Assessment of an active-cooling micro-channel heat sink device, using electro-osmotic flow

Marwan F. Al-Rjouba, Ajit K. Royb, Sabyasachi Gangulib,c, Rupak K. Banerjeea,*

a School of Dynamic Systems, University of Cincinnati, Cincinnati, OH, USA
b Thermal Sciences and Materials Branch (AFRL/RXBT), Air Force Research Laboratory, WPAFB, OH, USA
c University of Dayton Research Institute, Dayton, OH, USA

ARTICLE INFO

Article history:
Received 7 February 2011
Received in revised form 10 June 2011
Accepted 10 June 2011
Available online 6 July 2011

Keywords:
Electro-osmotic flow
Micro-scale heat exchanger
Hot spot cooling
Zeta-potential
EOF

ABSTRACT

Non-uniform heat flux generated by microchips causes “hot spots” in very small areas on the microchip surface. These hot spots are generated by the logic blocks in the microchip bay; however, memory blocks generate lower heat flux in contrast. The goal of this research is to design, fabricate, and test an active cooling micro-channel heat sink device that can operate under atmospheric pressure while achieving high-heat dissipation rate with a reduced chip-backside volume, particularly for spot cooling applications. An experimental setup was assembled and electro-osmotic flow (EOF) was used thus eliminating high pressure pumping system. A flow rate of 82 l/min was achieved at 400 V of applied EOF voltage. An increase in the cooling fluid (buffer) temperature of 9.6 °C, 29.9 °C, 54.3 °C, and 80.1 °C was achieved for 0.4 W, 1.2 W, 2.1 W, and 4 W of heating powers, respectively. The substrate temperature at the middle of the microchannel was below 80.5 °C for all input power values. The maximum increase in the cooling fluid temperature due to the joule heating was 4.5 °C for 400 V of applied EOF voltage. Numerical calculations of temperatures and flow were conducted and the results were compared to experimental data. Nusselt number (Nu) for the 4 W case reached a maximum of 5.48 at the channel entrance and decreased to reach 4.56 for the rest of the channel. Nu number for EOF was about 10% higher when compared to the pressure driven flow. It was found that using a shorter channel length and an EOF voltage in the range of 400–600 V allows application of a heat flux in the order of 10^4 W/m^2, applicable to spot cooling. For elevated voltages, the velocity due to EOF increased, leading to an increase in total heat transfer for a fixed duration of time; however, the joule heating also got elevated with increase in voltage.

1. Introduction

Intensive computing devices are projected to generate heat flux values that can exceed 2.5 × 10^6 W/m^2 [1]. With these high-heat generation rates the conventional cooling techniques have been found to be inadequate. Instead of forcing air to flow over fins, liquid can be forced to flow through channels that are in contact with the devices’ surface. Various methods are being used to drive a liquid through micro-channels, which include pressure driven techniques that use mechanical pumps or pressurized gases. Another approach is to use electro-osmotic flow pumping technique; this technique has no moving parts and needs less maintenance compared to mechanical pumps.

1.1. Electro-osmotic flow (EOF)

Solid surfaces develop a charge when brought into contact with an aqueous solution [2]. Due to the formation of surface charge, counter ions in the solution will be attracted toward the surface. The concentration of counter ions in the vicinity of the surface will be higher than in the fluid bulk, whereas co-ions concentration in the fluid bulk is higher than its concentration in the vicinity of the surface. The charged surface and the thin layer of counter ions that balances its charge are defined as the electric double layer (EDL) [2].

Ions in the vicinity of the charged surface are attracted to it and this restricts their motion (immobile ions). Ions away from the charged surface are not strongly affected by the charged surface. This leads to a higher mobility (mobile ions). When an electric field is applied between two points of the aqueous solution, the mobile ions start to move under the influence of the electric field. The ions motion drags other fluid particles due to viscous forces. This leads to a bulk fluid motion known as electro-osmotic flow.
Jiang et al. [3] designed a closed-loop two-phase micro-channel cooling system with an external EOF pump and a heat rejecter, a maximum of 7 ml/min coolant flow rate was achieved, with a chip temperature maintained less than 120 °C. Laser et al. [4] used the EOF through narrow deep micro-channels to reduce the temperature of the high-power density spots of microchips, 170 μL/min of coolant flow was achieved at 400 V of applied EOF voltage. Zhang et al. [5] tested a single micro-channel for the pressure distribution along the heating length and measured the temperature variations during the phase change, the flow was pressure driven using a syringe pump. Jung et al. [6] used a single micro-channel to study the heat transfer to nanofluids, and investigated the differences between nanofluids and pure water for convection heat transfer coefficient and friction factor. Eng et al. [7] designed an electro-osmotic flow silicon-based heat spreader that generated a coolant flow rate of 0.2 ml/min at 2 V/mm electric field; a 4 °C reduction in device temperature was achieved.

