International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:8, No:4, 2014

Effect of Baffles on the Cooling of Electronic Components

O. Bendermel, C. Seladji, M. Khaouani

Abstract**—**In this work, we made a numerical study of the thermal and dynamic behavior of air in a horizontal channel with electronic components. The influence to use baffles on the profiles of velocity and temperature is discussed. The finite volume method and the algorithm Simple are used for solving the equations of conservation of mass, momentum and energy. The results found show that baffles improve heat transfer between the cooling air and electronic components. The velocity will increase from 3 times per rapport of the initial velocity.

Keywords**—**Electronic components, baffles, cooling.

I. Introduction

WITH the advent of high speed computers, the heat dissipation in the chip has become a source of concern. The high temperature of the chip affects reliability and system performance. Therefore, the effective heat removal is necessary to ensure a better life of the system. Several studies have been conducted for the thermal management of electronic devices.

A. Hamouche et al. [1] made a numerical study of heat transfer by mixed convection in a horizontal channel with two heat sources. They found that increasing the separation distance, the height and width of the electronic components improves the rate of heat removal, and therefore improves the heat transfer in the channel. Y.P. Cheng et al. [2] proposed a new algorithm CLEARER to simulate the conjugate heat transfer. The influence of the openings on the substrate, the height of the heat sources and their distribution along the beam is studied. The results show that different configurations can improve the heat transfer. Sung-Min Kim et al. [3] made an analytical study on the effects of heat diffusion in heat sinks for electronic cooling applications. Analytical models are built for sinks with microchannel with triangular sections, trapezoidal, rectangular and diamond-shaped. The analytical results are compared with numerical models for the same configurations. This study shows that the analytical models provide accurate predictions for Biot numbers of practical interest. M.R. Sohel et al. [4] studied the thermal performance of a copper heat sink using three types of nanofluids. The nanofluids Al_2O_3-water, TiO_3-water and CuO-water are used in this analysis. The hydraulic diameter of the circular duct is 400 microns and the block size is 100 × 10 × 4mm. The results show that the thermal performance can be increased significantly by using CuO-water as a coolant. M. Liu et al. [5] made an experimental study on a heat sink with a matrix of 625 micro-fins. Using deionized water as a coolant, the flow is designed with a Reynolds number of from 60 to 800. The experimental data shows that the pressure drop and the average Nusselt number increases with increasing Reynolds number. G. Xia et al. [6] studied numerically the behavior of water in a heat sink with helical cavities and internal ribs for the Reynolds number ranging from 150 to 600. This study also offers empirical correlations of the friction coefficient and the average Nussselt number depending on the Reynolds number and the relative size of the rib. The results show that the Nusselt number is 1.3-3 times higher than the rectangular microchannel, while the friction factor is 6.5 times. T.C Hung et al. [7] carried out a numerical analysis to examine the characteristics of heat transfer from a heat sink double layer. The effects of substrate materials, coolants and geometrical parameters such as the number of channels, the channel width, substrate thickness and thermal resistance are discussed. The predictions show that the heat transfer performance of the heat sink is improved for a system with substrate materials with a higher thermal conductivity. A coolant with a high thermal conductivity and low dynamic viscosity also improves the heat transfer performance. In addition, the thermal resistance of the heat sink can be minimized by optimizing the geometric parameters. Finally, the results show that for the same geometric dimensions, the thermal performance of double-layer heat sink is better than single layer, by an average of 6.3%.

In our work, we will perform a numerical study of the air convection for cooling of electronic components in a rectangular channel with transverse baffles. The finite volume method is used in solving the equations of conservations of mass, momentum and energy.

II. Mathematical Formulation

A. Geometry of the Problem

The physical model is shown in Fig. 1. Four heat sources made of copper are attached to a support made of Teflon. To enhance heat transfer, we used the baffles attached on the upper part of the channel.

Fig. 1 Geometry of the problem

- Length of the channel L=100mm.

O. Bendermel is with the Faculty of Technology, University of Tlemcen, Algeria (e-mail: bendermelothman@yahoo.fr).

C. Seladji and M. Khaouani are with is with the Faculty of Technology, University of Tlemcen, Algeria.

- Height of channel H= 7mm.
- Length of the heat sources a= 10mm.
- Height of the heat sources h= 3mm.
- Distance between the leading edge of first heat source and inlet of channel S_0= 5mm.
- Distance between heat sources S= 7mm.

B. Governing Equations

Continuity equation:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0$$

Momentum equations:

$$\frac{\partial(\rho uu)}{\partial x} + \frac{\partial(\rho vu)}{\partial y} = -\frac{\partial P}{\partial x} + \eta\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$
$$\frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho vv)}{\partial y} = -\frac{\partial P}{\partial y} + \eta\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

Energy equation:

$$\frac{\partial(\rho uT)}{\partial x} + \frac{\partial(\rho vT)}{\partial y} = \frac{\lambda_L}{c_p}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)$$

In the heat sources:

$$\lambda_H\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + q = 0$$

C. Boundary Conditions

At the inlet:

- The initial temperature T_0= 20°C.
- The initial velocity U_0= 0.5 m/s.

