Heat Transfer Enhancement in Air Duct Flow By Micro-Channel Experimental And Numerical Investigation

Jalal M. Jalil
Ghada A. Aziz
Amjed A. Kadhim

Electromechanical Engineering Department, University of Technology, Baghdad, Iraq

Abstract. The heating of air flow through micro-channel was studied experimentally and numerically to examine the improvement in the thermal performance achieved by micro-channel. Numerically, laminar incompressible 3D steady-state Navier Stokes equations were solved by Finite volume method. Experimentally, a rig was built to investigate the cooling of air flow through micro-channel for different velocities and electric powers heater. The micro-channels block (length = 0.1, width = 0.05, height = 0.005 m) was manufactured from copper metal with 10 rectangular micro-channels (length = 0.1, width = 0.001, height = 0.0005 m). The performance of the microchannel was evaluated through exit air temperature value. The studied parameters numerically and experimentally were air velocities inside micro-channels (1 to 20 m/s) and heater powers (1 to 5 W). The comparison between numerical and experimental results was acceptable and reached 3% as maximum. Also the results were compared with other investigator results. High heat transfer coefficients values were achieved by micro-channel, reaches maximum value of 150 W/m² K at velocity air 20 m/s and heater power 5 W.

Keywords: rectangular channel, micro-channel, CFD, 3D simulation, heat transfer coefficient.

NOMENCLATURE

\( \text{cp} \) specific heat J/(kg K)
\( D_H \) hydraulic diameter (m)
\( H_{ch} \) channel height (m)
\( H_{HE} \) exchanger height (m)
\( K \) thermal conductivity (W/m K)
\( L \) channel length (m)
\( M \) mass flow rate (kg/s)
\( q \) heat transfer rate (W)
\( T \) temperature (K)
\( t_s \) separating wall thickness (m)
\( u \) x-component velocity (m/s)
\( V \) y-component velocity (m/s)
1. Introduction

Over the past years there have been an increased investigations in micro-channel flow, that has included all manufacturing and industry part; from space, medical application and the car world, it have accuracy and small design of this work tend to have small good cooling.

