INFLUENCE OF MICRO-CHANNEL INTERNAL FINS ON HEAT TRANSFER CHARACTERISTIC IN HEAT SINKS

Ahmed H. Abd El-Aziz*, Medhat M. Sorour, Mohamed A. Hassab, Wael M. El-Maghany

Mechanical Engineering Department, Faculty of Engineering, Alexandria University, Alexandria, Egypt.

*Corresponding author: ah_helmy007@hotmail.com

Abstract

A three-dimensional theoretical model has been carried out to predict the performance and temperature distribution and hydraulic characteristics for a heat sink with water-cooling in a circular mini/micro channels with internal fins at different operating and design parameters using ANSYS FLUENT 14.5 code. The model was validated with numerical and experimental published results. The conjugate heat transfer predicted by the model has a close agreement with the numerical and experimental results. As a case study, the effect of the fin length on the heat sink performance were explored. The simulation results showed that the optimum fin length that satisfied the best heat transfer criteria is 75% of the channel radius. However, the increasing of the fin length has a significant negative effect on the pressure drop.

Keywords: mini-channel heat sink, circular channel, internal fins, water-cooling.

NOMENCLATURE

D Diameter, m
R Radius, m
H Height, m
K Thermal conductivity, W/m K
h Convective heat transfer coefficient, W/m² K
L Length, m
W Width, m
P Pressure, Pa
T Temperature, K
t Thickness, m
V Velocity, m/s
x, y, z Cartesian coordinate directions
r, θ, z Cylindrical coordinate directions
u, v, w Velocity of x, y, z, directions respectively, m/s

©2015 RS Publication, rspublicationhouse@gmail.com
1. Introduction

In recent years, there are many researchers in heat transfer have been devoted to characterizing the flow in mini and microchannels. These channels have different characteristics than normal flow in heat exchangers and heater process devices. The concept of micro- and mini channels was at first proposed by Tuckerman and Pease [1] in heat sinks about three decades ago. They showed that the heat transfer could be enhanced by reducing the channel height down to micro scale. Wang and Peng [2] investigated experimentally the single-phase forced flow convection of water or methanol flowing through microchannels with a rectangular cross-section. Fedorov and Viskanta [3] developed a three-dimensional model to investigate flow and conjugate heat transfer in the microchannel-based heat sink for electronic packaging applications. Soodphakdee et al. [4] studied the heat transfer performance of heat sink with commonly used fin geometries (round, elliptical and square). Alam and Ghoshdastidar [5] presented a finite-
difference based numerical simulation of steady, laminar heat transfer in circular tubes fitted with four identical longitudinal fins having tapered lateral profiles. Qu and Mudawar [6] analyzed numerically a three-dimensional fluid flow and heat transfer in a rectangular microchannel heat sink using water as the cooling fluid. Guo and Li [7] focused on the size effect induced by the variation of dominant factors and phenomena in the flow and heat transfer as the scale device decreases. Owhaib and Palm [8] investigated experimentally the heat transfer characteristics of single-phase forced convection of R134a through single circular microchannels with 1.7, 1.2, and 0.8 mm as inner diameters. Lelea et al. [9] presented an experimental and numerical research on microchannel heat transfer and fluid flow. Li et al. [10] performed three-dimensional numerical simulations of the laminar flow and heat transfer of water in micro silicon channels with non-circular cross-sections (trapezoidal and triangular). Jiang and Xu [11] investigated experimentally forced convection heat transfer of air and water in bronze and pure copper mini-fin structures and mini-channel structures. Naphon and Khoneur [12] performed experiments to investigate the heat transfer characteristics and pressure drop in the microchannel heat sinks under constant heat flux conditions. Naphon and Wiriyasart [13] studied the liquid cooling in the mini-rectangular fin heat sink with and without thermoelectric for CPU. Foonget al. [14] conducted a numerical study to investigate the fluid flow and heat transfer characteristics of a square micro channel with four longitudinal internal fins. Shao et al. [15] optimized the configuration sizes of multi-layer microchannel heat sink in order to enhance the performance of the high flux chip. Shafeie et al. [16] presented a numerical study of laminar forced convection in heat sinks with micro pin-fin structure. Sharma et al. [17] are analyzed the performance of trapezoidal and rectangular microchannels and compared for two different coolants, liquid gallium, and water. Naphon and Nakharintr [18] studied the heat transfer characteristics of nanofluids cooling in the mini-rectangular fin heat sink. Nagarajan et al. [19] analyzed numerically using ANSYS FLUENT and ANSYS structural module a three-dimensional model of ceramic plate-fin high-temperature heat exchanger with different fin designs and arrangements. Alfaryjat et al. [20] numerically investigated the water flow and heat transfer characteristics are affected by the geometrical parameters of the microchannel. Hasan [21] investigated a micropin fin heat sink numerically with three fins geometries (square, triangular and circular). Esmaeilnejad et al. [22] investigated convection heat transfer and laminar flow of nano fluids with non-Newtonian base fluid in a rectangular micro channel numerically using two-phase mixture model. Lee et al. [23] used enhanced micro channel heat sink with a sectional oblique fin to modulate the flow in contrast to continuous straight fin.

