A NOVEL HYBRID MICROCHANNEL HEAT EXCHANGER CONFIGURATION AIMING FOR UNIFORM WALL TEMPERATURE

M. Omri [1], J. Barrau [2], S. Riera [2], L.G. Fréchette [1]
[1]Université de Sherbrooke, Dépt génie mécanique, 2500 boul. Université, QC J1K 2R1, Canada
[2]University of Lleida, Solar Energy Group, Edifici CREA, C/Pere de Cabrera, s/n, 25001 Lleida, Spain

Abstract: In this work, a numerical calculation and analysis of flows and heat transfer within a proposed novel heat exchanger are presented. This new configuration consists of a central impinging jet followed by rows of micro-channels of decreasing diameter. This design aims to make the wall temperature uniform by reducing the local thermal resistance along the flow path to compensate for the increase in flow temperature. Calculations of the flow and heat transfer in a 3D model were done as a function of flow rate with a uniform wall heat flux. The mean temperature on the active wall decreases from the inlet to the outlet, showing the effectiveness of the proposed approach. The jet impingement maintains low temperatures near the inlet area. The velocity distributions in the micro-channels indicate non uniformities due to the outlet flow from one row perturbing the inlet flow to the next. Due to the discrete nature of the layout, the wall temperature profile oscillated within a range of about 11°C.

Keywords: CFD, micro-channel, jet impingement, cooling, microfluidics

INTRODUCTION

For proper operation of concentrated photovoltaic cells and microelectronics, the device temperature should be uniform and thermal gradients limited. With the increasing size of solar cell arrays and complexity of multi-chip modules, managing the heat load over a large area is increasingly challenging.

Although micro-channel heat exchangers allow the removal of high heat rates [1], the increase in coolant flow temperature along the heat exchanger will typically lead to an increase in wall temperature. To make the temperature more uniform, fresh coolant can be provided at multiple locations across the surface, but with the added complexity of distributing the flows with manifolds. An alternate approach, presented here, consists of locally adapting the thermal resistance of the heat sink to compensate for the coolant flow temperature increase as it travels over the surface. In this work, a combination of jet impingement and variable diameter micro-channels will be studied to locally adapt the thermal resistance.

Agostini et al. [2] compared the performances of jet impingement and micro-channels technologies. Steinke et al. [3] indicate that micro-channels cause a large increase in temperature along the path of the fluid flow and are also responsible for a relatively large pressure drop. Also the jet impingement requires the use of arrays of jets to ensure a relatively good level of temperature uniformity across the base of the heat sink. But the interaction between adjacent jets reduces the local heat transfer coefficient. To remedy to this problem Aldabbagh et al. [4] propose the use of complex geometries that allow the extraction of the spent flow from the jets. A solution to lower the drawbacks of both technologies consists of using a hybrid jet impingement and micro-channel scheme [5, 6]. The approach proposed here consists of tailoring the local heat transfer coefficient by using impacting jets at the fluid inlet and variable diameter micro-channels along the flow path to compensate for the increase in flow temperature. This concept was initially proposed by Barrau et al. [7] and successfully demonstrated in millimeter-scale channels [8, 9]. Their measurements demonstrated the capacity of the proposed heat sink to achieve temperature profiles of the target object that may even decrease in the direction of the coolant flow, while maintaining the high heat removal capabilities inherent to micro-channel and jet impingement technologies [8].

This work presents a novel heat exchanger configuration that tends to make the wall temperature uniform even under uniform heat flux conditions and increasing flow temperature. First, the analytical design methodology is presented, followed by numerical CFD calculations to validate the design and better understand what defines the temperature distribution.

MICRO HEAT EXCHANGER DESIGN

The heat exchanger layout is shown in Fig. 2. The inlet flow forms a jet that impacts the center line of the heat exchanger. The coolant then flows laterally from both sides of the jet through five rows of microchannels. To increase the thermal performance of the heat sink, an optimization procedure of the hybrid cooling scheme has been developed. The main objective of this procedure is to improve the temperature uniformity of the cooled object, while maintaining the thermal resistance levels achieved in the previous work [8, 9].

The first step is to set the operating parameters such as T_{fi} (Inlet fluid temperature, K), q'' (dissipation rate W/m^2), Q (Flow rate m^3/s), T_s (Wall temperature K) and S (area of the heat sink m^2) for which the optimisation procedure will be applied. The geometric design parameters will then be calculated, including the number of rows (N) and the microchannel width (W) and fins thickness (W_f) for each row.