Vladimir Glazar et al [1] Authors’ main findings ,numerical analysis of fluid flow and heat transfer in the heat exchanger with microchannel coil (MCHX), 3D mathematical model has been defined and appropriate numerical simulation of heat transfer has been performed , as it gives more precise results. Isaev et al [2] Authors presented Enhancement of heat transfer in laminar air flow in narrow channels, with designing mini- and micro-channels of microelectronics cooling devices, compact miniature heat exchangers, air capacitors, radiators. Vortex heat transfer is considered using the computational domain of the narrow micro channel of dimensionless height 1, width 6 and length 4 at periodic boundary conditions. An oval-trench dimple of dimensionless width 1 and length 4.5 is located in the center of the heated wall at an angle of 45° to incoming flow. To solve the Navier–Stokes equations and the energy equation, . The Reynolds number is equal to 103. Sun, X. Y., et al [3] presented two types of DCHEs, desiccant coated microchannel heat exchanger (DMHE) and desiccant coated fin-and-tube heat exchanger (DFHE), are manufactured and tested. The heat transfer, mass transfer and pressure drop performances of two DCHEs are analyzed and compared by experimental results. DMHE shows lower heat capacity, higher heat and mass transfer capacity compared with DFHE. Mushtaq et al. [4] numerically studied the effect the size and shape of channels on the performance of counter-flow micro-channel in heat exchanger and used water as a cooling liquid. They reached that if the size of channels decreasing, both the effectiveness of heat exchanger and pressure drop will increase and they mention that the decision of increase or decrease the size of channels will depends on the application. Mathew and Hegab [5] analyzed theoretically performance of parallel flow microchannel in heat exchanger subjected to constant external heat transfer. The equation predicts temperature distributions as well as effectiveness for the heat exchanger. Moreover, the model equation used when the fluids subjected to equal or unequal external heat transfer. Costaschuk et al. [6] investigated experimentally flow of water through aluminum rectangular microchannel (hydraulic diameter equals 169 μm and a Reynolds number range from 230 to 4,740). The average Poiseuille number for the flows was 86.4, which is in good agreement with the theoretical value of 86.9. The average critical Reynolds number was found 2,370. Li and Wang [7] examined experimentally fluid flow and heat transfer of six array of louvered fins for a range of Reynolds no 400–1600 and subsequently developed. Sabbah et al. [8] mentioned that the calculation of heat transfer in Microchannels becomes difficult with complexity of the geometry of the Microchannels, requiring
3d analysis of heat transfer in both liquid and solid phases. CFD models were implemented in order to investigate and optimize the fluid flow and heat transfer in microchannel heat sinks. Ryu et al. [9] parametrically studied the procedure to improve the work of corrugated louvered fins. They have worked fin pitch, louver pitch, and louver angle as the three most important parameters. The JF factor of the resulting optimum model was improved 14–32% compared to the base model for Reynolds numbers in the range $0 \leq Re_{Lp} \leq 500$. Jang and Chen [10] used the conjugate gradient method to numerically investigate the thermal and hydraulic performance of a louvered fin heat exchanger with a variable louver angle for nine face velocities. Xia et al. [11] investigated the improved heat transfer effect in microsized channels. A numerical model has been set using CFD platform, hot flow is squeezed into the micro-channel and the heat is gradually transferred into the cold surface, top and bottom surfaces of the channel with rectangular area cross-sections. They reached that, heat transfer value per unit effective area is higher if the inlet area was decreased, and new design of heat exchanger is proposed based on these extra micro effects. Shinde and Lin [12] investigated the heat transfer and pressure drop of compact heat exchangers with louvered fins and flat tubes was conducted within a low air-side Reynolds number range of $20 < Re < 200$. They reached that better represented by two equations in two flow regimes (one for Reynolds $= 20 – 80$ and one for Reynolds $= 80 – 200$) than single regime equation in the format of power-law. Liu et al. [13] investigated experimentally the phase distribution of two-phase slug flow in horizontal multi-parallel micro channels, which was consisted of a header with hydraulic diameter of 0.667 mm and three parallel channels with hydraulic diameter of 0.5 mm, all of them have rectangular cross-sections. The inlet velocities of gas and liquid were from 0.130 to 1.47 m/s and from 0.128 to 0.900 m/s, respectively. They found that the phase distribution characteristics of two-phase flow in parallel channels highly depended on the inlet gas slug length and the inlet real velocity, a uniform phase distribution was achieved at high real velocity with short gas slugs. Shen et al. [14] investigated analytically the optimal location of internal vertical bifurcation integrated with a micro channel heat sink and the corresponding rectangular smooth micro channel is compared with those with internal vertical bifurcation. The simulation indicated that a clear inflection point of pressure gradient may prevail with the presence of internal vertical bifurcation. It was also found that the best thermal performance and the proposed optimal design of internal vertical bifurcation thermal performance without any pressure drop penalty.

To add more data on micro-channel literature, this work was organized to study micro-channel air heating experimentally and numerically. Numerically, laminar incompressible 3D steady-state Navier Stokes equations will solve by Finite volume method. Experimentally, a rig will built to investigate the heating of air flow through micro-channel for different velocities and electric powers heaters. The performance of the microchannel will evaluate through exit air temperature value.

2. Numerical Studies

In this study, the following assumptions have been made:

1- Three-dimensional
2- Steady-state and incompressible flow.
3- Conduction three-dimensional heat transfer in block of micro-channels.
4- Laminar flow (inlet Reynolds number is between is 500 to 2069).
5- All properties are evaluated at an average temperature.

The equations that describe the flow of air inside the micro-channel (continuity, momentum, and energy) were solved by finite volume method (SIMPLE algorithm) using Fortran 90 language (build in program). These equations are as follows [15, 16, and 17].

