Liquid cooling of a microprocessor: experimentation and simulation of a sub-millimeter channel heat exchanger

Julien Fontaine\textsuperscript{a}, Charles Gonzalez\textsuperscript{a, b}, Prashant Kumar\textsuperscript{b}, François Pigache\textsuperscript{a}, Pascal Lavieille\textsuperscript{a}, Frédéric Topin\textsuperscript{b}, and Marc Miscevic\textsuperscript{a}

\textsuperscript{a}University of Toulouse, LAPLACE (Laboratory on Plasma and Conversion of Energy), Toulouse, France; \textsuperscript{b}Aix-Marseille Université, CNRS, IUSTI, Marseille, France

ABSTRACT
A heat exchanger dedicated to the cooling of a microprocessor has been designed and realized. It consists of a bottom wall in contact with the processor and a cover that has been dug to a depth of 200 μm on one side and 1 mm on the other. Thus, by turning the cover, the hydraulic diameter of the channel can be changed. Both hydraulic and thermal performances of this heat exchanger have been experimentally tested. Three-dimensional numerical simulations were simultaneously carried out and good agreement was obtained. The influence of the distributor and the collector on the distribution of fluid flow and heat fluxes is emphasized. A new concept of micro-heat exchanger is proposed for the cooling of electronics devices for which wall to fluid heat exchange quality and pumping effect are critical. The ability of a liquid heat exchanger involving a dynamic deformation of one of its walls to cool a microprocessor is investigated. Three-dimensional transient numerical simulations of fluid flow and conjugate heat transfer were performed using commercial software. Effect of geometrical and actuation parameters has been explored, demonstrating the ability of such heat exchanger to simultaneously pump the fluid and enhance the heat transfer.

Introduction

The evacuation of the heat generated within a microprocessor is a crucial problem. It affects the user from various points of view: limitation of performance and maximum allowable temperature of the environment in which it can be used, drastic reduction in reliability and lifetime, energy consumption of the chip. It is therefore necessary to develop efficient cooling solutions even for high heat generation microprocessors and to keep it near its optimum operating temperature. Numerous works have been conducted to propose new techniques to enhance heat transfer and improve the existing ones [1]. For application in high dissipative electronics, air cooling appears to be increasingly inappropriate due to low thermal conductivity as well as low density and low heat capacity of this fluid. Thus, liquid or two-phase thermal management systems must be developed. Furthermore, in addition to the rapid increase in the power density, electronic packages are more and more miniaturized, implying to develop efficient cooling systems. Several cooling solutions can then be envisaged. Among these solutions, microchannel-based heat exchanger is often presented as one of the promising cooling technologies [2–8]. For example, very recently Kheirabadi and Groulx [8] have shown a device whose thermal resistance is varying from 0.105 K/W to 0.08 K/W when the coolant (water) flow rate varies from 1.1 l/min to 3.8 l/min in a microchannel heat sink in which channels were 0.5 mm wide and 2.3 mm tall.

Yang et al. [9] have proposed a general optimization process for the thermo-hydraulic performances of minichannel heat sinks. They reported empirical correlations extracted from literature and gave some recommendations about their use for practical design. Although many works have already been carried out, thermo-hydraulic behavior and performances of such miniature heat sinks are still to be explored [10, 11].

Among these microchannel cooling systems, those equipped with corrugated channels appear particularly interesting. Heat transfer and pressure drop in sinusoidal corrugated channels have been studied by Nishimura et al. [12]. These authors showed that transition between laminar and turbulent flows is obtained...
at lower Reynolds number (Re = 300) compared to straight channel due to unsteady vortex motion. As for flat channel, the friction factor is inversely proportional to Reynolds number in the laminar flow range, while it is independent of Reynolds number in the turbulent one. Extending this study, Nishimura et al. [13] studied flow patterns characteristics in symmetrical two-dimensional sinusoidal and arc-shaped corrugated channels at moderate Reynolds numbers (20 < Re < 300). They concluded that the transitional Reynolds number depends on the corrugation shape and is lower for the arc-shaped wall.

Niceno and Nobile [14] have conducted an extended numerical study on these corrugated channels. They found that these two corrugation shapes are ineffective in terms of heat transfer rate (slightly higher for sinusoidal compared to arc-shaped channel) as compared to flat channel in steady low Reynolds flow. The unsteady regimes appear at different Reynolds numbers for the two corrugation shapes: unsteady regime was observed at Re = 60 – 80 for arc-shaped channel and at Re = 175 – 200 for sinusoidal one. On the other hand, heat transfer rate increases significantly for both corrugation shapes, up to a factor of three, as a result of self-sustained oscillations. Moreover, this transfer rate was found to be higher for arc-shaped channel while the friction factor was smaller for sinusoidal channel.

Naphon [15] conducted a numerical study on various corrugation geometries arranged in in-phase and out-phase layouts (e.g., flat plate, arc-shaped, trapezoidal, and V-shaped) to enhance the thermal performances. He obtained that for a given air flow rate, V-shaped corrugated channel enhances most heat transfer. This enhancement is linked to boundary layer disruption. Most of the studies revealed an increase in overall thermal performance from 3 to 5 times depending on the working fluid. However, static corrugated channels have shown to increase significantly the pressure drop. In such corrugated (and grooved) minichannels, high heat transfer usually leads to high Reynolds number (mostly turbulent) flow.

Yang et al. [16, 17] compared three different designs of a micro-heat exchanger for use in a liquid cooling system. Best thermal performances were obtained with the chevron channel heat exchanger. Unfortunately, this heat exchanger was also the one generating the highest pressure drops.
On the other hand, Léal et al. [18] proposed a relatively simple model of dynamic corrugated minichannel to be operated at low Reynolds number. This has been achieved by deforming dynamically one of the channel walls. Very high heat transfer has been obtained compared to static corrugated channel (with a relative gain in heat transfer coefficient of about 400%). It appeared to be the first study to introduce dynamic wall in such minichannel heat exchanger.

