SSRG International Journal of Mechanical Engineering
ISSN: 2348-8360/ https://doi.org/10.14445/23488360/IJME-V11I7P111

Volume 11 Issue 7, 132-143, July 2024

Original Article

Optimizing Heat Transfer Efficiency: CFD Analysis of Microchannel Designs for Two-Wheeler Radiators

Santosh Laxman Pachpute[1], Kiran C. More[1]

[1]*Mechanical Engineering Department, School of Engineering & Technology, D. Y. Patil University, Maharashtra, India.*

[1]*Corresponding Author: santosh.phd2021@gmail.com*

Received: 18 May 2024 Revised: 20 June 2024 Accepted: 19 July 2024 Published: 30 July 2024

***Abstract* - *Heat exchangers are pivotal in engineering applications for transferring thermal energy between two fluids across a separation wall, especially within Microchannel Heat Exchangers (MCHXs). These devices, known for their compactness and lightweight design, offer exceptional heat transfer capabilities. Yet, a significant challenge arises in the misdistribution of two-phase fluids within these systems, particularly at the inlet header of evaporators, where this misdistribution affects the uniform flow into the microchannel tubes. Leveraging recent advancements in Computational Fluid Dynamics (CFD) for multiphase flow, this study embarks on a detailed exploration of the flow behavior inside MCHX headers, employing CFD modeling to scrutinize the heat transfer parameters of flat face versus curved face MCHX designs. Our methodology involves comprehensive CFD simulations to investigate the thermal dynamics within these exchangers, focusing on how design variations impact heat transfer efficiency. The analysis reveals a notable advantage of the curved face MCHXs: a higher local heat transfer coefficient. This enhancement is attributed to improved fluid mixing and an increase in surface interaction within the curved face design. Through detailed CFD analysis, this paper not only addresses the challenge of fluid misdistribution but also illuminates the potential of curved face MCHXs to enhance heat exchanger performance in practical applications significantly*.

Keywords* - *Microchannel heat exchanger, CFD, Heat transfer parameters, Two-wheeler radiator, MCHXs.

1. Introduction

The initial exploration into Microchannel Heat Sink (MCHS) technology focused on the dynamics of fully developed laminar flow within fin configurations, both with and without modifiable tip clearance [1]. This research underscores the pivotal role of microchannels in enhancing heat transfer efficiency. In these systems, a singular fluid medium commonly water or other liquids circulates through narrow tubes or encased channels while air navigates in a cross-current manner along adjacent fins [2]. The defining characteristic of these channels is their hydrodynamic diameter, which does not exceed 1 mm, optimizing the surface area for heat exchange [3].

The design ensures that fluid flow within the MCHX is evenly distributed across numerous minuscule passageways arranged in parallel to facilitate efficient heat dispersion [4, 5]. Heat transfer in such exchangers predominantly occurs via convection at the boundary layers adjacent to the walls, with the process taking an extended period to affect the fluid's core [6, 7]. The utilization of narrower tubes inherently diminishes the volume of fluid required, thereby enhancing the convective heat transfer efficiency due to a greater proportion of the fluid coming into direct contact with the channel walls.

This arrangement not only necessitates less fluid for the same heat transfer rate but also contributes to a reduction in flow velocity within each channel, optimizing thermal performance. Further efficiency gains are achieved through the strategic reduction of parallel channel lengths and the minimization of axial heat radiation, ensuring a more uniform flow distribution. These measures collectively contribute to a decrease in the liquid volume required by the pump, underscoring the system's operational efficiency [9].

Microchannel Heat Exchangers (MCHEs) have revolutionized thermal management systems by offering substantial efficiency and size benefits over traditional heat exchangers. According to industry reports, MCHEs require up to 30% less fluid and can achieve significant reductions in size and weight being up to 30% smaller and 60% lighter thereby enhancing overall system efficiency [12].

This compactness not only allows for the use of smaller fans, reducing energy consumption but also contributes to quieter operation, addressing noise pollution concerns [14]. The versatility in design afforded by MCHEs liberates engineers from traditional constraints, facilitating more innovative layouts without compromising on performance or efficiency [16].

The engineering advantage of MCHEs is further underscored by their smaller hydraulic radius, which facilitates a host of technical benefits, particularly in Heating, Ventilation, and Air Conditioning (HVAC) systems. The abundance of smaller conduits in MCHEs presents a larger surface area for fluid-to-wall contact compared to conventional tubes, substantially boosting heat transfer efficiency [17]. Some manufacturers report that this design can lead to a 20–40% increase in heat transfer efficiency over standard fin and tube exchangers [18].

Furthermore, the dynamics of two-phase flow within MCHEs have been extensively studied, revealing that flow patterns significantly vary with changes in mass flow rates and fluid quality [20]. At lower rates or quality, separated inflow patterns emerge, whereas churn flow dominates under conditions of higher mass transfer rates and quality.

Research has delved into the effects of different header designs, intake configurations, and insertion depths on flow behavior. Through a combination of analytical and numerical modeling, alongside experimental visualization, significant strides have been made in understanding the complexities of flow misdistribution and behavior in headers, paving the way for more refined and efficient thermal exchange systems [22].

1.1. CFD Applications in Heat Exchangers Design

A multitude of studies have proposed different methodologies aimed at enhancing thermal transfer capabilities, minimizing pressure losses, and reducing the dimensions of existing standard heat exchangers [24]. It has been found that in the laminar flow regime, the extent to which these augmentation techniques can increase heat transfer is largely unaffected by the conditions at the boundary walls and remains within the same magnitude range [25]. For situations requiring compact solutions, heat exchangers featuring tubes of various dimensions and configurations have been innovated, offering a high density of heat transfer area, approximately 700 square meters per cubic meter [26].

In the context of traditional air-to-liquid cross-flow heat exchangers, they are constructed using tubes of differing sizes and forms, with the airside resistance accounting for about 80% of the overall thermal resistance [27]. Some researchers suggest that the thermal resistance on the airside could be as high as 96%, underscoring the importance of selecting a suitable airside temperature correlation for the specific design and application of the heat exchanger [28]. These conventional heat exchangers are characterized by channel sizes of 6 mm and above, incorporating varied geometries, surface augmentations, and orientations to improve performance [29].

In microchannels, the heat transfer process is significantly influenced by the surface area designated for heat transfer, which linearly escalates with either the channel's Diameter (D) or its Hydraulic Diameter (Dh). On the contrary, the flow rate within the channel is defined by its cross-sectional area (Ac), which increases quadratically with D, following a D^2 relationship [30].

The heat transfer surface area to volume ratio (As/V) demonstrates an inverse relationship with the diameter, increasing as D decreases. Within the regime of laminar flow, the local heat transfer coefficient (h) inversely varies with the diameter, enhancing as the diameter diminishes ($h \alpha 1/D$). Consequently, a reduction in the channel diameter not only simplifies the unit but also elevates the heat transfer rate [31].

Widely used commercial CFD software, such as FLUENT, CFX, STAR CD, FIDAP, ADINA, CFD2000, and PHOENICS, plays a crucial role in the analysis and optimization of heat exchangers. These tools have been instrumental in examining critical aspects such as heat performance, pressure drop, fouling, and fluid misdistribution, leading to significant advancements in heat exchanger technology [33].

