

# Computation of Thermal and Hydraulic Performances of Minichannel Heat Sink with an Impinging Air Jet for Computer Cooling

**M'hamed Beriache**

Université des sciences et technologies Lille1, Polytech'Lille, USTL, LML, UMR 8107, 59655 Villeneuve d'Ascq, France  
Université Hassiba Benbouali de Chlef, BP 151, Hay Essalem, Chlef (02000), Algérie; E-mail: m.beriache@gmail.com

**Hassan Naji**

Laboratoire Génie Civil et géo-Environnement (LGCgE- EA 4525), F-62400 Béthune, Université d'Artois, France Université Lille Nord de France, F-59000 Lille, France; E-mail: hassane.naji@univ-artois.fr

**Ahmed Bettahar, Leila Mokhtar Saïdia**

Département de mécanique, Université Hassiba Benbouali, BP 151, Hay Essalem, Chlef (02000), Algérie; E-mail: bettahara@yahoo.fr, ms\_layla80@yahoo.fr

---

*Abstract: This research work presents a numerical simulation of the CPU (Central Processor Unit) heat sink with a parallel plate fin and impingement air cooling. The governing equations are discretized by using the finite difference technique. The objective of this article is to investigate the thermal and hydraulic performances of a heat sink with impinging air flow to evaluate the possibility of improving heat sink performances. The numerical simulations are done by a personal C++ developed code. The thermal and hydraulic characterization of a heat sink under air-forced convection cooling condition is studied. The hydraulic and thermal parameters, including velocity profiles, the distribution of static pressure, pressure drop and temperature distributions through the fins, the base heat sink and the heat sink body through the heat sink are analyzed and presented schematically. The results show that the heat transferred by the heat sink increases with impinging Reynolds number. The performance of the proposed model computed by the numerical calculation is high compared with literature results.*

*Keywords: electronics cooling; heat sink; air flow; impingement jet; thermal resistance; pressure drop*

---

# 1 Introduction

As electronic equipment becomes smaller and more advanced, it necessitates higher circuit integration per unit area, which in turn contributes to a rapid increase in heat generation. Thus, the effective removal of heat dissipations and maintaining the chip at a safe operating temperature have played important roles in insuring a reliable operation of electronic components [1]. There are many methods for electronics cooling, such as jet impingement cooling and heat pipe [2]. Conventional electronics cooling has normally used an impinging jet with a heat sink, showing superiority in terms of unit price, weight and reliability. Therefore, the most common way to enhance air-cooling is through the utilization of an impingement air jet on a mini or micro channel heat sink. In order to design an effective heat sink, some criteria such as thermal resistance, a low pressure drop and a simple structure should be considered.

When the literature is surveyed, a number of scholars have examined the air jet impingement on the heat sink geometry. Hilbert *et al.* [3] reported a novel laminar flow heat sink with two sets of triangular or trapezoidal shaped fins on the two inclined faces or a base. This design is efficient because the downward flow increases the air speed near the base of the fins, where the fin temperatures are highest. By having the cool air enter at the center of the heat sink and exit at the sides, the length of the fins in the flow direction is reduced so that the heat transfer coefficient is increased. Jang and Kim [4] conducted an experimental study of a plate fin heat sink subject to an impinging air jet. The geometry of a heat sink in impingement flow is similar to that shown in Figure 1. In this flow arrangement, the air enters at the top and exits at the sides. Based on experimental results, a correlation for the pressure drop and a correlation for its thermal resistance are suggested. They show that the cooling performance of an optimized microchannel heat sink subject to an impinging jet is enhanced by about 21% compared to that of the optimized microchannel heat sink with a parallel flow. Saini and Webb [5] developed a numerical model for predicting the pressure drop and thermal performance for impingement air flow in plate fin heat sinks. The model was then validated by experiments. The predicted pressure drop is 31% lower than the experimental data. El-Sheikh and Garimella [6] experimentally investigated the heat transfer enhancement of air jet impingement by using pin-fin heat sinks. In their study, the heat transfer coefficient, for both pinned and unpinned heat sinks, is only modestly dependent on the nozzle-to-target plate spacing ( $H/d$ ). They also found that the heat transfer coefficient increases as the nozzle diameter decreases at a fixed flow rate. Biber [7] carried out a numerical study to determine the pressure drop of single isothermal channel with variable width impingement flow. Many different combinations of channel parameters are studied, and a correlation for the average static pressure loss coefficient across the slot jet and for average Nusselt number from an isothermal channel was presented. Duan and Muzychka [8] developed an impingement flow thermal resistance model. The simple model