Very recently, Kumar et al. [19] extended this work and proposed a realistic 3D geometry of dynamic corrugated heat exchanger. In their study, the average height of the channel is fixed while the minimum gap (distance between the lowest point of the deformed wall and the fixed wall) varies as a function of the relative amplitude. Numerical studies have been conducted for a channel outlet pressure greater than the inlet one and for an imposed frequency of 50 Hz. They showed that heat transfer coefficient is proportional to wave amplitude and that such devices possess self-pumping capacity. This system can operate with negligible pressure drop, or even without an external pump.

The aim of this article is to determine the ability of this type of actuated liquid heat exchanger to cool a micro-electronic device. First, a reference (static) heat exchanger has been studied, from both experimental and numerical point of view. A systematic numerical study was then conducted considering a micro-heat exchanger using three actuators to deform dynamically one of its walls to understand the impact of operating parameters in enhancing thermo-hydraulic characteristics.

Reference heat exchanger

Experimental setup

The reference exchanger consists of two parts, the sole made of copper and the cover made of aluminum. Two grooves are machined in the sole and communicate with the inlet and outlet pipes (see Figure 1). An O-ring is placed at the periphery for sealing. The cover is a simple parallelepiped plate that has been dug to a depth of 200 $\mu$m on one face and 1 mm on the other. Thus, by turning the cover, the hydraulic diameter of the channel can be changed. The channel obtained for the circulation of the fluid after assembly is thus rectangular with a cross-section of 50 mm in width, 38 mm in length (distance between the distributor and the collector), and 200 $\mu$m or 1 mm in height (Figure 2).

In order to be able to determine the thermal performance independently of the thermal contact resistance between the exchanger and the microprocessor, three thermocouples were inserted into the sole at a distance of 1 mm from the surface in contact with the liquid. These thermocouples are K-type, sheathed in stainless steel, 500 $\mu$m in diameter. The holes in the sole were made by electro-erosion; they have a diameter of 600 $\mu$m and a length of 33 mm. Two other identical thermocouples are placed in inlet and outlet pipes along with pressure taps (diameter 4 mm) in order to measure the inlet and outlet bulk temperatures of the fluid. All thermocouples have been calibrated against a certified Pt100 RTD probe (accuracy of 0.05 °C). Thus, the relative accuracy of temperature measurements is estimated at 0.1 °C. A differential pressure sensor is positioned between the inlet and outlet pipes of the heat exchanger (the connections are located 25 cm upstream and downstream of the heat exchanger) and allows measuring the pressure loss with an accuracy better than 10 Pa.

The heat source is constituted by a microprocessor mockup supplied by Intel Company and fixed to the sole. This mockup is geometrically identical to a commercially available processor and generates similar and easily scalable thermal solicitations.

The heat exchanger is connected to a hydraulic circuit (see schematic in Figure 3). A counter-current tubular heat exchanger is used as cold source for the fluid loop. The fluid then passes through a Coriolis flowmeter allowing mass flow rate measurements in the range 0 up to 6 g/s with an accuracy of 0.02 g/s. We thus obtained a maximum Reynolds number of 240 within the heat exchanger for both considered thicknesses (hydraulic diameter, respectively, equal to
1960 µm and 398 µm). A variable speed gear pump is used for discharging the fluid into a precision valve. This latter allows us to vary the pressure drop in the circuit at the desired value in a simple manner. The fluid then flows toward the inlet of the heat exchanger prototype. A reservoir (open to the atmosphere) is connected to the hydraulic circuit and serves as an expansion vessel. The secondary fluid in this heat exchanger is thermostated water.

**Numerical simulation**

Simultaneously, direct numerical simulations of this heat exchanger were carried out using commercial software Star-CCM+. The virtual prototype was constructed using the CAD model of the assembly (bottom wall, top cover, and channel). The conjugate heat transfer problem was solved in the transient regime, and heat conduction is considered in all solid parts. Note that a perfect thermal contact between top and bottom parts of the assembly was considered. For the channel as well as inlet and outlet manifolds, incompressible laminar and transient flow and heat transfer were solved using the classical combination of continuity, momentum, and energy equations (here simply written for constant properties):

\[
\nabla \cdot \mathbf{u} = 0
\]

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = \rho g - \nabla P + \mu \mathbf{\Delta u}
\]

\[
\rho C_p \left( \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \rho \mathbf{u} \cdot \frac{\partial \mathbf{u}}{\partial t} + \frac{\rho}{2} (\nabla ||\mathbf{u}||^2) \cdot \mathbf{u} = k_f \Delta T + \frac{\partial P}{\partial t} + \theta
\]

where \( \theta \) is the volumetric energy source due to viscous dissipations. This term is taken into account during all calculations; however, in the present work, it is negligible compared to the wall heat transfer.

The physical properties of the fluid were considered to vary with the temperature (water using the models of IAPWS-IF97 (International Association for the Properties of Water and Steam, Industrial Formulation 1997), while the solid part properties are supposed constant (copper for the base and aluminum for the top cover). The numerical resolution used a segregated approach with implicit second-order temporal discretization and ad hoc relaxation factors in order to obtain adequate convergence behavior. 1,691,272 and 1,852,205 meshes were used for the 1-mm and 200-µm heat exchanger, respectively. For both cases, an unstructured trimmer – octree mesh type – is used in solid parts while structured hexahedral anisotropic cells are used in the channel with 16 up to 20 cells along the height of the channel. This mesh is
refined in the vicinity of all solid–fluid interfaces and near inlet and outlet of the channel. Mesh convergence effects were checked, and no additional improvement is obtained with further refinement.

Both mass flow rate and fluid temperature were imposed at the inlet of the heat exchanger; outlet pressure is imposed at the atmospheric one. A uniform heat flux is imposed on the bottom wall of the sole, on an area of $30 \times 30 \text{mm}^2$ as in the experiments. The rest of the solid walls are considered adiabatic (as experimental results show that thermal losses are negligible).