Continuity equation

Continuity equation
\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) + \frac{\partial}{\partial z} (\rho w) = 0
\]  

(1)

Momentum equations

X direction (U Momentum)

\[
\frac{\partial}{\partial x} (uu) + \frac{\partial}{\partial y} (uv) + \frac{\partial}{\partial z} (uw) - \frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)
\]  

(2)

Y direction (V Momentum)

\[
\frac{\partial}{\partial x} (vu) + \frac{\partial}{\partial y} (vv) + \frac{\partial}{\partial z} (vw) = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)
\]  

(3)

Z-direction (W Momentum)

\[
\frac{\partial}{\partial x} (uw) + \frac{\partial}{\partial y} (vw) + \frac{\partial}{\partial z} (ww) = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)
\]  

(4)

Energy equation

\[
\frac{\partial}{\partial x} (uT) + \frac{\partial}{\partial y} (vT) + \frac{\partial}{\partial z} (wT) = \frac{k}{\rho C_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)
\]  

(5)

The boundary conditions for the block and air flow are shown in the Table 3. The boundary conditions for air are shown in Table 1

Table 1 Boundary Conditions

| Variable | Wall | Inlet | Outlet |
|----------|------|-------|--------|
| Velocity | \( u, v, w = 0 \) | \( u = u_{in}, v = w = 0 \) | \( \left[ \frac{\partial u, v, w}{\partial n} = 0 \right] \) |
| Temperature | \( \left[ \frac{\partial T}{\partial n} = 0 \right] \) | \( T = T_{in} \) | \( \left[ \frac{\partial T}{\partial n} = 0 \right] \) |

By solving the above governing equations, the velocities and temperature distribution is determined in the air flow.
For the block, one-tenth of the block was studied (symmetry) by solving energy equation in the block as shown in Fig. 1. The energy equation was solved by thermal resistances. For example,

Node 1,

\[ q_w = -\frac{k_w A_w}{\Delta z} (T_{i,j,k} - T_{i,j,k-1}) \] (6)

\[ q_e = h_{in} * A e * (T_{i,j,k} - T_{air}) \] (7)

\[ q_n = -\frac{k_w A_n}{\Delta y} (T_{i,j,k} - T_{i,j+1,k}) \] (8)

\[ q_s = -\frac{k_w A_s}{\Delta y} (T_{i,j,k} - T_{i,j-1,k}) \] (9)

\[ q_t = -\frac{k_w A_t}{\Delta x} (T_{i,j,k} - T_{i+1,j,k}) \] (10)

\[ q_b = -\frac{k_w A_b}{\Delta x} (T_{i,j,k} - T_{i-1,j,k}) \] (11)

\[ C_s = \frac{k A_s}{\Delta y}, \quad C_e = h_{in} * A e, \quad C_t = \frac{k A_t}{\Delta x} \]

\[ C_n = \frac{k A_n}{\Delta y}, \quad C_w = \frac{k A_w}{\Delta z}, \quad C_b = \frac{k A_b}{\Delta x} \] (12)

\[ C = C_s + C_n + C_e + C_w + C_t + C_b \] (13)

\[ T_{i,j,k} = T_{i,j,k} \ast (1 - w) + w \ast \left[ c_s T_{(i,j-1,k)} + c_n T_{(i,j+1,k)} + c_e T_{air} + c_w T_{(i,j,k-1)} + c_t T_{(i+1,j,k)} + c_b T_{(i-1,j,k)} + q_{(i,j,k)} \right] / C \] (14)

The energy equation for the block was solved through a subroutine within the main program which solves Navier-Stokes equations in air flow simultaneously.
3. Experimental Studies

A schematic diagram of the test facility (block) is shown in Figs. 2, 3 and 4, which was made from copper. The overall dimensions are 50 mm wide x 5 mm high x 100 mm long. The block has ten micro-channels. The dimensions of single rectangular micro-channel are 1 mm wide x 0.5 mm high x 100 mm long (Tables 1 and 2). The working fluid is air; the inlet temperature was maintained at 26°C for all tests by control circuit. Air was drawn from the room through a blower and then through micro-channel equipped with a programmable variable speed drive (digital). Electric heater, with PID controllers, was installed in the bottom of the block of the microchannels. The electric power heater values vary between 1 to 5 W, while the velocity of the air inside the micro-channel varies between 2 and 20 m/s.