Bengt Sundén has significantly contributed to the field by highlighting the versatility of CFD in addressing various challenges within heat exchangers. He proposed potential models for CFD analysis and conducted simulations on Plate Heat Exchangers (PHE) with different corrugation patterns, showcasing the breadth of issues CFD can tackle. The subsequent sections delve into various topics related to heat exchangers, categorizing them by type for detailed discussion [34].

A primary factor contributing to the inefficiency of heat exchangers is the uneven flow of fluid, which can stem from several design flaws, such as poor distributor design, plate corrugations, header layout, and the configuration of inlet/outlet ports. To address this issue, a local Nusselt-Rayleigh correlation formula was developed, where x represents the distance from the top of the annulus:

$$Nu_x = 0.46 \text{ x } (Ra_x \text{ x } X/D_{\text{hydraulic}})^{0.28} \quad \text{for } Ra = Ra_x <= 10^{11} \tag{1}$$

This equation helps in understanding the heat transfer dynamics in specific conditions. Furthermore, the degradation of working fluids due to high temperatures or contact with the exchanger walls leads to the formation of byproducts that adhere to the walls, diminishing the heat transfer coefficient.

This fouling necessitates periodic, sometimes frequent, maintenance to ensure efficient operation. The issue of fouling, particularly relevant in industrial applications, has been extensively studied using CFD techniques, especially within the context of PHEs [35].

The passage of fluid through a heat exchanger results in a loss of pressure. This loss is significantly influenced by the design of the heat exchanger core or matrix and the components involved in fluid flow, including inlet/outlet ports, headers, nozzles, and ducts, among others [36]. Flow distribution devices are primarily responsible for this pressure loss, which in turn critically affects the heat transfer efficiency of the exchanger [37].

CFD has facilitated the study of this issue. Factors such as the ratio of water to air and the design of the tubes play roles in the reduction of pressure [38]. The coefficient of predicted pressure loss has been found to correlate directly with the mass flow rate ratio of water to AIR while showing an inverse relationship with the transverse pitch [39].

Additionally, it has been noted that upstream air interference can increase the pressure loss by 17%; conversely, bypassing this air leads to a reduction in pressure loss [35]. The degree of pressure loss per unit length can be ascertained by inputting the geometric parameters of the model into a CFD program [40]. Both experimental and simulation studies have confirmed that the inefficiencies related to pressure in any heat exchanger can be precisely analyzed using a straightforward model represented in a CFD program.

Investigating parameters such as the Nusselt Number, Prandtl Number, and Friction Factor, which influence the overall heat transfer coefficient of a heat exchanger, is a critical approach to evaluating the efficiency of an exchanger's performance [41]. Since its inception, CFD has shown its efficacy in researching thermal characteristics and optimizing the design of heat exchangers [42].

It has been used to research various changes, evaluate outcomes, and display the ideal variable combination to guarantee peak performance [43]. This research aims to analyze the impacts of several heat transfer characteristics that are evaluated, such as the distribution of pressure, temperature, and heat flux.

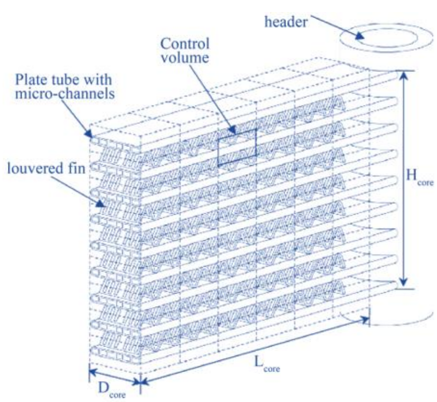


Fig. 1 Microchannel heat exchangers [11]

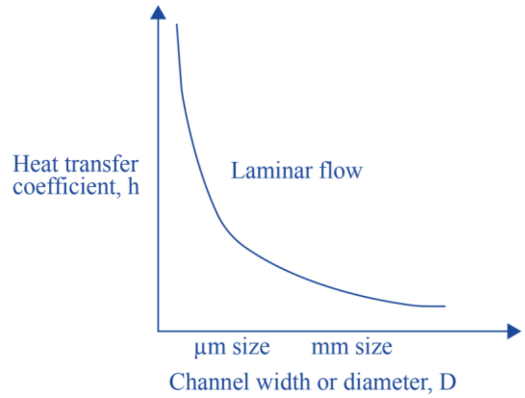


Fig. 2 Effect of channel size D on the heat transfer coefficient (h) in laminar flow regime [32]

2. Literature Review

The literature review initiates a comprehensive examination of the current understanding related to the heat transfer capabilities of microchannel heat exchangers with both flat and curved faces. By conducting a comparative analysis through computational simulations, researchers aim to unravel the intricate interplay between geometric configuration and convective heat transfer. This review endeavors to synthesize and analyze relevant studies, elucidating the underlying mechanisms that govern heat transfer in these two distinct microchannel designs.

In recent years, the demand for efficient heat transfer technologies has grown significantly across various industries, driven by the ever-increasing need for enhanced thermal management in modern engineering applications. Microchannel heat exchangers, characterized by their compact size and high surface area-to-volume ratios, have emerged as promising candidates to meet these demands [16]. The design and optimization of microchannel heat exchangers hold substantial potential for improving heat transfer performance in a range of applications.

The thermal transport capabilities of a hybrid nanofluid composed of water, Alumina oxide (Al_2O_3), and Aluminum nitride (AlN) within semiconductor devices, specifically across six circular microchannel heat sinks [44]. The simulation utilizes a volume fraction ranging from 1% to 4% to assess the influence on thermal resistance, pressure drop, Nusselt number, heat transfer coefficient, and Darcy friction factor. The analysis employs a single-phase, laminar, inert, and steady-state flow approach, numerically resolved through the finite volume method using Ansys Fluent, a leading CFD software.

This methodological approach underscores the potential of hybrid nanofluids in enhancing the cooling efficiency of microchannel heat sinks critical in the operation of chips with electronic functions. In a related analytical and comparative study by Mohanty et al. (2020), a CFD model of a heat exchanger with counterflow is meticulously evaluated [45]. This research delivers an in-depth thermodynamic assessment of the counterflow heat exchanger under various operational and geometrical conditions, offering a detailed simulation of the temperature gradient profile along its length. The refined design of the heat exchanger, incorporating an altered flow direction, demonstrates significantly improved performance, with computational errors ranging from 0.66% to 1.004% for temperature changes and 0.83% to 1.05% for mass flow rate adjustments, thereby highlighting the effectiveness of design optimizations in heat exchange systems.

To further contextualize and support this research, recent studies have demonstrated the importance of nanofluid application in thermal management systems. For example, a study explores the thermal conductivity enhancement of nanofluids containing copper oxide and water, indicating significant improvements in heat transfer efficiency suitable for cooling applications in electronic devices [46]. Additionally, a study by Liu et al. (2006) on the application of nanofluids in heat exchangers illustrates the broad potential of nanotechnology in achieving superior thermal performance across various industrial sectors [47]. These studies collectively affirm the critical role of advanced CFD analysis and nanotechnology in the development of efficient thermal management solutions for high-performance electronic systems.

Panda et al. (2020) undertook a detailed investigation into the performance of MCHX heads through a meticulously designed multiphase flow CFD simulation grounded in experimental validation [23]. The research highlights the application of the Hybrid Eulerian Multiphase (EMP) methodology, elaborating on the comprehensive CFD modeling process, which includes a rigorous sensitivity analysis of various parameters and sub-models. Experimental validation is achieved using thermal infrared imaging to ensure the reliability of the CFD model.