Tiselj et al. [8] studied the effect of axial conduction on heat transfer in micro-channels, where experimental and numerical approaches were adopted. It was shown that the bulk fluid and the channel walls’ temperatures do not vary linearly along the channel. The most significant changes in the temperature gradient were reported in the axial flow direction. In the flow direction the effect of axial heat flux was maximum at the channel inlet and minimum at outlet. Yin and Bau [9] studied the performance of micro-heat exchangers theoretically. The micro-heat exchanger was optimized to achieve a minimum heat resistance. Gillot et al. [10] designed a micro-heat sink to handle the heat generated by a module of insulated gate bipolar transistor (IGBT); a thermal resistance of 0.08–0.12 K/W was achieved for the module of IGBT chips.

Sze et al. [11] measured the zeta-potential for an electro-osmotic flow in a parallel plate micro-channel. Using Smoluchowski equation and current time relationship, zeta potentials for glass and PDMS were evaluated. Husain and Kim [12] numerically studied the performance of an EOF driven flow and a pressure driven flow in a microchannel heat sink with wavy channels. The study showed an increase in heat transfer to the cooling fluid for the pressure driven flow due to recirculation. They found that applying EOF flow increased the flow rate but eliminated the effect of channel waviness due to its uniform flow profile. Dasgupta et al. [13] of our group studied the effects of the applied-electric field and micro-channel wetted-perimeter on electro-osmotic velocity. These numerical and experimental studies found that, as the wetted perimeter increases, the electro-osmotic velocity decreases, and the electro-osmotic flow increases.

Studies of heat removal techniques using fluid flow through micro-channels, reported in recent literature, can be classified into three main pumping categories, which are pressure driven pump, external EOF pump, and integrated EOF pump. These are identified by the type of technique used to drive the fluid flow through the micro-channels. In the case of pressure driven flow, fluid is forced through the channels by an external mechanical pump or compressed gas. An external EOF pump can also be used to drive the fluid flow through electrically charged micro-channels. The micro-channels of the heat removal device can itself be used as an integrated EOF pump. For devices that fall in the first two categories, there is the need for an external pump that occupies larger volume and needs higher power for its operation. For devices that fall in the third category, the electrically charged microchannels serve both as a heat exchanger and an integrated pump. Hence the volume and power requirement are reduced. The maximum heat flux removal capacity and the cooling fluid flow rate for the three categories vary. A brief comparison between the cooling capacities of the three categories is presented in the following paragraphs.

1.2. Pressure driven flow

A recent study by Chiu et al. [14] used a pressure driven flow which removed a maximum heat flux of 1.57 × 10^5 W/m². The maximum flow rate of the cooling fluid was 300 ml/min. In another study by Ornatskii and Viyarskii [15] the maximum heat flux removed using micro-channel fluid flow was 4 × 10^7 W/m². Due to the higher flow rates of the cooling fluid, pressure driven flows are typically capable of removing higher heat flux values. The convective heat transfer due to pressure driven flow is typically limited by the parabolic nature of the velocity profile in the developed region. Due to the size and power limitations of modern electronic devices, such pressure driven pump has major challenges because of the high operating pressure needing better sealing mechanism, larger pump size, and higher pumping power. The reliability of such a system with moving parts is a major operational issue.

1.3. External EOF pump

Using an external EOF pump Jiang et al. [3] removed a 3.8 × 10^7 W/m² of heat flux and achieved a maximum cooling fluid flow rate of 4 ml/min. This system eliminated the need of high pressure mechanical pump and replaced it with an EOF pump. While such external EOF pumping system can be more reliable

### Nomenclature

- **Across**: cross sectional area of the channel, m²
- **B**: bias error
- **DAQ**: data acquisition system
- **DI**: de-ionized
- **E**: electric field, V/m
- **EDL**: electric double layer
- **EOF**: electro-osmotic flow
- **h**: heat transfer coefficient, W/m² K
- **h_c**: cooling fluid (buffer enthalpy) at the channel inlet, J/kg
- **h_o**: cooling fluid (buffer enthalpy) at the channel outlet, J/kg
- **m**: mass, kg
- **p**: pressure, Pa
- **p**: precision error
- **PDMS**: polydimethyl siloxane
- **T**: temperature