From previous works, there are few studies on the circular microchannels and fewer on the internal fins. Therefore, the aims of this study are to create a 3D numerical
model for circular microchannels with internal fins based on the conjugate heat transfer mechanisms of conduction and convection.

2. Description of The Designed Cooling Model

A numerical study was undertaken. A tool has been developed to study the heat transfer between the heat sink and the fluid in a mini/microchannel. The proposed domain in the study is a heat sink with four internal finned one-row microchannels. The heat sink with one row finned microchannels model consists of a 50 mm long and a square cross section with a single circular microchannel with a diameter of 1 mm with 2 of the diameter pitch (square cross section 2 mm × 2 mm). There are four internal fins equally distributed along the perimeter of the channel cross section, each has a length of 0.125, 0.25, 0.325, 0.375 and 0.425 mm, which are studied as shown in Fig. 1. In addition, one horizontal fin and one vertical fin are undertaken. All these cases with fin thickness equal to 0.1 mm. The heat sink is made from Aluminum and water is used as the cooling fluid.

Next figures show the three dimensions models geometries of the cases with the boundaries of the computational domain, where symmetrical boundary conditions are considered around the vertical centerline.

![Fig. 1.a: Geometry of single row finned channel](image)

![Fig. 1.b: \(L_{\text{fin}} = 0.375\) mm](image)

3. Numerical model

To analyze the thermal and flow characteristics of this model, the following assumptions are made:
- Coolant fluid is water and laminar.
- Steady state conditions
- Uniform wall heat flux.
- Negligible radiation heat transfer.
- Incompressible fluid.
- Variable thermos-physical properties.
- Negligible viscous heat dissipation.
- A body force is neglected, and no external force is applied.
- No heat generation.

The fundamental governing equations are continuity, momentum and energy equations that are derived from basic principles of heat and fluid flow solved using ANSYS FLUENT version 14.5. The equations are posed to implement SIMPLE (Semi-Implicit Method for Pressure-Linked equation) algorithm. Consequently, the Navier-Stoke’s equations are solved. A cylindrical coordinate system is used to simulate the model convection within the circular channel while Cartesian coordinate system is used to simulate the conduction within the solid substrate domain.

**Fluid Domain**

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial V_r}{\partial r} + \frac{1}{r} V_r + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} = 0 \quad (1)$$

Moreover, the momentum (Navier-stokes Equation) equation can be written as:

$$V_r \frac{\partial V_r}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_r}{\partial \theta} + V_z \frac{\partial V_r}{\partial z} - \frac{1}{r} V_\theta^2 = -\frac{1}{\rho_f} \frac{\partial P}{\partial r} + V \left[ \frac{\partial^2 V_r}{\partial r^2} + \frac{1}{r} \frac{\partial V_r}{\partial r} + \frac{1}{r^2} \frac{\partial^2 V_r}{\partial \theta^2} - \frac{V_r}{r^2} - \frac{2}{r^2} \frac{\partial V_r}{\partial \theta} \right] \quad (2)$$

$$V_r \frac{\partial V_\theta}{\partial r} + \frac{1}{r} V_\theta + \frac{\partial V_\theta}{\partial \theta} + V_z \frac{\partial V_\theta}{\partial z} = -\frac{1}{\rho_f} \frac{\partial P}{\partial \theta} + V \left[ \frac{\partial^2 V_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial V_\theta}{\partial r} + \frac{1}{r^2} \frac{\partial^2 V_\theta}{\partial \theta^2} - \frac{V_\theta}{r^2} - \frac{2}{r^2} \frac{\partial V_\theta}{\partial \theta} \right] \quad (3)$$

