EXPERIMENTAL AND NUMERICAL STUDY OF TRANSIENT ELECTRONIC CHIP COOLING BY LIQUID FLOW IN MICROCHANNEL HEAT SINKS

Jingru Zhang¹, Tiantian Zhang², Shaurya Prakash³, and Yogesh Jaluria¹

¹Mechanical & Aerospace Engineering Department, Rutgers University, Piscataway, New Jersey, USA
²Department of Strategy and Development, China Three Gorges Corp., Beijing, P. R. China
³Department of Mechanical Engineering, The Ohio State University, Columbus, Ohio, USA

Cooling of electronic chips has become a critical aspect in the development of electronic devices. Overheating may cause the malfunction or damage of electronics. The time needed for heat removal is particularly important in a wide range of electronic systems, such as switching circuits. Thus, it is important to characterize the transient behavior of the system and determine the response. Most studies in the literature have focused on steady-state circumstances and the transient effects have not been considered in the detail needed. In this article, an experimental system and a numerical model were developed to test the effects of different parameters and their influence on the transient electronic chip cooling by liquid flow in microchannel heat sinks. The temperature change with time of the system for different heat fluxes at different flow rates was determined, from which the response time is obtained. Three different configurations of multi-microchannel heat sinks were tested during the experiment. Numerical models were then developed to simulate the transient cooling for two of these configurations. A good agreement between the experimental data and numerical results showed that single-channel models are capable of simulating the thermal behavior of the entire heat sink by applying appropriate assumptions and boundary conditions. The experimental results can then be used to improve the numerical models and vice versa.

1. INTRODUCTION

The semiconductor industry has seen a rapid pace of growth in its products for the past four decades [1]. Electronic devices and systems have been reduced in size, while their functions and complexity continue to increase at an amazing rate. As a consequence, the power density has risen rapidly and the heat input has increased...
dramatically. For all electronic devices, effective cooling systems must be designed to remove excessive heat and the temperature must be controlled for every component to ensure reliable and stable performance and prevent any failure or malfunction [2, 3]. The heat sink considered here is a finned structure with multi-microchannels, and is applied as an effective cooling device for electronic chips [4]. It is cooled largely by forced convection in the liquid flow through the microchannels, though forced air cooling is the most commonly used approach for heat removal because of convenience and design simplicity. However, the pumping power requirements for gases are often much larger for the same thermal performance compared with liquid. The power dissipated from electronic chips is conducted through the substrate of the heat sink to the fins and to the coolant [5].

Liquid cooling applied in heat sinks with multiple microchannels is believed to be a promising thermal management for electronic chips [6]. Two-phase flow patterns and the influence of boiling flow on heat transfer were studied by Zhang et al [7]. They designed single and multiple channel devices to study bubble formation and flow regimes in microchannels, being different from those in larger channels. Compared with two phase flow, single phase cooling systems have less complicated physical phenomena and the system is easier to control. There are several research groups that have worked on single phase liquid flow in microchannels [8–10]. Wu and Cheng [11] experimentally studied laminar convective heat transfer in silicon microchannels with different surface conditions. They discussed the influence of surface roughness, surface hydrophilic properties, and geometric parameters. Qu and Mudawar [12] investigated the pressure drop and heat transfer characteristics of a single microchannel heat sink made of copper, both experimentally and numerically. They found good agreement between the measured data and corresponding numerical predictions using the conventional Navier-Stokes equation and the energy equation. Wei and Joshi [13] observed the velocity profiles inside silicon microchannels using micro resolution particle image velocimetry. They also carried out numerical simulations to study the effects of sidewall slope on the heat transfer.

Electronic cooling problems are currently considered mainly for steady-state conditions. Assuming the power to the electronic chip to be constant and, after the electronic system is turned on and kept running for a long period of time, the temperatures of the electronic chips and the cooling devices are expected to reach steady state.