The study contrasts the performance of two distinct designs of vertical headers: the traditional Tube-Insertion header and its derivative, the Loop header, which incorporates an additional loop. Findings indicate that the Loop header offers superior performance in terms of fluid distribution. The sensitivity analysis extends to examining the effects of critical design variables such as header loop diameters, input mass flow rates, and tube insertion depths, aiming to optimize fluid distribution. The primary contribution of this paper is the provision of an in-depth simulation framework to address the challenge of header maldistribution in MCHXs, offering significant insights for enhancing thermal management solutions. In 2021, Meral et al. embarked on an examination of a crossflow microchannel heat exchanger using CFD simulations. Their study produced an aluminum crossflow microchannel heat exchanger with dimensions of 50x50x3 mm³, incorporating two plates in a crossflow setup. Each analyzed geometry featured square microchannels with dimensions of 490 μm in both width and depth. Experimental setups for heat and fluid flow were meticulously arranged, paralleled by simulations conducted in ANSYS Fluent V15 to model the heat transfer and fluid dynamics within these microchannels. The outcomes from the CFD simulations were then systematically compared with the experimental data [21]. This comparison highlighted a noteworthy congruence between the experimental findings on heat transfer and the simulation data.

In a concurrent study by David O. Ariyo et al. (2021), the focus was on the design and construction of two-phase stacked microchannel heat exchangers tailored for dissipating high heat fluxes from microelectronic devices. This investigation involved validating a CFD algorithm by comparing its predictions with experimental results from well-established sources, specifically evaluating its efficacy in modeling subcooled flow boiling essential for cooling microelectronics. The simulations and subsequent optimizations were conducted at heat fluxes of 1.1×10^7 W/m² and 1.2×10^7 W/m². A pivotal aspect of their research was examining the Critical Heat Flux (CHF) in an optimally designed two-stacked microchannel heat sink, functioning at a heat flux of 1.2×10^7 W/m² under both counterflow and parallel flow arrangements. Their findings demonstrated robust CHF performance for both configurations, with the counterflow setup showing superior effectiveness [19].

Investigating the Influence of Geometry on Micro-Channel Heat Exchangers, a study by J. Derek and colleagues (2019) delves into the significance of geometry on the thermo-hydraulic modeling of a Laser Additive Manufacturing (LAM)-built Micro-Channel Heat Exchanger (MCHE) featuring various cross-sections. Through the application of the CFD code ANSYS CFX, the research conducts a 3D steady-state CFD analysis to evaluate the channel's temperature profiles, pressure drop, velocity distribution along its length, and the variation of the friction factor with respect to the Reynolds number.

To authenticate the simulation's accuracy, the results for conventionally manufactured Plate-and-Frame Heat Exchangers (PCHE) are juxtaposed with prior findings, showcasing a consistency within a 5% margin. Furthermore, the study extends to the examination of MCHEs with square and circular cross-sections, in contrast to the traditional semi-circular geometries. This numerical investigation reveals that the square cross-section leads in terms of heat transfer efficiency and exhibits the lowest pressure drop, velocity variance across the channel, and friction factor [15]. A CFD and effectiveness-NTU-based co-simulation approach for analyzing flow mal-distribution in microchannel heat exchanger headers was investigated by Long Huang and colleagues (2014).

This method integrates a detailed effectiveness-based finite volume model for tube-side heat transfer and fluid flow with a comprehensive header simulation using CFD. The co-simulation concept was demonstrated through a case study involving ten tubes in an MCHX specifically tailored for an automotive R134a condenser. Experimental data supported the validation of the approach. After the experiments, the input header was dissected to assess the intricate header geometry accurately using the proposed CFD header model. The comparison of simulation predictions with experimental data showed favorable results. The introduced co-simulation method accurately predicts overall heat exchanger efficiency and provides detailed insights into the fluid flow dynamics [13].

Yanhui Han and colleagues (2012) explored the advancements in the design of micro-channel heat exchangers for air conditioning systems. With their increasing application in the cooling and refrigeration sectors, micro-channel heat exchangers have attracted global attention in research focusing on their theoretical aspects and practical challenges. Precise prediction of pressure loss and heat transfer characteristics is crucial before designing microchannels, especially as there is no standardized industry protocol for their production and an evolving theoretical base that could offer reliable design guidance [10].

Nonetheless, Mesbah G. Khan and colleagues (2011) provide a comprehensive survey of potential applications for microchannel heat exchangers. This paper not only sets the stage for future research by outlining necessary study parameters and pinpointing areas needing further inquiry, but it also evaluates the existing literature on the progress and potential of microchannel technologies. From the insights garnered through this review, an experimental air-to-liquid crossflow apparatus has been developed and operationalized. This setup is designated for examining fluid flow and heat transfer characteristics across different microchannel samples using varied working fluids [8].

2.1. Research Gap

The design and optimization of MHEs play a crucial role in achieving efficient thermal management, which is vital for enhancing overall system performance. While considerable research has been conducted on microchannel heat exchangers, a noticeable research gap exists in the specific area of comparative analysis between flat face and curved face microchannel heat exchangers using CFD simulations. A comprehensive comparative analysis would provide insights into the thermal performance differences between flat face and curved face microchannel heat exchangers under varying operational conditions. Such insights are crucial for selecting the appropriate design for specific applications, leading to improved heat exchanger efficiency. The comparative analysis would serve as a platform for validating CFD models against experimental data for both flat face and curved face microchannel heat exchangers. This validation is crucial for establishing the accuracy and reliability of simulation tools for future design and optimization studies by addressing this research gap by conducting a thorough comparative analysis and finding out the effective microchannel in a heat exchanger.

3. Methodology

3.1. Problem Statement

Efficient heat exchangers are essential in a variety of industrial applications, such as electronics cooling, automobile engines, and HVAC systems. Because of its small size and strong heat transfer capabilities, microchannel heat exchangers have received much attention. However, with little extensive research available, the option between flat face and curved face microchannel heat exchangers remains a vital design issue. The issue at hand is the necessity to use CFD simulations to understand better and compare the heat transfer performance of flat face and curved face microchannel heat exchangers. There is currently a scarcity of comprehensive research that thoroughly investigates and quantifies the variations in heat transfer properties between these two kinds of microchannel layouts. This information gap impedes the development of optimized heat exchanger designs and limits their potential uses in a variety of sectors. Analyzing the fluid flow pressure drop and heat transfer coefficients in both kinds of microchannel heat exchangers to learn more about the underlying processes that influence heat transfer. The use of CFD to compare flat face and curved face microchannel heat exchangers is critical for providing engineers and researchers with useful insights into the most effective design for certain applications. This understanding will help in the creation of more efficient and ecologically friendly heat exchangers, hence contributing to developments in thermal management technologies across a wide range of sectors. In this study, for model designing purposes, we used 3D software Unigraphics NX -8, and for analyzing purposes, we used Ansys Fluent.

3.2. Dimensions Used for Assembly

At the center position of the rectangular duct, a microchannel is fitted for experimental work. The microchannel of Aluminum material and Right-side header and Left side header in Microchannel heat exchanger are made by 3D printing technique of Acrylonitrile Butadiene Styrene (ABS) Materials. For flat face Microchannel heat exchanger overall dimension is 142 × 28 × 131 mm, and for the Curved face Microchannel heat exchanger overall dimension is 139.5 × 40.5 × 125.3 mm in section along with left and right headers.

