Association Journal of Interdisciplinary Technics in Engineering Mechanics, Vol. 3, No. 1, 2025
ISSN: 3049-0898

CFD Analysis of Turbulent Flow in Microchannels for Heat Exchanger Applications

Blessing Kabasa[1]

[1]Department of Electrical Engineering, University of Zimbabwe, Harare, Zimbabwe. E-mail: kabasa.bles.in@gmail.com

Abstract--- Microchannel (MC) heat exchangers (MCHXs) are tiny, lightweight, and capable of delivering superior Heat Transfer (HT) efficiency. A significant issue with MCHXs is the uneven dispersion of two-phase refrigeration from the Header Area (HA) into the MC tubing. This problem is widespread in evaporation programs' intake HAs, where two-phase condensate is supplied into the MCHX tubes. Recent advancements in multiple-phase flow Computational Fluid Dynamics (CFD) have enabled the visualization and analysis of intricate multiphase flow patterns inside the MCHX HAs. This article uses the Hybrid Eulerian Multiphase (EMP) framework to simulate condensate dispersion in a two-phase fluid simulation. This model integrates the advantages of the Traditional EMP model and the Volume of Fluid (VOF) approach, with the former identifying scattered phases and delineating phase separation surfaces. The simulation area is partitioned into three phases. Two environments use scattered phase simulation, where a single stage is distributed inside another and vice versa. The third phase utilizes a comprehensive interface methodology to distinguish between the two stages. This study discusses implementing the Mixed EMP technique, providing a thorough explanation of the generated CFD simulation, involving the sensitivity evaluation of the sub-models and variables. The model is confirmed empirically by infrared thermal imagery. The flow pattern is analyzed, and the factors contributing to the uneven distribution are discovered. Design research is conducted for two vertical HAs: a Tube-Insertion HAs and a Loop HAs, which incorporates a loop into the former kind. The Loop HA demonstrates enhanced distribution efficiency. A sensitivity study is conducted to comprehend the impact of characteristics essential for enhancing shipping, including tube-insertion dimension, intake mass flow rate, and HAs loop diameters. This work aims to provide an exhaustive simulation methodology for the Has' uneven distribution issue in MCHXs.

Keywords--- CFD, Microchannels, Heat Exchanger, Turbulent Flow.

Received: 02 - 12 - 2024; Revised: 20 - 01 - 2025; Accepted: 10 - 02 - 2025; Published: 31 - 03 - 2025

I. Introduction

Microchannel (MC) (Deng et al., 2021) heat exchangers (MCHXs) (Li et al., 2024) have tubes with hydraulic widths ranging from a few hundred microns to millimeters. MCHXs are small and demonstrate exceptional Heat Transfer (HT) (Dwivedi et al., 2023) efficiency due to an improved surface space to volume proportion and the amplified influence of frictional and shear stresses on HT at the small scale. Additional advantages of MCHXs involve reduced air-side pressure decrease, fewer refrigerant charges relative to traditional finned-tube heat exchangers (HX) (Herbinger & Groulx, 2022), and a fully aluminum structure contributing to weight reduction. Owing to the reduced dimensions of the tubes and ports, refrigerant can't be directly injected into the flow pathways via distributors and blood vessels, as frequently occurs in finned-tube HT systems. Header Areas (HA) (Li et al., 2023) are the intermediary distribution mechanism between the distributors and the MC tubing. When an MCHX is used in the evaporation unit of a heat pump structure, the cooling fluid entering its intake is in a two-phase state. The uneven dispersion of two-phase refrigeration in input HAss is a significant factor contributing to the deterioration of HT efficiency in MCHXs.

In vaporizers, refrigerant uneven distribution leads to a decline in thermal transfer efficiency, resulting in undesirable Superheated Zone (SHZ) (He et al., 2024) with low approaching temperatures in some tubing. Research conducted trials using R745 in one-pass MC vaporizers with horizontal HAss and noted an efficiency decline of up to 25% attributable to maldistribution. Therefore, it is crucial to concentrate on the design optimization of MCHX connections to mitigate uneven distribution and subsequent performance deterioration.

This research proposes a multiphase Computational Fluid Dynamics (CFD) (Tsang et al., 2023) simulation framework using the Hybrid EMP technique to forecast the two-phase refrigerant dispersion in the intake HAs of MCHXs. CFD minimizes the human effort and time necessary for experiments while facilitating the visualization of comprehensive flow dynamics (Rutkowski et al., 2021). The deficiency in precision in CFD modelling diverges from the findings of actual flow dynamics. This strategy seeks to address the problem and is used in research on the MCHX HA layout. It aids in forecasting diverse multiphase flow characteristics by capturing phase distribution and distinct connections, enhancing the precision of two-phase flow representations.

