for HTGRs is done in the paper. The bundle of helically coiled tubes that make up the heat exchanger are placed in a small, modular configuration. Studies to ascertain the heat exchanger's functioning were assessed in relation to that of a typical shell-and-tube heat exchanger based on the characteristics of heat transfer and pressure fall. The results showed that the novel heat exchanger design achieved increased heat transfer rates and decreased pressure decreases than the conventional design, indicating that it could enhance HTGR systems' effectiveness and performance [11].

Miss. Manasi et al. (2023) investigated A shell and tube heat exchanger's design and performance are covered in "Performance Analysis of a Shell and Tube Heat Exchanger for Steam Condensation". A type of heat exchanger called a shell and tube heat exchanger includes a collection of tubes inside a cylindrical shell. The fluids that need to be heated or cooled through the tubes and the shell, respectively. When steam condenses, hot steam is forced through the tubes while cooling water or another fluid passes through the shell, chilling the steam and causing it to condense. The performance evaluation of a particular shell and tube heat exchanger for steam condensation is discussed in the article. The scientists performed studies to evaluate the heat exchanger's efficiency under various operating settings, including steam flow rate and cooling water flow rate, as well as pressure drop and overall performance. The outcomes demonstrated that the heat exchanger performed well overall, and had a great heat transfer coefficient and a little pressure drop. It transferred heat quickly as well. The authors also provided recommendations for optimizing the design and operation of the heat exchanger to improve its performance further [12]. Brough et al. (2020) investigated the outcomes of Experiment 1 when the temperature of the emissions was kept at about 4.08×10^1 K [13]. B. Xu et al. (2020) investigated temperature variations between the experiments were 2.2 K, 2.5 K, and 3.5 K. [14]. I. Korobiichuk et al. (2022) investigated that in the channel formed by plates, the maximum speed w = 1.127 m/s [15].

2. ANSYS SIMULATION

ANSYS is a powerful tool for real-time simulation. Numerous researchers have used ANSYS software for simulation before production [16-30]. This work evaluates the velocity and temperature using the heat exchanger's ANSYS fluid flow (fluent) simulation. The simulation's parameter selections are crucial for assuring a realistic representation of real-world circumstances and relevant outcomes.

The size of a heat exchanger body can grow up to 1.2 m. The defeature size of the exchanger is 1.336 10^{-3} m. As seen in Fig. 3, the ANSYS Workbench design modeller generates a curve with a minimum radius of 3.0456 10^{-3} m.

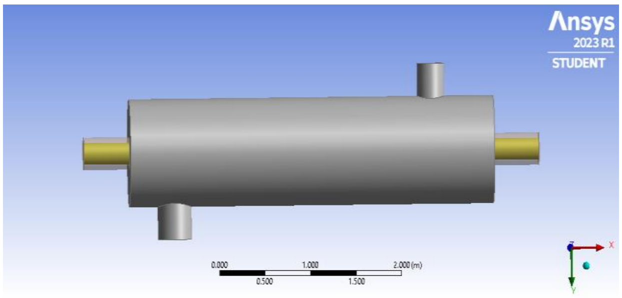

Fig. 3. Geometry of heat exchanger.

The heat exchanger's water liquid moves at a 3 m/s beginning speed. The material qualities for real-time simulation are selected by the ANSYS engineering data source. The heat exchanger has a 1 m diameter. The heat exchanger's length, the viscosity of 0.001003 N-S/m^2, and the outlet pressure of 0 Pa are just a few of the details included in the heat exchanger material composition. Water liquid with an initial velocity of 2.5 m/s is the simulation material. The heat exchanger has a density of 998.2 kg/m^3 and a volume of 3.2672 m^3. Additionally, the heat exchanger has 1.657 m^2 of surface area and a thickness of 1 m. The heat exchanger, matching geometry and element size, has been perfect for precise modelling. The mesh setup includes the coarse relevance centre, medium smoothing, and coarse span angle centre. The mesh geometry consists of 41986 nodes and a total of 155638 elements. Fig. 4's basic heat exchanger diagram showing meshing.