### Greek symbols

- **ψ**: wall surface charge, C/m²
- **ε**: permittivity, C/V m
- **ρ**: density, kg/m³
- **ρ_e**: bulk charge density, C/m³
- **z**: zeta potential, V
- **ζ**: stream function
- **σ**: surface tension, N/m
- **λ**: thermal conductivity, W/m K
- **φ**: applied voltage, V
- **k**: bulk conductivity, S/m
- **q**: energy loss from the reservoir, J
- **C**: specific heat, J/kg K
- **f**: cooling fluid (buffer) diffusivity
- **ρ_m**: density of coolant, kg/m³
- **μ**: water viscosity, Pa s
- **ε**: electric permittivity, C/m
- **e**: electron charge, C

### Examples

- **T** is the temperature maintained less than 120 °C.
- **h** is the heat transfer coefficient, W/m² K.
- **p** is the pressure, Pa.
than mechanical pumps, it could not adequately reduce the size of the system.

1.4. Integrated EOF pump

For these systems the flow channels with cooling fluid serve both as a heat exchanger and an integrated EOF pump. For most integrated micro-heat exchangers reported in literature, the amount of heat flux that can be removed from such device was lower than pressure driven and external EOF pumps. The cooling fluid flow rate was comparatively less for integrated EOF systems. Laser et al. [4] designed an integrated system that achieved 170 μl/min of cooling fluid flow but the maximum heat flux that the device could remove was not reported. In another study, Eng et al. [7] designed an EOF heat spreader which removed a 5.5 W of power. Though the power for the integrated systems was lower, they were very effective for localized (hot spot) cooling, where the heat flux is very high and the area is extremely small. The total power generated by these hot spots can be effectively removed by the integrated micro-heat exchangers, which is the focus of this study. Integrated EOF micro-heat exchangers are simple, reliable and do not need any additional fabrication processes.

In this pilot study, a device that falls into the third category (integrated EOF pump) of microchannel fluid flow systems was designed, fabricated, and tested. The device is a micro-scale heat exchanger with an integrated EOF pump designated for hot spot cooling applications. The initial design of the device consisted of an array of micro-channels that were connected by two reservoirs. These channels and reservoirs were etched on a silicon substrate. Various sequential processes were conducted on the substrate, which included cleaning, spin coating, soft baking, photolithography, development, hard baking, and etching. The substrate was then bonded to a polydimethyl siloxane (PDMS) cover. An experimental flow loop was designed that included a voltage power supply and a data acquisition system (DAQ) for temperature measurement. As a validation for the experimental data, numerical calculations were also performed for a single channel. Numerical results were compared with the experimental data.

2. Methodology

2.1. Experimental methods

The actively cooled micro-channel heat sink device was designed by introducing 20-parallel-micro-channels, where each channel had a 300 μm width, and 3 cm length. The channels depth was 100 μm. These channels were connected by two reservoirs at both ends. Fig. 1A shows a drawing of the actively cooled micro-channel heat sink device having multiple micro-channels and a single reservoir with dimensions.

A 2” (0 0) silicon substrate was selected. Channels and reservoirs were etched on it. Silicon etching was performed using KOH (potassium hydroxide) solution and the etching process was monitored to get the exact channel depth. Fig. 1B shows an SEM photograph of the etched channels, the photograph has a 50× magnification. At the end of the etching process a silicon dioxide layer was thermally grown on the channels surface. The silicon dioxide layer thickness was about 1 μm. This layer helped in creating a hydrophilic channel surfaces allowing EOF to occur. A similar heat spreader design by Eng et al. [7] used a silicon dioxide layer with a thickness of 250 nm.

Instead of conventional fluid pumping techniques, EOF principle was implemented for which a power supply was required. A PDMS cover was introduced as a thermal insulator and to seal the top of the channels. A foil type DC powered heater was selected and attached to the bottom surface of the substrate. To vary the heat flux, a variable voltage DC power supply was used to power the heater. Voltages, currents, and temperatures were recorded using a DAQ system. A photograph of the actively cooled micro-channel heat sink device is shown in Fig. 1C. The wires, shown in Fig. 1C, were connected to a DC heater and to a thermocouple that was attached to the bottom surface of the silicon substrate. A T-type surface thermocouple [SA1XL-T (Omega Engineering, Inc.)] was used to measure the substrate temperature at the middle of the microchannel heat exchanger. The temperature of the cooling fluid at the inlet and outlet reservoir was measured using T-type thermocouples, HYP1-30-1/2-T-G-60-SMWPW-M (Omega Engineering, Inc.).

The EOF was applied to the cooling fluid (buffer) using two platinum (99.95%) electrodes; the electrodes were wires with circular cross section of 0.5 mm diameter and 12.5 cm length. The electrodes were immersed at the center of each reservoir without touching the top surface of the silicone substrate. The dimensions and configuration of the inlet and outlet cooling fluid reservoirs are shown in Fig. 2A.