**Numerical actuated prototype**

In the studied configuration, the lower wall was fixed and subjected to a constant heat flux along the imprint of heating zone while the dynamically deformed upper wall (also called “membrane” in the following) and side walls were kept adiabatic. Several actuators were employed to dynamically deform the upper wall of a rectangular cross-section microchannel in order to simultaneously enhance heat transfer and pump the fluid.

The length of the actuated zone ($L$) is 49 mm while width ($W$) is 35 mm. The channel gap ($g$) is fixed and is 10 $\mu$m while the average height of the channel ($\delta$) is linked to the wave amplitude employed for wall deformation and the channel gap. The lengths of input and outlet zones have been chosen long enough to establish the hydrodynamic. Furthermore, the no-slip condition was imposed on the walls.

The influences of relative amplitude ($A_0$), frequency ($f_r$), and wavelength ($\lambda$) on thermo-hydraulic performances are studied for an imposed value of the pressure difference between outlet and inlet of the exchanger equal to zero.

Only three actuators were employed in the present study. The channel cross-section shape is presented in Figure 4 where the actuator positions can be described using $x_1$, $x_2$, $x_3$, $x_4$, $x_5$, and $x_7$. The positions $x_0$ and $x_7$ are fixed to external inlet and outlet zones as dampers. These small zones make the fluid to enter and leave in laminar condition. The microchannel geometry is composed of four main parts: inlet zone, heated surface where the heat power is imposed, outlet zone, and membrane. This simplified geometrical configuration, dedicated to study the impact of actuators’ operating parameters (three actuator case), does include sidewall damping effects on the lateral side of the actuated zone. Along the main flow axis, the membrane displacement is linearly interpolated between actuators pair and between actuator and fixed zone.

Three-dimensional numerical simulations were carried out using the commercial software Star-CCM+. This software allows modeling fluid flow and heat transfer in dynamically deformed structures. The flow was considered transient, three dimensional, and laminar. The working fluid was liquid water (properties taken from IAPWS-97 at 27°C) whose all thermo-physical properties were supposed constant for systematic studies. The imposed heat power on the imprint (heated zone) was 65 W (note that only half of the device is modeled due to symmetry).

The results presented focus on the “active zone” namely between $S_1$ and $S_2$ sections (see Figure 4). All representative global thermo-hydraulic quantities were calculated in this zone.

The incompressible laminar and transient conjugate flow and heat transfer problems were solved as for the static heat exchanger. The numerical resolution used a segregated approach with implicit second-order temporal discretization and ad hoc relaxations factors in order to obtain adequate convergence behavior. The displacement of the top membrane is created by:

1. fixing it in inlet and outlet zone and displacing the upper wall at chosen height $\delta$ (average height of the channel),
2. adding the movement of the three actuators (Eq. 4),

![Figure 4. Cross-section of dynamically corrugated microchannel. The lower wall is subjected to constant heat flux between $x_1$ and $x_7$, and the gap $g$ is the minimum height of the channel. Upper profiles are drawn for two different amplitudes and same gap.](image)
3. in between two actuators or one actuator and border, the displacement is simply interpolated.

The displacement of each actuator is given by the following function $Act_i$ for $i = 1, 2, 3$ and numbered sequentially along main flow axis:

$$Act_i (x, t) = A_i \cdot \cos (2\pi f_i t + \Phi_i)$$

where $\Phi_1 = 0^\circ$, $\Phi_2$, $\Phi_3$ are phase of actuators’ solicitations.

In the studied configuration, the vertical plane passing through the central axis of the channel is a plane of symmetry, so only half of the channel was simulated (i.e., $W/2 = 17.5$ mm). The imposed power on the modeled half-imprint was thus 32.5 W to carry out numerical simulations (see Figures 4 and 5). This allows us to reduce the number of mesh cells to optimize computation time by dividing the simulated configuration by two compared to its actual size. Note that the fluid flow was considered in $x$ direction while the width and height of the channel were considered in $y$ and $z$ directions, respectively.

Meshing strategy was similar to the one used by Kumar et al. [19]. It was rather tricky to create a convenient mesh in such highly anisotropic geometrical configuration subjected to large amplitude deformations (up to 98% of channel height) while keeping a computational time low enough to carry out parametric study without sacrificing precision or convergence of the dynamic transfer.

The deformed channel mesh is presented in Figure 5 in case of an actuator size of 11 mm and a phase shift between two consecutive actuators of $120^\circ$ ($\Phi = 0^\circ, -120^\circ, -240^\circ$).

In case of dynamic corrugated channel, surface-averaged and volume-averaged thermo-hydraulic quantities for all operating configurations were calculated as follows. The Reynolds number and hydraulic diameter are defined by:

$$Re = \frac{\rho |\vec{u}| D_h}{\mu}, \quad D_h = \frac{1}{k} \int_0^L \frac{4S}{p} d\lambda$$

where $S$ is the flow fluid passage area along the heated zone and $p$ is the perimeter of the passage section, respectively. Note that these quantities are averaged over a time period.

Following the method used in Léal et al. [18], the heat transfer coefficient is obtained by the spatial and temporal averaged fluid temperature and wall temperature over the width and between sections $S_1$ and $S_2$ (separated by a distance $L$) as well as over one period ($\tau = 1/f_r$):

$$\bar{T}_w = \frac{1}{\tau \omega WL} \int_t^{t+\tau} \int_{S_1}^{S_2} T_w(x, y, t) dx \ dy dt$$

$$\bar{T}_{mf} = \frac{1}{\tau \omega WL} \int_t^{t+\tau} \int_{S_1}^{S_2} \int_0^{L} T_{mf}(x, y, z, t) dx \ dy dz dt$$

In the following, the global heat transfer coefficient and Nusselt number across the channel (between $S_1$ and $S_2$) are defined as follows:
When NTS value as a reference value for both these quantities. In order to obtain an optimized NTS value, we performed for amplitude $A_0 = 85 \mu m$ and frequency $f = 20$ Hz. We initialized both corrugated shape and fluid flow. Then, calculations were carried out until a periodic stationary regime is reached. This latter point is checked comparing the temporal evolutions of global heat transfer and flow characteristics: calculations were performed until the heat transfer coefficient values between two consecutive time periods differ by less than 1%. Consequently, an additional time period is added to extract all instantaneous and time-averaged values of all physical quantities.