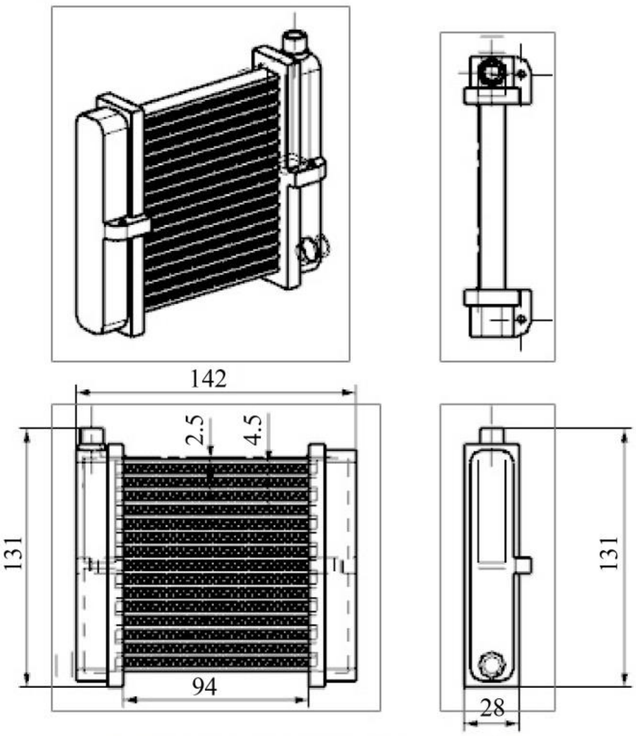


Fig. 3 Dimensions used for flat surface heat exchanger

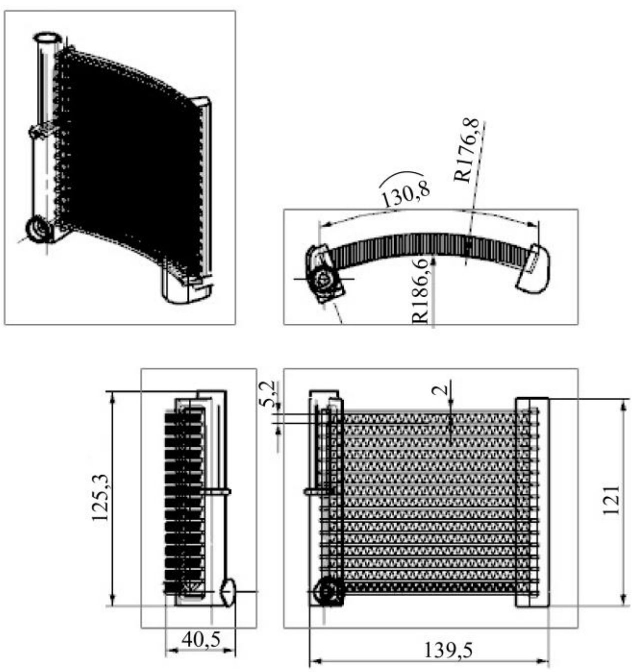


Fig. 4 Dimensions used for curved surface heat exchanger

When the requirements call for a larger heat exchanger or an increased heat transfer area, especially when specific features are needed, it may be necessary to design a heat exchanger tailored to these specific requirements. In these situations, one encounters the challenge of sizing the heat exchanger appropriately.

3.3. Modeling

Modeling a flat and curved surface heat exchanger using Siemens Unigraphics NX (also known as NX) involves several stages that begin with the creation of 3D geometry for the heat exchanger components and conclude with a fully comprehensive model suitable for simulation and analysis. Create a flat and curved surface that represents one side of the heat exchanger first. We did this using drawing and extrusion tools. Check that the size and form are correct. Create the channels or tubes through which the fluid will flow by using the appropriate directions.

This is determined by the design of the heat exchanger. We added fins to cylindrical or rectangular tubes using the modeling tools in Unigraphics NX-8. Check that they are properly positioned and have the correct proportions. On your heat exchanger, make openings or connectors for fluid inflow and exit. These should be parallel to the channels or tubes. Use the assembly capabilities in Unigraphics NX-8 to put together all the heat exchanger's components, including the flat surface, channels, fins, inlets, outlets, and support structures.

To make full use of Unigraphics NX-8's capabilities and features, it is critical to refer to its user manuals and tutorials during the modeling process. Ensure that the model follows technical standards and best practices for heat exchanger design and analysis. The ANSYS software and its capabilities for conducting thermal scenarios, calculating heat transfer laws, and modeling fluid movement in the microchannel. The geometry and specifications of the flat face and curved face microchannel heat exchangers used in the simulations are shown in the figure. Detail the assumptions and boundary conditions used in the simulations to ensure accurate and realistic results.

In the simulation setup the steps taken to set up the simulations in ANSYS, including the meshing strategy, selection of governing equations, and convergence criteria included. In this study, we compared the flat face heat exchanger model with the curved face heat exchanger model and found an effective model. Meshing is commonly employed in software-based simulations for Finite Element Analysis (FEA).

It plays a crucial role in determining the precision of the simulation as well as the computational resources needed to conduct it. Curved face microchannel heat exchanger divides geometry into a total number of Nodes: 427718 and Elements: 223800. Flat face microchannel heat exchanger divides geometry into a total number of Nodes: 263112 and Elements: 39034.

3.4. Numerical Calculations on Convective Heat Transfer

Convective heat transfer can be quantified using the average Nusselt number (Nu_H), which is based on the fin pitch (H) and defined by the formula:

$$Nu_H = h_H/k_{air} \qquad (2)$$

Here, h_H represents the average heat transfer coefficient and can be calculated as:

$$h_H = q_{num} / (A_0 \Delta T_{lm}) \qquad (3)$$

In this equation, $q$num stands for the total heat transferred from the fins to the airflow, $A0$ is the total area through which heat transfer occurs, and ΔTlm is the log-mean temperature difference. The log-mean temperature difference is calculated using the inlet and wall temperatures compared to the outlet temperature as follows:

$$\Delta T_{lm} = ((T_i - T_w) - (T_o - T_w))/\ln((T_i - T_w)/(T_o - T_w)) \qquad (4)$$

Where, T_o is the air temperature at the outlet.

The average pressure drop across the designated area can also be represented in terms of the Fanning friction factor (f), detailed as:

