JTE HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

Numerical Simulation of Heat Transfer Characteristics in a Finned Flat-Tube Microchannel Heat Exchanger

Thanh Hao Nguyen, Thanh Tinh Tran, Thanh Trung Dang*, Hoang Tuan Nguyen
Ho Chi Minh City University of Technology and Education, Vietnam

*Corresponding author. *Email:* trungdang@hcmute.edu.vn

ARTICLE INFO

Received:	10/04/2025
Revised:	11/05/2025
Accepted:	02/06/2025
Published online:	24/09/2025

KEYWORDS

Microchannel;

COMSOL Multiphysics 6.2;

Heat exchanger;

Heat transfer characteristics;

Finned flat-tube.

ABSTRACT

Heat transfer behaviors of a finned flat-tube microchannel heat exchanger have been numerically simulated. The microchannel device model was computed, designed, and simulated in this work. Water serves as the thermal working fluid for the design model; it has a temperature range from 40, 45, 50, 55 to 60 degrees Celsius and a fixed mass flow rate of 0.028 kg/s within the heat exchanger. The fluid that absorbs heat is air, which flows perpendicular to the heat exchanger's exterior. The study using COMSOL Multiphysics 6.2 software evaluated the effects of inlet parameters, including the feedwater temperature and mass flow rate, on the heat transfer characteristics of the sample. Numerical simulation results of the heat transfer characteristics of a finned flat-tube microchannel heat exchanger were validated by experimental data. The results demonstrated high cooling effectiveness, a characteristic velocity profile, and vortex formation in the first pass. The simulation model showed good agreement with experimental trends (with an approximately 8% deviation), proving useful for design and optimization while clarifying the operating mechanism and the role of numerical simulation. Key contributions include demonstrating significant cooling effectiveness, with water temperature reducing from 60 °C to below 38.6 °C after six passes. The simulations also revealed a flow velocity distribution consistent with fluid dynamics theory, observed vortex formation at the microchannel inlet and outlet, and noted a non-uniform temperature distribution across the fins.

Doi: https://doi.org/10.54644/jte.2025.1877

1. Introduction

Current developments in technology have propelled the need for miniaturized, space-efficient products, leading to the rapid development of micro-cooling systems. Microchannel heat exchangers, particularly the finned flat-tube variant, have emerged as a leading solution due to their enhanced heat transfer efficiency and compact design, achieved through a large surface contact area between fluids. These systems are now widely utilized in industries like aerospace, biomedicine, and electronics for their superior thermal performance and operational efficiency.

About finned flat-tube microchannel heat exchanger's pressure loss and heat transfer properties, Glazar et al. [1] used the response surface approach to optimize a finned flat-tube micro heat exchanger. The response surface method, as illustrated in Figure 1, was used to investigate the effects of four geometrical parameters on the heat capacity per volume and mass, pressure loss on the air and water surfaces, heat transfer efficiency, and compactness: fin pitch, microtube pitch, number of microchannels, and baffle wall thickness. A finned flat-tube microchannel heat exchanger was the subject of an experimental investigation by Dasgupta et al. [2], using a 50:50 ethylene glycol mixture at 74 °C as the hot fluid and air at varying temperatures (23 °C, 33 °C, 43 °C) as the cold fluid. They varied the air velocity to achieve air-side Reynolds numbers (Re) ranging from 750 to 3165, while keeping the Reynolds number (Re) on the liquid side at 200. The findings demonstrated a relationship between the air-side Reynolds number and the air-side Nusselt number (Nu_a), which is represented as $Nu_a = 0.861Re_a^{0.269}$. The obtained Nusselt numbers were higher compared to other studies, possibly due to the flat surfaces of the heat exchanger allowing for longer air residence times, ensuring uniform temperature distribution.

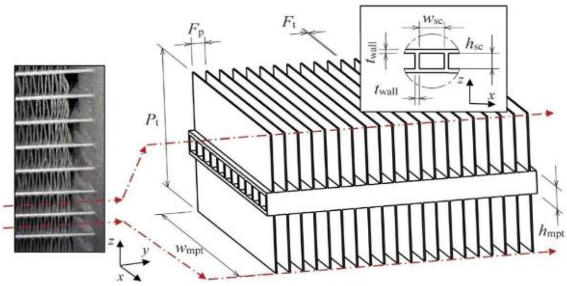


Figure 1. *A finned flat-tube microchannel heat exchanger model according to research [1].*

Oh et al. [3] found that flat-type microchannel heat exchangers have the maximum heat transfer efficiency after comparing several types of microchannel heat exchangers (MHEs), while U-shaped MHEs are suitable for high heat capacity. Efficiency increases with air flow rate but decreases slightly with inlet temperature. With a focus on finned flat-tube microchannel heat exchangers, an innovative technology that improves heat transfer efficiency and minimizes system space, Kumar et al. [4] examined the impact of liquid phase properties on flow and heat transfer in microchannel heat exchangers. The study examined heat transfer properties and pressure drop by modifying the water flow rate and take temperature. The findings demonstrated that while water temperature impacts the performance index and pressure drop, water flow rate has a major impact on heat efficiency.

The heat transfer properties and pressure loss of microchannel heat exchangers for various fluids are discussed. A rough-surfaced microplate heat exchanger's single-phase water flow was experimentally examined by Nilpueng and Wongwises [5], varying Reynolds numbers (1,300 - 3,200) and surface roughness (0.936 - 3.312 μm). Their findings revealed that increasing surface roughness enhanced both the heat transfer coefficient (4.46% to 17.95%) and pressure loss (3.90% to 19.24%) compared to smooth surfaces, illustrating the trade-off between improved heat transmission and higher pressure drop. A rectangular microchannel with a hydraulic diameter of 0.68 mm was used to experimentally study the pressure loss properties of R134a refrigerant boiling flow by Keepaiboon et al. [6]. They found that frictional pressure loss significantly influenced total pressure loss, with density increases raising the frictional loss gradient and saturation temperature increases lowering it. Heat flux had minimal impact. In order to improve the design of microchannel heat exchangers, they suggested a novel correlation. The case of manifold microchannel heat exchangers is examined. Andhare et al. [7] explored a micro-grooved/microchannel plate heat exchanger using single-phase water, detailing its design, experimental setup, and numerical simulation. At a low mass flow rate (20 g/s), experiments showed a high overall heat transfer coefficient (almost 20,000 W/(m²K)), validating its counter-flow performance against ε-NTU correlations. The study highlights the effectiveness of micro-grooves/microchannels in improving plate heat exchanger performance for industrial applications like refrigeration and electronics. Wang et al. [8] investigated unsteady boiling flow in a parallel microchannel with three inlet/outlet configurations (manifold, unrestricted, and restricted). They found that the restricted inlet (Type C) achieved the highest heat flux but also resulted in the highest-pressure loss within channels that are 30 mm long and have a hydraulic diameter of 186 μm.