II. Background

The primary aim of this study is to improve the cross-sectional shape of the cooling device to maximize HT from the small channels' fluid and minimize surface temperatures. The design was an internal fin configuration, including four fins of identical cross-sectional area positioned at the middle of each opening side. The purpose of including fins is to enhance the total contact surface area and facilitate increased HT, while keeping other parameters constant.

The enhanced contact region will augment the entire coefficient of HT and diminish the maximum wall temperatures due to a more uniform flux distribution. Investigations revealed that the variable tube design optimizes airflow dispersion to enhance the HT efficiency relative to traditional MCHX. The variable shape concept will save material in MC thermal exchangers, hence decreasing the cost of the HX. Studies conducted computational simulations using N-structure to examine the efficacy of HT and pressure loss in MC HT systems. They concluded that the MC's architecture influences the pressure differential across the cold side inflow and outflows. To investigate the impact of diverse geometry on MC heating elements, this research used a numerical technique using COMSOL Multiphysics. Different geometries were analyzed, and the findings were contrasted with typical MC heat sinks (HS) using clean water as the working liquid. They determined that an optimal number of tunnels would ensure flow uniformity and improve HT. Research was performed on an aluminum crossed-flow MC warmth exchanger, which included square cross sections. CFD models corroborated the results of experiments.

HT modeling is crucial in several engineering fields, including energy storage gadgets, fuel cells, HT systems, porous materials, and meteorology. MC warmth sinks represent one of the more energy-efficient methods for heating and cooling equipment. Research examined the thermal conductivity characteristics of a dual-layer MC thermal sink. Their findings demonstrated that the heating capacity of the double-layered MC heat absorber surpassed that of the single-layered variant. Research investigated using nano-fluids to enhance thermal efficiency in a double-layered MC HS. Reducing the channel quantity or dimensions enhances the channel's heating efficiency. Studies adjusted the morphological properties of double-layered tiny HSs, revealing that elevated pumping power adversely impacts the efficacy of the thermal exchanges. Research examined the thermal transfer in a dual-layer parallel flow MC HS. It was shown that for all examined Reynolds numbers, a reduced middle-rib diameter enhances heating efficiency.

III. CFD Analysis of Flow in MC for HX

Fig. 1 illustrates the wireframe shape of an elementary model of an MCHX with dimensions of 40 mm by 40 mm. The HS is composed of aluminum, using water as the liquid cooling medium. It displays the three-dimensional solid model of the fundamental MCHX, which serves as the initial model for analysis. A pocket-like enclosure accommodates and secures micro-electronic components such as Integrated Circuits (ICs). The pocket sizes are engineered to closely match the specifications of the integrated circuit according to Intel specifications. The MC design was imported into Ansys Workbench, boundary requirements were applied, meshing was conducted, and a mesh-independent test was executed. A steady-state CFD study was conducted under the assumptions of laminar and indestructible flow conditions. Temperature outlines are created and examined during post-processing. Subsequent calculations are performed to ascertain the bulk median temperature and mean HT ratio to evaluate the effectiveness of the fundamental model with different geometries.

The solid region used a tetrahedral mesh, whereas the fluid domain mainly employed a hexahedral mesh. The rationale is that a continuous flow benefits from a structured mesh, such as a hexahedral shape, rather than an unstructured mesh, like a tetrahedral shape. Individual component body scaling enhanced the mesh integrity in the liquid and solid domains. The emphasis was on minimizing the fluid mesh dimension in the flux zone to get precise findings. Fig. 1 illustrates the mesh developed for the investigation. Approximately 0.64 million pieces were produced for the basic mesh. A mesh-independent test was performed using a greater number of pieces.