The hydraulic diameter,

\[ D_h = \frac{4A}{P} = \frac{4*W_{ch}*H_{ch}}{2(W_{ch} + H_{ch})}. \]  

(15)
Reynolds number,

$$\text{Re} = \frac{\rho v D_h}{\mu} \quad (16)$$

To calculate the convective heat transfer coefficient of the flow,

$$Q = m \cdot C_p \cdot (T_{ain} - T_{aout}) \quad (17)$$

The averaged heat transfer coefficients were defined as

$$h = \frac{Q}{A_s \cdot (T_w - T_b)} \quad (18)$$

Where $$A_s = (2H + 2W) L$$. The channel wall temperature, $$T_w$$, is assumed to be uniform and equal to the average of the readings from the six thermocouples located in the copper block.

$$T_b \text{(Average bulk mean temperature)} = \frac{T_{in} + T_{out}}{2}$$

Figure 2: Schematic of Microchannels
Table 2 Dimensional geometry of the micro-channel

| No of channels | $L_{ch}$ | $w_{ch}$ | $H_{ch}$ |
|----------------|----------|----------|----------|
| 10             | 100 mm   | 1 mm     | 0.5 mm   |

Table 3 Dimensional geometry of the micro-channels block

| length  | width | thickness |
|---------|-------|-----------|
| 100 mm  | 50 mm | 5 mm      |

4. Results and Discussions

In the experimental part, the studied parameters were; air velocity (1 to 20 m/s) and the heater powers (1 to 5 W). Fig. 5 shows the variation of air outlet temperature with inlet air velocities for various heater powers. The temperature decreases as the inlet air velocities increases from 2 to 20 m/s. The temperature increases as heater power increases. The micro-channel shows great performance; the temperature of the air starts with 26 °C at inlet and reaches 80 °C at outlet with heater power 5.5 W in only 10 cm length. Fig. 6 shows the variation of heat transfer coefficient with various inlet air velocities at variable heater powers. The heat transfer coefficient increases with increases of velocity reaching value of 150 W/m² K which is very high value for air to achieve. Fig. 7 shows variation of heat transfer coefficient with Reynolds number for different heater powers. The figure shows that the flow is laminar, although the flow is laminar, the heat transfer coefficient reaches 150 W/m² K. This value usually cannot be achieved even with turbulent flow for air. Fig. 8 shows the variation of air outlet temperature with inlet air velocity for various power heaters (numerical results). Fig. 9 shows the comparison between the experimental and numerical results of the outlet air temperature with inlet air velocities for various heater powers. The comparison was good. The numerical value was higher than experimental value because of small losses which usually ignored in numerical solution.
(insulated walls). Fig. 10 shows that the flow field of air through micro-channel at power = 5 W, $U_{ch} = 5$ m/s, the boundary layer was built even in small length (10 cm). Fig. 11 shows the isothermal contours of air flow inside micro-channel. The developing of thermal boundary layer was clear in three distances. The temperature reaches 80°C as average at the end for velocity 2 m/s and power 5 W. The outlet air temperature is nearly uniform in cross-section which means that the heat reaches the core of the flow (main advantage of the micro-channel). Fig. 12 shows comparison of the present results with Ref [18] results for the variation of Nusselt numbers with Reynolds number. The present study fits well with Ref [18] results.

5. Conclusions

The conclusions that can be drawn from the present study are:

1. Good agreement with deviation of about 2-3% between numerical and experimental results for outlet temperature.
2. The outlet air temperature is nearly uniform in cross-section.
3. The outlet air temperature from micro-channel was increased as velocity decreased.
4. CFD is a powerful tool to analyze and model to predict the temperature and velocity distribution in the flow of the microchannel and also to predict temperature of the body of the microchannel.
5. The value of heat transfer coefficient was reached high value 150 W/m² K, which cannot be reached without micro-channel.
Figure 5: Variation of air outlet temperature with inlet air velocities for various power heaters

Figure 6: Variations of heat transfer coefficient with inlet air velocities at different power heaters
Figure 7: Variation of heat transfer coefficient with Reynolds number for different power heaters

Figure 8: Numerical variation of air outlet temperature with inlet air velocities for different power heaters
Figure 9: Comparison between the experimental and numerical results of the variation of air outlet temperature with inlet air velocities for 3 and 4 W power heaters

Figure 10: Flow field through micro-channel at power = 5 W, $U_{ch} = 5$ m/s
Figure 1: Isothermal contours of air flow at power = 5 W, $U_{in} = 5$ m/s

HD = 0.3 and 0.9 mm [18], 0.66 mm present study

Figure 12: Comparison between numerical Nusselt number with Reynold number with different hydraulic diameters of present and other research results [18].
References

[1] Glazar, V., Trp, A., Lenic, K., & Torbarina, F. (2019). Numerical analysis of heat transfer in air-water heat exchanger with microchannel coil. In E3S Web of Conferences (Vol. 95, p. 02004). EDP Sciences.