Concerning the impact of size and geometric shape on the properties of heat transfer and pressure loss in microchannel heat exchangers. Huang et al. [9] studied microchannel heat exchangers with cavities of different shapes. They carried out studies to examine how the geometries of the collecting cavity and the open cavity affected the flow and heat transfer properties. The findings demonstrated that adding cavities to the microchannel can both increase its capacity for heat transfer and decrease pressure loss when compared to a straight channel. In order to model the flow characteristics and optimize the shape for a two-phase flow cooler, Ariyo and Bello-Ochende [10] ran a numerical simulation with an intake temperature of 25°C, they employed aluminum as the heat sink material and deionized water as the coolant. Heat flux (between 100 and 1,200 W/cm²) and velocities (between 0.1 and 4.5 m/s) were employed in the ANSYS modeling and optimization. Two-phase flow outperforms single-phase flow in rectangular microchannels at low Reynolds numbers and high heat flux, according to comparisons between the two types of flow. The thermal resistance falls with a rise in the optimal Reynolds number. It was discovered that the pressure loss determined the microchannels aspect ratio, optimum diameter, and axial length. Increased channel dimensions have a considerable influence on velocity profiles, according to a combined computational and experimental investigation on microchannel heat exchangers by Zhou et al. [11], while the hydraulic inlet length remains independent of channel size. They found that the average Nusselt number improves with wider and shallower channels, and friction coefficients decrease with wider spacing and reduced channel dimensions. Based on Nusselt number, friction, and thermal performance metrics, they determined that microchannels with heights of 1.0 - 1.5 mm, widths of 0.4 - 0.6 mm, and spacings of 0.6 - 0.8 mm offer optimal heat exchange performance. Boyea et al. [12] investigated a 4,000 W microchannel condenser with manifolds using R134a and R245fa, finding condenser efficiency dependent on manifold shape, achieving a convective heat transfer coefficient of 60,000 W/(m²K) and a 7 kPa pressure drop with R134a. Arie et al. [13] optimized single-phase flow in manifold microchannels without considering pressure drop and later extended this work [14], [15] to develop the optimal geometry of a flat-tube microchannel heat exchanger. Through numerical simulation and testing, the effects of fin shape (straight vs. V-shaped) on the heat transfer and heat dissipation time of a CO_2 air conditioning microchannel evaporator were investigated by Nguyen and Dang [16]. They found that fin shape had very little influence on the heat transfer process when the air-side heat transfer coefficient and heat transfer area remained constant. However, V-shaped fins significantly reduced computational time and memory requirements compared to straight fins. A typical heat flux of approximately 1,550 W/m² was observed at an evaporation temperature of 10 °C. The fin shape is described as in Figure 2.

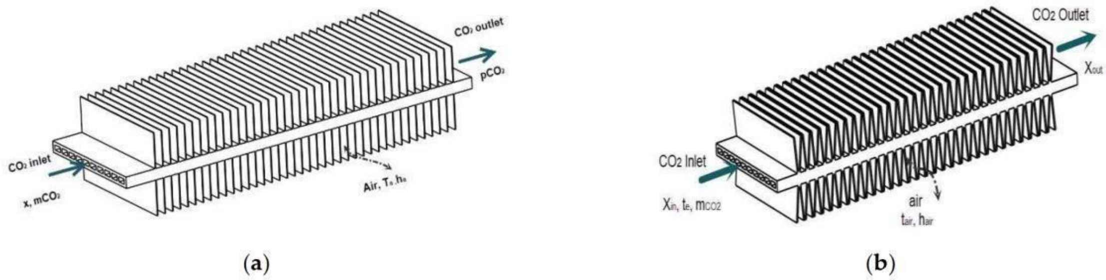


Figure 2. *Shape of straight heat sink (a) and V-shaped heat sink (b) according to [12].*

Existing studies focusing on heat transfer characteristics, pressure loss, and flow regimes, primarily on single or multiple channels within a tube, have predominantly relied on experimental methods and numerical simulations of two-phase flow. However, there's a lack of in-depth numerical simulations of single-phase flow to accurately assess local losses and related heat transfer characteristics. Numerical simulation and experimental studies that determine heat transfer characteristics and pressure loss within the specified temperature range not only clarify the operational mechanisms of microchannel heat exchangers but also provide a crucial scientific foundation for optimizing their design and operation. The advancement of numerical simulation tools, particularly COMSOL software, has opened up opportunities for more detailed research compared to traditional experimental approaches.

2. Materials and Methods

In this study, based on the study of reputable scientific works, this research identifies the problems that have and have not been solved to propose potential research directions. COMSOL Multiphysics 6.2 is used to construct and simulate a mathematical model of heat transfer and flow in a finned flat-tube microchannel heat exchanger. An experiment previously studied is used to validate the simulation results and evaluate the actual performance. Data from the simulation and experiment are processed and analyzed to draw conclusions.