is suitable for heat sink parametric design studies. The analytical model is developed for the low Reynolds number laminar flow and heat transfer in the inter fin channels of impingement flow plate fin heat sinks. The accuracy of the predicted thermal resistance was found to be within 20% of the experimental data at channel Reynolds numbers less than 1200. Duan and Muzychka [9] proposed a simple impingement flow pressure drop model based on developing laminar flow in a rectangular-channels heat sink. The validity of the model is verified by an experimental test. Measurements of the pressure drop were performed with heat sinks of various impingement inlet widths, fin spacing, fin heights and airflow velocities. It was found that the predictions agreed with experimental data within 20% at a channel Reynolds number less than 1200.

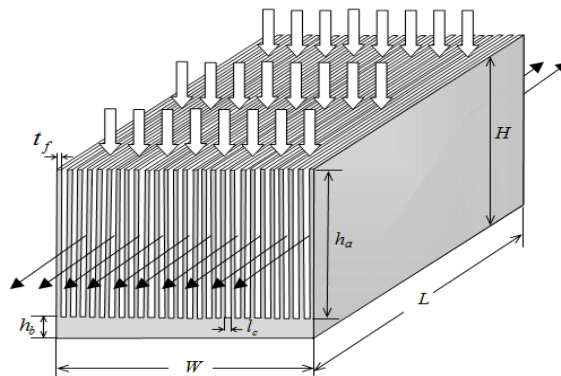


Figure 1

Schematic diagram showing the physical model

Duan and Muzychka [10] performed an experimental investigation of the thermal performance with four heat sinks of various impingement inlet widths, fin spacing, fin heights and airflow velocities. They developed a heat transfer model to predict the thermal performance of impingement air cooled plate fin heat sinks for design purposes. Shah *et al.* [11] demonstrated the results of a numerical analysis of the performance of an impingement heat sink designed for use with a specific blower as a single unit. The effects of the shape of the heat sink fins, particularly near the center of the heat sink, were examined. It was found that the removal of fin material from the central region of the heat sink enhances the thermal as well as the hydraulic performance of the sink. Shah *et al.* [12] extended the previous work by investigating the effect of the removal of fin material from the end fins, the total number of fins, and the reduction in the size of the hub fan. The reduction in the size of the hub of the fan is found to be a more uniform distribution of the air inside the heat sink, particularly near the center of the module. In general, there are many testing processes for heat sinks which must be introduced in an effort to obtain the thermal and hydraulic performance of heat sinks. If we take advantage of the numerical simulation to obtain some probable optimal design parameters before running experiments, the cost and research time can be reduced.

In the works cited above, some analytical and empirical models were developed and verified experimentally; others are based on CFD modeling, and they were mainly based on analyses of individual channels subjected to different boundary conditions. The literature survey has not revealed any articles that deal in detail with the flow field and heat transfer in heat sinks. In order to better reflect the flow field and temperature distribution in the channels and the body of the heat sink respectively, the present work complements the detailed flow field, pressure drop predictions and temperature distribution in the plate-fin heat sink with impingement air flow. An important focus of this study is to examine the capability of Navier-stokes equations treated by our developed code based on finite difference method (FDM) to effectively represent the fluid flow and heat transfer behavior in mini-channel heat sinks. In this paper, the numerical simulation of plate-fin heat sinks with confined impingement cooling in thermal-fluid characteristics will be investigated. The objective of this study is focused on the impingement flow plate-fin geometry. The research objectives are to develop a simple model for predicting thermal and hydraulic performances of a plate-fin heat sink for impingement air cooling.