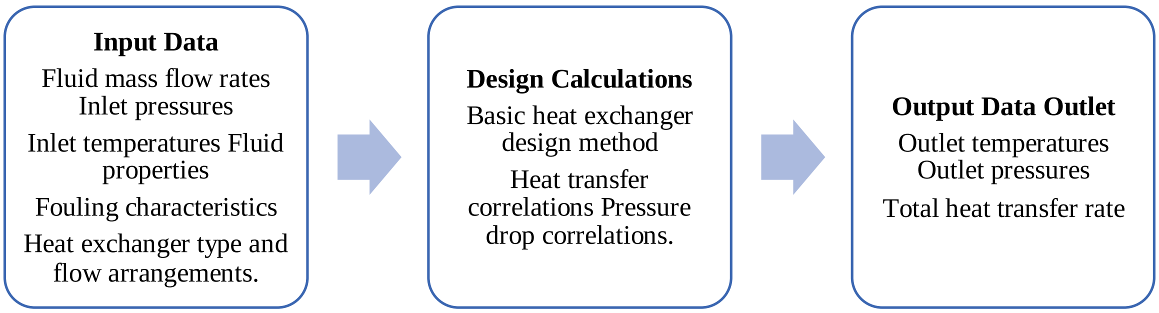


Fig. 5 Data input and output for thermo-hydraulic sizing of heat exchangers

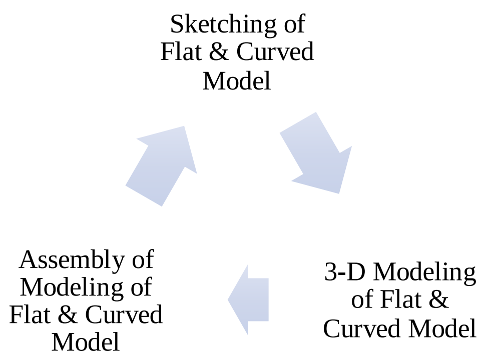
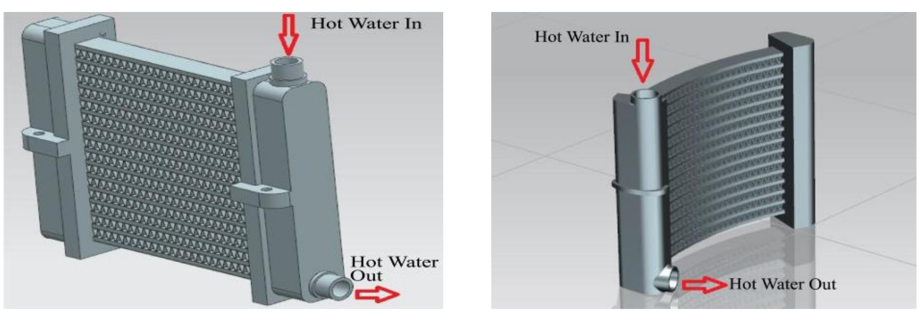


Fig. 6 Modeling process of a flat and curved surface heat exchanger using Siemens Unigraphics NX



Flat Surface Heat Exchanger **Curved Surface Heat Exchanger**

Fig. 7 Modeling of a flat and curved surface heat exchanger

$$F = \Delta P/(1/2pUin2).\ A_c/A_o \qquad (5)$$

Where, ΔP indicates the pressure drop and Ac is the minimum flow area. A useful parameter for comparing the efficiency of two fin shapes is the Performance Evaluation Criteria (PEC), which is described by:

$$PEC = ((Nu/Nu_{ref}))/(f/f_{ref})1/3 \qquad (6)$$

Where, Nu_{ref} and f_{ref} are obtained from the established data.

4. CFD Analysis and Results

4.1. Temperature

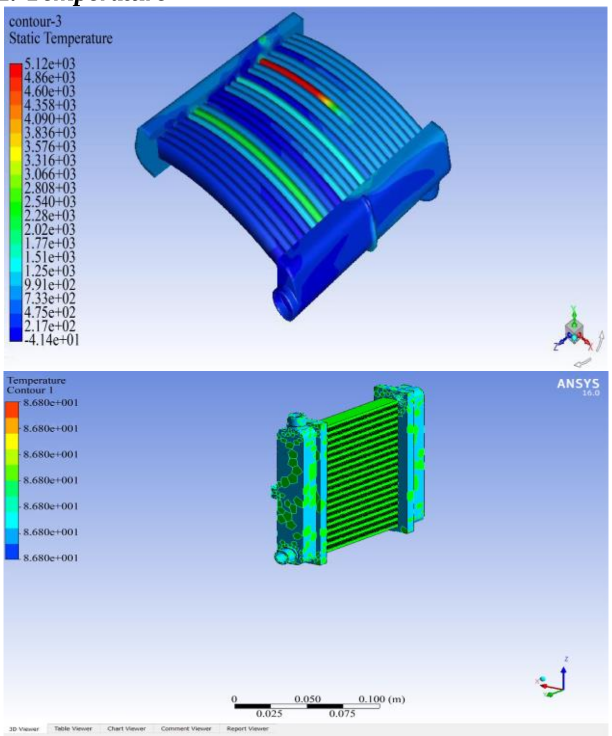


Fig. 8 The CFD of the flat model, the maximum temperature occurs 5120 K, and the curved model temperature is 86K

4.2. Density

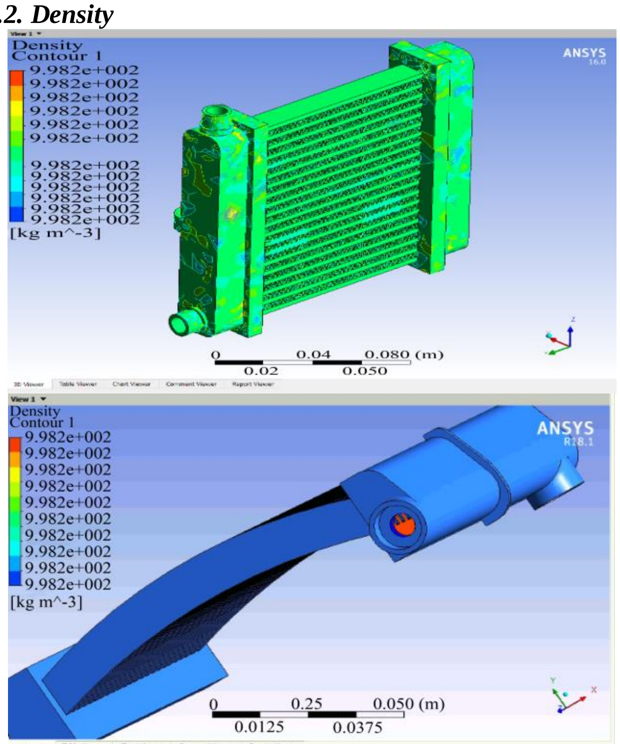


Fig. 9 The CFD of the maximum density that occurs in a flat model is 982 kg/m^3 and the curved model density is 998.2Kg/m^3

4.3. Pressure

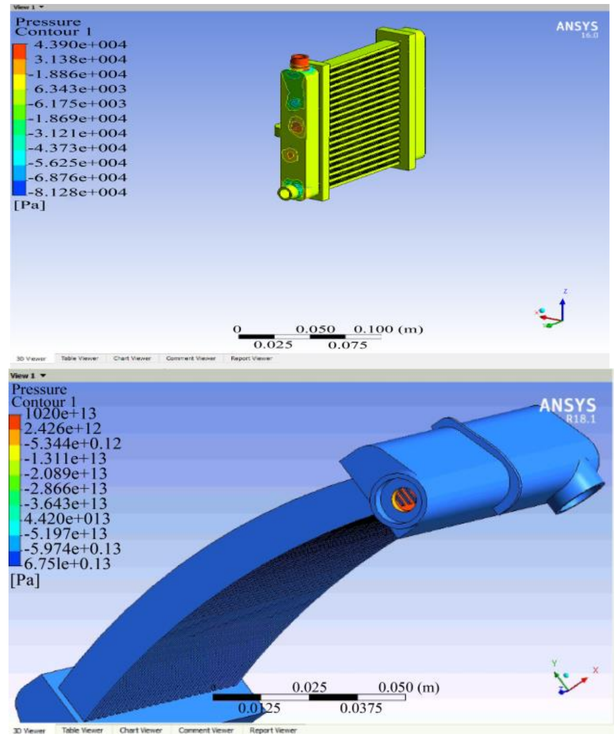


Fig. 10 The CFD of the flat model, the maximum pressure occurs at 43900 Pa and curved model pressure 1.020×10^{13}Pa