2.1. Thermophysical properties of microchannel heat exchangers

The thermal capacity, or the amount of heat transferred via the heat exchanger, is determined using the following formula for both the water and air sides:

The rate of heat transfer via the water side Q_w is calculated as follows

$$Q_w = m_w c_{p,w}(t_{w,i} - t_{w,o}) \tag{1}$$

The air-side Q_a heat transfer rate is calculated as follows

$$Q_a = m_a c_{p,a}(t_{a,o} - t_{a,i}) \tag{2}$$

The heat exchanger's actual heat transfer efficiency is determined by

$$\varepsilon_a = \frac{Q_a}{Q_w} \tag{3}$$

It is computed what the heat exchanger's heat flux is

$$q = \frac{Q_a}{A} \tag{4}$$

The heat exchanger's air-side heat exchanger area is identified as follows

$$A = \frac{Q_a}{k \cdot \Delta t_{lm}} \tag{5}$$

The following formula is used to get the overall heat transfer coefficient

$$k = \frac{1}{\frac{1}{\alpha_a} + \frac{\delta}{\lambda} + \frac{1}{\alpha_w}} \tag{6}$$

Logarithmic mean temperature difference

$$\Delta t_{lm} = \frac{\Delta t_{max} - \Delta t_{min}}{ln\frac{\Delta t_{max}}{\Delta t_{min}}} \tag{7}$$

The Reynolds number was determined using the following equation

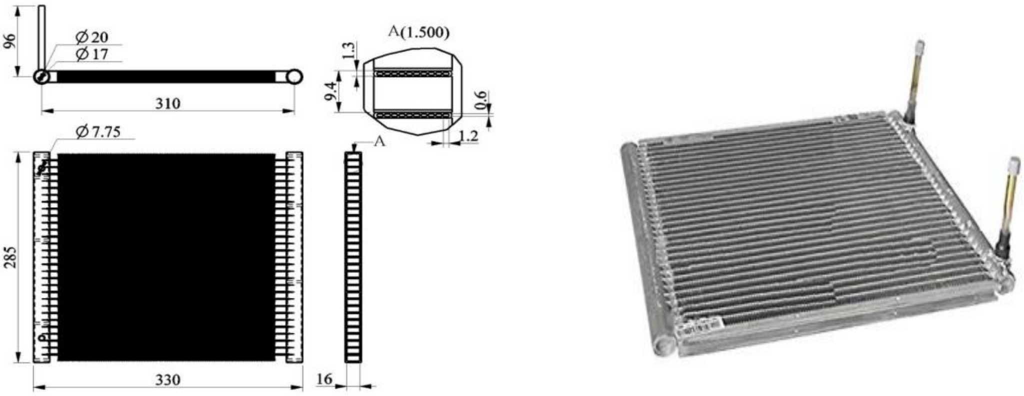
$$Re = \frac{\rho w D_h}{\mu} \tag{8}$$

Where: Q is the heat transfer rate (kW); m is the mass flow rate (kg/s); c_p is the specific heat capacity (kJ/(kg·K)); t are temperatures (The water temperatures at the inlet and outflow are denoted by $t_{w,i}$, $t_{w,o}$, respectively; $t_{a,o}, t_{a,i}$ are the outlet and inlet air temperature, respectively); q is heat flux (W/m^2); A is the heat exchanger's air-side heat exchanger area (m^2); k is overall heat transfer coefficient (W/(m^2·K)); α is the coefficient of convective heat transfer (W/(m^2 ·K)); δ is the heat exchanger plate's thickness (m); λ is the coefficient of heat conductivity (W/(m·K)); Δt_{lm} is the difference in logarithmic mean (°C); ρ is the density (kg/m^3); w is the fluid's velocity (m/s); D_h is the hydraulic diameter (m); μ is the dynamic viscosity (Pa·s);

Theoretical studies show that, with a heat capacity of 2.6 kW and a water mass flow rate of 0.028 kg/s, the water outlet temperature from the coil is 33.8°C, with the water input temperature to the heat exchanger set at 60°C. 30°C is the fixed temperature for the room air, which is also the heat exchanger's entrance temperature. Based on the experimental studies of Glazar et al. [1], which investigated different microchannel geometries with heat exchange efficiencies ranging from 60% to 95%, we choose an initial heat exchange efficiency of 87% for our heat exchanger, which translates to a 2.27 kW heat output on the air side. We determine the air outlet temperature from the coil to be 40.4°C with a 0.217 kg/s air mass flow rate through the coil, which is based on typical manufacturer requirements for the aforementioned power output, k = 108 W/(m^2 ·K) is the total heat transfer coefficient, and the logarithmic mean difference is Δt_{lm} = 9.6 °C. The following characteristics were used to construct a microchannel heat exchanger with channel diameters less than 1 mm, starting with a heat transfer area of A = 2.5 m². The heat exchanger's total dimensions (W x H x D) are 330x285x16 mm. Fins that were

JTE HCMUTE
ISSN: 2615-9740
JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

V-shaped and had dimensions of 1.1 mm for pitch, 0.1 mm for thickness, and 8.1 mm for height were chosen. The heat exchanger has six passes and 29 flat tubes in total. For the first, second, third, fourth, fifth, and sixth passes, there are 5, 6, 6, 5, 4, and 3 flat tubes, respectively. These passes are stacked on top of each other. Ten rectangular microchannels, each measuring 1.2 mm by 0.6 mm, are present in each flat tube. The material used to manufacture the heat exchanger is aluminum with a thermal conductivity coefficient λ = 202.5 W/(m·K). There is a 300 µm thickness between the microchannels. This heat exchanger has an intake for this heat exchanger unit. The intake headers of this heat exchanger have an inner diameter of 17 mm and an outer diameter of 20 mm. This heat exchanger unit's layout is seen in Figure 3.



(a) *(b)*

Figure 3. *Image of model design: (a) on SolidWorks software; (b) Actual image of model.*

2.2. Mathematical Model

The governing equations used in numerical simulations include [17]:

The continuity equation

$$\nabla \cdot (\rho \boldsymbol{u}) = 0 \tag{9}$$

The momentum equation

$$\rho(\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = \nabla \cdot [-p\boldsymbol{I} + \boldsymbol{K}] + \boldsymbol{F} \tag{10}$$

And heat transfer equations

$$\boldsymbol{q} = -k\nabla T \tag{11}$$

Where: the absolute temperature is denoted by T (K), pressure is denoted by p (Pa), the fluid velocity is denoted by $\boldsymbol{u}$ (m/s), the turbulence viscosity is denoted by μ_T (Pa·s), the fluid's external forces are denoted by $\boldsymbol{F}$ $(\mathrm{N/m^3})$, the density is denoted by ρ $(\mathrm{kg/m^3})$, the symbol for the specific heat capacity at constant pressure is c_p (J/(kg·K)), the viscous stress tensor (Pa) is denoted by $\boldsymbol{K}$, the identity matrix by $\boldsymbol{I}$, and the heat flux is denoted by $\boldsymbol{q}$ $(\mathrm{W/m^2})$.