At the outlet:

- The outlet pressure $P=P_{atm}$.

The upper wall and the substrate are adiabatic.

III. Results and Discussions

The finite volume method, based on the algorithm SIMPLE, is used to solve the governing equations system with boundary conditions.

A. Choice of Mesh

Numerical experiments with mesh grids have been made to the number of nodes 5274, 6580 and 8266. Fig. 2 shows that the variation of temperature is very small for the position x = 44mm. In our study, we use the grid with 6580 nodes.

B. Contours of the Axial Velocity

Fig. 3 shows the contours of the axial velocity for two configurations: with and without baffles for initial velocity U_0 = 0.5m/s. For the configuration without baffles, there are areas of recirculation between heat sources, against the velocity increases in this section in the configuration with baffles. Then, the heat can be transported to the air by forced convection effectively.

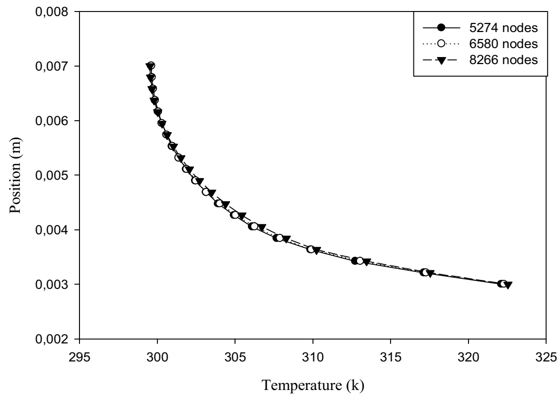
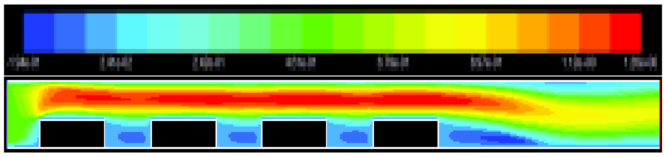
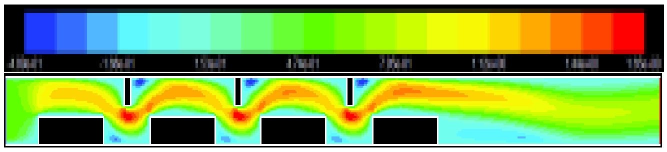


Fig. 2 Validation of the mesh

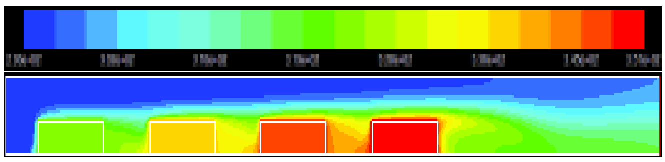


(a)

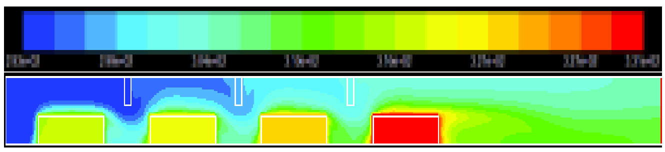


(b)

Fig. 3 Contours of axial velocity (a) without baffles, (b) with baffles



(a)



(b)

Fig. 4 Contours of temperature (a) without baffles, (b) with baffles

C. Contours of the Temperature

Fig. 4 shows the contours of temperature for both configurations for U_0= 0.5 m/s. The configuration with baffles led to a reduction of the maximum temperature of 20°C.

D. Velocity Profiles:

Fig. 5 shows the profiles of the axial velocity at x = 10 mm. There is a small difference between the velocity profiles for this section. For the other sections (Figs. 6, 7 and 8), the velocity becomes important for the configuration with baffles.

E. Temperature Profiles:

Fig. 9 shows the temperature profiles at x = 10mm (before the first chicane). There is a difference in the maximum temperature of 7°C. For the other sections, the configuration with baffles give high temperatures in the upper part of the channel, and at lower temperatures of the electronic components with respect to the configuration without baffles.