4.4. Wall Shear

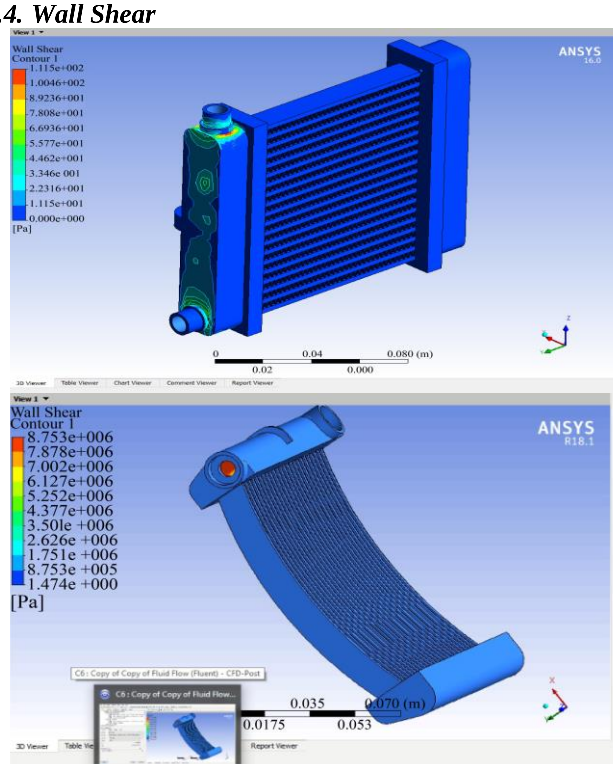


Fig. 11 The CFD of the curved model, the maximum wall shear occurs at 111.5 Pa, and the flat model wall shear is 8.753 ×10^6 Pa

5. Results and Discussion

5.1. Temperature

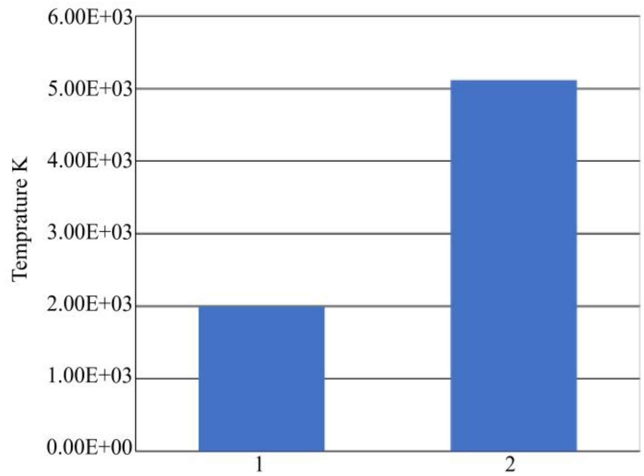


Fig. 12 Temperature profile

The above Figure 12 shows the Flat Model and the Curved Model. The latter shows temperature data. The Flat Model has a temperature of 2,000 units, but the Curved Model has a far higher temperature of 5,120 units, according to the numbers, which are stated in scientific notation. The temperature simulations or measurements shown in these figures show that there is a substantial variation in temperature between the two models, with the Curved Model being much warmer than the Flat Model.

5.2. Density

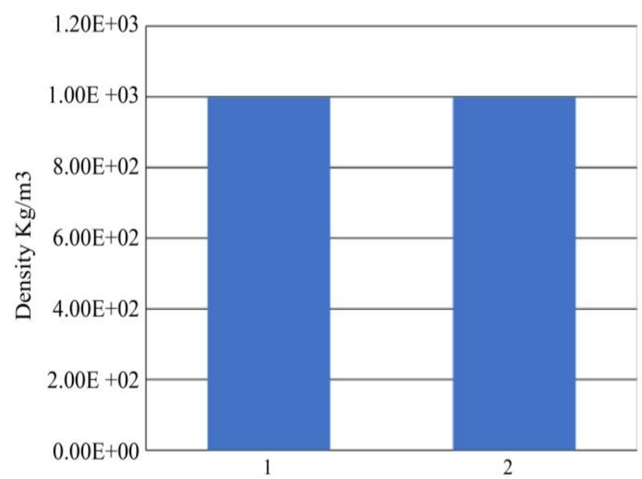


Fig. 13 Density profile

The above Figure 13 shows the Flat Model and the Curved Model. According to scientific notation, both models have the same density, which is around 998 units, or 9.98 $\times 10^2$. This implies that there is no appreciable variation in density between the two models; their density values are the same.

5.3. Pressure

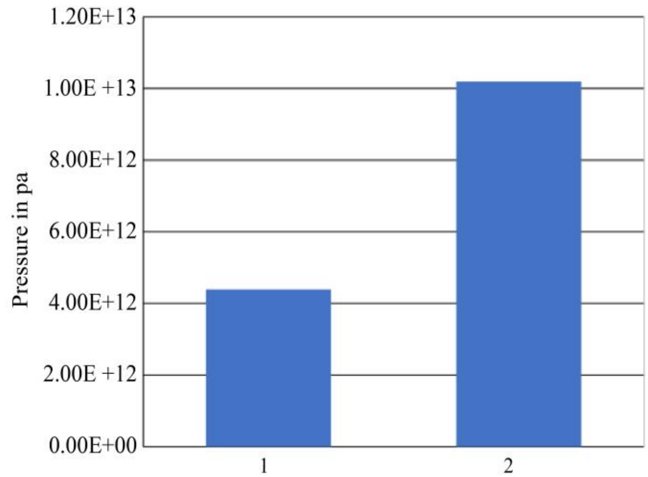


Fig. 14 Pressure profile

The above Figure 14 shows the Flat Model and the Curved Model. Scientific notation is used to express the data. The Flat Model shows a pressure of around 4.39 trillion Pascals (4.39×1012), but the Curved Model shows a far greater pressure of about 10.2 trillion Pascals (1.02×1013). The data suggests that there are variations in the internal pressure characteristics between the Flat Model and the Curved Model, with the former having a much greater pressure level than the latter.

5.4. Wall Shear

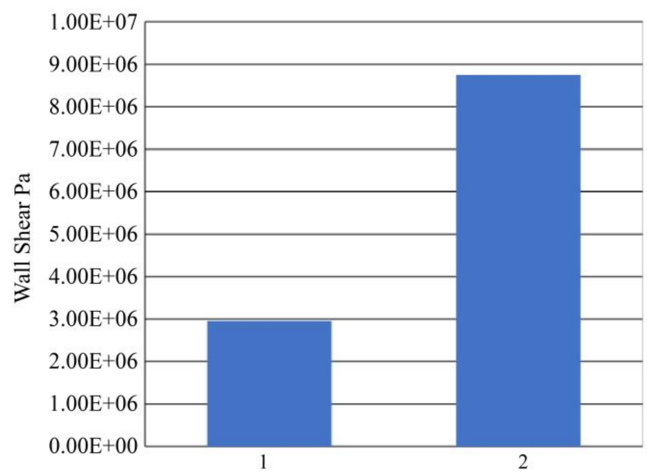


Fig. 15 Wall shear profile

Figure 15 shows the Curved Model and the Flat Model wall shear Pascals (Pa). The Curved Model has a far greater wall shear of around 8.75 million Pascals (8.75$\times 10^6$), in scientific notation, than the Flat Model, which has a wall shear of about 2.95 million Pascals (2.95E$\times 10^6$). According to this data, there seems to be a significant difference in wall shear between the Curved Model and the Flat Model, suggesting that different forces are operating on their different surfaces.