Referring to Fig. 2A, the heights of the inlet and outlet reservoirs are 8 mm each. During the experiments the maximum difference between cooling fluid levels in both the inlet and outlet reservoirs was 3 mm or less. This elevation difference will produce a pressure of 0.029 kPa. Electro-osmotic flow is associated with significantly higher equivalent pressures. Laser et al. [4] reported a 10 kPa of pressure generated by a similar micro-heat exchanger. Also Chen and Santiago [16] designed and tested a planar electroosmotic pump which had similar difference in fluid levels between the input and the output of the pump. Thus, it can be noted that the hydrostatic pressure generated by the level difference in the fluid reservoirs is negligible.

For conducting the experiment, a special mounting system was designed and fabricated using Plexiglas material. This mounting system helped in localizing the major components and kept the micro-channel heat sink device in place. Fig. 2B shows the block
diagram of the experimental flow loop. The setup included the EOF voltage power supply, a DC power supply, a DAQ system, thermocouples, and the actively cooled micro-channel heat sink device. The channel walls were treated with 1M NaOH (sodium hydroxide) solution in order to increase the surface charge. Then, the micro-channels within the heat sink device were filled with the cooling fluid (0.4 mM borax buffer, Ricca Chemical Company). Properties of the cooling buffer are shown in Table 1. The 0.4 mM borax buffer (cooling fluid) was used to conduct all the heat transfer experiments. Borax buffer was used by Laser et al. [4] as a cooling fluid in a similar microchannel heat sink. For the current design, the flow rate produced by borax buffer was higher when compared to the flow rate produced by phosphate buffer or distilled water.

2.2. Numerical methods

A 3D model of a single channel was constructed using CFD-GEOM (ESI-CFD Inc., Huntsville, AL). Structured meshing was used with 180,000 cells. A similar 3D numerical model was investigated by Husain and Kim [17] where a microchannel heat sink was studied for the optimum dimensions. The constructed channel had dimensions of 300 μm width, 3 cm length, and 100 μm depth which were equal to the physical channel dimensions. The boundary conditions used were zero pressures at the channel inlet and outlet, zero voltage at the outlet and EOF voltage value at the inlet ranging from 100 to 400 V. No slip condition was used for channel walls and the experimentally obtained zeta potential was used as a wall property. Constant wall heat flux was used as a wall boundary condition. Governing equations were solved using a multi-physics finite volume solver. The algebraic multi-grid (AMG) solver was used to solve for the steady state coupled heat transfer, flow, and electric field through the channel. The solution was tested for mesh independency by doubling the mesh size. Velocities and temperatures matched within 1% for both mesh sizes. The convergence criterion of the solutions was $10^{-6}$.

2.3. Governing equations

Electro-osmotic flow through the channel is governed by the continuity equation (1) and Navier–Stokes equation (2) [18]

$$\nabla \cdot \vec{u} = 0 \quad (1)$$

$$\rho(\vec{u} \cdot \nabla \vec{u}) = \mu \nabla^2 \vec{u} - \nabla p + \vec{f}_e \quad (2)$$

where $\vec{u}$ is the cooling fluid (buffer) velocity (m/s), $\rho$ is the cooling fluid (buffer) density (kg/m$^3$), $\mu$ is the dynamic viscosity of the cooling fluid (buffer) (Pa s), $p$ is the pressure inside the channel (Pa), and $\vec{f}_e$ is electro-osmotic force induced by the applied electric field ($\vec{f}_e = \rho_e \cdot \vec{E}$, where $\rho_e$ is the bulk charge density, C m$^{-3}$ and $\vec{E}$ is the electric field, V m$^{-1}$). Eq. (2) was written in terms of the applied electric potential, $\Phi$, (V) and the wall surface charge, $\psi$, (C m$^{-2}$) by Comandur et al. [19] of our lab as shown in Eq. (3)

$$\rho(\vec{u} \cdot \nabla \vec{u}) = \mu \nabla^2 \vec{u} - \nabla p + \epsilon(\nabla^2 \psi + \nabla \Phi) \quad (3)$$

where $\epsilon$ is the cooling fluid (buffer) permittivity (C/V m).

Steady state heat equation (4) was solved with a constant wall heat flux boundary condition at a constant inlet temperature of the cooling fluid. The solution was obtained for steady state thermal distribution through the channel

$$0 = \kappa \nabla^2 T - \vec{u} \cdot \nabla(T) \quad (4)$$

where $\kappa$ is the cooling fluid (buffer) thermal diffusivity (m$^2$/s), $T$ is the cooling fluid (buffer) temperature.
3. Results

The results are presented in three major subsections. First, using experimental data, calculation of zeta potential is presented for subsequent assessment of the EOF characteristics using numerical technique. Second, the cooling fluid (buffer) flow rate and velocity are presented and discussed. Third, the outlet cooling fluid temperature with change in power is discussed. While presenting the cooling fluid temperature data, the influence of voltage difference and associated joule heating on EOF and heat transfer is discussed. Subsequently, Nusselt number was calculated and plotted versus the non-dimensional channel length.