## 2 Physical and Mathematical Model

### 2.1 Physical Model

The physical model of this study is illustrated in Fig. 1. In the configuration shown above, a heat sink with rectangular mini-channels with hydraulic diameter  $D_h=0.0029\text{ m}$  is heated from the bottom with the power ( $Q$ ) generated by the CPU, which is absorbed by the sink and released to the atmosphere through the fins topped by an axial fan that blows air in impingement flow to dissipate heat in the atmosphere with a constant flow rate  $\dot{V}$  ( $\text{m}^3/\text{s}$ ). The sink is made of aluminium of  $\lambda =237\text{W/m.K}$ . The power dissipation from the P4 CPU is set to 80W [13]. The following assumptions are made in order to model the heat transfer and fluid flow in the heat sink (2D fluid flow and 3D heat transfer) [14]: (1) steady state; (2) the fluid is assumed to be incompressible (Although air is a compressible fluid, for the range of flow velocities considered in this study, incompressible flow assumption is valid as long as the Mach number is smaller than 0.3 [15]); (3) laminar flow, because of the small fin spacing and low air flow rates; (4) constant fluid and solid properties; (5) negligible viscous dissipation; (6) negligible radiation heat transfer. The radiation effect is important in the natural convection system; it can easily be ignored when convection becomes a significant term. The dimensions of the heat sink considered in this work are listed in Table 1. Only a quarter of the heat sink was included in the computational domain, in view of the symmetry conditions.

Table 1  
Geometry of the heat sink used in the computation

$H(m)$	$h_a(m)$	$h_b(m)$	$L(m)$	$l_c(m)$	$N_f$	$t_f(m)$	$W(m)$
0.036	0.032	0.004	0.082	0.0015	27	0.001	0.066

## 2.2 Mathematical Model

The governing equations of continuity, momentum and energy are solved numerically using a finite difference scheme with boundary conditions as follows. The velocity is zero on all wall boundaries, and as flow is assumed to be hydrodynamically developed [9].

(1) Continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

(2) Momentum equation

▪ for the fluid flow:

$$\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$\rho \left( u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

▪ for the solid :

$$u = v = 0 \quad (4)$$

(3) Energy equation

▪ for the fluid:

$$\rho \cdot c_p \left( u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = \lambda_f \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (5)$$

▪ for the solid :

$$\lambda_s \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) = 0 \quad (6)$$

The assumption of constant physical properties for the solid and the air allows decoupling the hydrodynamic equations of heat equations. Thus, solving the energy equation may be conducted after the convergence of computing hydrodynamics. In addition, the energy equation is linear; it converges faster than the Navier-Stokes. The equations were solved using a C++ developed code.

The developed code is based upon the finite differences approach. The discretized equations are solved iteratively using the Gauss-Seidel method.

The Reynolds number of the impingement jet is defined as:

$$\text{Re} = \frac{|V_0| \cdot D_h}{\nu} \quad (7)$$

$$\text{Where } D_h = \frac{4A}{P_m} = \frac{4 \cdot h_a \cdot l_c}{2h_a + l_c} \quad (8)$$

The average convection heat transfer coefficient  $h$  is calculated by

$$h = \frac{Q}{A_h(T_s - T_a)} \quad (9)$$

The average Nusselt number  $Nu$  is calculated by

$$Nu = \frac{h \cdot D_h}{k_a} \quad (10)$$

For very small distances from the rectangular duct inlet, the effect of curvature on the boundary layer development is negligible. Thus, it should approach the classical isothermal flat plate solution for developing flow in the entrance region of rectangular ducts ( $L^* \rightarrow 0$ ) [10].

$$L^* = (L/2)/(D_h \text{ Re Pr}) \quad (11)$$

$$Nu = 0.664 \text{ Re}^{1/2} \text{ Pr}^{1/3} \quad (12)$$

Assuming  $\text{Pr}=0.707$  for air, [16].

The thermal resistance of the heat sink is calculated by

$$R_{th} = \frac{T_{\max} - T_{\infty}}{Q} \quad (13)$$

Note that the boundary conditions of this problem are stated in Figures 2 and 3.

This enables us to set properties in the solid and the fluid regions appropriately and to solve the conjugate conduction convection problem [16]. The iterations

were terminated when the residuals for the continuity, momentum equations were 1% of the characteristic flow rate and were 0.1% for the energy, the solution is converged.