Since temperature uniformity is the objective, T_s is considered constant along the whole flow path. The outlet temperature of the fluid is defined by energy conservation across each row for a uniform heat flux as:

$$T_{f_{out},i} = T_{f_{in},i} + q'' \cdot S_i \,/\, (\rho \cdot C_p \cdot Q) \qquad (1)$$

The local heat transfer coefficients of each section, h_i, for i=1..N, are then calculated from the desired wall temperature:

$$h_i = q'' / (T_s - T_{f_i}) \qquad (2)$$

Although a more uniform temperature distribution would be obtained with an infinite number of sections, five sections on each side of the jet will be chosen here to limit the geometric complexity and the effect of entrance length on the local heat transfer distribution.

The previous experimental and numerical studies [8, 9], made at millimetre and micrometric scales, provide data for h as a function of hydraulic diameter, d_h, accounting for entrance effects (Fig. 1). The following correlation is fit to this data, for the operating conditions in Table 1:

$$d_h = 185416 h^{-1.88} \qquad (3)$$

The height of the micro-channels will be considered constant along the whole flow path, and taken to be 300 μm for microfabrication considerations. This implies that the width of the channels en each section, W_i, can be calculated from the hydraulic diameter definition for open rectangular sections:

$$W_i = 2 \cdot H \cdot d_{h_i} / (4 \cdot H - d_{h_i}) \qquad (4)$$

Similarly, the minimum fin thickness, W_f, will be set to 100 μm for good heat conduction and ease of micro-fabrication. The chosen parameters are summarized in Table 1 and the calculated geometry in Table 2 and Fig. 2.

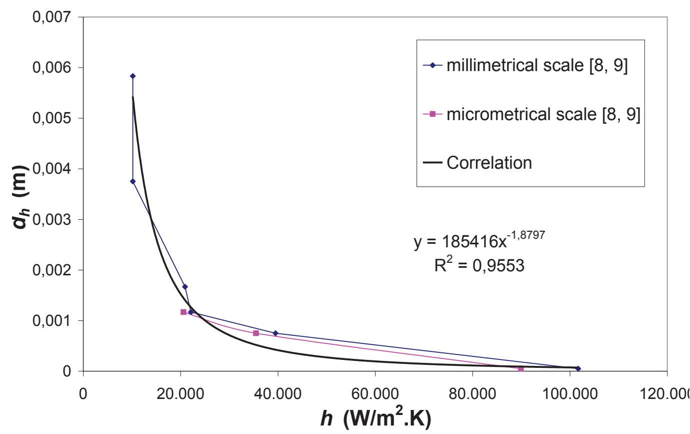

Fig. 1: Relation between hydraulic diameter and local heat transfer coefficient: q''= 50 W/cm^2, Q= 8.73.10^{-6} m^3/s.

Table 1: Setup conditions of the optimisation procedure

	Parameter	Unit	Value
Setup of the optimisation case	Tf_{in}	K	293
	q''	W/cm^2	50
	Q	cm^3/s	8,73
	T_s	K	313
	S	cm^2	5
Design constraints	N	-	5
	H	μm	300
	W_f	μm	100

Table2: Results of the optimisation procedure (Setup conditions of table 1)

Parameter	Unit	Values S_1 (i=1)	S_2 (i=2)	S_3 (i=3)	S_4 (i=4)	S_5 (i=5)
Tf_i	K	20	22,9	25,1	26,8	28,2
h_i	W/m^2.K	25000	29303	33605	37908	42210
dh_i	μm	1003	744	575	459	375
W_i	μm	1528	490	276	186	136