3.1. Calculation of zeta potential ($\zeta$) and bulk conductivity ($\lambda$)

An important parameter to be determined for the EOF analysis is the zeta potential ($\zeta$), which is the electric potential difference between the charged channel surface and the electrolyte solution. The zeta potential ($\zeta$) was measured experimentally using the setup shown in Fig. 3A [2].

Borax buffers with different concentrations (0.4 mM borax buffer and 4 mM borax buffer) were added to the two reservoirs shown in Fig. 3A. The left reservoir represents the fluid with higher concentration while the right represents the buffer having lower concentration. By applying a voltage between the two reservoirs flow was initiated in the direction shown. This setup is similar to our recently published results [19]. When the higher concentration buffer continuously replaced the lower concentration buffer, the conductivity of the medium increased, leading to an increase in EOF current. Using Eq. (5) by Sze et al. [11] which was based on Helmholtz–Smoluchowski equation [2], zeta potential can be evaluated

$$\zeta = \frac{\mu \cdot \text{Slope} \cdot L}{\varepsilon_r \varepsilon_0 F c A_{\text{cross}} (\lambda_{b2} - \lambda_{b1})}$$

where $\mu$ is the water viscosity (N s m$^{-2}$), Slope is the slope of the EOF current time curve that needs to be measured, $L$ is the channel length (m), $\varepsilon_r$ is the relative permittivity of water, $\varepsilon_0$ is the vacuum permittivity (C/V m), $E_c$ is the electric field across the channel (V/m), $\lambda_{b2}$ is the bulk conductivity (S/m) of the higher concentration solution, and $\lambda_{b1}$ is the bulk conductivity (S/m) of the lower concentration solution. These conductivities need to be measured. Table 1 shows a summary of the parameters used in the numerical calculations.

The bulk conductivity of the cooling fluid was measured using the simple experiment shown in Fig. 3B. A tube with a known internal diameter and a specific length was filled with the higher and lower concentration buffers separately. Using two electrodes connected to a high precision electrical resistance meter, the resistance of the fluid filling the tube section was measured. Knowing the tube cross-sectional area and length, the cooling fluid resistivity was calculated. The cooling fluid bulk conductivity was calculated as the reciprocal of its resistivity. The bulk conductivity of the lower concentration borax buffer (0.4 mM) was $7.66 \times 10^{-3}$ S m$^{-1}$ and the bulk conductivity for the higher concentration borax buffer (4 mM) was $29.5 \times 10^{-3}$ S m$^{-1}$.

Fig. 4 shows the EOF current variations with time and the Slope that was used in Eq. (5). Using the above conductivity values and the Slope, a $-95$ mV value was obtained for zeta potential for the present design of the micro-channel heat sink device. Laser et al. [4], used silicon dioxide as coating and borax buffer as a cooling fluid. They obtained a $-25$ mV of zeta potential. Also Sze et al. [11] reported a range of $-68$ mV to $-110$ mV of zeta potential for PDMS surfaces. Obtaining the value for zeta potential was critical for performing the numerical calculations for heat transfer and flow through the micro-scale heat exchanger channels.

3.2. Cooling fluid (buffer) flow rate in microchannels

The cooling fluid flow rate through the micro-scale heat exchanger was 12.3 $\mu$L/min for each 2 V/mm of applied electric field. The maximum flow rate achieved was 82 $\mu$L/min at 400 V of applied EOF voltage. Reynolds number (Re) was 0.1, 0.2, 0.3, and 0.4 for 100 V, 200 V, 300 V, and 400 V, respectively. Flow rate and flow velocity values for different applied electric field are presented in Table 2. It can be noticed from Table 2 that the flow rate is proportional to the applied EOF voltage. As the electric field increases, the EOF velocity increases and thus, flow rate increases through the microchannels.