$$V_r \frac{\partial V_z}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_z}{\partial \theta} + V_z \frac{\partial V_z}{\partial z} = -\frac{1}{\rho_f} \frac{\partial P}{\partial z} + V \left[ \frac{\partial^2 V_z}{\partial r^2} + \frac{1}{r} \frac{\partial V_z}{\partial r} + \frac{1}{r^2} \frac{\partial^2 V_z}{\partial \theta^2} + \frac{\partial^2 V_z}{\partial z^2} \right] \quad (4)$$

Finally, the energy equation can be written as:

$$V_r \frac{\partial T_f}{\partial r} + \frac{V_\theta}{r} \frac{\partial T_f}{\partial \theta} + V_z \frac{\partial T_f}{\partial z} = \alpha \left[ \frac{\partial^2 T_f}{\partial r^2} + \frac{1}{r} \frac{\partial T_f}{\partial r} + \frac{1}{r^2} \frac{\partial^2 T_f}{\partial \theta^2} + \frac{\partial^2 T_f}{\partial z^2} \right] \quad (5)$$

**Solid Domain**

Assuming steady-state conditions and constant properties of the wall, the energy equation for this region is simplified as follows:

$$\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} + \frac{\partial^2 T_s}{\partial z^2} = 0 \quad (6)$$
Boundary Conditions
The heat sink is idealized as a constant heat flux boundary condition at the bottom wall. Heat transport in the unit cell is a conjugate problem which combines heat conduction in the solid and convective heat transfer to water.
It has been assumed a constant heat flux (q) at the bottom wall. The other wall boundaries of the solid region are assumed perfectly insulated with zero heat flux except the top wall is assumed a convective wall (h_{conv} = 100 \text{ w/m}^2\cdot\text{k} & T_{conv} = 298 \text{ K}) as shown in Fig.2. Water flowing through the channel at constant temperature 313 K. The water flow inlet velocities are taken from Reynolds number, which changes from case to another. Moreover, water is an exit at the atmosphere pressure.

![Boundary Conditions of heat sink](image)

Fig.2: Boundary Conditions of heat sink

**Hydraulic Boundary Conditions** of a water-cooled heat sink incremental element can be given as:
At channel inlet (z=0): p=p_{inlet}, u=0, v=0, w=w_{inlet}
At channel outlet (z=L): p=p_{atm}, u=u_{out}, v=v_{out}, w=w_{out}

**Thermal boundary conditions** can be given as:
At the bottom surface of heat sink (y = 0): \[ -K_s \frac{dT_s}{dy} = q_o \] (7)
At the top surface of heat sink (y = H): \[ -K_s \frac{dT_s}{dx} = h(T_s - T_\infty) \] (8)
At the left surface of heat sink (x = 0): \[ -K_s \frac{\partial T_s}{\partial x} = 0 \] (9)
At the right surface of heat sink (x = W): \[ -K_s \frac{\partial T_s}{\partial x} = 0 \] (10)
At the front surface of heat sink ($z = 0$): for channel $T = T_{\text{in}}$, for solid $-K_s \frac{\partial T_s}{\partial z} = 0$ (11)

At the back surface of heat sink ($z = L$): for channel $-K_f \frac{\partial T_f}{\partial z} = 0$, for solid $-K_s \frac{\partial T_s}{\partial z} = 0$ (12)

Grid dependence and validation

A variable sized grid system is generated in the domain for the numerical calculation. To ensure mesh-independent results, a series of trial solutions is conducted for several mesh configurations in the axial and radial directions. For example, a case of 1 mm diameter microchannel with fin length 0.375 mm and $\text{Re} = 1023$. According to the mesh sizes which had been studied from 0.000025 m to 0.000035 m as a maximum mesh size, many numbers of cells had been generated in the domain (2468180 cells), (3104640 cells), (3844740 cells), (4969508 cells) and (6583290 cells). In all the cases, the meshes are non-uniform in both directions, and denser meshes are considered in regions of high gradients, e.g., near the channel wall in the radial direction.