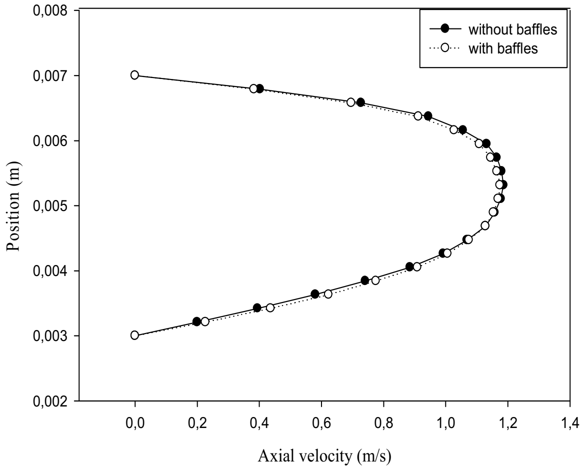


Fig. 5 Axial velocity profiles (x= 10 mm)

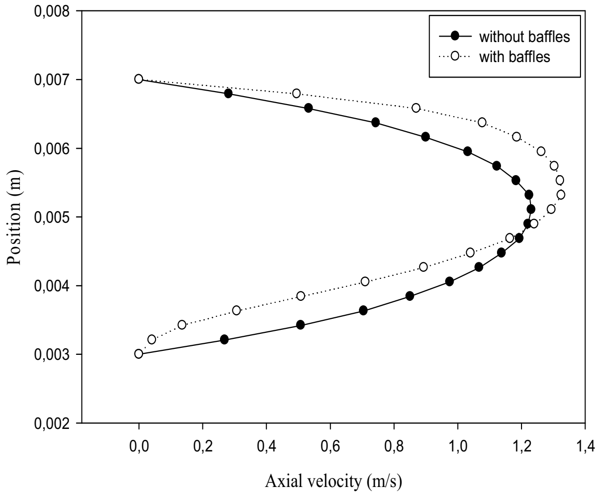


Fig. 6 Axial velocity profiles (x= 27 mm)

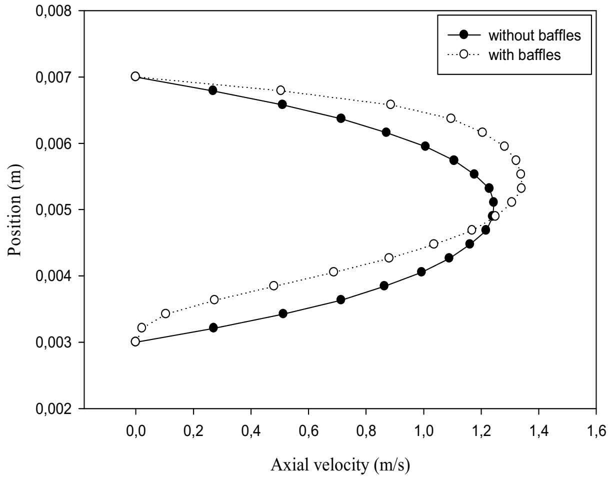


Fig. 7 Axial velocity profiles (x= 44 mm)

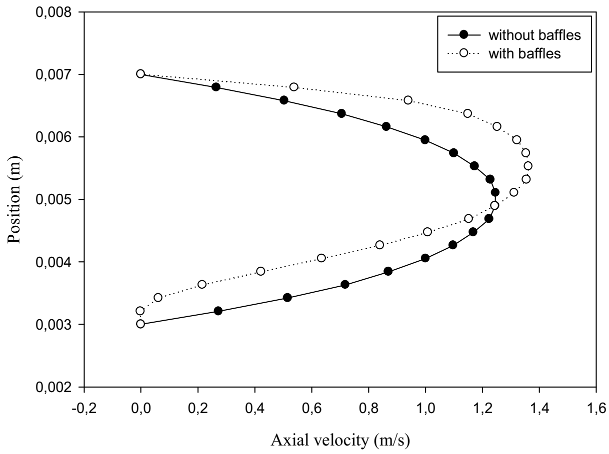


Fig. 8 Axial velocity profiles (x= 61 mm)

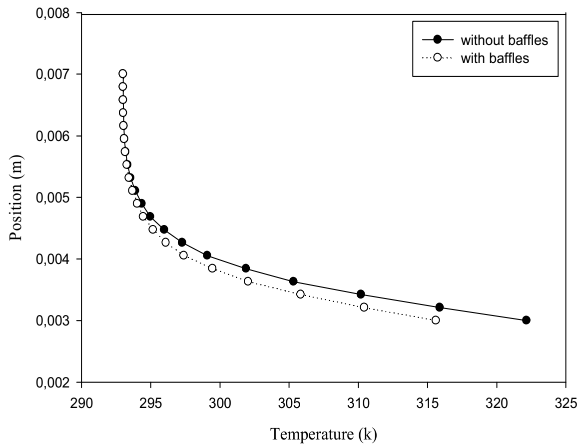


Fig. 9 Temperature profiles (x= 10 mm)