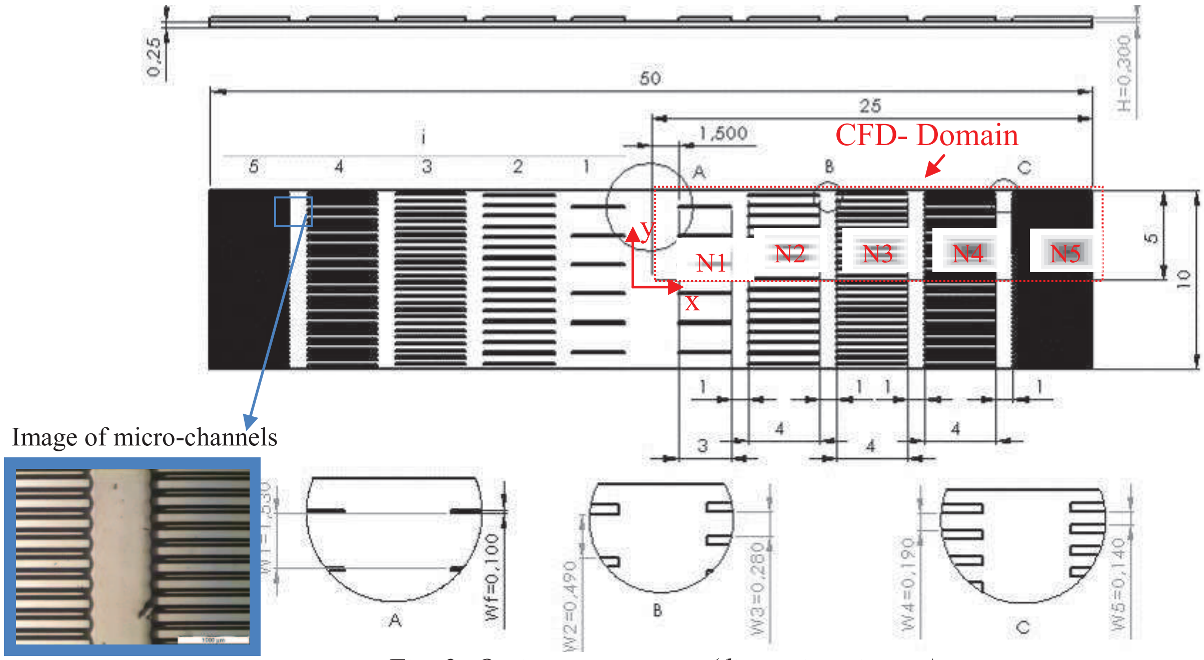

Fig. 2: Optimum geometry (dimensions in mm)

MODEL DESCRIPTION

To predict the wall temperature of the proposed heat exchanger under a uniform heat flux, a three-dimensional conjugate heat transfer model us used. One quarter of the device is modeled given the symmetry (Fig. 2). It contains five rows (N1 to N5) of micro-channels that are identical in each row, with dimensions shown in Fig. 2. The solid material is aluminum and a uniform heat flux is applied on the active wall (50 W/cm^2). The inlet temperature is 293 K and two different mass flow rates will be used: case 1 with 2.19 g/s and case 2 with 0.871 g/s. The Reynolds number based on the hydraulic diameter varies from 200 to 700, suggesting that the entry lengths will not be negligible.

The Navier-Stokes equations are solved in the fluid domain with the assumption of laminar flow. The energy equation is solved in the fluid and solid domains. The coupled partial differential equations describing the flow field are discretized with the finite volume method, using a commercial solver (Fluent Inc.). Second order upwind discretization is used. The rectangular staggered grid is non-uniform in both directions within the fluid domain: it is finer near the walls where gradients are more important. First, a grid independence study was done for each micro-channel to obtain the minimum required grid, and after a validation of the whole domain is done, for a total number of seven millions nodes used. Convergence is declared when the cumulative residuals for all conservation equations are less than at least 10^{-9}.

RESULTS AND DISCUSSION

This section describes the results from the 3D CFD calculation. Overall, the flow in the micro-channels becomes hydrodynamically developed and is well distributed among the channels in a row.

Figure 3 indicates the developed velocity profiles of all the micro-channels for the two different mass flow rates. The velocity gradients at the walls are larger for the smaller channels, suggesting that the heat transfer coefficient will increase along the flow. For the highest flow rate (case 1), irregularities in the velocity profiles can be observed. This is mainly due to the transverse position of the micro-channels with respect to each other: wakes from one row of micro-channels will perturb the inlet flow to the next. The relatively flat velocity profiles are due to the channel height being smaller than its width (N1, N2).

The temperature distribution is indicated in the figure 4, taken in the same plane as Figure 3. The water at low temperature (jet impact) enters the first row of micro-channels where the heat transfer is localized within the boundary layers of the fins. In the first row, the temperature difference between the mean flow temperature and the mean fin temperature is important (19 °C). This difference decreases along the flow path and becomes less than 5°C in the last row. This justifies the need for higher *h* and/or wetted area near the outlet to accommodate the uniform heat flux with lower temperature difference.