Fig. 3 presents the dependence of average heat transfer coefficient and the pressure drop in the number of cells. It can be shown that the last two numbers of cells of 4969508 and 6583290 give nearly identical average heat transfer coefficient. But still there is a small difference in the value of pressure drop about 30 Pa between these two numbers of cells. According to that, and to satisfy the highest possible degree of accuracy in the pressure drop, the maximum number of cells (6583290) was taken in the numerical study. Although that has a negative effect, represents the much computational time that the ANSYS spends to satisfy the convergence criteria.

![Fig. 3: Mesh dependence of the numerical solution](image-url)
In order to validate the computational modeling of microchannel heat transfer, some initial runs are performed, and the resulted data has been compared with experimental data of Lelea et al. [9]. As Fig.4 shows, the calculated local Nusselt number at the channel is compared with that experimentally obtained by Lelea et al., at Re equals 754.

![Graph showing comparison of local Nu versus distance from entrance between the present study and Lelea et al. [9]](image)

This comparison leads to good agreement between the results obtained numerically, and the experimental ones are proved with accepted percentage deviation that equals 2%.

### 4. Result and discussion

The studied micro channel geometry is a one-row constant channel diameter at 1 mm with internal fins. The studied geometrical parameter item is the fin length ($L_{\text{fin}}$ from 0.125 to 0.425 mm) with a variation in the number of fins (four fins distributed equally around the parameter of the channel (perpendicular to each other), one vertical and one horizontal fin). To study the effect of the fin length change, other parameters should be constant, such as; the fin thickness is 0.1 mm, the pitch is 2D, the mass flow rate is 1.026 g/s and the heat sink is made of Aluminum.

Next table shows the result parameters for the different studied cases, such as the hydrodynamic, thermal entry lengths, the maximum temperature at the bottom surface of the heat sink, the average heat transfer coefficient and the pressure drop at a subjected heat flux equals 20 w/cm².
Table 1: Calculated parameters values for different fin lengths.

<table>
<thead>
<tr>
<th>Fin Length (mm)</th>
<th>Hydraulic Diameter (mm)</th>
<th>Re</th>
<th>Hydrodynamic Developing Length (mm)</th>
<th>Thermal Entrance Length (mm)</th>
<th>Max. Temp. @ the bottom surface (k)</th>
<th>Average Heat Transfer Coefficient (kW/m².k)</th>
<th>Pressure Drop (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.125</td>
<td>0.71026</td>
<td>1517</td>
<td>53.87</td>
<td>226.25</td>
<td>339.596</td>
<td>8.22</td>
<td>3547.11</td>
</tr>
<tr>
<td>0.25</td>
<td>0.5332</td>
<td>1222</td>
<td>32.58</td>
<td>136.84</td>
<td>331.4</td>
<td>8.83</td>
<td>6711.05</td>
</tr>
<tr>
<td>0.325</td>
<td>0.4566</td>
<td>1095</td>
<td>25</td>
<td>105</td>
<td>327.626</td>
<td>10.31</td>
<td>9751.28</td>
</tr>
<tr>
<td>0.375</td>
<td>0.4138</td>
<td>1023</td>
<td>21.17</td>
<td>88.9</td>
<td>326.277</td>
<td>11.27</td>
<td>12128.3</td>
</tr>
<tr>
<td>0.425</td>
<td>0.3763</td>
<td>960</td>
<td>18.06</td>
<td>75.86</td>
<td>326.874</td>
<td>10.77</td>
<td>13630.9</td>
</tr>
<tr>
<td>One Vertical</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.555</td>
<td>1273</td>
<td>35.33</td>
<td>148.37</td>
<td>331.165</td>
<td>9.08</td>
<td>6504.81</td>
<td></td>
</tr>
<tr>
<td>One Horizontal</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.555</td>
<td>1273</td>
<td>35.33</td>
<td>148.37</td>
<td>331.496</td>
<td>9.1</td>
<td>6503.83</td>
<td></td>
</tr>
</tbody>
</table>