<table>
<thead>
<tr>
<th>NOMENCLATURE</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Symbol</strong></td>
</tr>
<tr>
<td>H</td>
</tr>
<tr>
<td>k</td>
</tr>
<tr>
<td>P</td>
</tr>
<tr>
<td>Q</td>
</tr>
<tr>
<td>q</td>
</tr>
<tr>
<td>t</td>
</tr>
<tr>
<td>T</td>
</tr>
<tr>
<td>ΔTH</td>
</tr>
<tr>
<td>ΔTss</td>
</tr>
<tr>
<td>u</td>
</tr>
<tr>
<td>ρ</td>
</tr>
<tr>
<td>Δ</td>
</tr>
<tr>
<td>τ</td>
</tr>
<tr>
<td>μ</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Subscripts</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
</tr>
<tr>
<td>i</td>
</tr>
<tr>
<td>s</td>
</tr>
</tbody>
</table>
When the thermal equilibrium condition is met, the rates of heat being transferred by conduction, convection, and radiation all remain constant. For a large electronic system, it may take a long operation time before the system becomes steady. However, only a few minutes may be needed for the cooling system of a small electronic chip to reach steady state. Also, the maximum heat load usually appears at the start up or the shutdown of the electronic system. In order to control the temperature and prevent the electronics from overheating, it is important to study the transient heat transfer behavior.

Transient cooling or heating conditions will occur and need to be considered for electronic systems when the power is turned on or shut off, or when there is a change in the power load on the system. The temperature increase of the electronic chip will be linear if it is completely insulated. The temperature rise with respect to time can be determined by the heat load, the mass and the specific heat. When the power to the electronic system is turned on and a given cooling method is applied at the same time, the temperature rise with respect to time will depend on the heat generating rate and the heat removal rate. The temperature increase rate is expected to decrease until it reaches steady state for an electronic chip with a well-designed cooling system. The temperature rise will be faster when the power is initially turned on. Then, the slope of temperature versus time will gradually decrease until steady state is reached. Unfortunately, there are not many existing transient temperature results in the electronic system cooling literature. Hence, in our study, we will present some fundamental results on the transient characteristics for electronic chip cooling.

2. DEVICE FABRICATION AND PACKAGING

The study reported here uses wet etching methods for fabricating microchannels. In contrast to dry etching, wet etching is relatively economical and the chemical mixtures are easier to prepare. One down side of the wet etching process is the challenge in controlling the undercutting and sidewall profile, which tend to be more controllable with dry etching. However, for the results and overall trends reported here, wet etching provides a relatively quick method to fabricate and test microchannel heat sinks. In this work, 30% potassium hydroxide solution (KOH) was used as an etchant to etch 0.085 - 0.115 Ω-cm p-type (110) oriented silicon wafers from University Wafer masked with silicon nitride to create microchannels. KOH etching is orientation-dependent, and this anisotropic etching scheme allows tailoring of sidewall profiles. The fabrication and packaging process is illustrated in Figure 1 as a process flow diagram.

As shown in Figure 1, bare silicon in Figure 1a was cleaned by RCA-1 solution (water: 27% ammonium hydroxide: 30% hydrogen peroxide = 4:1:1) to remove organic contamination, followed by Hydrogen fluoride (HF) dip to remove native oxide on the silicon in order to reduce undercutting of the nitride mask (Figure 1b). The 790 Unaxis plasma enhanced chemical vapor deposition (PECVD) system reacted gases in a (radio frequency) (RF) induced plasma to deposit a 2000 Å silicon dioxide (SiO₂) and a 2500 Å silicon nitride (Si₃N₄). The system uses SiH₄ (160 m³ = min) and N₂O (720 m³ = min) for the SiO₂ deposition, the corresponding operating temperature, pressure, and RF are 300°C, 900 mTorr, and 19 W, respectively. The deposition rate is 1000 Å every three minutes. For the Si₃N₄ deposition, the available gas are SiH₄ (200 m³ = min), NH₃ (47 m³ = min), and N₂ (900 m³ = min). The operating temperature, pressure and RF are 250°C, 900 mTorr, and 25 W,
respectively. The Si$_3$N$_4$ deposition rate is 100 Å/min. Buffered oxide etch 7:1 was used to open window of silicon nitride (Figure 1d) after conventional ultraviolet (UV) photolithography defines the microchannel pattern on the photo resist (Figure 1c). The silicon wafer was then dipped into an AZ400T solution (Figure 1e) for half an hour to remove the rest of the photo resist before it dissolved in the KOH solution (Figure 1f). A magnetic stirrer was used to agitate the KOH solution to prevent the etch rate variation from the top to the bottom; 10% - 15% isopropanol was added to KOH solution to improve the etch uniformity. A PDMS layer is bonded on top the silicon microchannels after treating it with oxygen plasma at 200 W for 15 s at room temperature. Openings were punched in the PDMS for fluid connections. Figure 2 shows the scanning electron microscope (SEM) images of the fabricated microchannel.
3. EXPERIMENTAL SETUP