Furthermore, for the turbulent flow in the current simulation, a universal $k - \varepsilon$ model was used with the Conjugate Heat Transfer. The $k - \varepsilon$ model's equations are shown below.

The model for the turbulent viscosity is as follows

$$\mu_T = \rho C_\mu \frac{k_T^2}{\varepsilon} \tag{12}$$

For k, the transport equation is as follows

$$\rho(\boldsymbol{u} \cdot \nabla)k = \nabla \cdot \left[\left(\mu + \frac{\mu_T}{\sigma_k}\right)\nabla k\right] + p_k - \rho\varepsilon, \tag{13}$$

where p_k is the production term

$$p_k = \mu_T\left[\nabla \boldsymbol{u}: \left(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T - \frac{2}{3}(\nabla \cdot \boldsymbol{u})^2\right)\right] - \frac{2}{3}\rho_k \nabla \cdot \boldsymbol{u} \tag{14}$$

The following is the transport equation for ε

$$\rho(\boldsymbol{u} \cdot \nabla)\varepsilon = \nabla \cdot \left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + C_{\varepsilon 1}\frac{\varepsilon}{k_T}p_k - C_{\varepsilon 2}\frac{\varepsilon^2}{k_T}, \tag{15}$$

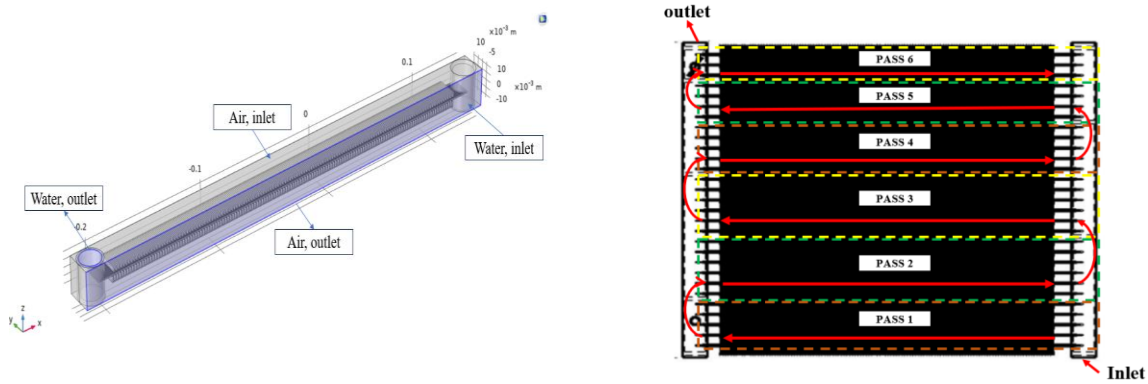
where k_T is the turbulent viscosity.

2.3. Numerical Simulation

COMSOL Multiphysics is a software program used in engineering that simulates flow and thermodynamics. It enables users to model and examine issues pertaining to thermodynamic sensors and fluid flow. COMSOL Multiphysics's flow simulation and thermodynamic analysis are essential elements. Due to its tiny fins and microchannels, the microchannel heat exchanger is very complicated and challenging to mimic at the same time. Assuming that all tubes within each pass have similar flow and thermal characteristics, selecting a representative flat tube helps reduce complexity in simulation and computation. And for this problem, the middle flat tube of each pass is chosen for simulation. Six passes of a finned flat-tube microchannel heat exchanger, therefore equate to six simulation periods. This method is employed to lower the amount of computational power. The input condition parameters for the first run are listed in Table 1. The first pass's numerical simulation findings serve as the input conditions for the second pass, and so on, until all six passes have been numerically simulated. Table 1 lists a few parameters for the inlet and outflow. There are certain starting values included in the inlet parameters.

Table 1. *Measured parameters and inlet parameters.*

Pass Number	The parameter of the inlet	The outlet parameter
1 (5 flat tubes)	$T_{1,i}$=X°C; T_a=30°C; v_a=3.5 m/s; $m_{w,1}=\frac{Y}{5}$ (kg/s)	$T_{1,o}$ (°C)
2 (6 flat tubes)	$T_{2,i}$=$T_{1,o}$; T_a=30°C; v_a=3.5 m/s; $m_{w,2}=\frac{Y}{6}$ (kg/s)	$T_{2,o}$(°C)
3 (6 flat tubes)	$T_{3,i}$=$T_{2,o}$; T_a=30°C; v_a=3.5 m/s; $m_{w,3}=\frac{Y}{6}$ (kg/s)	$T_{3,o}$(°C)
4 (5 flat tubes)	$T_{4,i}$=$T_{3,o}$; T_a=30°C; v_a=3.5 m/s; $m_{w,4}=\frac{Y}{5}$ (kg/s)	$T_{4,o}$(°C)
5 (4 flat tubes)	$T_{5,i}$=$T_{4,o}$; T_a=30°C'; v_a=3.5 m/s; $m_{w,5}=\frac{Y}{4}$ (kg/s)	$T_{5,o}$(°C)
6 (3 flat tubes)	$T_{6,i}$=$T_{5,o}$; T_a=30°C; v_a=3.5 m/s; $m_{w,6}=\frac{Y}{3}$ (kg/s)	$T_{6,o}$(°C)



(a) *(b)*

Figure 4. *Model: (a) A finned flat-tube with a straight fin; (b) A finned flat-tube microchannel model.*