From Table 1, it can be seen that reducing the characteristic mesh size from 3 to 2 mm does not impact strongly on the global properties. The percentage differences between the properties are, respectively: 0.14% for $m$, 0.0024% for $T_W$, 0.006% for $T_{mf}$, 0.47% for $<h>$, 0.0095% for $T_f$, and 0.0095% for $T_{fo}$. On increasing the mesh size, these quantities vary significantly while reducing it further do not lead to any additional improvement. Based on these observations and the percentage difference in the properties, we chose the BMS (base mesh size) equal to 2 mm to perform the systematic studies.

To perform the numerical simulations in terms of time effectiveness as well as the precision, it is very important to balance the time step duration in order to ensure a sufficient accuracy and an optimal tracking of prescribed actuators movement without reaching prohibitive computational time. We test the following number of time steps per period (NTS): 20, 50, 75, and 100. Based on these time steps, global characteristics have been presented in Table 2. It can be observed that heat transfer coefficient values decrease with increase in NTS. On the other hand, the wall to fluid temperature difference $\Delta Tm = T_W - T_{mf}$ increases with increase in NTS.

In order to obtain an optimized NTS value, we have calculated the percentage variation of $<h>$ and $\Delta Tm$ for different time steps based on the lowest value as a reference value for both these quantities. When NTS = 50, we observed a variation of $<h>$ equal to 2.84% while the variation in $\Delta Tm$ is 6.14%.

The global results of different characteristics do not vary strongly when increasing NTS. Moreover, higher value of NTS increases the computational time while the properties vary within a difference of 3%. Hence, NTS = 50 has been chosen to study the impact of amplitudes and frequencies on pumping and heat transfer performances.

### Hydraulic results

#### Hydraulic behavior of the reference heat exchanger

A first series of experimental tests and numerical simulations were carried out without powering the processor, at a temperature of $20^\circ$C, in order to characterize the heat exchanger from hydraulic point of view. The pressure losses obtained as a function of the mass flow rate are reported in Figure 6 for the two considered channel thicknesses ($200 \mu m$ and $1 mm$). Globally, experiments and numerical simulations exhibit the same trend. Nevertheless, significant discrepancies are found between experimental and numerical results, especially for the 1-mm channel. These discrepancies can be explained by the fact that the pressure losses in the numerical simulations are determined between the inlet and the outlet of the heat exchanger, while in the experiments the tapings for the pressure measurement are located at $25 \text{cm}$ upstream and $25 \text{cm}$ downstream of the prototype. Thus, additional pressure losses are taken into account in the experiments (regular pressure drop in the inlet and outlet tubes, as well as singular pressure drops in the connections between the tubes of the loop and the heat exchanger).

Furthermore, in the 1-mm channel heat exchanger, the flow distribution appears less homogeneous than in the $200-\mu m$ channel heat exchanger, as it is shown in Figures 7 and 8, respectively. Such a flow maldistribution variation according to the channel hydraulic diameter is not surprising: it has widely been highlighted. For example, Kheirabadi and Groulx [4] numerically calculated variation of the mean velocity

### Table 1. Presentation of global thermal and flow properties for different mesh sizes for number of time step per period NTS = 50.

| BMS  | $m$ (g/s) | $T_W$ (K) | $T_{mf}$ (K) | $<h>$ (W/m$^2$K) | $T_f$ (K) | $T_{fo}$ (K) |
|------|-----------|------------|--------------|------------------|-----------|-------------|
| 3 mm | 1.11      | 312.51     | 307.87       | 13829            | 301.1     | 313.8       |
| 2 mm | 1.10      | 312.51     | 307.85       | 13765            | 301.1     | 313.8       |

### Table 2. Presentation of global thermal and flow properties for different time steps for a characteristic mesh size BMS = 2 mm.

| TS   | $m$ (g/s) | $T_W$ (K) | $T_{mf}$ (K) | $<h>$ (W/m$^2$K) | $T_f$ (K) | $T_{fo}$ (K) |
|------|-----------|------------|--------------|------------------|-----------|-------------|
| 20   | 1.14      | 311.7      | 307.3        | 14610            | 300.9     | 312.6       |
| 50   | 1.10      | 312.5      | 307.9        | 13765            | 301.1     | 313.8       |
| 75   | 1.10      | 312.7      | 308.0        | 13615            | 301.2     | 314.1       |
| 100  | 1.06      | 312.8      | 308.0        | 13384            | 301.2     | 314.2       |
Figure 6. Experimental and numerical pressure losses as a function of the mass flow rate for the two channel thicknesses.

Figure 7. Calculated streamlines in the 1-mm channel heat exchanger in adiabatic configuration. The liquid temperature is 20°C. Color online (Blue: 0 – Red: 0.1 m/s).

Figure 8. Calculated streamlines in the 200-μm channel heat exchanger in adiabatic configuration. The liquid temperature is 20°C. Color online (Blue: 0 – Red: 0.25 m/s).
from a channel to another in an array of parallel microchannels connected to a single manifold at the inlet and at the outlet. Variation up to 52% was highlighted for the 1 mm microchannel width, while this variation was only 16% for the 0.25 mm microchannel width. In the present study, the flow distribution in the 1-mm channel heat exchanger could be homogenized by optimizing the distributor and collector headers as suggested, for instance, by Saeed and Kim [20].

From these figures, it can be speculated that the main part of the pressure drop in the 200-\(\mu\)m channel heat exchanger is the regular pressure drop in the channel itself, while in the 1 mm the main part of the pressure drop corresponds to the pressure drop in the singularities and in the manifolds.