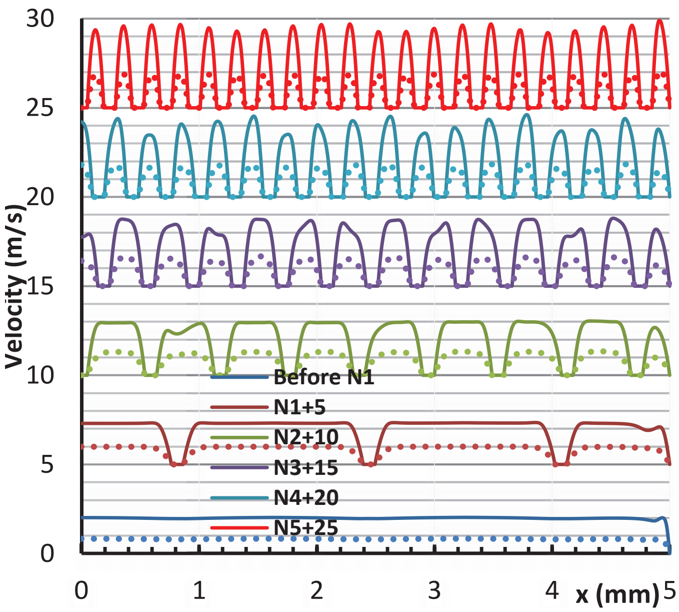

Fig. 3. Velocity profiles of the michrochannels at the mid length (ligne is case 1 and dashed is case 2).

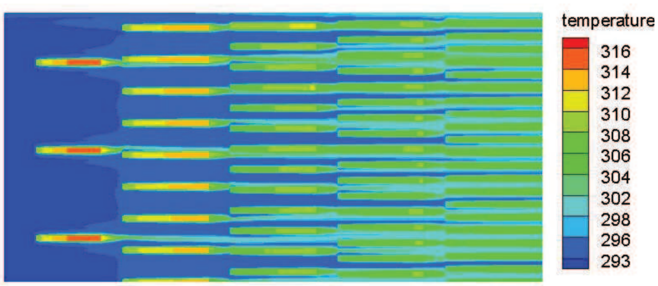

Fig. 4. Temperature contours of the flow adjacent to the active wall (case 1)

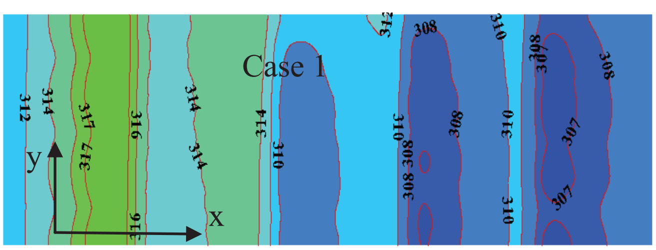

Fig. 5. Temperature contours on the active wall (case 1).

Figure 5 indicates the temperature contours on the active wall. The temperature variation in the transverse direction (along y) is negligible and does not exceed 1°C; this variation is a consequence of the velocity distribution in the micro-channels which presented some irregularities as indicated in Fig. 3. The variation of the temperature band is essentially along the x axis where the impact of the jet is seen (312 K zone in Fig. 5) followed by the impact of micro-channels rows. The maximum variation in this direction (x) does not exceed 11°C.

Figure 6 shows the wall temperature profiles from the impinging jet to the outlet, for the two cases. The mean temperature of each profile does not increase along the flow direction indicating the ability of the proposed geometry to maintain a quasi-uniform temperature on the active wall. The slight decrease suggests that the heat exchanger configuration was somewhat overdesigned.

By changing the inlet mass flow (case 1 to case 2), the temperature profile is essentially shifted; this validates the idea to consider any mass flow inlet to design the heat exchanger. On this figure, locations of the micro-channel rows are indicated by the labels and vertical lines. At *x*=0 (symmetry axis of the jet), the temperature is significantly reduced (312K in case 1 and 323 K in case 2), showing the effectiveness of the impinging jet. After (x=0.0015 m), the flow enters the first row of micro-channels where the temperature increases gradually, but then decrease at the outlet of the row (0.0045≤x≤0.0055m). At the inlet of the second row of micro-channels, the heat transfer coefficient and the wetted surface are substantially larger. This locally reduces the thermal resistance and the local wall temperature. This makes the temperature at the outlet of the first row greater than that at the inlet of the next row. Consequently, the decrease of wall temperature between rows is natural in this case.