Referring to Table 2 the magnitude of the flow velocities ranged from $5.65 \times 10^{-4}$ m/s at 100 V to $22.59 \times 10^{-4}$ m/s at 400 V of applied EOF voltage. The increase in velocity was linearly proportional to the magnitude of the applied EOF voltage. Husain and Kim [20] showed a linear relationship between the cooling fluid flow rate and the electric potential for a similar microchannel cooling system. This linearity can be explained using Helmholtz–Smoluchowski equation which states that for a solution with constant permittivity and viscosity the velocity is linearly proportional to the applied electric field.
3.3. Heat transfer in microchannels

The difference between the experimentally measured outlet and inlet reservoir temperatures are shown in Fig. 5. These figures represent the data measured by the DAQ system with time. The temperatures were measured inside the inlet and outlet cooling fluid reservoirs. However, it is more important to evaluate the temperature at the channel inlet and outlet. Using the measured temperature of the cooling fluid in the reservoir, an energy balance was conducted to obtain the temperature of the cooling fluid at the channel outlet by accounting for the change in the cooling fluid enthalpy, internal energy, and its flow rate. The increase in the cooling fluid temperature, shown in Fig. 5, for the period 0 to \(~180\) s was due to joule heating. After \(~180\) s the heater was switched on. This way we were able to delineate the joule heating effect from the heat removed by the microchannels from the heater.

The outlet reservoir and channels connected to it are shown in Fig. 6. The dotted rectangle shows the control volume under consideration. Eq. (6) represents the thermodynamic energy balance for the control volume

$$h_i = \frac{u_2 m_2 - u_1 m_1 + q}{m_2 - m_1}$$  \hspace{1cm} (6)

where $h_i$ is the cooling fluid enthalpy (J/kg) at the channel outlet; $u_1$ and $u_2$ are the initial and final internal energies (J/kg) of the cooling fluid in the reservoir; $m_1$ and $m_2$ are the initial and final cooling fluid masses (kg) in the outlet reservoir; and $q$ is the energy loss from the reservoir to the surroundings (J). Using $h_i$ value at the end of the channel the experimental cooling fluid temperature was evaluated.

The difference between the experimental channel outlet and inlet fluid temperatures at different heating power values for 400 V of applied EOF voltage are shown in Fig. 7A. The 400 V of EOF voltage achieved the highest cooling fluid flow rate and hence had the best heat removal performance. Heating power values were applied as follows: 0.4 W (heating power #1), 1.2 W (heating power #2), 2.1 W (heating power #3), and 4.0 W (heating power #4). The cooling fluid (buffer) temperature at the inlet reservoir was maintained at 23 °C. For heating power #1 the measured outlet temperature difference ($T_{\text{channel outlet}} - T_{\text{channel inlet}}$) was 9.6 °C,
and the numerically calculated temperature difference at the outlet was 10.0 °C. For heating power #2 the measured outlet temperature difference was 29.9 °C, and the numerically calculated temperature difference at the outlet was 32.1 °C. For heating power #3 the measured outlet temperature difference was 54.3 °C, and the numerically calculated temperature difference at the outlet was 58.6 °C. For heating power #4 the measured outlet temperature difference was 80.1 °C, and the numerically calculated temperature difference at the outlet was 89.0 °C.

The differences between the experimental and numerical cooling fluid outlet temperatures were 1.2%, 4.1%, 5.6%, and 8.6% for heating powers #1, #2, #3, and #4, respectively. It may be noted that higher heating power values were not applied to avoid boiling of the cooling fluid in the outlet section of the microchannels. The increase in the difference between the experimental and numerical temperatures at the outlet with the increase in heat flux was possibly due to the higher heat losses from the micro-scale heat exchanger to the surroundings. At higher heat flux values the cooling fluid temperature increased, which in turn, elevated the temperature gradient leading to a higher heat transfer to the surroundings.

Fig. 7B shows the substrate and the cooling fluid temperatures for different heating powers. It may be noted that the substrate temperatures were measured at the middle of the channels (~1.5 cm from the channel entrance). For comparison the cooling fluid temperatures at the middle of the channels were also plotted on the same figure. The substrate temperature at the middle was 29.0 °C, 39.8 °C, 57.0 °C, and 80.5 °C for 0.4 W, 1.2 W, 2.1 W, and 4.0 W, respectively. Similarly, the cooling fluid temperature at the middle of the channels was 27.8 °C, 38.9 °C, 50.2 °C, and 63.1 °C for 0.4 W, 1.2 W, 2.1 W, and 4.0 W, respectively.

To quantify the joule heating due to the applied EOF voltage, temperature rise due to joule heating was delineated from the temperature rise due to heat flux. The EOF voltage was turned-on while keeping the heater off. Fig. 8 shows the increase in the cooling fluid temperature for EOF voltages ranging from 100 V to 400 V. The increments in the cooling fluid temperature for 100 V, 200 V, 300 V, and 400 V of applied EOF voltage were 1.2 °C, 1.8 °C, 2.6 °C, and 4.5 °C, respectively. It is evident that there is an increase in the cooling fluid temperature from inlet to outlet reservoirs during the application of the EOF voltage without the application of the heating power. This increase in temperature was proportional to the increase in EOF voltage. As the EOF voltage increases the electric current passing through the cooling fluid increases, leading to a higher joule heating and an increase in the temperature of the cooling fluid.