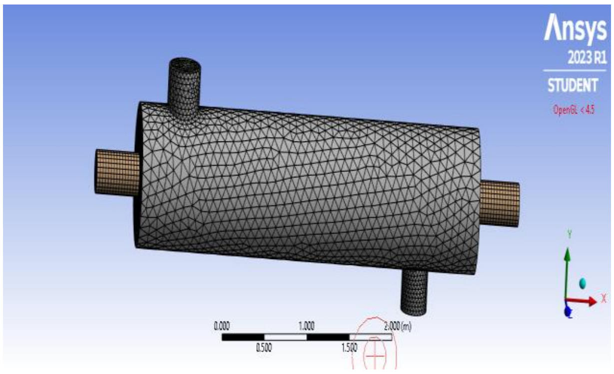

Fig. 4. Mesh Analysis of the heat exchanger.

The heat exchanger is utilized in industrial settings for various purposes as it transfers heat from hot fluid to cool fluid. In the simulation, the entrance velocity can range from 3 m/s to 2.5 m/s to 2 m/s and many more values. Hot and cold water temperatures are 2.8 × 10^1 K and 3.5 × 10^2 K, respectively, when a fluid is moved from hot to cold. The results of the simulation are shown using the velocity and temperature values.

3. RESULTS AND DISCUSSIONS

The heat exchanger is built with a water inle, outlet, and tubes for the purpose of heat exchanger simulation. The heat exchanger simulation uses three input variables: thickness, length, and heat exchanger. A simulation of a heat exchanger uses the water's convection, conduction, and radiation to move heat from hot to cold. The heat exchanger results have velocity- and temperature-like characteristics, a crisp value of 0 Pa, a length of 1 m, and a thickness of 0.3 m. ANSYS fluid fluency developed the heat exchanger's geometry. Fig. 5 shows the velocity contour mapping computation.

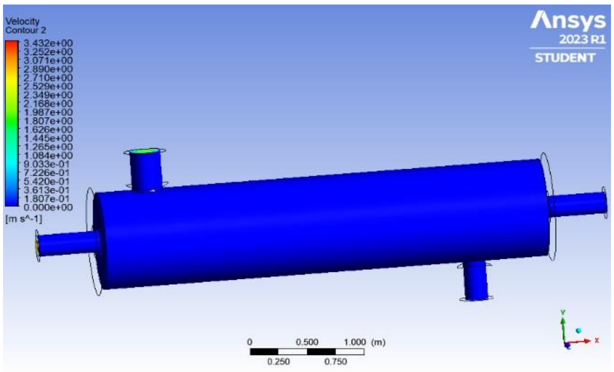

Fig. 5. Velocity contour.

The heat exchanger's beginning velocity is 3 m/s when the hot water and cold water are both 15 °C. To confirm simulation results, the heat exchanger, and the ANSYS fluid flow programme are also used to compute results. We determined the heat exchanger's final values for temperature and velocity and compared those results to our earlier discoveries. Through the use of ANSYS, the heat exchanger's temperature and velocity in this investigation were determined to be 3.5 102 K and 3.43 m/s, respectively. Through the intake pipe, heat is quickly transported from the heat exchanger's hot fluid to the cold fluid. There are various uses for this hot-to-cold fluid transformation, but industrial settings benefit the most. The speed of the water and its temperature are inversely related to one another. In other words, the temperature of the fluid will fall as the stream's velocity increases. Results from the simulation and the traditional formulation quite closely agree. The previous researchers, D. Brough et al. (2020), B. Xu et al. (2020), and I. Korobiichuk et al. (2022), used ANSYS simulation software to find the results for temperature and velocity, which were 4.08 101 K, 2.2 K, 2.5 K, and 3.5 K, and speed V = 1.127 m/s, respectively. In this work, the results for temperature and velocity are 3.525 102 and 3.43 m/s, respectively. This essay perfectly shows how well the findings of this inquiry and earlier studies agreed