To assess the respective contributions of the channel and the singularities on the total pressure drop, an analytical estimation can be done. Assuming a Poiseuille flow in the inlet and outlet tubes, as well as in the channel, the pressure drop can be calculated as follows:

\[
\Delta p = \frac{64}{R_e D_t} \frac{1}{2} \rho U_i^2 + \frac{64}{R_e D_h} \frac{1}{2} \rho U_c^2 + \sum_i \xi_i \frac{1}{2} \rho U_c^2
\]

with:

\[R_e D_t = \frac{\rho U_i D_t}{\mu}, \quad R_e D_h = \frac{\rho U_c D_h}{\mu} \quad \text{with} \quad D_h = \frac{4e l}{2(e + l)} \approx 2e\]

The last term in Eq. (10) represents the pressure drop due to all the singularities. The value of \(\sum_i \xi_i\) is adjusted to reproduce the experimental pressure drop of the 1-mm channel heat exchanger: \(\sum_i \xi_i \approx 5\) (see Figure 9).

The global pressure losses of the device equipped with the 200-\(\mu\)m channel heat exchanger are 2 to 2.5 times higher than those measured with the one using the 1-mm channel heat exchanger, while the hydraulic diameter is 5 times lower. This behavior can be explained by analyzing the part of the regular pressure losses in comparison with the singular pressure losses corresponding to the different changes in flow direction and cross-sections. These relative contributions are shown in Figure 10. For the 1-mm channel heat exchanger, the pressure losses in the actual channel are negligible, less than 2% of the total pressure losses. For the 200-\(\mu\)m channel, these pressure losses in the channel become preponderant and represent 60 to 70% of the total pressure losses. The superimposed graph in Figure 10 compares the numerical pressure drop along each minichannel with the theoretical values deduced from established Poiseuille flow model. A relatively good agreement is obtained in both cases, the numerical value being globally about 10% higher than the theoretical one, this is due to the inlet-outlet effect along with maldistribution.

**Hydraulic behavior of the virtual dynamic prototype**

Local fields are presented in Figure 11 for the dynamic heat exchanger with \(A_0 = 100 \, \mu\)m and \(f_r = 10\) Hz for arbitrary time during periodic stationary regime. During expansion of the dynamic channel, the fluid moves in the longitudinal direction because of the successive expansion and contractions. Wall movement induces also vertical (i.e., toward heated wall) displacement of the fluid. Characteristic times associated to vertical and horizontal fluid displacement are similar and significantly lower than the actuation one.

Mass flow rate is proportional to both frequency and amplitude (see Table 3). High frequency leads to fast
movement of the corrugated wall, which in turn enhances the mass flow. The latter is therefore directly proportional to the frequency (Figure 12, top). Increasing amplitude (at constant gap) leads to increase in the channel volume, thus having more fluid in the channel. This behavior also enhances mass flow rate in the system.

**Figure 10.** Comparison between regular and singular pressure drops in the 1-mm channel (top) and 200-μm channel (bottom) heat exchanger. $\Delta p_{\text{tube}}$ represents the pressure drop in the 25-cm upstream and 25-cm downstream tube.

**Figure 11.** Instantaneous velocity field: for $A_0=100 \, \mu m$ and $f_r=10 \, \text{Hz}$. Online color range (Blue: 0 – Red: 1 m/s). Grayscale: (light gray 0 – Dark 1 m/s). The vertical scale is magnified 100 times. Channel height is 10 μm below center actuator and total length is 7 cm.
which is also directly proportional to the amplitude (Figure 12, bottom). Note that increasing amplitude (at constant gap) adds mainly volume to the channel near the membrane and increases average height.

Influence of gap size has been tested on mass flow rate and heat transfer for amplitude 100 \( \mu \text{m} \) and frequency 10 Hz. The initial gap was fixed at 10 \( \mu \text{m} \) and then varied up to 35 \( \mu \text{m} \). In Figure 13, it can be easily observed that mass flow rate decreases with increasing gap. This indicates that global efficiency of such device is largely affected by the minimum constriction height.

**Thermal performance**

**Reference heat exchanger**

In order to determine the heat performance of the two configurations of the heat exchanger, specific tests were carried out. The experimental protocol consists

| \( A \) (\( \mu \text{m} \)) | \( f_r \) (Hz) | \( m \) (g/s) | \( \Delta T_m \) (K) | \( h \) (W/m\(^2\)K) | \( T_f \) (K) | \( T_{in} \) (K) |
|---|---|---|---|---|---|---|
| 100 | 10 | 0.65 | 5.8 | 11054 | 301.75 | 321.94 |
| 100 | 20 | 1.31 | 5.03 | 12756 | 301.06 | 311.94 |
| 100 | 30 | 1.96 | 4.56 | 14071 | 300.74 | 308.35 |
| 100 | 40 | 2.64 | 4.24 | 15154 | 300.60 | 306.46 |
| 100 | 50 | 3.34 | 4.01 | 16007 | 300.50 | 305.32 |
| 85 | 10 | 0.55 | 5.43 | 11808 | 301.88 | 325.87 |
| 85 | 20 | 1.10 | 4.66 | 13765 | 301.12 | 313.80 |
| 85 | 30 | 1.68 | 4.2 | 15261 | 300.83 | 309.63 |
| 85 | 40 | 2.18 | 3.91 | 16381 | 300.62 | 307.40 |
| 75 | 10 | 0.49 | 5.1 | 12580 | 301.94 | 329.17 |
| 75 | 20 | 0.97 | 4.39 | 14606 | 301.22 | 315.65 |
| 75 | 30 | 1.46 | 4 | 16045 | 300.87 | 310.79 |
| 60 | 10 | 0.39 | 4.62 | 13879 | 302.09 | 336.41 |
| 60 | 20 | 0.77 | 4 | 16067 | 301.37 | 319.66 |
| 60 | 30 | 1.18 | 3.53 | 18167 | 301.00 | 313.50 |
| 60 | 40 | 1.62 | 3.33 | 19285 | 300.77 | 310.16 |

**Figure 12.** Influence of (top) frequency for \( A_0 = 100 \mu \text{m} \) and (bottom) amplitude for \( f_r = 10 \text{Hz} \) on mass flow rate.
in imposing a water flow rate by means of the gear pump and then supplying the microprocessor simulator with an electrical power such that the water temperature difference between the inlet and the outlet is equal to 10 ± 0.2 °C. The inlet temperature of the water in the exchanger is imposed at 23.5 °C for all the tests. The deviations between the applied electric power and the heat flux received by the water (quantified by multiplying the mass flow rate with the enthalpy variation) vary between 0 and 7% and are on average 4%.