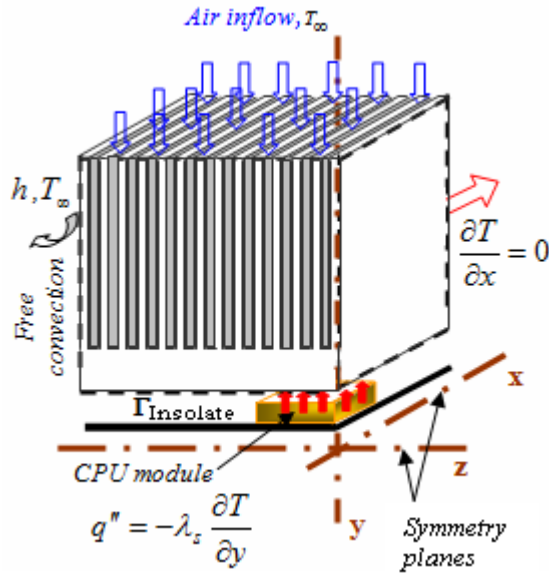


Figure 2  
Thermal boundary conditions

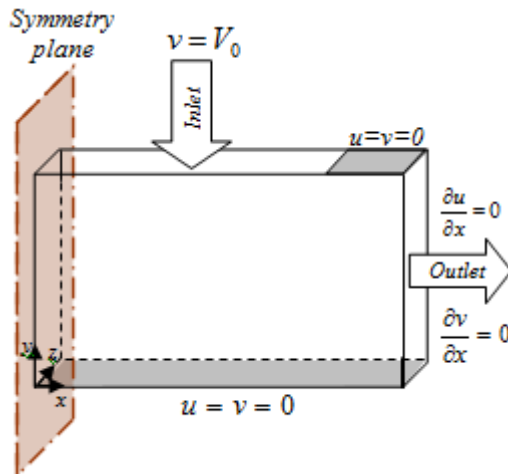


Figure 3  
Hydrodynamics boundary conditions

### 3 Results and Discussion

The model is validated with the numerical results taken on the studied heat sink with the operating parameters listed in Table 2.

Table 2  
Operating parameters of the model, [13]

Parameter	Value	Description
$V_0$	5 m/s	inlet velocity (m/s)
$h$	10 W/m <sup>2</sup> .K	Natural convection Coefficient (both sides of heat sink), [17]
$T_\infty$	27 °C	ambient temperature.
$Q$	80 W	power of heat source (W)
$A_{Die}$	14x16 mm <sup>2</sup>	Die Surface of CPU.
$A_{CPU}$	31x 31 mm <sup>2</sup>	Surface of CPU.

Figure 4 shows the distribution of static pressure through the channel of the heat sink (XY plane). The static pressure increases gradually up to the stagnation zone at the center where the velocity is the lowest, then decreases in the direction of the air outlet. A low pressure drop is located at the exit channel where the velocity is highest. The numerical results (Figures 4 and 5) show that the pressure drops lie in a range of 5-62 Pascal, which is not so large. It indicates proper fin spacing. This is an advantage that improves heat transfer with less energy pumping. The numerical results of the pressure drop are compared with experiment data for a similar geometry and flow velocities for impingement flow in the laminar regime obtained by Duan and Muzychka, [9] and Saini and Webb [5] as shown in Figure 5. Overall, the trend is very good. The results are in good agreement in view of the simplicity of the model.