As discussed earlier, numerical temperature distributions for the microchannel were numerically calculated using a finite volume (FV) solver (ESI-CFD Inc., Huntsville, AL). A sample of the numerical temperature contours near the outlet of the channel for heating power #1, #2, #3, and #4 is shown in Fig. 9. Temperature distributions for 0.4 W, 1.2 W, 2.1 W, and 4.0 W are presented by Fig. 9A, B, C and D, respectively.

Nusselt number (Nu) variations and values for the microchannel are presented in Fig. 10A and Table 3, respectively. The bulk temperature was evaluated at various sections through the
microchannel. Using the difference in the channel’s wall temperature and the fluid bulk temperature and knowing the wall heat flux the heat transfer coefficient \( h \) was evaluated. Nusselt number was calculated using the \( h \) value, the channel hydraulic diameter, and the fluid thermal conductivity. Nusselt number was 5.39 at the channel entrance and deceased to a constant value of 4.56 for the rest of the channel as shown in Fig. 10A.

For cross-checking the Nu values, an alternate finite volume (FV-Alternate) solver (ANSYS-Fluent, ANSYS, Inc., Canonsburg, PA) was used to calculate the numerical temperature distribution through the microchannel. To mimic the EOF, a microchannel with slip walls was constructed and uniform velocity was used as the inlet boundary condition instead of electric potential. Using a similar approach, Nu number was evaluated for the microchannel at the same flow and heat transfer conditions. Nusselt number was 5.48 at the channel entrance and deceased to a constant value of 4.56 for the rest of the channel as shown in Fig. 10A. The Nu results of the FV and FV-Alternate solvers matched within 1%.

To compare Nu numbers for EOF and the conventional pressure driven flow (parabolic velocity profile), numerical calculations of the temperature distribution were performed for the pressure driven case. Both FV and FV-Alternate were used for the numerical calculations. For the FV solver, Nu number was 5.19 at the channel entrance and deceased to a constant value of 4.35 for the rest of the channel. For the FV-Alternate solver, Nu was 5.21 at the channel entrance and deceased to a constant value of 4.35 for the rest of the channel as shown in Fig. 10A. These Nu numbers matched within 1%. It can be noticed that Nu number for the EOF flow was 5% higher than the case of the pressure driven flow.

Using the analytical method described in Chapter 4 of Burmeister [21], analytical Nu numbers for EOF and pressure driven flow were evaluated and summarized along with the numerical Nu numbers in Table 3.

<table>
<thead>
<tr>
<th></th>
<th>EOF flow</th>
<th>Pressure driven flow</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD-finite volume (FV) (^a)</td>
<td>4.56</td>
<td>4.35</td>
<td>5.0</td>
</tr>
<tr>
<td>CFD-finite volume (FV-Alternate) (^b)</td>
<td>4.56</td>
<td>4.35</td>
<td>5.0</td>
</tr>
<tr>
<td>Analytical (^c)</td>
<td>4.50</td>
<td>4.07</td>
<td>10.6</td>
</tr>
<tr>
<td>Shah et al. [22]</td>
<td>–</td>
<td>3.95</td>
<td>–</td>
</tr>
</tbody>
</table>

\(^a\) CFD-ACE (ESI-CFD Inc., Huntsville, AL).
\(^b\) ANSYS-Fluent (ANSYS, Inc., Canonsburg, PA).
\(^c\) Evaluated by authors.
\(^d\) Linearly interpolated between 2.5 and 3.33 channel aspect ratio using Table 44 from Shah et al. [22] (aspect ratio of the channel under investigation = 3).
near the walls. For the pressure driven flow, velocity profile de-
forms to a parabolic profile, where the velocity is zero at the wall as the flow develops. The higher velocity and its gradient at the wall in the case of EOF increase the convective heat transfer coeffi-
cient, which leads to a higher Nu number. The increase in Nu number leads to an increase in heat transfer to the cooling fluid for the EOF case.

The difference between the numerically and analytically calcu-
lated Nu numbers for the pressure driven flow was within 6.8%. Shah et al. [22] reported Nu number of 3.95 for a pressure driven flow using similar geometry, heat transfer, and flow conditions. The analytical value of Nu number for the pressure driven flow matches the Nu number reported by Shah et al. [22] within 3%.