The mean temperature $<T_w>$ of the bottom wall of the heat exchanger is evaluated by performing a second-order polynomial regression of the measured wall temperatures and then integrating this regression between the inlet and the outlet of the channel (i.e., between the outlet of the distributor and the inlet of the collector, on a distance of 38 mm). It should be noted that this calculation represents only an estimation of the mean temperature because it does not take into account the two-dimensional distribution of the surface temperature (variation in the direction perpendicular to the main axis of the flow). The average fluid temperature $<T_f>$ is estimated by performing an arithmetic mean between the inlet and outlet water bulk temperatures. Considering the operating conditions described above, this average temperature of the water in the heat exchanger is substantially constant for all the experiments and is equal to 28.5 °C. An “apparent” thermal conductance $G$ of the heat exchanger is then defined as follows:

$$G = \frac{m c_p (T_{out} - T_{in})}{\langle T_w \rangle - \langle T_f \rangle}$$

(11)

The variations of this apparent overall thermal conductance as a function of the mass flow rate are shown in Figure 14 for both considered channel thicknesses. The thermal conductance of the 200-μm channel heat exchanger is substantially higher than that of the 1 mm one: enhancement up to 70% is achieved for mass flow rate approximately equal to 3 g/s. For such a mass flow rate, the overall thermal conductance of the 200-μm channel heat exchanger is almost 15 W/K.

Let us consider three typical application cases using water as the working fluid, and a maximum allowable fluid temperature difference of 10 °C between inlet and outlet. The severe environment case will impose a cold source at a temperature relatively close to the maximum allowable processor temperature (e.g., 70 °C) along with limitations on the flow rate. In such conditions, heat flux up to 125 W can be removed from the microprocessor with a mass flow rate of 3 g/s. The temperature of the bottom wall of the heat exchanger would be only about 8 °C higher than the fluid mean temperature. For a mass flow rate of about 5.4 g/s, up to 225 W could be removed with a wall to fluid temperature difference of less than 20 °C (e.g., automotive applications). Finally, increasing the mass flow rate up to 9 g/s, the actuated channel allows to extract 375 W with a wall to fluid temperature of 30 °C. This device is thus also an efficient solution for data center-type applications. Moreover, contrary to static mini/microchannels which generate prohibitory high pressure drop and fluid maldistributions when operated in parallel, the active heat exchanger – due to the pumping effect – constitutes a best-suited device for low pressure losses and well-distributed flow in parallel branches of a fluid network. It can thus be concluded that such a heat sink can be efficiently integrated to the different constraints of applications involving microprocessor thermal management.

Numerical simulations have been performed in the same conditions than the experiments (i.e., inlet water temperature equal to 23.5 °C, heat flux adjusted to obtain a temperature difference between inlet and outlet of 10 °C). Virtual temperature sensors are placed at the same locations than the experimental ones and data are extracted from the numerical temperature field. The same procedure used in the experiments is then applied to calculate the apparent overall thermal conductance; numerical results are reported along with the experimental ones (Figure 14).

Adequacy between experimental and numerical results is excellent for the 200-μm channel heat exchanger, while the numerical simulations slightly overestimate the overall thermal conductance in the case of the 1-mm channel heat exchanger. As mentioned in the hydraulic behavior section, the flow in the 1-mm channel is non-uniformly distributed. This
maldistribution of the flow implies that the temperature field of the fluid is farther from a one-dimensional field than in the case of the 200-μm channel (see Figures 15 and 16). Calculating the mean temperature of the fluid by integrating a second-order polynomial trend curve as described above is therefore less precise in the case of the 1-mm channel than in the case of the 200-μm channel. In the case of the 1-mm channel heat exchanger, the flow field is more sensitive to the deviation in the imposed inlet conditions and geometry compared to the real ones. It is thus not surprising to obtain a better match between numerical results and experiments in the case of the best-distributed flow.

**Numerical prototype**

Due to movement of the channel wall, cold fluid is moved near the heated wall successively at every location; this boundary layer disruption increases sharply heat transfer along with mixing that takes place within the core flow. On temperature field, successive “hot”
and “cold” plumes could be seen (Figure 17). Near inlet cold fluid fills the pocket during expansion, then is driven toward heated surface, and eventually transferred below the next actuated zone. The constricted zone acts as a barrier separating inlet and outlet parts. This leads the fluid to fill expansion zone while being mixed efficiently; it is then transferred in the next pocket over time. These “bursts” repeat cyclically along with mixing, thus increasing heat transfer.

Heat transfer coefficient depends on both imposed frequency and amplitude (see Table 3). Increasing frequency increases the mixing effect and the fluid velocity. Thus, it results in high heat transfer coefficients varying roughly proportionally to the frequency (see...
On the other hand, increasing amplitude—while keeping the gap (minimum height below actuators) constant—leads to increase the average height of the channel and thus reduces the heat transfer coefficient. This one is thus found to decrease linearly with the amplitude (see Figure 18, bottom). Nevertheless, the heat transfer coefficient remains very high (greater than 10,000 W/m²K, see Table 3) for all tested cases. This is due to two main effects: the average fluid thickness increases, and the fluid velocities decrease. Mainly the most constricted zones become less and less efficient as the gap increases. As these zones (which cross all the channel length) are the place where the most efficient heat transfer takes place, it is not surprising to observe a net decrease in heat transfer coefficient. Two main conclusions could be drawn from these results: although the heat transfer coefficient decreases when increasing channel height, for sub-millimeter cases it remains high enough to produce a very small thermal resistance compared to the other involved in a chip cooling assembly (e.g., contact resistances). On the other hand, the flow increases with channel height and thus extracting relatively high heat power with limited inlet-outlet temperature difference will be more easily achieved using larger channel height. Moreover, required pumping power and mechanical effort and constraints will be more easily meet for channel with “large” characteristic dimension, i.e. in the order of one millimeter rather than 100 µm.