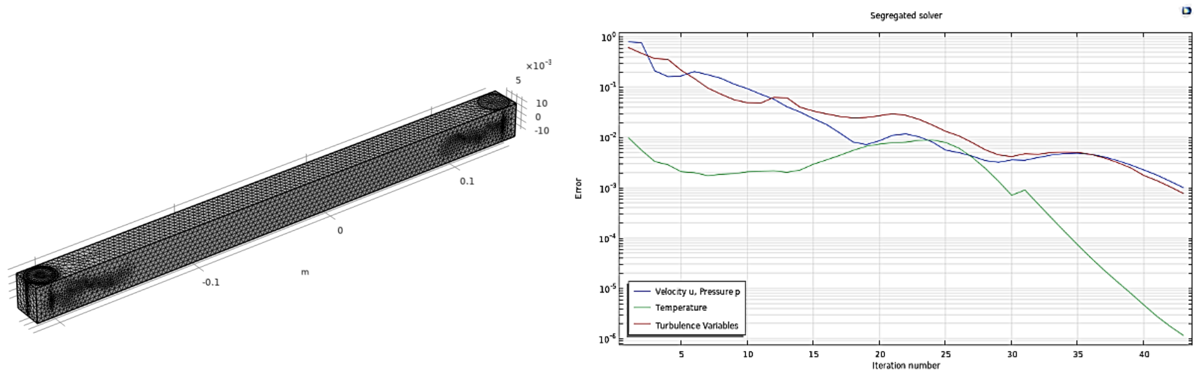
JTE HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

The entrance and output of a finned flat-tube microchannel heat exchanger are indicated by the subscripts i and o, respectively. T_a is the room temperature of the air, and m_w is the average water mass flow rate in a pass. The water temperature input is denoted by "*X*", which is 40, 45, 50, 55, and 60 degrees Celsius, respectively. The heat exchanger's incoming water flow rate is constant at 0.028 kg/s, denoted as "*Y*".

The microchannel heat exchanger's flat-tube form is seen in Figure 4. Through convective heat transmission, the air is outside while the water runs within the flat tube.

As seen in Figure 5, the next stage is to evaluate the mesh quality and mesh convergence after putting the geometry construction into practice and establishing the input conditions.



(a) *(b)*

Figure 5. *Mesh quality of model: (a) Mesh Model; (b) Mesh Solution Convergence.*

Mesh quality is critical for accurate COMSOL 6.2 simulations. COMSOL offers flexible meshing options. This model used boundary conditions and tetrahedral meshing. The required mesh quality is an average index of 0.5-0.6 and a minimum index below 0.1. Table 2 details the mesh quality.

Table 2. *Summarizes the mesh quality.*

	Coarse	Normal	Fine	Finer
Number of elements	1200590	2812681	4950975	13449078
Minimum element quality	0.01032	0.006431	0.011	0.01896
Average element quality	0.4986	0.4883	0.5637	0.6625
Water outlet temperature at pass 1	55.9 (°C)	55.7 (°C)	55.2 (°C)	54.8 (°C)

The "Fine" mesh satisfied the requirements for numerical analysis, comprising 4,950,975 elements with an average quality of 0.5637 and a minimum quality of 0.011. The robustness of the simulation's convergence is demonstrated in Figure 5, where the temperature, turbulence, pressure, and velocity variables all converged within a 0.001 error range using the Generalized Minimal Residual Method (GMRES) solver. This approach takes a finite number of steps before producing an approximate solution to the linear problem.

3. Results and Discussion

Sequential numerical simulations were run, with the results of one pass feeding into the next. The simulation results of a finned flat-tube microchannel heat exchanger are presented and analyzed in this work. A water mass flow rate of 0.028 kg/s and inlet temperatures of 40, 45, 50, 55, and 60 degrees Celsius were used in six simulations. Heat transfer characteristics at the maximum investigated intake temperature are highlighted by a detailed analysis of the case with an air velocity of 3.5 m/s, an ambient temperature of 30 °C, and an inlet water temperature of 60 °C shown in Table 3 and Figure 6.

Table 3. *Numerical simulation results for 6 passes.*

Passing Number	The inlet temperature (℃)	The outlet temperature (℃)
1 (5 flat tubes)	$T_{1,i} = 60$	$T_{1,o} = 55.2$
2 (6 flat tubes)	$T_{2,i} = T_{1,o} = 55.2$	$T_{2,o} = 51.4$
3 (6 flat tubes)	$T_{3,i} = T_{2,o} = 51.4$	$T_{3,o} = 46.1$
4 (5 flat tubes)	$T_{4,i} = T_{3,o} = 46.1$	$T_{4,o} = 43.2$
5 (4 flat tubes)	$T_{5,i} = T_{4,o} = 43.2$	$T_{5,o} = 40.2$
6 (3 flat tubes)	$T_{6,i} = T_{5,o} = 40.2$	$T_{6,o} = 38.6$

Figure 6 shows the numerical simulation results of a finned flat-tube microchannel heat exchanger for 6 passes of the water temperature field in COMSOL software.

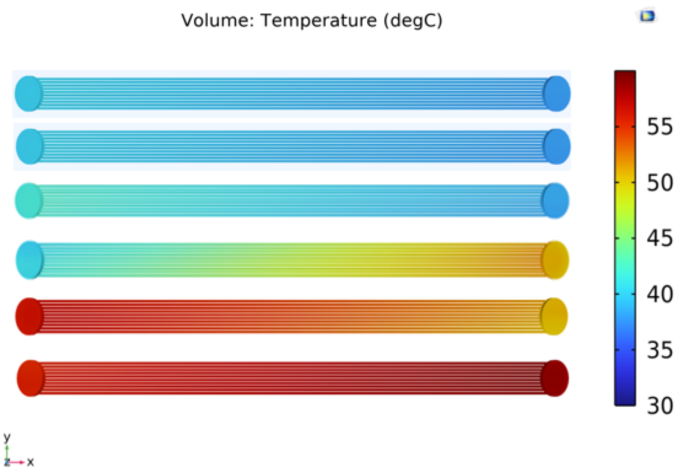


Figure 6. *Simulated water temperature results at 6 passes.*

To assess the overall heat exchange efficiency over the six passes, we will specifically examine the heat transfer properties of the flat tube in the first pass (pass 1), where the most important heat exchange takes place, and the outlet water temperature in the last pass (pass 6).