3.1. Facility

For experiments, a commercial miniature Kapton heater (model number: HK5573R15.7L12F) from Minco was attached using a conductive epoxy underneath the microchannel heat sink to simulate the heat released by an electronic chip. The heat flux provided by the heater was controlled by regulating electrical current and voltage of a DC power supply. T-type thermocouples (12 in total) were used to measure the temperature of the thermal system. Four thermocouples were attached on the back of the heat sink, while eight were attached on the heater, four on the back and four on the front facing the heat sink, shown in Figure 3b. On the back of the heat sink, the microchannel area (the area between two dashed lines) was treated as four portions, and each portion has a thermocouple at its center. All the data were collected by the data acquisition system including SCXI system, which consists of SCXI-1000 chassis, SCXI-1100 multiplexer module and the SCXI 1300 terminal block. The SCXI system is used to connect the measurement devices to PCI-6040E DAQ card from National Instrument. The DAQ software is Labview.

As shown in Figure 3a, distilled water was used as the coolant due to its large heat capacity (4186 J/kg · K, one of the best among liquids). The standard pump 11 plus syringe pump from Harvard Apparatus was used to drive the flow. The programmable aspects of the syringe pump also allowed use of the pump as a flow meter and a valve since specific flow rates can be dialed in for continuous

![Figure 3. Schematic of the experimental arrangement (color figure available online).](image-url)
measurements. A filter with 1 micrometer mesh element was used after the syringe to remove any residual impurities suspended in the cooling water. The heat sink test device is packaged in an acrylic chamber with through holes at the bottom to allow for natural convection on the back side of the heater.

### 3.2. Calibration and Data Collection

All the thermocouples were carefully calibrated and found to have an accuracy of ±0.8°C. Three different heat sink configurations were designed and fabricated as shown in Figure 4, including straight rectangular channels (Figure 4a), U-shaped channels (Figure 4b), and serpentine channels with counter-flow heat exchanger

![Figure 4](image-url)

**Figure 4.** Sketch of different multi-microchannel heat sinks considered in the study. (a) Straight channel, (b) U-shaped channel, and (c) serpentine channel with counter flows (color figure available online).
configuration (Figure 4c). The serpentine channels, whose configuration is shown in Figure 4c, have two sets of inlets and outlets. This counter flow design is likely to increase the heat transfer because of the large temperature difference between adjacent parallel channels compared to other configurations. An example of this is the temperature difference between the water flow toward outlet B and water inflow at inlet A. All the multi-microchannel heat sinks have the same 1 cm² surface area (inlet and outflow reservoirs are not included).

The syringe pump was turned on before the DC power supply, and the system was run for a while to ensure open bubble-free channels with no leakage in the test system. Then, the power supply was turned on and data recording was started.

4. RESULTS AND DISCUSSION

4.1. Experimental Results

All the experimental results shown here were obtained from the experimental facility described in the preceding section. The multi-microchannel heat sinks investigated have the same surface area of 1 x 1 cm². Each microchannel has a nominal width of 50 μm, a depth of 60 μm, and a fin thickness of 200 μm. The number of channels is 41, 19, and 38 for rectangular, U-shaped and serpentine channel configurations, respectively. The heat transfer characteristics are studied in terms of heat flux, temperature difference, and thermal resistance. The average of the outputs from the four thermocouples attached at the back of the heat sink was taken to indicate the temperature. Similar averages were applied to obtain the temperatures at the top and bottom of the heater. The y axis of Figure 5 shows the temperature difference between the heat sink and the ambient temperature as well as the temperature difference between the heater and the ambient. The straight channel heat sink (Figure 5a) shows a more uniformly distributed temperature, and it has the lowest temperature difference from the bottom of the heater to the heat sink, whereas the serpentine channel has the largest.

The temperature differences increase with increasing heat flux almost linearly at constant flow rate for all configurations. Straight channels shown in Figure 5a gave the lowest temperature rise compared with the other two configurations. There is a large temperature drop from the top of the heater to the bottom of the heat sink, which is caused by the large thermal resistance at the heater/heat sink interface $\theta_i$. However, $\theta_i$ is difficult to reduce in practice with the set-up containing a distinct heating element. In practical applications, the heat sink can be integrated with the electronic chips to eliminate $\theta_i$.