A global thermal conductance of the virtual prototype can be derived from these heat transfer coefficient values. The variation of this conductance is reported in Figure 19 along with the experimental ones of the reference heat exchanger for both 1 mm and 200 µm channel thickness. When the channel wall is actuated, a sharp heat transfer enhancement is obtained: for a mass flow rate of 1 g/s, the thermal conductance of the virtual prototype is almost the triple than the one of the 200-µm reference heat exchanger (corresponding thus to an enhancement of approximately 200%). For a mass flow rate of 3 g/s, the enhancement is almost 100%.

The thermal conductance of the virtual prototype appears thus very high. For example, as a comparison point, Kheirabadi and Groulx [8] have developed several prototypes of very compact heat exchangers for the cooling of electronics. They obtained a thermal conductance of 26 W/K considering a serpentine channel with a mass flow rate of 18 g/s (leading to a pressure drop of 14 700 Pa). This thermal conductance is very close from the one obtained with the virtual prototype with a mass flow rate of 3 g/s (with an imposed zero pressure difference). So, the actuated heat exchanger appears as a good candidate when efficient cooling is needed.

**Conclusions**

An experimental setup and a numerical tool have been built in order to analyze hydrothermal performances of
heat sink with low diameter channel. For the 1-mm channel heat exchanger, it was found that the main contribution in the pressure drop is due to the singularities and the tubes upstream and downstream of the heat exchanger, and that the flow is badly distributed in the channel. For the 200-μm channel, the main contribution in the pressure drop is due to the pressure loss in the channel itself, and the maldistribution is not significant. From heat transfer point of view, the 200-μm channel heat exchanger allows reaching thermal conductance up to 15 W/K with a very low mass flow rate of 3 g/s. This thermal performance demonstrates that such a liquid cooling heat sink could be efficiently used for the thermal management of electronic chip like microprocessor. Nevertheless, important improvements in the thermo-hydraulic performance could be obtained by optimizing the internal design.

Integration of the pumping function within a heat exchanger can be obtained considering dynamic morphing of at least one of the wall of this heat exchanger. In addition, this dynamic morphing may conduct to a significant heat transfer enhancement, making the concept particularly interesting for embedded thermal management systems. A numerical tool has been developed that allows determining the thermo-hydraulic performances of such a heat exchanger. To meet the application constrains, the dynamic deformation has to be realized with a small number of actuators. So, only three actuated zones are considered in the present work. It has then been established that:

- The mass flow rate is mainly controlled by the frequency and amplitude of the deformation, as well as the gap;
- High heat transfer coefficient values can be obtained, up to 20,000 W/m²K;
- The corresponding thermal conductance variation according to the mass flow rate is almost twice the one of the 200-μm reference static heat exchanger. In addition to the integration of the pumping function within the heat exchanger, important heat transfer enhancement is thus obtained.

**Acknowledgment**

This work has been realized in the framework of the CANOPEE project and supported by the “Fond Unique Interministériel (FUI) – 18th call for projects.” We gratefully acknowledge contributions from all project partners.

**Notes on contributors**

**Julien Fontaine** graduated in electrical engineering and automatics from the engineering school ENSEEIHT at Toulouse, France, in 2014. He is currently a PhD student at the Federal University Toulouse Midi-Pyrénées, France. The field of his research is piezoelectric actuation dedicated to micropumping and the improvement of heat transfer.

**Charles Gonzalez** graduated in electrical engineering and automatics from the engineering school ENSEEIHT at Toulouse, France, in 2016. Since graduation, he is mainly involved in electrical engineering and mechatronics in the field of heat transfer enhancement and photovoltaic solar plants exploitation.

**Prashant Kumar** graduated in computational fluid dynamics (CFD) from Trinity College in Dublin, Ireland, in 2010. He obtained his PhD entitled “Investigation of Kelvin-like solid foams for potential engineering applications: An attractive set of geometrical and thermo-hydraulic properties” in 2014 from Aix-Marseille Université, France. He then involved in CFD analysis of dynamically deformed heat exchangers in IUSTI until 2017. He then joined EPSILON-Alcen in Toulouse.

**François Pigache** is an Associate Professor at LAPLACE, Toulouse University, France, since 2006. Before joining the LAPLACE, he obtained his PhD in 2005 from the Université Sciences et technologies de Lille, France. He is involved in research in the areas of multiphysics modeling, the optimization of the piezoelectric transformers, and alternative applications of the electroactive materials for the surface functionalization.

**Pascal Lavieille** has been an Assistant Professor in the LAPLACE Laboratory, University Paul Sabatier of Toulouse, France, since 2002. He obtained his PhD in 2001 from the University H. Poincaré in Nancy, France. He is a specialist in optical diagnostic for the determination of temperature fields and measurement of film thickness. His research interests are focused on phase change (condensation, boiling, capillary phenomena, wettability) and two-phase cooling systems.

**Frédéric Topin** is an Associate Professor at IUSTI, Aix-Marseille Université, France, since 1996. He obtained his PhD in 1995 from the Université de Provence in Marseille, France. He is involved in research in the areas of transport phenomena in porous media (structure – flows/transfers – reactions), dynamic fluid
structure interaction, pore scale modeling of flow and phase change in solid foams, and coupled phenomena such as bio fuel cells or stability of superconductor cables.

Marc Miscevic is an Associate Professor at LAPLACE, Toulouse University, France, since 1999. Prior coming to LAPLACE, he obtained his PhD in 1997 from the Université de Provence in Marseille, France. He is involved in research in the areas of active method of heat transfer enhancement, phase change in microchannels and microgravity (condensation, boiling, capillary phenomena, wettability), and two-phase cooling systems.