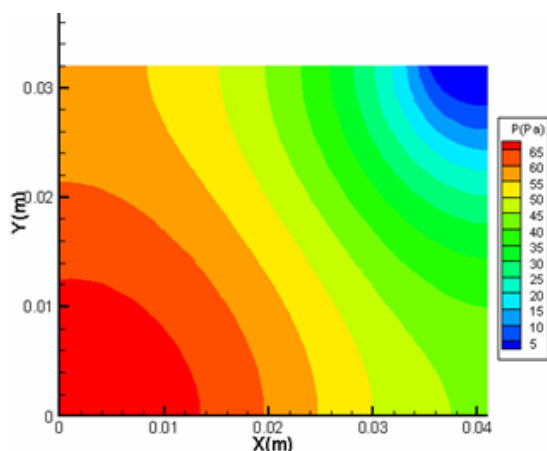


Figure 4

Static pressure contours through the channel



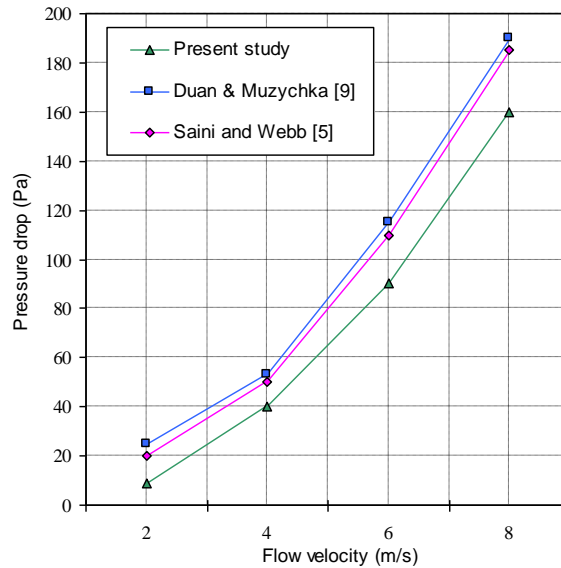


Figure 5

Pressure drop through the mini-channel as a function of cooling velocity ( $V_0$ )

Figure 6 shows the average velocity field in the channel of the heat sink. The results show clearly that the velocity slows down by going to the base until a breakpoint in the center. This velocity increases rapidly in the direction of the exit channel and reaches higher values than those at the entrance; it is the boundary layer phenomenon. These conclusions are in excellent agreement with those of Biber [7].

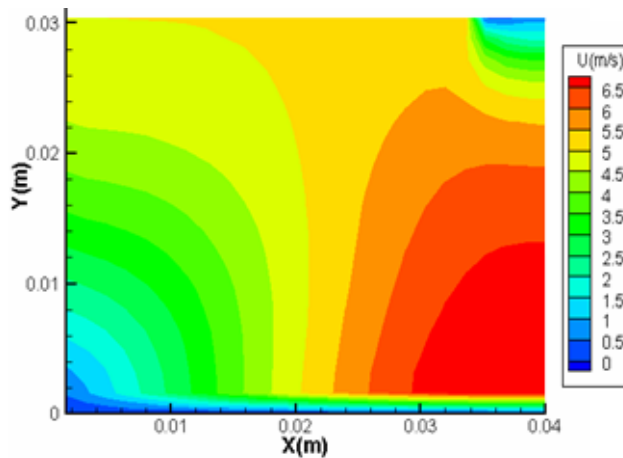


Figure 6

Velocity contours through the channel

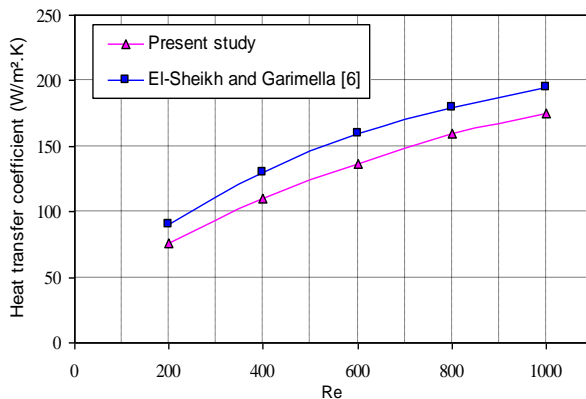


Figure 7

Variation of convective heat coefficient as a function of cooling velocity ( $V_0$ )

From Figure 7, the heat transfer coefficient is seen to increase as the jet velocity increases. Consequently, heat dissipation from heat sinks may be enhanced by increasing the cooling velocity. The obtained results are in good agreement with those of El-Sheikh and Garimella [6]. On the other hand, increasing jet velocity affects negatively the pressure drop and consequently flow by pass results.

The results of the temperature profile in Figures 8, 9 and 10 show a peak ( $T_{max}$ ) at the heat source, which is obvious, but also a high temperature gradient in the center of the sink, where it should be a low airflow following, while the corner points have a minimum temperature. Figure 10 depicts the contours of temperature in the sink at the central cross-section. The maximum temperature is always observed in the heat source; this temperature under the mentioned conditions is about 46°C, which is less than the temperature restricted by manufacturer.