6. Conclusion and Future Scope

Car radiators and air conditioners are only two examples of the many industrial goods that use MCHXs. MCHXs have been put in several pieces of equipment with limited space because of their small construction and good heat transfer performance. For this reason, CFD is often used to analyze and improve MCHX performance since it takes less time and costs less to do research. Using CFD, the heat transfer performance of the flat surface and curved surface heat exchanger is investigated. Experimental findings corroborate the reference flat surface and curved surface results. The air in the curved surface model prefers to flow between louvre fins, increasing heat transfer and increasing air velocity. Consequently, boosting heat transfer may counteract the pressure decrease caused by the louvre fin domain's vortices.

The air conditioning system may benefit from this study's application to boost efficiency and lower operating costs. In comparison to the Flat Model, the Curved Model's wall shear is approximately 196.61% higher. Approximately 132.81% higher than in the Flat Model, the internal pressure in the Curved Model. Approximately 156% higher than the Flat Model's temperature is the Curved Model's. Compared to the heat exchanger design parameters found in earlier literature, the proposed updated design yields superior results. Higher pressure and velocity magnitudes are seen along both model contours, as well as temperature. We found that the curved surface heat exchanger is superior to the flat surface heat exchanger based on the analysis findings. Furthermore, the use of novel materials with superior thermal characteristics in the building of counterblow heat exchangers may result in noticeable and enhanced improvements in heat transfer. By considering the Nussle and Reynolds numbers, stressing models to improve the system's flow characteristics, and comparing the outcomes with those of other computational techniques, the current work may be further improved.

Nomenclature

MCHXs - *Microchannel Heat Exchangers*
CFD - *Computational Fluid Dynamics*
Nu - *Nusselt Number*
ΔTlm - *Log-Mean Temperature Difference*
q_{num} - *Total Heat Transfer from Fins to Airflow*
F - *Fanning Friction Factor*
ΔP - *Pressure Drop*
Ao - *Total Heat Transfer Area*
Ac - *Minimum Flow Area*
PEC - *Performance Evaluation Criteria*

References

[1] Anupam Dewan, and Pankaj Srivastava, "A Review of Heat Transfer Enhancement through Flow Disruption in a Microchannel," *Journal of Thermal Science*, vol. 24, pp. 203-214, 2015. [CrossRef] [Google Scholar] [Publisher Link]

[2] Heeseung Kang et al., "Numerical Investigation and Design Optimization of a Novel Polymer Heat Exchanger with Ogive Sinusoidal Wavy Tube," *International Journal of Heat Mass Transfer*, vol. 166, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[3] Gege Song et al., "Reviews: Applications of Optimization Algorithm for Microchannel and Microchannel Heat Sink on Heat Transfer," *International Journal of Heat and Fluid Flow*, vol. 108, 2024. [CrossRef] [Google Scholar] [Publisher Link]

[4] A. Ram, "Performance Enhancement of Mobile Air Conditioning System Using Variable Geometry Micro Channel Heat Exchanger (MCHX)," *International Journal of Engineering Sciences & Research Technology*, vol. 5, pp. 808-819, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[5] Zhi-Qiang Yu, Mo-Tong Li, and Bing-Yang Cao, "A Comprehensive Review on Microchannel Heat Sinks for Electronics Cooling," *International Journal of Extreme Manufacturing,* 2023. [CrossRef] [Google Scholar] [Publisher Link]

[6] Long Huang et al., "A Computational Fluid Dynamics and Effectiveness-NTU based Co-simulation Approach for Flow Mal-distribution Analysis in Microchannel Heat Exchanger Headers," *Applied Thermal Engineering,* vol. 65, no. 1-2, pp. 447-457, 2014. [CrossRef] [Google Scholar] [Publisher Link]

[7] Binghuan Huang, Haiwang Li, and Tiantong Xu, "Experimental Investigation of the Flow and Heat Transfer Characteristics in Microchannel Heat Exchangers with Reentrant Cavities," *Micromachines,* vol. 11, no. 4, pp. 1-15, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[8] Jahar Sarkar, and Souvik Bhattacharyya, "Application of Graphene and Graphene-Based Materials in Clean Energy-Related Devices Minghui," *Archives of Thermodynamics*, vol. 33, no. 4, pp. 23-40, 2012. [Google Scholar]

[9] Yongjia Wu et al., "Enhanced Thermal and Mechanical Performance of 3D Architected Micro-channel Heat Exchangers," *Heliyon,* vol. 9, no. 3, pp. 1-19, 2023. [CrossRef] [Google Scholar] [Publisher Link]

[10] Yanhui Han et al., "A Review of Development of Micro-Channel Heat Exchanger Applied in Air-Conditioning System," *Energy Procedia*, vol. 14, pp. 148-153, 2012. [CrossRef] [Google Scholar] [Publisher Link]

[11] Seok-Won Seo, Huee-Youl Ye, and Kwan-Soo Lee, "Design of a Micro-Channel Heat Exchanger for Heat Pump Using Approximate Optimization Method," *Korean Journal of Air-Conditioning and Refrigeration Engineering*, vol. 24, no. 3, pp. 256–264, 2012. [CrossRef] [Google Scholar] [Publisher Link]