As indicated from table 1, it can be seen that for all studied cases the hydrodynamic fully developed lengths were established inside the channel length (50 mm) except for fin length equals 0.125 mm because its high Re number and hydraulic diameter ($L_h, \text{laminar} = 53.87 \text{ mm}$). Otherwise, the thermal developing is indicated as a developing flow over the entire channel length for all the cases. In addition, the fin length has a noticeable effect on the pressure drop; the relation is proportional between the two parameters. It is observed that the value of pressure drop of one vertical or horizontal fin is almost the same value of 0.25 mm fin length. Fig. 5 presents the effect of fin length change on the temperature distribution along the center line of the bottom surface which subject directly to the heat flux in the Z direction. As it shown, from finless micro channel to fin length equals 0.375 mm micro channel, when the fin length increases and due to the decreasing in the hydraulic diameter and the cross section area, the inlet velocity increases to keep the mass flow rate without any change between the studied cases. So in turn that causing the decrease of the Re number, as shown in the table 1, the maximum temperature at the center of the bottom surface which occurs at the end of the channel (under the water exit) decreases. For finless micro channel, the maximum temperature is 343.441 k while when the fin length reaches to 0.375 mm, the maximum temperature decreases to 326.277 k. That is because, as shown in Fig. 6, the heat transfer coefficient increases proportionally to the fin length until the fin length reaches 0.375 mm. When the fin length increases more than that (such as 0.425 mm) the maximum temperature along the center of the bottom surface starts to increase (326.874 k) and the heat transfer coefficient
decreases, this result can be explained by Fig. 7, which shows the temperature difference (between the average fluid temperature and surface mean temperature) distribution in Z direction for different fin lengths. This figure indicates that the temperature difference decreases with the increases of the fin length until it equals 0.375 mm, after that by the increasing of the fin length, the temperature difference starts to increase which cause the decrease of the heat transfer coefficient and in turn, the maximum temperature at the center of the bottom surface increases. Both of the one vertical fin and the one horizontal fin are almost identical, and all the results of these two cases are so close, which mean the positions of the fins is not an effective parameter on the thermal and hydraulic criteria. In addition, the results of the both cases are almost the same results of 0.25 mm fin length, which lead to that the total fins length is the major parameter, no matter how this length is distributed or how the fins are positioned. So from these results, it is clearly that, the optimum fin length that satisfied the best heat transfer criteria is 0.375 mm, which, in general, equals 75% of the channel radius.

![Graph showing temperature distribution along the centerline of the bottom surface (Z direction) for many fin lengths.](image)

**Fig. 5:** Temperature distribution along the centerline of the bottom surface (Z direction) for many fin lengths.
Fig. 6: Local heat transfer coefficient in Z direction for many fin lengths.

Fig. 7: Temperature difference (between the average fluid temperature and surface mean temperature) distribution in Z direction for many fin lengths.
Fig. 8 shows the maximum heat flux which the bottom surface of the heat sink can be subjected to it for many fin lengths to keep the condition of that the maximum heat sink temperature does not exceed the practical allowable electronic processor temperature limit of 343 k. As it shown in Fig. 8, the maximum allowable heat flux, which the heat sink is subjected to it, occurs at 0.375 mm fin length (45 w/cm²). That result matches with the results of heat transfer coefficient with the fin lengths.

![Fig. 8: Maximum heat flux against fin length.](image)

Fig. 9 shows the average heat transfer coefficients occur for many fin lengths, the maximum average heat transfer coefficient occurs at 0.375 mm fin length (11.27 kw/m².k). As shown in the table.

![Fig. 9: Average heat transfer coefficient against fin length.](image)
Fig. 10 shows the pressure drop occurs for many fin lengths, the relation is proportional to the fin length and the pressure drop.

5. Conclusion

In this research, different geometric parameters in microchannel cooling of electronic boards and electronic equipment subjected to high heat flux were studied. The numerical software used was ANSYS FLUENT 14.5 commercial CFD package. The maximum temperature of the bottom surface that is subjected to the heat flux was evaluated. Other parameters such the local heat transfer coefficient, the pressure drop, the maximum heat flux that the bottom surface can subject to it and the average heat transfer coefficient were also considered. The main conclusions were as follows:

- The optimum fin length that satisfied the best heat transfer criteria is 75% of the channel radius.
- Less than that fin length, when the fin length increases, the maximum temperature at the center of the bottom surface that occurs at the end of the channel decreases and heat transfer coefficient increases.
- More than that fin length, when the fin length increases, the maximum temperature at the center of the bottom surface that occurs at the end of the channel increases and heat transfer coefficient decreases.
- The total fins length is the major parameter, no matter how this length is distributed or how the fins are positioned.
- The fin length has a noticeable effect on the pressure drop; it has a proportional relation with the pressure drop.
References