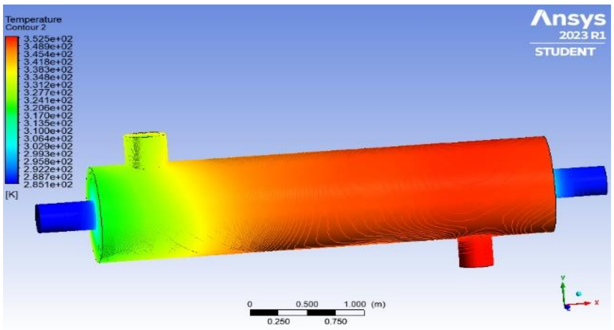

Fig. 6. Temperature contour.

upon. Fig. 6 displays the outcomes of the temperature contour mapping.

In a heat exchanger, temperature and velocity have an inverse relationship to one another. From the input to the output, the fluid's velocity progressively increases while the temperature decreases. Therefore, we draw the conclusion that temperature and velocity are inversely related. Figure 7 shows the graph between temperature and velocity.

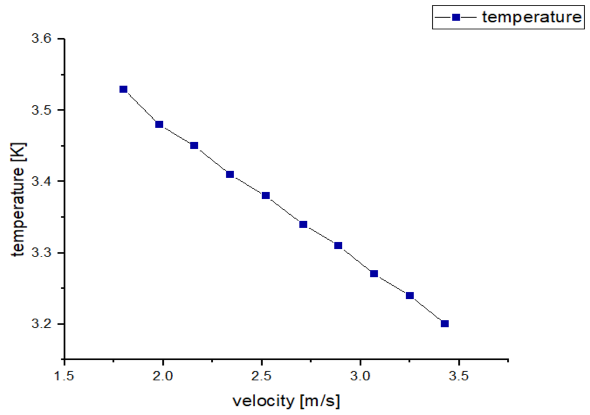

Fig. 7. Graph between temperature and

In the circular graph, the vertical numbers and percentage values represent, respectively, temperature and velocity. The vertical values' black color indicates a maximum velocity of 3.43 m/s at the heat exchanger's inlet side. Vertical numbers in orange indicate a minimum velocity of 1.8 m/s at the heat exchanger's outflow side. Thus, we have concluded that the temperature rises from black to orange. The circular graph's black colour shows the heat exchanger's inlet side's lowest temperature of 9.5%. The heat exchanger's outlet side displays a maximum temperature value of 10.5% in orange. In a circular graph, we have looked at how the fluid's velocity in the heat exchanger steadily increases from black to orange. As a result, the heat exchanger's fluid's temperature and velocity are inversely related to one another. Fig. 8 depicts the temperature and velocity results.

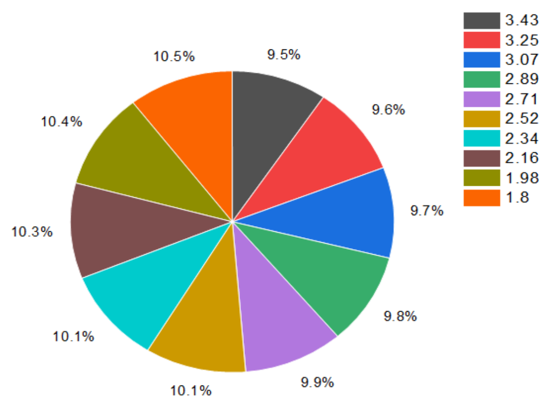

Fig. 8. Circular graph plotted between temperature and velocity.