References

1. L. Léal et al., “An overview of heat transfer enhancement methods and new perspectives: focus on active methods using electroactive materials,” Int. J. Heat Mass Transf., vol. 61, pp. 505–524, 2013. DOI: 10.1016/j.ijheatmasstransfer.2013.01.083.
2. X. L. Xie, Z. J. Liu, Y. L. He, and W. Q. Tao, “Numerical study of laminar heat transfer and pressure characteristics in a water-cooled minichannel heat sink,” Appl. Therm. Eng., vol. 29, no. 1, pp. 64–74, 2009. DOI: 10.1016/j.applthermaleng.2008.02.002.
3. J. F. Tullius, R. Vajtai, and Y. Bayazitoglu, “A review of cooling in microchannels,” Heat Transfer Eng., vol. 32, no. 7–8, pp. 527–541, 2011. DOI: 10.1080/01457632.2010.506390.
4. A. C. Kheirabadi, and D. Groulx, “A comparison of numerical strategies for optimal liquid cooled heat sink design,” presented at the Proceedings of the ASME 2016 Summer Heat Transfer Conference, Paper HT2016-7076, Washington DC, USA, July 10–14, 2016.
5. A. C. Kheirabadi, and D. Groulx, “Cooling of server electronics: a design review of existing technology,” Appl. Therm. Eng., vol. 105, pp. 622–638, 2016. DOI: 10.1016/j.applthermaleng.2016.03.056.
6. S. M. Sohel Murshed, and C. A. Nieto de Castro, “A critical review of traditional and emerging techniques and fluids for electronics cooling,” Renew. Sust. Energ. Rev., vol. 78, pp. 821–833, 2017. DOI: 10.1016/j.rser.2017.04.112.
7. A. Habibi Khalaj, and S. K. Halgamuge, “A review on efficient thermal management of air- and liquid-cooled data centers: from chip to the cooling system,” Appl. Energy, vol. 205, pp. 1165–1188, 2017. DOI: 10.1016/j.apenergy.2017.08.037.
8. A. C. Kheirabadi, and D. Groulx, “Experimental evaluation of a thermal contact liquid cooling system for server electronics,” Appl. Therm. Eng., vol. 129, pp. 1010–1025, 2018. DOI: 10.1016/j.applthermaleng.2017.10.098.
9. X. H. Yang, S. C. Tan, Y. J. Ding, and J. Liu, “Flow and thermal modeling and optimization of micro-mini-channel heat sink,” Appl. Therm. Eng., vol. 117, pp. 289–296, 2017. DOI: 10.1016/j.applthermaleng.2016.12.089.
10. T. Dixit, and I. Ghosh, “Review of micro- and mini-channel heat sinks and heat exchangers for single phase fluids,” Renew. Sust. Energ. Rev., vol. 41, pp. 1298–1311, 2015. DOI: 10.1016/j.rser.2014.09.024.
11. I. A. Ghani, N. A. C. Sidik, and N. Kamaruzaman, “Hydrothermal performance of microchannel heat sink: the effect of channel design,” Int. J. Heat Mass Transf., vol. 107, pp. 21–44, 2017. DOI: 10.1016/j.ijheatmasstransfer.2016.11.031.
12. T. Nishimura, Y. Ohori, and Y. Kawamura, “Flow characteristics in a channel with symmetric wavy wall for steady flow,” J. Chem. Eng. Japan, vol. 17, no. 5, pp. 466–471, 1984. DOI: 10.1252/jcej.17.466.
13. T. Nishimura, S. Murakami, S. Arakawa, and Y. Kawamura, “Flow observations and mass transfer characteristics in symmetrical wavy-walled channels at moderate Reynolds numbers for steady flow,” Int. J. Heat Mass Transf., vol. 33, no. 5, pp. 835–845, May 1990. DOI: https://doi.org/10.1016/0017-9310(90)90067-5.
14. N. Niceno, and E. Nobile, “Numerical analysis of fluid flow and heat transfer in periodic wavy channels,” Int. J. Heat Fluid Flow, vol. 22, no. 2, pp. 156–167, 2001. DOI: 10.1016/S0142-727X(01)00074-1.
15. P. Naphon, “Effect of wavy plate geometry configurations on the temperature and flow distributions,” Int. Com. Heat Mass Transfer, vol. 36, no. 9, pp. 942–946, 2009. DOI: 10.1016/j.icheatmasstransfer.2009.05.007.
16. C. Y. Yang, C. T. Yeh, W. C. Liu, and B. C. Yang, “Advanced micro-heat exchangers for high heat flux,” Heat Transfer Eng., vol. 28, no. 8–9, pp. 788–794, 2007. DOI: 10.1080/01457630701328676.
17. C. Y. Yang, and W. C. Liu, “Development of a mini liquid cooling system for high-heat-flux electronic devices,” Heat Transfer Eng., vol. 32, no. 7–8, pp. 690–696, 2011. DOI: 10.1080/01457632.2010.509773.
18. L. Léal, F. Topin, P. Lavieille, L. Tadrist, and M. Miscevic, “Simultaneous integration, control and enhancement of both fluid flow and heat transfer in small scale heat exchangers: a numerical study,” Int. Com. Heat Mass Transfer, vol. 49, pp. 36–40, 2013. DOI: 10.1016/j.icheatmasstransfer.2013.10.004.
19. P. Kumar, K. Schmidmayer, F. Topin, and M. Miscevic, “Heat transfer enhancement by dynamic corrugated heat exchanger wall: numerical study,” J. Phys.: Conf. Series, vol. 745, pp. 32–61, 2016. DOI: 10.1088/1742-6596/745/3/032061.
20. M. Saeed, and M. H. Kim, “Header design approaches for mini-channel heatsinks using analytical and numerical methods,” Appl. Therm. Eng., vol. 110, pp. 1500–1510, 2017. DOI: 10.1016/j.applthermaleng.2016.09.069.