After 6 simulations of 6 passes in a finned flat-tube microchannel heat exchanger. In the case of pass 6, the simulated output of pass 5 is the inlet temperature. The analysis of pass 1 (Figure 7 - inlet) and pass 6 (Figure 8 - outlet) shows a significant decrease in water temperature. In pass 1, the water enters at 60°C and exits at 55.2°C. By pass 6, the outlet water temperature has decreased to below 38.6°C, indicating the cooling effectiveness of the heat exchanger.

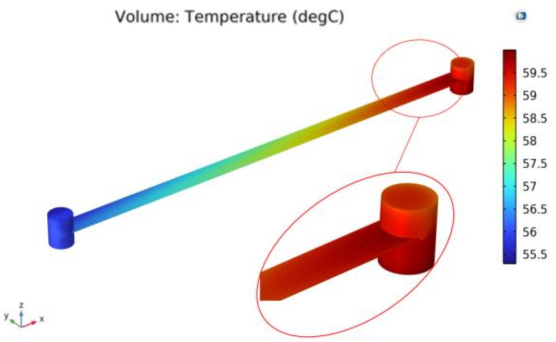


Figure 7. *Temperature distribution of the inlet water region at pass 1.*

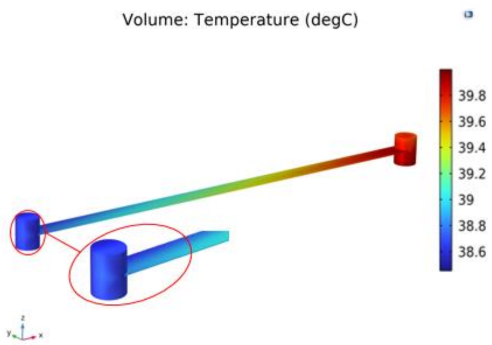


Figure 8. *Temperature distribution of the outlet water region at pass 6.*

JTE
HCMUTE
ISSN: 2615-9740
JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

Figures 9 and 10 present the simulation results of flow velocity in a finned flat-tube microchannel in pass 1. Figures 9 and 10 depict the flow velocity within a flat tube. Figure 9 shows a longitudinal cross-section of the tube, where lower velocities (blue) are observed near the walls and the highest velocities (red) are concentrated at the center, clearly illustrating the boundary layer effect. Figure 10 provides the velocity profile across the tube's cross-section, revealing that the maximum velocity is located in the central elliptical or rounded-rectangular region, decreasing towards the walls. This velocity distribution, with the highest velocity at the center and the lowest at the tube walls, is characteristic of flow in a flat tube and aligns with fluid dynamics theory regarding the influence of wall friction.

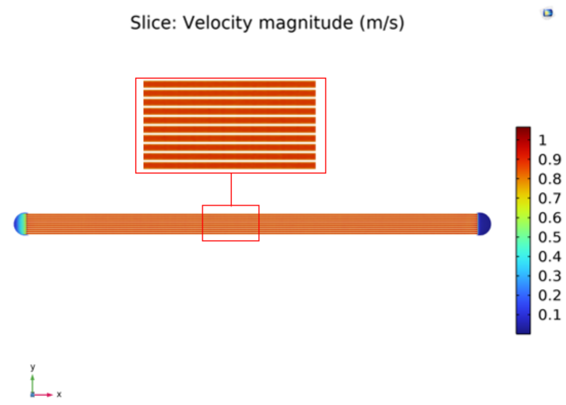


Figure 9. *Flow velocity in flat tube cross-section xy-planes at pass 1.*

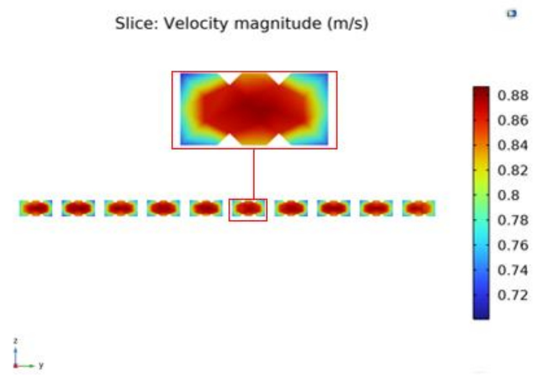


Figure 10. *Flow velocity profile in the flat tube cross-section yz-planes at pass 1.*

Figure 11 focuses on the velocity field of the water flow, illustrating the formation of complex vortex regions at the inlet caused by the channel structure. As the flow develops along the straight channel, the streamlines become more ordered, indicating a more stable flow. The colors on the streamlines would quantify the magnitude of the velocity at different locations. Conversely, Figure 12 depicts the temperature distribution across the microchannel heat sink fins. The region with the highest temperature is likely near the heat source, and the temperature gradually decreases along the length of the heat sink fins due to the heat conduction process. The heat sink fin structure, with its large surface area, helps to enhance heat exchange with the water flow.

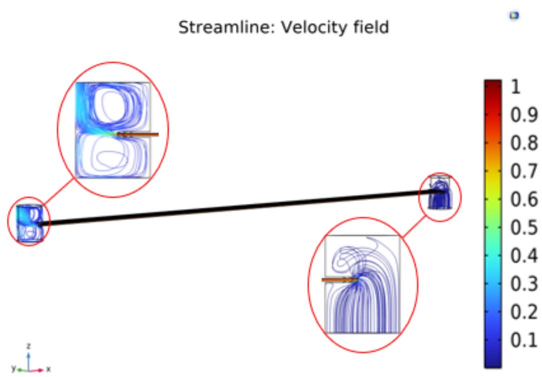


Figure 11. *Velocity distribution of the water flow in the microchannel at pass 1.*