4. CONCLUSION

ANSYS Fluent software was used in this study to model and simulate a heat exchanger. The simulation results of the heat exchanger of velocity at the inlet and temperature at the outlet are 3.43 m/s and 3.52×10^2 K, respectively. The simulation results and experimental data showed good agreement, showing the model's accuracy and dependability. Investigations into the impacts of various parameters on heat exchanger performance found that the inlet temperature and velocity greatly impacted the heat exchanger's flow and heat transfer characteristics. Information obtained from the simulation of the heat exchanger's flow and heat transfer mechanisms can be used to enhance the device's performance and optimize its design. While other researchers employed aluminium material in their studies, this study uses copper material to achieve superior outcomes. Copper is a better heat conductor than aluminium because it has a higher thermal conductivity. Because of this, it is a strong option for applications where heat transfer must happen swiftly and effectively. Copper is a superior material for applications that will be exposed to hostile environments since it is more corrosion-resistant than aluminium. The findings of this study show how useful the ANSYS Fluent software is for modelling and simulating heat exchangers, as well as how it might help create more effective and environmentally friendly energy systems.

5. REFERENCES

[1] J. Berce, M. Zupančič, M. Može, and I. J. A. T. E. Golobič, "Infrared thermography observations of crystallization fouling in a plate heat exchanger," vol. 224, p. 120116, 2023.

[2] R. Guráš, M. Mahdal, and M. J. A. T. E. Bojko, "Heat transfer analysis of a liquid cooling device using thermoelectric elements," vol. 226, p. 120274, 2023.

[3] A. Hassan et al., "An experimental and numerical study on the impact of various parameters in improving the heat transfer performance characteristics of a water-based photovoltaic thermal system," vol. 202, pp. 499-512, 2023.

[4] S.-P. Yu, Y.-F. Lue, T.-P. Teng, H.-K. Hsieh, and C.-C. J. A. S. Huang, "Enhanced heat transfer performance of the tube heat exchangers using carbon-based nanofluids," vol. 11, no. 17, p. 8139, 2021.

[5] M. Ghaderi, S. M. Hosseini, and M. J. A. T. E. Sorin, "Assessment of waste heat recovery potential in greenhouse ventilation systems using dynamic pinch analysis," vol. 227, p. 120363, 2023.

[6] S. Abdullah et al., "Technological Development of Evaporative Cooling Systems and its Integration with Air Dehumidification Processes: A review," p. 112805, 2023.

[7] Y. Zeng et al., "Numerical Simulation Study on Flow Heat Transfer and Stress Distribution of Shell-and-Tube Superheater in Molten Salt Solar Thermal Power Station," vol. 10, no. 5, p. 1003, 2022.

[8] R. J. I. J. o. H. Deeb and M. Transfer, "New correlations for predicting convective heat transfer of single and multi-row heat exchangers employing staggered drop-shaped tubes," vol. 202, p. 123689, 2023.

[9] M. Hashemi-Tilehnoee, S. M. Seyyedi, E. P. del Barrio, and M. J. J. o. E. S. Sharifpur, "Heat transfer intensification of NEPCM-water suspension filled heat sink cavity with notches cooling tubes by applying the electric field," vol. 59, p. 106492, 2023.

[10] V. Zmrhal, P. Zelenský, J. Boháč, M. Barták, M. Kučera, and R. Vavřička, "Determination of the sensible heat effectiveness and pressure loss of a rotary regenerative heat exchanger using CFD," in Building Simulation, 2023, pp. 1-19: Springer.

[11] H. V. Malapur, S. N. Havaldar, and G. A. J. M. T. P. Anderson, "Heat gain in an internally dimpled tube heat exchanger-a numerical investigation," vol. 72, pp. 1530-1536, 2023.

[12] M. M. B. Gangurde, S. V. Bhaskar, and B. D. Popatrao, "Design and Analysis of Shell and Tube Type of Heat Exchanger."

[13] D. Brough et al., "An experimental study and computational validation of waste heat recovery from a lab scale ceramic kiln using a vertical multi-pass heat pipe heat exchanger," vol. 208, p. 118325, 2020.