The transient response time can be determined if the temperature rise during the heating cycle and the steady state temperature are known. The basic equation is as follows:

$$\frac{\Delta T_H}{\Delta T_{ss}} = \frac{T_r - T_i}{T_s - T_i} = e^{-t/\tau}$$  \hspace{1cm} (1)

Where $\Delta T_H$ is the temperature rise that occurs during the heating cycle, $\Delta T_{ss}$ is the temperature rise required to ready steady state condition, $T_r$ is temperature at the
characteristic thermal response time, $T_i$ is initial steady state temperature, $T_f$ is the temperature at final steady state, $\tau$ is the time constant, and $t$ is the characteristic thermal response time.

Figure 5. Temperature difference versus heat flux for different configurations. (a) Straight channel, (b) U-shaped channel, and (c) serpentine channels (color figure available online).
It is convenient to evaluate a thermal design in terms of the time constant $\tau$. When the time constant is known, it is possible to obtain the thermal response of the system. A convenient reference point is one time constant. When $t$ is equal to the time constant, Eq. (1) becomes as follows.

$$\frac{T_r - T_i}{T_s - T_i} = \frac{1}{e}$$

where $e = 2.718$. This shows that one time constant represents a temperature increase that is 63.2% of the steady state temperature rise. The response time can be obtained from the temperature data recorded at different locations during the experiment, as shown in Figures 6–8. The accuracy of the response time is $\pm 5 \text{s}$.

It is found that the response time of the heat sink is larger than that of the heater. For example, when the flow rate is 0.3 ml/min, the response time for the U-shaped microchannel heat sink, based on the temperature measurements at its back, is 107 s, and 81 s for the heater at the top. Thus, the sink has a slower response and would take longer to become steady. This was caused mainly by the large thermal resistance at the heater and heat sink interface. The conduction at the interface and silicon substrate causes the response delay of the heat sink as well.

The response time has a decreasing trend with an increasing flow rate. The response time $t$ was found to be 92 s for a U-shaped channel when the flow rate is 0.4 ml/min, and heat flux is 1286 W/m², as shown in Figure 6a from measurements at the back of the heat sinks. It is 121 s and 110 s for straight channels and serpentine channels, respectively. The U-shaped channel heat sink took a shorter time to reach steady state, and it responded faster and removed heat faster than the other two configurations. This is caused by the flow structure and the ratio of the surface area between the solid (silicon) and the liquid (distilled water). The edge outside the arc area of the U-shaped channel has a large solid area compare with other configurations. The serpentine channels had the lowest heat removal rate within the experimental range. The difference in the response time between the straight channel heat sink and the serpentine channel heat sink becomes shorter with increasing flow rate at the top surface of the heat sink, even though they still took a longer time to reach steady state conditions compared to the U-shaped channel. Figure 7 shows the results from measurements taken at the top surface of the heater, and trends similar to those in Figure 6 are seen.

Figure 8 shows the response time for different flow configurations, with different heat fluxes but the same flow rate. The flow rate was held at 0.4 ml/min, and the response time was found to vary between 83 s and 101 s for a U-shaped channel when the heat flux is increased. Straight channels and serpentine channels had a longer response time, compared with the U-shaped channel. Overall, the influence of the heat flux to the response time is smaller than the influence of the flow rate.

4.2. Numerical Results

Numerical models were established to simulate the transient process for the straight microchannel and U-shaped microchannel heat sinks. The experimental data facilitated validation of the numerical model, which leads to an expansion of the domain for design and optimization. In addition, numerical results can be used to improve the experimental system and vice versa.
4.3. Straight Channels

In order to simply the problem and reduce the computational time, a single channel model, as shown in Figure 9, is first established. This model is then extended to simulate the heat transfer phenomena in the entire heat sink with multiple microchannel by appropriate assumptions and boundary conditions. The governing

Figure 6. Response time at the back of the heat sink versus flow rate for heat flux values of (a) 1286 W/m² and (b) 5270 W/m² (color figure available online).
equations, along with the relevant boundary conditions, were solved by the commercial CFD code CFX12. Assuming density \( \rho \) to be constant and the flow to be an incompressible Newtonian flow, the governing equations are as follows.

Continuity:

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

Momentum:

\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot [\mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)]
\]

Energy:

\[
\nabla \cdot (\rho u h) = \nabla \cdot (k \nabla T) + u \cdot \nabla p + \nabla \cdot \sigma
\]

Where \( \nabla \) is the vector operator, and is defined as follows.

\[
\nabla = \left[ \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right]
\]
Figure 8. Response time at different heat fluxes. (a) At the back of the heat sink (flow rate $Q = 0.4 \text{ ml/min}$), and (b) at the top surface of the heater (flow rate $Q = 0.4 \text{ ml/min}$) (color figure available online).