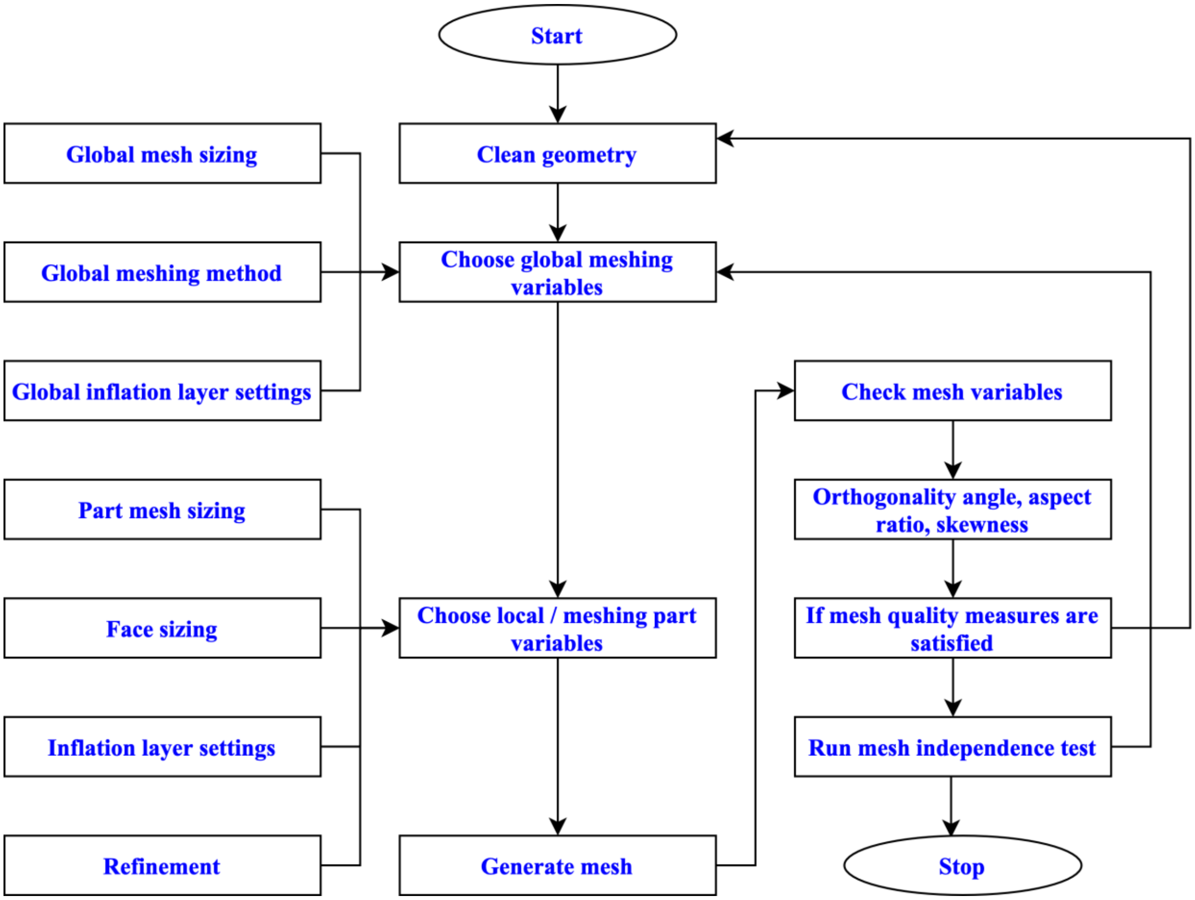

Figure 1: Flowchart of the Workflow

IV. Empirical Verification of the CFD Model

After creating the CFD approach, simulations were carried out with a 23-tube Tube-Insertion HA for experimental validation. This study uses the contrast between infrared thermal photographs as a criterion for validating CFD models, instead of using observed variables indicative of maldistribution, such as void percentage in a container, tube output weight flow rate, speed, or quality. Although it is theoretically feasible to quantify these values within the tubes and their connections, constructing such measuring apparatuses and acquiring data are complex endeavors. Intricate mechanisms directly influence the dispersion of the refrigerant. Although this article does not address such metrics, they are being considered for future research. Future identification of appropriate testing methodologies is essential, and such measurements must be conducted to enhance validation accuracy and foster more trust in the CFD methodology.

This research validates the findings by contrasting the SHZ dispersion between a CFD model and the experimental results. The proportions of the SHZ downstream from every tube have been displayed to get this pattern in the experiment. The dispersion was presumed to resemble the temperature dispersion acquired using thermal infrared imaging with an infrared camera during the examination. The SHZ is shown in red and delineated by a black barrier to emphasize its distribution pattern. The distribution characteristic of the SHZ is similar in both modeling and reality. The peaks and minima of the profile roughly coincide in both instances.

The proportion of every tube's length occupied by the SHZ was computed. Although this proportion was easily accessible for the simulation information, it was calculated manually by determining it from the infrared heat photographs in the case of the real-world study. The disparity in the percentage of SHZ width between the numerical simulation and the experimental results characterizes the mistake. The inaccuracy for all tubes was determined by subtracting the proportion of SHZ in the computer model from that in the actual test. The variances for every pipe are then summed. The mean deviations for every tube are 3.68%, 5.23%, and 3.34% for HA intake mass flow rates of 37 kg/h, 58 kg/h, and 76 kg/h, respectively. The mean errors are approximately five percent or lower for all three intake flow rate scenarios. Consequently, the simulation model has been empirically verified.