[2] Isaev, S. A., Leontiev, A. I., Milman, O. O., Popov, I. A., & Sudakov, A. G. (2019). Influence of the depth of single-row oval-trench dimples inclined to laminar air flow on heat transfer enhancement in a narrow micro-channel. International Journal of Heat and Mass Transfer, 134, 338-358.

[3] Sun, X. Y., Dai, Y. J., Ge, T. S., Zhao, Y., & Wang, R. Z. (2019). Heat and mass transfer comparisons of desiccant coated microchannel and fin-and-tube heat exchangers. Applied Thermal Engineering, 150, 1159-1167.

[4] Hasan, M. I., Shkarah, A. J., Ali, M. S., & Rageb, A. M. A. (2010). Using of Microencapsulated Phase Change Material Suspension to Enhance the Performance of Microchannel Heat Exchanger. Thi-Qar University Journal for Engineering Sciences, 1(2), 1-12.

[5] Mathew, B., & Hegab, H. (2010). Application of effectiveness-NTU relationship to parallel flow microchannel heat exchangers subjected to external heat transfer. International Journal of Thermal Sciences, 49(1), 76-85.

[6] Costaschuk, D., Elsnab, J., Petersen, S., Klewicki, J. C., & Ameel, T. (2007). Axial static pressure measurements of water flow in a rectangular microchannel. Experiments in fluids, 43(6), 907-916.

[7] Li, W., & Wang, X. (2010). Heat transfer and pressure drop correlations for compact heat exchangers with multi-region louver fins. International Journal of heat and mass transfer, 53(15-16), 2955-2962.

[8] Sabbah, R., Farid, M. M., & Al-Hallaj, S. (2009). Micro-channel heat sink with slurry of water with micro-encapsulated phase change material: 3D-numerical study. Applied Thermal Engineering, 29(2-3), 445-454.

[9] Ryu, K., Yook, S. J., & Lee, K. S. (2014). Optimal design of a corrugated louvered fin. Applied thermal engineering, 68(1-2), 76-79.

[10] Jang, J. Y., & Chen, C. C. (2015). Optimization of louvered-fin heat exchanger with variable louver angles. Applied Thermal Engineering, 91, 138-150.

[11] Xia, L., & Chan, Y. (2015). Investigation of the enhancement effect of heat transfer using micro channel. Energy Procedia, 75, 912-918.

[12] Shinde, P., & Lin, C. X. (2017). A heat transfer and friction factor correlation for low air-side Reynolds number applications of compact heat exchangers (1535-RP). Science and Technology for the Built Environment, 23(1), 192-210.

[13] Liu, Y., Sun, W., & Wang, S. (2017). Experimental investigation of two-phase slug flow
distribution in horizontal multi-parallel micro-channels. Chemical Engineering Science, 158, 267-276.

[14] Shen, H., Wang, C. C., & Xie, G. (2018). A parametric study on thermal performance of microchannel heat sinks with internally vertical bifurcations in laminar liquid flow. International Journal of Heat and Mass Transfer, 117, 487-497.

[15] Awbi, H. B. (2002). Ventilation of buildings. Routledge.

[16] Versteeg, H. K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: the finite volume method. Pearson education.

[17] Patankar, S. V., Minkowycz, W. J., & Sparrow, E. M. (1980). Series in computational methods in mechanics and thermal sciences. McGraw-Hill Book Company, New York, 1-197.

[18] Sahar, A. M., Wissink, J., Mahmoud, M. M., Karayiannis, T. G., & Ishak, M. S. A. (2017). Effect of hydraulic diameter and aspect ratio on single phase flow and heat transfer in a rectangular microchannel. Applied Thermal Engineering, 115, 793-814.