[12] Changye Huang et al., "Review on the Characteristics of Flow and Heat Transfer in Printed Circuit Heat Exchangers," *Applied Thermal Engineering*, vol. 153, pp. 190-205, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[13] M. Valdes, J.G. Ardila Marín, and M. Holguin, "Evaluation of the Minimum Required Length to Study Improved Heat Exchangers via Computational Fluid Dynamics," *Journal of Physics Conference Series, Workshop on Modeling and Simulation for Science and Engineering*, Pereira, Colombia, vol. 1403, pp. 1-5, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[14] N. Piroozfam, A. Hosseinpour Shafaghi, and S.E. Razavi, "Numerical Investigation of Three Methods for Improving Heat Transfer in Counter-Flow Heat Exchangers," *International Journal of Thermal Science*, vol. 133, pp. 230-239, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[15] J. Derek et al., "Investigating the Effect of Geometry on Micro-Channel Heat Exchangers Using CFD Analysis," *Advances in Fluid and Thermal Engineering, Lecture Notes in Mechanical Engineering*, pp. 401-408, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[16] Arslan Saleem, and Man-Hoe Kim, "Air-side Thermal Hydraulic Performance of Microchannel Heat Exchangers with Different Fin Configurations," *Applied Thermal Engineering*, vol. 125, pp. 780-789, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[17] Chirag Maradiya, Jeetendra Vadher, and Ramesh Agarwal, "The Heat Transfer Enhancement Techniques and Their Thermal Performance Factor," *Beni-Suef University Journal of Basic Applied Sciences*, vol. 7, no.1, pp. 1-21, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[18] Wenguang Li, Weihong Li, and Zhibin Yu, "Heat Transfer Enhancement of Water-cooled Triply Periodic Minimal Surface Heat Exchangers," *Applied Thermal Engineering*, vol. 217, pp. 1-21, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[19] David O. Ariyo, and Tunde Bello-Ochende, "Constructal Design of Two-Phase Stacked Microchannel Heat Exchangers for Cooling at High Heat Flux," *International Communications in Heat and Mass Transfer*, vol. 125, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[20] Hemant Naik, S. Harikrishnan, and Shaligram Tiwari, "Numerical Investigations on Heat Transfer Characteristics of Curved Rectangular Winglet Placed in a Channel," *International Journal of Thermal Science*, vol. 129, pp. 489-503, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[21] Zeynep Küçükakça Meral, and Nezaket Parlak, "Experimental Research and CFD Simulation of Cross Flow Microchannel Heat Exchanger," *Journal of Thermal Engineering*, vol. 7, no. 2, pp. 270-283, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[22] Lokanath Mohanta et al., "Numerical Analysis of Fluid Flow and Heat Transfer in Wavy and Hybrid-Slit-Wavy Fin-and-Tube Heat Exchangers," *Science and Technology for the Built Environment*, vol. 25, no. 6, pp. 767-775, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[23] Kanishka Panda, Tomoki Hirokawa, and Long Huang, "Design Study of Microchannel Heat Exchanger Headers Using Experimentally Validated Multiphase Flow CFD Simulation," *Applied Thermal Engineering*, vol. 178, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[24] Muhammad Saeed et al., "Performance Enhancement of a C-shaped Printed Circuit Heat Exchanger in Supercritical CO_2 Brayton Cycle: A Machine Learning-based Optimization Study," *Case Studies in Thermal Engineering*, vol. 38, pp. 1-23, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[25] Yifeng Hu, David P. Yuill, and Amir Ebrahimifakhar, "The Effects of Outdoor Air-side Fouling on Frost Growth and Heat Transfer Characteristics of a Microchannel Heat Exchanger: An Experimental Study," *International Journal of Heat Mass Transfer*, vol. 151, pp. 1-42, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[26] F. Ayad, R. Benelmir, and M. Idris, "Thermal-Hydraulic Experimental Study of Louvered Fin-and-Flat-Tube Heat Exchanger Under Wet Conditions With Variation of Inlet Humidity Ratio," *Applied Thermal Engineering*, vol. 183, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[27] Jian Wen et al., "Optimization Investigation on Configuration Parameters of Serrated Fin in Plate-Fin Heat Exchanger Using Genetic Algorithm," *International Journal of Thermal Sciences*, vol. 101, pp. 116-125, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[28] Arafat A. Bhuiyan, and A.K.M. Sadrul Islam, "Thermal and Hydraulic Performance of Finned-Tube Heat Exchangers Under Different Flow Ranges: A Review on Modeling and Experiment," *International Journal of Heat and Mass Transfer*, vol. 101, pp. 38-59, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[29] Mohammadreza Kadivar, David Tormey, and Gerard McGranaghan, "CFD of Roughness Effects on Laminar Heat Transfer Applied to Additive Manufactured Minichannels," *Heat and Mass Transfer*, pp. 1-15, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[30] Inderjot Kaur, and Prashant Singh, "State-of-the-Art in Heat Exchanger Additive Manufacturing," *International Journal of Heat and Mass Transfer*, vol. 178, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[31] Qinguo Zhang et al., "Research on Low-Temperature Heat Exchange Performance of Hydrogen Preheating System for PEMFC Engine," *International Journal of Hydrogen Energy*, vol. 45, no. 55, pp. 30966-30979, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[32] Md Mesbah-ul Khan, "*Experimental Investigation of Heat Transfer and Pressure Drop Characteristics of Water and Glycol-Water Mixture in Multi-Port Serpentine Microchannel Slab Heat Exchangers*," Electronic Theses and Dissertations, University of Windsor, pp. 1-530, 2011. [Google Scholar] [Publisher Link]

[33] L. Berrin Erbay et al., "Numerical Investigation of the Air-Side Thermal Hydraulic Performance of a Louvered-Fin and Flat-Tube Heat Exchanger at Low Reynolds Numbers," *Heat Transfer Engineering*, vol. 38, no. 6, pp. 627-640, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[34] Chuan Sun et al., "Thermal Enhancement of Fin and Tube Heat Exchanger with Guiding Channels and Topology Optimisation," *International Journal of Heat and Mass Transfer*, vol. 127, no. C, pp. 1001-1013, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[35] Zuoqin Qian, Qiang Wang, and Junlin Cheng, "Analysis of Heat and Resistance Performance of Plate Fin-and-Tube Heat Exchanger with Rectangle-Winglet Vortex Generator," *International Journal of Heat and Mass Transfer*, vol. 124, p. 1198-1211, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[36] V. Pandey, P. Kumar, and P. Dutta, "Thermo-Hydraulic Analysis of Compact Heat Exchanger for a Simple Recuperated sCO_2 Brayton Cycle," *Renewable and Sustainable Energy Reviews*, vol. 134, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[37] Tomas Venegas et al., "Critical Review and Future Prospects for Desiccant Coated Heat Exchangers: Materials, Design, and Manufacturing," *Renewable and Sustainable Energy Reviews*, vol. 151, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[38] Shimaa Zahran et al., "Heat Transfer Augmentation through Rectangular Cross Section Duct with One Corrugated Surface: An Experimental and Numerical Study," *Case Studies in Thermal Engineering*, vol. 36, pp. 1-14, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[39] S.K. Sarangi et al., "Analysis and Optimization of the Curved Trapezoidal Winglet Geometry in a High-Efficiency Compact Heat Exchanger," *International Journal of Thermal Science*, vol. 164, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[40] Saari Jussi, *Heat Exchanger Thermal Design Guide*, LUT University, 2010. [Publisher Link]

[41] Daniel Bacellar et al., "Design Optimization and Validation of High-Performance Heat Exchangers Using Approximation Assisted Optimization and Additive Manufacturing," *Science and Technology for the Built Environment*, vol. 23, no. 6, pp. 896-911, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[42] Seungjoon Baik et al., "Study on CO_2 – Water Printed Circuit Heat Exchanger Performance Operating under Various CO_2 Phases for S-CO_2 Power Cycle Application," *Applied Thermal Engineering*, vol. 113, pp. 1536-1546, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[43] Kiatbodin Wanglertpanich et al., "A Study of Curved Louver Fin Configuration for Heat Transfer Enhancement," *Journal of Applied and Computational Mechanics*, vol. 8, no. 2, pp. 754-763, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[44] Yu-Ming Chu et al., "CFD Analysis of Hybrid Nanofluid-Based Microchannel Heat Sink for Electronic Chips Cooling: Applications in Nano-Energy Thermal Devices," *Case Studies in Thermal Engineering*, vol. 44, pp. 1-15, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[45] S. Mohanty, O. Prakash, and R. Arora, "Analytical and Comparative Investigations on Counter Flow Heat Exchanger Using Computational Fluid Dynamics," *Journal of Computational and Applied Research in Mechanical Engineering*, vol. 10, no. 1, pp. 139-152, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[46] M.L.G. Ho, "A Review on Nanofluids Coupled with Extended Surfaces for Heat Transfer Enhancement," *Results in Engineering*, vol. 17, pp. 1-27, 2023. [CrossRef] [Google Scholar] [Publisher Link]

[47] M.S. Liu et al., "Enhancement of Thermal Conductivity with CuO for Nanofluids," *Chemical Engineering & Technology*, vol. 29, no. 1, pp. 72–77, 2006. [CrossRef] [Google Scholar] [Publisher Link]