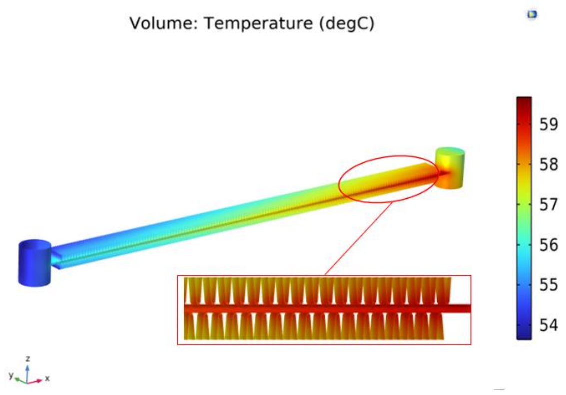


Figure 12. *Thermal distribution across microchannel heat sink fins at pass 1.*

In this study, the simulation results of a finned flat-tube microchannel heat exchanger have compared with the experimental results in [18]. Simulation and experimental data demonstrate that the exit water temperature and heating capacity increase as the intake water temperature rises, with a constant inlet water flow rate of 0.028 kg/s and input temperatures between 40 and 60 °C. In contrast to the experiment, the simulation forecasts outlet temperatures (Figure 13) and heating capacity (Figure 14) to be about 8% higher. This disparity may result from different conditions, measurement mistakes, or flaws in the model. Notwithstanding these variations, the simulation shows strong predictive power, particularly when considering the general trend. When comparing simulation and experimental results, measurement uncertainty should be taken into account, as indicated by error bars in the experimental data. Simulating six passes reduces computational costs and simplifies management compared to simulating the entire finned flat-tube microchannel heat exchanger. However, this simplification might overlook complex flow patterns (recirculation, velocity variations) in real multi-pass heat exchangers, potentially causing inaccuracies in predicting heat transfer performance.

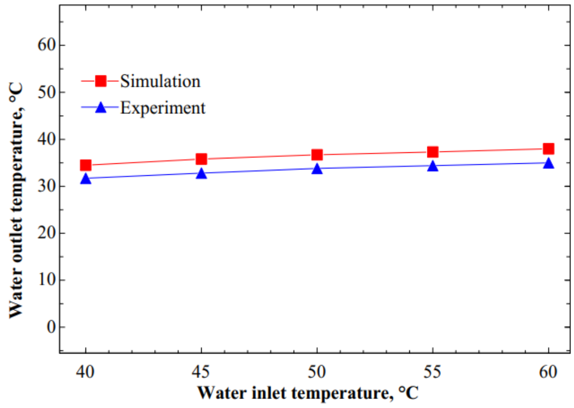


Figure 13. *The simulation and experiment's output water temperatures are compared using different intake water temperatures.*

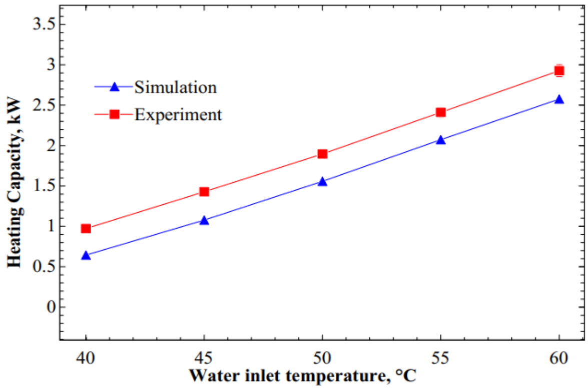


Figure 14. *Comparison of the simulation and experimental heating capacities for different inlet water temperatures.*

4. Conclusions

This study successfully simulated the heat transfer characteristics of a finned flat-tube microchannel heat exchanger using numerical methods with COMSOL Multiphysics 6.2 software, focusing on the influence of water inlet temperature and mass flow rate. The simulation results demonstrated a significant cooling effectiveness of the heat exchanger, with water temperature decreasing from 60°C to below 38.6°C after six passes. Flow analysis in the first pass showed a velocity distribution consistent with fluid dynamics theory, and vortices were observed at the microchannel inlet and outlet. Non-uniform temperature distribution on the fins was also noted.

The research indicated that increasing both inlet temperature and mass flow rate led to higher heating capacity and heat flux. Temperature simulations confirmed efficient heat exchange, while flow simulations revealed non-uniform velocity distribution due to channel configuration and friction.

Despite minor discrepancies (simulations predicted approximately 8% higher outlet temperature and power compared to experimental results, possibly due to model imperfections, measurement errors, or differing conditions), the simulations generally showed good agreement with experimental trends. This research clarifies the operating mechanisms of microchannel heat exchangers, providing a scientific basis for their design and optimization. The advancement of numerical simulation tools like COMSOL opens avenues for more detailed studies than traditional experimental methods. Future work will investigate other scenarios with varying mass flow rates (0.016, 0.020, 0.024, 0.032 kg/s) and inlet temperatures (40, 45, 50, 55°C).

Acknowledgments

In this study, we are grateful to Ho Chi Minh City University of Technology and Education (HCMUTE) for their assistance, time, and use of their facilities.

Conflict of Interest

The authors declare no conflict of interest.

REFERENCES

[1] V. Glazar, A. Trp, and K. Lenic, "Optimization of air-water microchannel heat exchanger using response surface methodology," *International journal of heat and mass transfer*, vol. 157, pp. 119887, 2020, doi: 10.1016/j.ijheatmasstransfer.2020.119887.

[2] Dasgupta, E. Sarbadaman, F. A. Siddiqui, and A. Fartaj, " Experimental study on air side heat transfer and fluid flow characteristics of microchannel heat exchanger, "*SAE International Journal of Materials and Manufacturing*, vol. 4, no. 1, pp. 1198-1210, 2011.

[3] S. H. Oh, S. H. Lee, D. Lee, S. H. Moon, and Y. Kim, "Air-side heat transfer and pressure drop characteristics of flat-type, U-and V-shaped microchannel condensers for refrigerator applications, " *International Journal of Heat and Mass Transfer*, vol. 176, pp.121460, 2021.

[4] R. Kumar and S. P. Mahulikar, " Physical effects of variable thermophysical fluid properties on flow and thermal development in Micro-Channel," *Heat Transfer Engineering*, vol. 39, no. 4, pp. 374-390, 2017.

[5] K. Nilpueng and S. Wongwises, "Experimental study of single-phase heat transfer and pressure drop inside a plate heat exchanger with a rough surface," *Experimental Thermal and Fluid Science*, vol. 68, pp. 268-275, 2015.