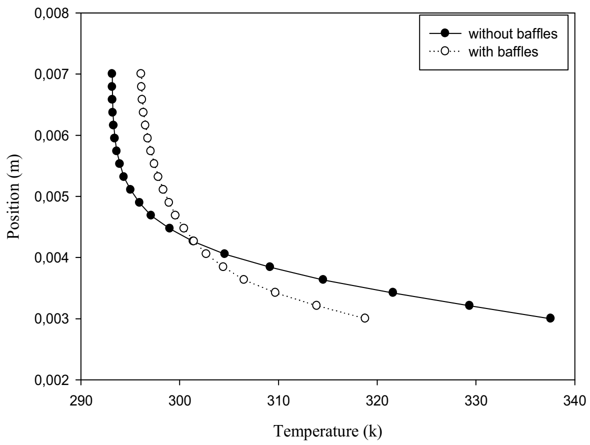


Fig. 10 Temperature profiles (x= 27 mm)

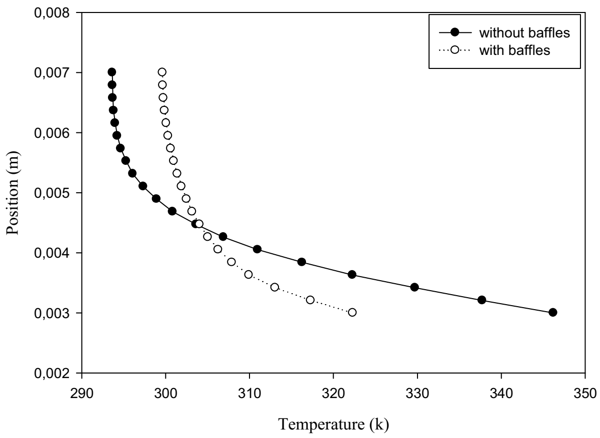


Fig. 11 Temperature profiles (x= 44 mm)

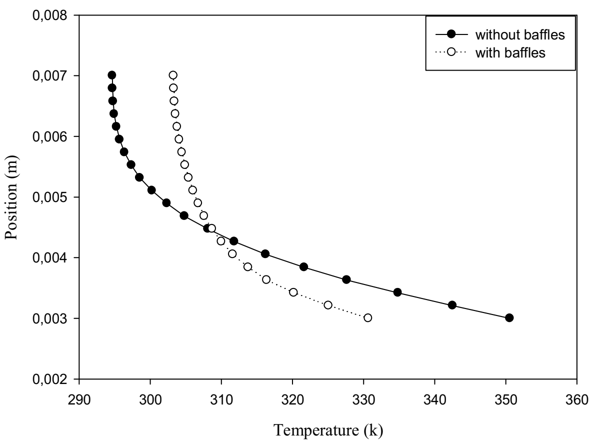


Fig. 12 Temperature profiles (x= 61 mm)

IV. Conclusion

In this work, we made a numerical study of the flow of air in a horizontal channel with baffles and electronic components. The finite volume method based on the SIMPLE algorithm is used for solving governing equations.

This study shows that the use of baffles in the top of the channel will improve the heat transfer between the air and electronic components. The velocity will increase from 3 times per rapport of the initial velocity.

This study can be extended for different initial velocity and different heights of baffles.

References

[1] A. Hamouche, R. Benssaih, "Mixed convection air cooling of protruding heat sources mounted in a horizontal channel", *International Communication in Heat and Mass Transfer*.36, 841-849 (2009).

[2] Y.P. Cheng, T.S. Lee, H.T. Low, " Numerical simulation of conjugate heat transfer in electronic cooling and analysis based on field synergy principle", *Applied Thermal Engineering*.28, 1826-1833 (2008).

[3] S.M. Kim, I. Mudawar, "Analytical heat diffusion models for different micro-channel heat sink cross-sectional geometries", *International Journal of Heat and Mass Transfer* .53, 4002–4016 (2010).

[4] M.R. Sohel, R. Saidur, Mohd Faizul Mohd Sabri, M. Kamalisarvestani, M.M. Elias, A. Ijam," Investigating the heat transfer performance and thermophysical properties of nanofluids in a circular micro-channel", *International Communications in Heat and Mass Transfer* .42, 75–81 (2013).

[5] M. Liu, D. Liu, S. Xu, Y. Chen, "Experimental study on liquid flow and heat transfer in micro square pin fin heat sink", *International Journal of Heat and Mass Transfer*.54, 5602–5611 (2011).

[6] G. Xia, Y. Zhai, Z. Cui, "Numerical investigation of thermal enhancement in a micro heat sink with fan-shaped reentrant cavities and internal ribs", *Applied Thermal Engineering*.58, 52-60 (2013).

[7] T.C. Hung, W.M. Yan, W.P. Lee, "Analysis of heat transfer characteristics of double-layered microchannel heat sink", *International Journal of Heat and Mass Transfer*.55, 3090–3099 (2012).