The inaccuracies in the SHZ arise from uncertainty in both models and experiments. The former involves instrument unpredictability, setup mistakes, human manipulation of equipment, and manual evaluations, including the SHZ in all tubes. The challenges in simulation might be attributed to the assumptions and hypotheses used, which fail to mirror the actual event accurately. One critical component in multiphase interface modeling is the Interaction Length Scale (ILS). A user-defined procedure is established to assign the potential ILS values pertinent to various sections of the HA; it does not guarantee an exact duplication of the ILS values for each bubble or droplet present in the actual situation. Hypotheses such as using a streamlined homogeneous framework for two-phase solutions and substituting the outlet pipes with a porous material to reduce pressure add to the inaccuracies. They are crucial for ensuring convergence and optimizing simulation duration.

V. Conclusion

This research conducts an extensive computational and practical investigation of the uneven distribution of two-phase refrigerant in the intake HAs of MCHXs. A multiphase CFD system is constructed with the Hybrid EMP approach. This is a groundbreaking use of this strategy to address the issue of uneven distribution of the HA. The current two-phase CFD systems employ either VOF or Traditional EMP methods; however, this methodology identifies distinct phase separation boundaries and the distribution of one phase inside another, integrating the advantages of both VOF and Traditional EMP approaches. The CFD model is thoroughly elucidated and verified by experimental methods. The modeling results indicate that the proportion of SHZ on the HX surfaces aligns well with the infrared heating pictures obtained from the trials. The research further examines refrigeration flow patterns and analyzes the impacts of tube installation depth and intake mass flow velocity in a tube-insertion type HAs. A Loop HA is subsequently introduced and contrasted with the Tube-Insertion HAs. A sensitivity analysis is conducted to determine the size of the Loop HAs.

The computer simulations facilitate predicting flow dynamics inside the HAs and enhance comprehension and contrast among the HA layouts. This enables the designer to refine the essential design variables before manufacturing and evaluating prototypes. This work's multiphase CFD variable analysis refines the CFD modeling applicable to additional multiphase flow scenarios. At the same time, the layout parameter research can be utilized to optimize HA design across various applications. Future endeavors will focus on creating a comprehensive HX simulation tool that encompasses air-side computations, condensate water drainage, flow boils in MCs, flow expansions in the expanding gate, and the fluctuations of oil and refrigeration within the compressor part.

References

[1] Deng, D., Zeng, L., & Sun, W. (2021). A review on flow boiling enhancement and fabrication of enhanced microchannels of microchannel heat sinks. *International Journal of Heat and Mass Transfer*, *175*, 121332. https://doi.org/10.1016/j.ijheatmasstransfer.2021.121332.

[2] Li, F., Shi, K., Sun, X., Yue, B., Huang, D., Zhao, R., & Zhao, Y. (2024). Performance evaluation of a vertical-finned microchannel and a fin-tube heat exchangers under wet and frosting conditions. *International Journal of Refrigeration*, *162*, 158-168. https://doi.org/10.1016/j.ijrefrig.2024.03.011.

[3] Dwivedi, A., Khan, M. M., & Pali, H. S. (2023). A comprehensive review of thermal enhancement techniques in microchannel heat exchangers and heat sinks. *Journal of Thermal Analysis and Calorimetry*, *148*(23), 13189-13231. https://doi.org/10.1007/s10973-023-12451-3.

[4] Herbinger, F., & Groulx, D. (2022). Experimental comparative analysis of finned-tube PCM-heat exchangers' performance. *Applied Thermal Engineering*, *211*, 118532. https://doi.org/10.1016/j.applthermaleng.2022.118532.

[5] Li, Y., Roux, S., Castelain, C., Fan, Y., & Luo, L. (2023). Design and optimization of heat sinks for the liquid cooling of electronics with multiple heat sources: a literature review. *Energies*, *16*(22), 7468. https://doi.org/10.3390/en16227468.

[6] He, Z., Zhang, Q., Wei, Z., Liao, X., Wu, X., Zhang, J., & Tan, Y. (2024). Modeling Method for Overheated Zone and Two-Phase Zone of Dry Shell-and-Tube Evaporator in Ship Air Conditioning. *Processes*, *12*(2), 379. https://doi.org/10.3390/pr12020379.

[7] Tsang, T. W., Mui, K. W., & Wong, L. T. (2023). Computational Fluid Dynamics (CFD) studies on airborne transmission in hospitals: A review on the research approaches and the challenges. *Journal of Building Engineering*, *63*, 105533. https://doi.org/10.1016/j.jobe.2022.105533.

[8] Rutkowski, D. R., Roldán-Alzate, A., & Johnson, K. M. (2021). Enhancement of cerebrovascular 4D flow MRI velocity fields using machine learning and computational fluid dynamics simulation data. *Scientific reports*, *11*(1), 10240. https://doi.org/10.1038/s41598-021-89636-z.