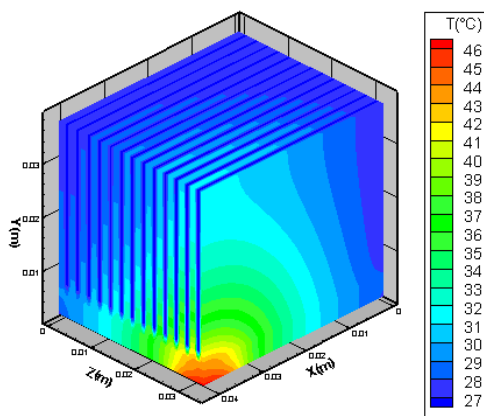


Figure 8

Temperature distribution through the heat sink, (3D)

The temperature distribution through the base of the sink is presented in Figure 9. The results are obtained by the developed code. The hottest spot ( $T_{max}=46^{\circ}C$ ) is located in the center of the base of the sink, as well as the pressure drop. This is due primarily to the presence of the heat source at this level, and as well because the cooler fluid has the lowest traffic there (stagnation zone).

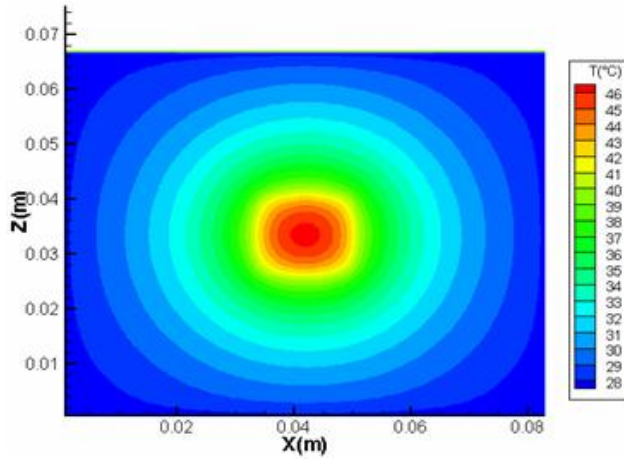


Figure 9

Contours of temperature distribution on the base of the heat sink

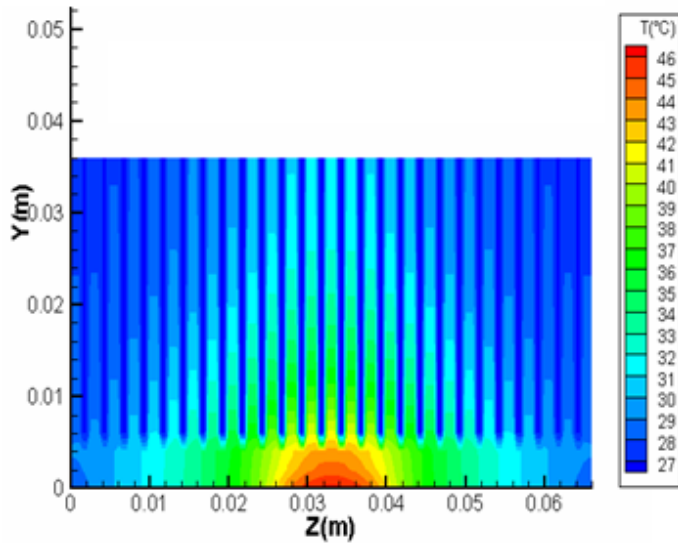


Figure 10

Temperature distribution in the sink at the central cross-section

In addition, heat is dissipated in both longitudinal and transverse directions, where it appears that the length of fin beyond a limit of 38 mm does not contribute to a decrease in temperature, and so fin length can be reduced. However, as regards width, the temperature can be dissipated through greater width. These conclusions, drawn in present work, are found to be in good agreement with conclusions drawn by Shah *et al.* [11] and Jang and Kim [4]. The cooling performance of the minichannel heat sink subject to an impinging jet can be evaluated in terms of the thermal resistance [4]. The temperature difference source-air is about 19 °C, so the thermal resistance  $R_{th}$  is calculated from Eq. (13) based on numerical results, it is of 0.2375 °C/W under fixed operating parameters of the model summarized in Table 2.

Figure 11 shows that thermal resistance is decreased if the flow velocity is increased. To check the validity of the results for thermal resistance of the minichannel heat sink subject to impinging jet, a comparison is made with Saini and Webb's [5] experimental data, and Duan and Muzychka's [9] analytical model. Overall, the trend is very good.