[14] B. Xu, H. Zhang, Z. J. I. J. o. H. Chen, and M. Transfer, "Study on heat transfer performance of geothermal pile-foundation heat exchanger with 3-U pipe configuration," vol. 147, p. 119020, 2020.

[15] I. Korobiichuk, V. Mel'nick, V. Shybetskyi, S. Kostyk, and M. J. E. Kalinina, "Optimization of heat exchange plate geometry by modeling physical processes using CAD," vol. 15, no. 4, p. 1430, 2022.

[16] M. J. Afzal, M. W. Ashraf, S. Tayyaba, M. K. Hossain, and N. J. M. Afzulpurkar, "Sinusoidal microchannel with descending curves for varicose veins implantation," vol. 9, no. 2, p. 59, 2018.

[17] M. A. TAYYABA and A. AKHTAR, "Simulation of a Nanoneedle for Drug Delivery by Using MATLAB Fuzzy Logic."

[18] Y. Wu, Q. Yue, Y. Gao, Z. Ren, and B. J. J. o. e. s. Gao, "Performance of bimetallic nanoscale zero-valent iron particles for removal of oxytetracycline," vol. 69, pp. 173-182, 2018.

[19] B. E. Logan et al., "Microbial fuel cells: methodology and technology," vol. 40, no. 17, pp. 5181-5192, 2006.

[20] M. J. Afzal, F. Javaid, S. Tayyaba, M. W. Ashraf, and M. K. Hossain, "Study on the Induced Voltage in Piezoelectric Smart Material (PZT) Using ANSYS Electric & Fuzzy Logic," 2020.

[21] M. J. Afzal, F. Javaid, S. Tayyaba, M. W. Ashraf, C. Punyasai, and N. Afzulpurkar, "Study of charging the smartphone by human movements by using MATLAB fuzzy technique," in 2018 15th International Conference on Electrical Engineering/Electronics, Computer, Telecommunications and Information Technology (ECTI-CON), 2018, pp. 411-414: IEEE.

[22] M. J. J. B. v. Afzal, "Study of constricted blood vessels through ANSYS fluent," vol. 27, pp. 11-2020.

[23] M. Afzal, F. Javaid, S. Tayyaba, A. Sabah, and M. J. B. Ashraf, "Fluidic simulation for blood flow in five curved Spiral Microchannel," vol. 65, no. 1, 2019.

[24] M. J. Afzal, S. Tayyaba, M. W. Ashraf, M. K. Hossain, and N. Afzulpurkar, "Fluidic simulation and analysis of spiral, U-shape and curvilinear nanochannels for biomedical application," in 2017 IEEE International Conference on Manipulation, Manufacturing and Measurement on the Nanoscale (3M-NANO), 2017, pp. 190-194: IEEE.

[25] M. J. Afzal, S. Tayyaba, M. W. Ashraf, M. K. Hossain, M. J. Uddin, and N. J. M. Afzulpurkar, "Simulation, fabrication and analysis of silver-based ascending sinusoidal microchannel (ASMC) for the implant of varicose veins," vol. 8, no. 9, p. 278, 2017.

[26] M. J. Afzal, S. Tayyaba, M. W. Ashraf, and G. Sarwar, "Simulation of fuzzy based flow controller in ascending sinusoidal microchannels," in 2016 2nd International Conference on Robotics and Artificial Intelligence (ICRAI), 2016, pp. 141-146: IEEE.

[27] M. T. Khan, M. J. Afzal, F. Javaid, S. Tayyaba, M. W. Ashraf, and M. K. Hossain, "Study of tip deflection on a copper-steel bimetallic strip by fuzzy logic and ansys static structural," 2021.

[28] M. Sufian et al., "Catalytic Converter Simulation for Pressure and Velocity Measurement," 2022.

[29] Z. ul Qasmi et al., "ANSYS simulation of Temperature of Cooling System in Li-ion Battery," 2022.

[30] M. S. Arif et al., "Laminar Flow Analysis of NACA 4412 Airfoil Through ANSYS Fluent," 2022.