[6] C. Keepaiboon, P. Thiangtham, O. Mahian, A. S. Dalkılıç, and S. Wongwises, "Pressure drop characteristics of R134a during flow boiling in a single rectangular micro-channel," *International Communications in Heat and Mass Transfer*, vol. 71, pp. 245-253, 2015.

[7] R. S. Andhare, A. Shooshtari, S. V. Dessiatoun, and M. M. Ohadi, "Heat transfer and pressure drop characteristics of a flat plate manifold microchannel heat exchanger in counter flow configuration, " *Applied Thermal Engineering*, vol. 96, pp. 178-189,2015

[8] G. Wang, P. Cheng, and A. Bergles, "Effects of inlet/outlet configurations on flow boiling instability in parallel microchannels," *International Journal of Heat and Mass Transfer*, vol. 51, no. 9-10, pp. 2267-2281, 2007.

[9] B. Huang, H. Li, S. Xia, and T. Xu, "Experimental investigation of the flow and heat transfer performance in micro-channel heat exchangers with cavities," *International Journal of Heat and mass transfer*, vol. 159, pp. 120075, 2020.

[10] J. Zhang, D. O. Ariyo, and T. Bello-Ochende, "Constructal design of subcooled microchannel heat exchangers," *International Journal of Heat and Mass Transfer*, vol. 146, pp. 118835, 2020.

[11] F. Zhou, W. Zhou, Q. Qiu, W. Yu, and X. Chu, "Investigation of fluid flow and heat transfer characteristics of parallel flow double-layer microchannel heat exchanger," *Applied Thermal Engineering*, vol. 137, pp. 616-631, 2018.

[12] D. Boyea, A. H. Shooshtari, S. V. Dessiatoun, and M. M. Ohadi, "Heat Transfer and Pressure Drop Characteristics of a Liquid Cooled Manifold-Microgroove Condenser, " *ASME Proceedings, ASME 2013 Heat Transfer Summer Conference*, Paper No. HT2013-17781, pp. V003T23A003, 2013.

[13] M. A. Arie, A. H Shooshtari, S. V. Dessiatoun, M. M. Ohadi, and E. A. Hajri, "Simulation and Thermal Optimization of a Manifold Microchannel Flat Plate Heat Exchanger," *In ASME 2012 International Mechanical Engineering Congress and Exposition*, pp. 209-220, 2012.

[14] M. A. Arie, A. H. Shooshtari, S. V. Dessiatoun, and M. M. Ohadi, "Thermal optimization of an air-cooling heat exchanger utilizing manifold-microchannels, " *Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm)*, pp. 807-815, 2014.

[15] M. A. Arie, A. H. Shooshtari, S. V. Dessiatoun, Al-Hajri, and M. M. Ohadi, "Numerical modeling and thermal optimization of a single-phase flow manifold-microchannel plate heat exchanger," *International Journal of Heat and Mass Transfer*, vol. 81, pp. 478-489, 2015.

[16] H. T. Nguyen and T. T. Dang, "The Effect of Fin Shape on the Heat Transfer and the Solution Time of a Microchannel Evaporator in a CO2 Air Conditioning System— A Numerical Investigation," *Micromachines*, vol. 13, no. 10, p. 1648, 2022.

[17] COMSOL Software, M. Heat Transfer Module, Version 6.2, 2023, Inc. 100 District Avenue Burlington, MA 01803, USA. Available online: https://www.comsol.com/heat-transfer-module (accessed on 10 March 2025).

[18] N. B. Nguyen, T. T. Dang, H. T. Nguyen, T. H. Nguyen, and R. Munsin," Experimental Study on Heat Transfer Characteristics and Pressure Drop of a Fin and Microchannel Flat-Tube Heat Exchanger, " *In Lecture notes in networks and systems*, pp. 368-380, 2024.

Thanh Hao Nguyen received his B.S. degree in Thermal Engineering Technology at HCMC University of Technology and Education (HCMUTE), Vietnam, in 2021, studying for a master's degree; Department of Thermal Engineering; Faculty of Vehicle and Energy Engineering, at the same school. Currently, he is a Visiting Lecturer at the College of Technology II, Vietnam.
Email: 2391002@hcmute.edu.vn. ORCID: https://orcid.org/0009-0000-9516-1517

Thanh Tinh Tran received his B.S. degree in Aerospace Engineering at HCMC University of Technology (HCMUT), Vietnam, in 2008, and a Ph.D. degree in Fluid Mechanics from École Nationale Supérieure de Mécanique et d'Aérotechnique, France, in 2013. Currently, he is a Lecturer at the Faculty of Vehicle and Energy Engineering, Ho Chi Minh City University of Technology and Education. His main research interests are turbulence modeling, aerodynamic optimization, mixing, computational fluid dynamics, and nuclear fusion.
Email: tinhtt@hcmute.edu.vn. ORCID: https://orcid.org/0000-0001-6842-9266

Thanh Trung Dang, PhD, APE, is an Associate Professor of the Department of Thermal Engineering (the Former Head of the Department of Thermal Engineering and Former Vice Dean of Faculty of Vehicle and Energy Engineering) at Ho Chi Minh City University of Technology and Education (HCMUTE), Vietnam. He received his B.S. and M.S. in the Department of Thermal Technology at Vietnam National University Hochiminh City – University of Technology (HCMUT), Ph.D. in the Department of Mechanical Engineering, Chung Yuan Christian University (CYCU), Taiwan. His main research interests are nano/microscale heat transfer, energy and sustainable development, industrial refrigeration and air conditioning, and energy economics.
Email: trungdang@hcmute.edu.vn. ORCID: https://orcid.org/0000-0003-2286-1509

Hoang Tuan Nguyen received his B.S. and M.S degrees in Thermal Engineering at HCMC University of Technology and Education (HCMUTE), Vietnam, in 2015 and 2019, respectively, and is studying for a Ph.D. degree in the Department of Mechanical Engineering; Faculty of Mechanical Engineering, at the same school. Currently, he is a Lecturer at the Faculty of Vehicle and Energy Engineering, Ho Chi Minh City University of Technology and Education. His main research interests are nano/micro channel heat exchanger, industrial refrigeration and air conditioner. Email: tuannh@hcmute.edu.vn. ORCID: https://orcid.org/0009-0002-3815-0437