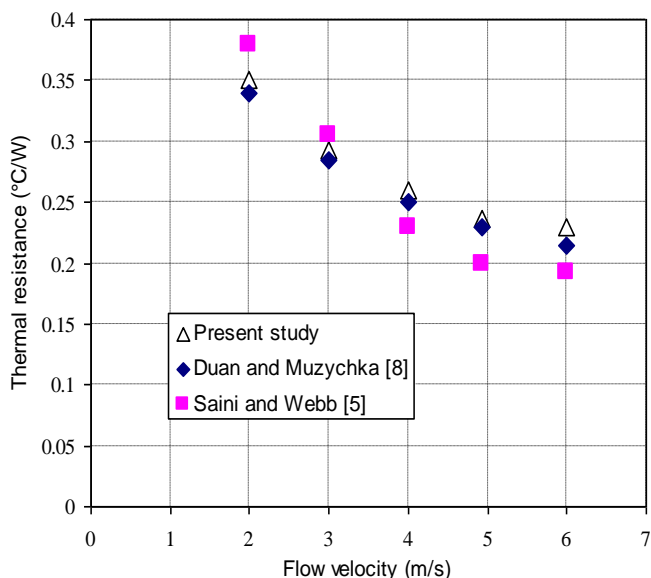


Figure 11  
Thermal resistance comparison

The obtained numerical results revealed a detailed picture of the flow and temperature distributions in the entire domain. They have provided a larger amount of data than an experiment can generally provide. Therefore, these results also show that there is potential for optimizing the geometric design, specifically for improving the low heat transfer and flow within the stagnation zone, as mentioned above.

## Conclusion

In this research work, thermal and hydraulic characterization have been carried out on a parallel plate heat sink, considering real operating parameters. The developed code was validated with available results, and the obtained results show that impingement air flow on mini-channel heat sink appears to have advantages for heat transfer. Thermal performances numerically carried out show that the chip delivers up to 80 W, which is adequately cooled by the studied heat sink as recommended by the manufacturer. The increase in the Reynolds number improves the convective heat transfer coefficient, which provides better thermal performance. The results also show that there is potential for optimizing the geometric design. This study should serve designers involved in the cooling of electronic components. In other words, it can be used to select or design heat sinks for effective thermal management in electronic assemblies. The obtained results have been favourably compared with available results in literature.

## Nomenclature

$A$	- section of the channel (m <sup>2</sup> )
$A_h$	- heat transfer area (m <sup>2</sup> )
$C_p$	- specific heat coefficient (J/kg K)
$D_h$	- hydraulic diameter (m)
$h$	- convective heat transfer coefficient (w/m <sup>2</sup> .K)
$h_b$	- height of the base (m)
$H$	- height of a minichannel fin (m)
$h_a$	- height of a minichannel fin (m)
$\lambda$	- thermal conductivity of a aluminium (W/m.K)
$K_a$	- thermal conductivity of air (W/m.K)
$L$	- length of the base of a minichannel heat sink (m)
$l_c$	- channel width (m)
$L^*$	- dimensionless thermal developing flow length
$\dot{m}$	- mass flow rate (kg/s)
$N_f$	- number of fins
$n$	- number of channels
$Nu$	- Nusselt number
$p$	- pressure (Pa)
$Q$	- power of heat source (W)
$q''$	- heat flux (W/m <sup>2</sup> )
$Re$	- Reynolds number
$R_{th}$	- thermal resistance (°C/W)
$t_f$	- fin thickness (m)
$T$	- temperature (°C)
$u$	- velocity in the x-direction (m/s)
$V$	- velocity in the y-direction (m/s)

$V_0$	- inlet velocity (m/s)
$\dot{V}$	- air flow rate (m <sup>3</sup> /s).
$W$	- width of the base (m)
$w_c$	- channel width (m)
$\rho$	- air density (kg/m <sup>3</sup> )
$\mu$	- dynamic viscosity (Ns/m <sup>2</sup> )
$x, y, z$	- space variables
$\Delta x, \Delta y$	- cell widths

### Acknowledgment

The authors are thankful to the Algerian ministry of higher education and research for providing financial support to carry out this research.