Figure 9. Sketch of a straight microchannel heat sink model. (a) General configuration and (b) computational domain (color figure available online).
For Newtonian incompressible flow, the stress $\sigma$, is given by the following.

$$\sigma = 2 \mu S = \mu [\nabla u + (\nabla u)^T]$$

The steady state model for a straight channel is well established, and the results indicated pretty good agreement with experimental data [16]. This comparison was used for validating the model and the numerical scheme.

The same numerical model was applied in CFX and a time-dependent solution was obtained instead of solving the steady flow problem. The time step was set as 1 s, and five iterations were used per each time step. The average temperature of the heat sink was monitored during the calculations and, after it reached steady state, the temperature distribution was found to agree closely with the one obtained from the steady state model. The steady state modeling work had already been validated by comparing the obtained values with the experiment data [16], which means that the transient model has the same thermal resistance as the experiment results with the same flow rate and heat flux.

The time constant $\tau$ of a heat sink depends on its heat capacity and thermal resistance. In order to have the single channel model describe the transient heating process of multiple microchannels via time constant, it must have equivalent thermal capacity and thermal resistance as the multiple microchannel heat sink applied in the experiment. Since the numerical model has the same thermal resistance as the experimental system under the same boundary conditions, the thermal capacity, which is the product of weight and specific heat, is the factor that can be improved.

The rectangular microchannel heat sink used in the experiment had 41 channels. Hence, in the model, the density of the silicon substrate and fins was increased by 41 times, which makes the numerical model to have the same mass and thermal capacity as the multiple microchannels heat sink used in the experiment.

The comparison of experimental and numerical is shown in Figure 10. The differences between numerical response time and experimental data are within 15% if the heat flux is kept as a constant. For example, when the flow rate was 0.4 ml/min, the response time obtained from experiment was 121 s, and it was 100 s according to numerical calculations. The difference between numerical data and experiment results was larger in Figure 10b, in which the flow rate was fixed and heat flux varied from 307 to 2737 W/m². For instance, when the flow rate was 0.4 ml/min, the numerical results were 28% larger than the experiment data. This might be caused by the inaccuracy in the estimation of the heat loss from the bottom and the sides of the heat sink, calculated by using empirical equations [16].

4.4. U-Shaped Channel

The same methodology has been applied to the simulation of a U-shape channel heat sink [17]. The model for a single U-shaped channel with computational domain shown in Figure 11b was established and then extended to simulate the multi-channel heat sink heat transfer system. Since the actual heat sink shown in Figure 11a had 19 channels, after increasing the mass of the single U-shaped channel by 19 times, the numerical data showed good agreement with experimental data (Figure 12). When flow rate is 0.35 ml/min, the difference between experimental and numerical data is only 1 s. The numerical solution also showed that the response
Figure 10. Comparison of numerical results and experimental data for straight channels. (a) Constant heat flux $q = 1286 \text{ W/m}^2$, and (b) constant flow rate $Q = 0.4 \text{ ml/min}$ (color figure available online).

Figure 11. Sketch of U-shaped microchannel heat sink and the computational model. (a) General configuration and (b) computational domain (color figure available online).
time remained between 80 s and 100 s, when the flow rate was kept at 0.4 ml/min (Figure 12b). This means that the heat sink has a reliable ability to bring down the temperature of the entire system under constant flow rate.

5. CONCLUSION

Three different configurations of multiple microchannel heat sinks were designed and fabricated. The transient heat transfer phenomena of heat sinks applied for heat removal were studied experimentally and numerically. The response time was used as a critical parameter to evaluate the thermal behavior of different heat sinks under different boundary conditions. According to the experimental data and numerical results, the main conclusions are given below.

The U-shaped channel has a shorter response time compared with straight and serpentine channel heat sinks. The heater reaches steady state condition faster than the straight heat sink. The response time decreased with an increase in the flow rate. The response time of a heat sink with a fixed flow rate tends to be stable for different heat fluxes.
A good agreement between the experimental data and numerical results showed that a single channel model is capable of simulating the transient heat transfer process of the entire heat sink with multiple channels by applying appropriate assumptions. It also showed that the thermal capacity and thermal resistance are two critical factors that will influence the response time of the heat transfer system. The transient heat transfer features in rectangular and U-shaped multiple microchannels heat sinks were reproduced successfully with one rectangular microchannel and one U-shaped microchannel model.

REFERENCES