Using experimental data for heating powers #1, #2, #3, and #4, an analytical study was conducted to assess the influence of micro-
channel geometry and other parameters for determining the max-
imum heat flux that can be applied without causing boiling and evaporation of the cooling fluid. Bubble formation due to high tem-
perature needs to be avoided as these could interrupt the EOF. It was found that the channel length and the applied EOF voltage are the critical factors for determining the maximum operational heat flux. Other factors such as zeta-potential had somewhat less effect on heat flux. Fig. 10B shows the variation of the channel length, the applied EOF voltage, and the maximum heat flux that can be applied. Referring to Fig. 10B, it can be noted that reducing the channel length and increasing the applied EOF voltage allows application of higher values of heat flux. Shorter channels and higher EOF voltages reduce the travel time of the cooling fluid through the channels. Shorter travel time prevents the cooling fluid from evaporation and boiling. For example heat flux of 5 × 10⁴ W/ m² can be applied using a 1 cm long channel and EOF voltages in the range of 400–600 V. To reduce the effects of joule heating it is recommended to use lower values of EOF voltage. Such shorter channels allow the micro-scale heat exchanger to be used for managing localized hot-spots.

4. Conclusions

A micro-scale heat exchanger was designed and tested for dif-
ferent powers, and EOF voltages. Temperature differences of the cooling fluid between the microchannel outlet and inlet, as well as the substrate temperature were measured. The variation of Nusselt number was plotted along the non-dimensional channel length. Nusselt number for the EOF flow was 4.56 whereas it was 4.35 for the pressure driven flow. Due to the plug flow profile associated with the EOF Nusselt number of the EOF was about 10% higher than the pressure driven flow (parabolic profile) flow.

It is evident that the joule heating increased with the increase in applied EOF voltage. This was observed when the EOF voltage was applied without the application of heat flux. This helped in deter-
mining the influence of joule heating with the increase in EOF volt-
age. For higher EOF voltages (e.g., 400 V), the cooling fluid temperature got elevated appreciably due to joule heating which needs to be minimized. For a fixed heating power, the applied EOF voltage increases the electro-osmotic velocity leading to some increase in heat transfer while elevating the joule heating. Choosing an optimum range of EOF voltage helps reducing joule heating while increasing the overall heat transfer.

Appendix A

A.1. Error analysis

Cooling fluid temperatures were measured using a T-type ther-
mcouple HYP1-30-1/2-T-G-60-SMWPW-M (Omega Engineering, Inc.). Cooling fluid (buffer) temperatures were measured with an accuracy of ± 1.1 °C. Substrate temperature was measured using T-type surface thermocouple SA1XL-T (Omega Engineering, Inc.), with an accuracy of ±1.1 °C. DC voltages and currents were measured using a digital multimeter HHM16 (Omega Engineering Inc.). Heater and EOF DC voltages were measured with an accuracy of ±0.25% V. Heater DC current was measured with an accuracy of ±2.0% A. Cooling fluid (buffer) temperature, substrate temperature, and EOF current were recorded using a DAQ OMB-DAQ55 (Omega Engineering Inc.). The accuracy of the DAQ system is ±0.015% of the reading.

Heater power was calculated by multiplying the heater voltage by the current according to Ohm’s law (P = VI). The uncertainty of measuring heater power can be evaluated using Moffat [23], where the total uncertainty U is given by:

\[ U = \left( B_0^2 + P_0^2 \right)^{1/2} \]  

where \( B_0 \) is the bias error and \( P_0 \) is the precision error. The bias er-
or is zero for the experiments conducted. The only source of error is the precision error. The propagation of uncertainty for a mea-
sured quantity \( r \) can be determined using Kline and McClintock equation [24]:

\[ \left( \delta r \right)^2 = \sum_{i=1}^{n} \left( \frac{\partial r}{\partial a_i} \delta a_i \right)^2 \]  

where \( a \) is a variable(s) used in measuring \( r \).

For heater power \( P = VI \), the precision error \( P_0 \) would be:

\[ \left( \frac{P_0}{P} \right)^2 \left( \frac{P}{V} \right)^2 \quad \text{and} \quad \left( \frac{P_0}{I} \right)^2 \left( \frac{I}{P} \right)^2 \]  

For \( V = 10 \) V and \( I = 0.23 \) A, \( P_0 = \pm2.5 \% \), \( P_0 = \pm0.46 \% \), the precision error in measuring power is:

\[ \left( \frac{P_0}{P} \right)^2 \left( \frac{10}{0.025} \right)^2 + \left( \frac{0.0046}{0.23} \right)^2 = 0.0202 \]  

The precision error in measuring the heater power is ±2.02% of the measured power value. Following the same procedure, the precision error in measuring zeta potential was ±0.25% of the measured zeta potential value.

References