### References

- [1] Y. T. Yang and H.S. Peng, "Numerical Study of Pin-Fin Heat Sink with Un-Uniform Fin Height Design", *Int. J. Heat and Mass Transfer*, 2008, Vol. 51, pp. 4788-4796
- [2] K. Nishino, M. Samada, K. Kasuya, and K. Torii, "Turbulence Statistics in the Stagnation Region of an Axisymmetric Impingement Jet Flow", *Int. J. Heat Fluid Flow*, 1996, Vol. 17, pp. 193-201
- [3] C. Hilbert, S. Sommerfeldt, O. Gupta and D. J. Herrell, "High Performance Micro-Channel Air Cooling", *Proceedings of the 6<sup>th</sup> Annual IEEE Semiconductor Thermal and Temperature Measurement Symposium*, Feb. 6-8, Scottsdale, Arizona, 1990, pp. 108-113
- [4] S. P. Jang and S. J. Kim, "Fluid Flow and Thermal Characteristics of a Microchannel Heat Sink Subject to an Impinging Air Jet", *J. Heat Transf.*, 2005, Vol. 127, pp. 770-777
- [5] M. Saini and R.L. Webb, "Validation of Models for Air-cooled Plane fin Heat Sinks Used in Computer Cooling", *Proc. 8<sup>th</sup> Intersoc. Conf. Therm. Thermomech. Phenom. Electronic Syst.*, (ICTTPES'02), Pennsylvania State Univ., University Park, PA, USA, 2002, pp. 243-250
- [6] H. A. El-Sheikh and S. V. Garimella, "Enhancement of Air Impingement Heat Transfer Using Pin-Fin Heat Sinks", *IEEE Trans. Compon. Pack. Technol*, 2000, Vol. 23, No. 2, pp. 300-328
- [7] C. R. Biber, "Pressure Drop and Heat Transfer in an Isothermal Channel with Impinging Flow", *IEEE Transac. On Comp. and Packag. Tech. – Part A*, 1997, Vol. 20, No. 4, pp. 458-462
- [8] Z. P. Duan, and Y. S. Muzychka, "Impingement Air-cooled Plate Fin Heat Sinks Part II-Thermal Resistance Model", *Proceedings of 9<sup>th</sup> Int. Soc. Conf. Therm. Phenom. Electronic Syst.*, 2004, pp. 436-443

- 
- [9] Z. P. Duan, and Y. S. Muzychka, "Impingement Air-cooled Plate Fin Heat Sinks Part I-Pressure Drop Model", Proceedings of 9<sup>th</sup> Int. Soc. Conf. Therm. Phenom. Electronic Syst., 2004, pp. 429-435
- [10] Z. P. Duan and Y. S. Muzychka, "Experimental Investigation of Heat Transfer in Impingement Air-cooled Plate Fin Heat Sinks", J. Electron. Packag, 2006, Vol. 128, pp. 412-418
- [11] A. Shah, B. G. Sammakia, K. Srihari and K. Ramakrishna, "A Numerical Study of the Thermal Performance of an Impingement Heat Sink Fin Shape Optimisation", IEEE Trans. Compon. Packag. Technol., 2004, Vol. 27, No. 4, pp. 710-717
- [12] A. Shah, B. G. Sammakia, K. Srihari and K. Ramakrishna, "Optimization Study for a Parallel Plate Fin Impingement Heat Sink", J. Electron. Packag., 2006, Vol. 128, pp. 311-318
- [13] Intel® Corporation, Intel® Pentium® 4 processor on 90 nm process thermal and mechanical design guidelines, Design Guide, 2004
- [14] P. S. Lee and S. V. Garimella,; Thermally Developing Flow and Heat Transfer in Rectangular Microchannels of Different Aspect Ratios, Proc. Int. J. Heat Mass Transf., 2006, Vol. 49, pp. 3060-3067
- [15] R. L. Panton,; Incompressible Flow. John Wiley and Sons, 1984
- [16] S. V. Patankar,; Numerical Heat Transfer and Fluid Flow. Washington: Hemisphere, 1980
- [17] J. P. Holman,; Heat Transfer. 8<sup>th</sup> SI Metric Edition. New York: McGraw-Hill Book Co., 1996
- [18] Builder C++,; User's Guide. Borland Software Corporation, 2002