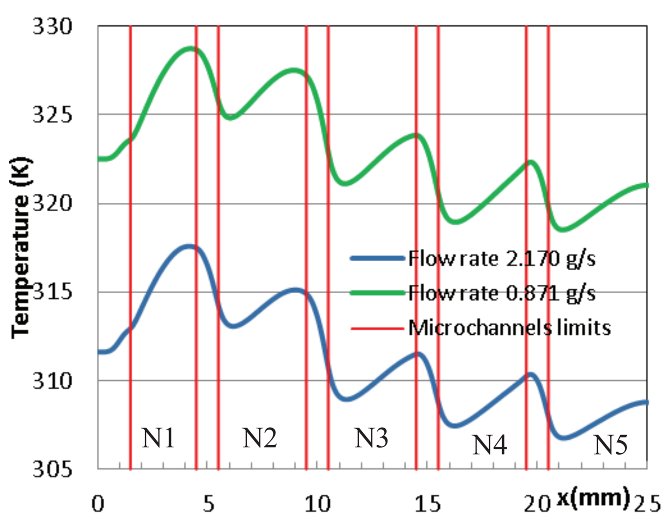

Fig. 6. Temperature profiles at the midline of the active wall

The main difference between the two flow rates studied is the variation in average flow temperature. For example, the outlet average flow temperature is 301 K for the high flow rate and 312 for the lowest case, for an inlet temperature of 293. Even though the exit temperature changes by more than 10 K between the two cases, the heat exchanger configuration is able to maintain the wall temperature variation in the same range. This shows robustness of the approach.

To improve the wall temperature uniformity, the trend in average wall temperature first needs to be flat. To do so, the reduction in hydraulic diameter from row to row should be less aggressive. Then, the oscillations in temperature from row to row should be reduced by increasing the number of rows, while respecting fabrication constraints.

CONCLUSIONS

This paper presented a novel heat exchanger and a numerical analysis of the laminar flow through this device. Numerical calculations using Fluent were done with two different mass flow rates. The maximum temperature variation across the surface is 11°C, mainly due to high heat transfer coefficients at the entrance of the micro-channels. The velocity profiles in each micro-channel of a given row are not identical due to the non-uniform flow coming from the previous row and the relative transverse positioning of the micro-channels. This created a maximum variation of 1°C of the active wall temperature in the transverse direction. The principal variation of the temperature on the active wall (max 11°C) is in the direction of the flow. Also, the transverse mean temperature profile on this surface decreases along the flow direction, indicating the success in gradually increasing the wetted surface and the heat transfer coefficient to compensate for the increase in flow temperature. The geometry presented here could be improved by using the CFD calculation in the design process since it gives a better estimate of the local heat transfer coefficient.

References

[1] M. G. Khan and A.Fartaj, A review on microchannel heat exchangers and potential applications, Int. J. Energy Res. 35 (2011) 553–582

[2] B. Agostini, M. Fabbri, J.E. Park, L. Wojtan, J.R. Thome, B. Michel, State of the art of high heat flux cooling technologies, H. Transfer Engineering 28 (2007) 258-281.

[3] M.E. Steinke, S.G. Kandlikar, Single-phase liquid heat transfer in plain and enhanced micro-channels, International Journal of Thermal Sciences 45 (2006) 1073-1083.

[4] L.B.Y. Aldabbagh, I. Sezai, Three-dimensional numerical simulation of an array of impinging laminar square jets with spent fluid removal, International Journal of Thermal Sciences 43 (2004/3) 241-247.

[5] Mudawar, I., 2001, "Assessment of high-heat-flux thermal management schemes", IEEE Trans.Components and Packaging Tech. 24 122-141.

[6] M.K. Sung, I. Mudawar, Experimental and numerical investigation of single phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme, Int. J. of Heat and Mass Transfer 49 (2006) 682-694

[7] J. Barrau, J.I. Rosell, M. Ibañez, Spain Patent P24787ES00, 2009.

[8] J. Barrau, D. Chemisana, J. Rosell, L. Tadrist, M. Ibañez. An Experimental Study of a New Hybrid Jet Impingement/micro-channel Cooling Scheme. Applied Thermal Engineering 30 (2010) 2058-2066

[9] J. Barrau, M. Omri, D. Chemisana, J. Rosell, M. Ibañez, L. Tadrist.. Numerical study of a hybrid jet impingement/micro-channel cooling scheme. Applied Thermal Engineering, 33–34 (2012) Pages 237-245.